

Airpak 3.0 User's Guide

May 2, 2007

Copyright © 2007 by Fluent Inc.
All Rights Reserved. No part of this document may be reproduced or otherwise used in
any form without express written permission from Fluent Inc.

Airpak, FIDAP, FLUENT, FLUENT for CATIA V5, FloWizard, GAMBIT, Icemax, Icepak,
Icepro, Icewave, Icechip, MixSim, and POLYFLOW are registered trademarks of Fluent
Inc. All other products or name brands are trademarks of their respective holders.

CATIA V5 is a registered trademark of Dassault Systèmes. CHEMKIN is a registered
trademark of Reaction Design Inc.

Portions of this program include material copyrighted by PathScale Corporation
2003-2004.

Fluent Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766

Using This Manual

What's In This Manual

The **Airpak User's Guide** tells you what you need to know to use **Airpak**. The first chapter provides introductory information about **Airpak**, as well as a sample session, and the second chapter contains information about the user interface. The next 30 chapters explain how to use **Airpak**. Each chapter focuses on a specific topic or problem setup step and, as far as possible, presents the relevant information in a procedural manner. The last chapter provides information about the theory behind **Airpak's** physical models and numerical procedures.

The index allows you to look up material relating to a particular subject or a specific **Airpak** menu item, button, panel, or option. The idea is to help you find answers to your questions quickly and directly, whether you are a first-time user or an experienced user.

A brief description of what's in each chapter follows:

- Chapter 1, **Getting Started**, describes the capabilities of **Airpak**, gives an overview of the problem setup steps, and presents a sample session that you can work through at your own pace.
- Chapter 2, **User Interface**, describes the mechanics of using the user interface.
- Chapter 3, **Reading, Writing, and Managing Files**, contains information about the files that **Airpak** can read and write, including hardcopy files.
- Chapter 4, **Importing and Exporting Model Files**, provides information on importing IGES files, IFC files, and other files created by commercial CAD packages into **Airpak**, and exporting **Airpak** files in various formats.
- Chapter 5, **Unit Systems**, describes how to use the standard and custom unit systems available in **Airpak**.
- Chapter 6, **Defining a Project**, describes how to define a project for your **Airpak** model.
- Chapter 7, **Building a Model**, contains information about how to set up your model in **Airpak**.
- Chapter 8, **Blocks**, contains information about block objects and how to add them to your **Airpak** model.

- Chapter 9, Fans, contains information about fan objects and how to add them to your **Airpak** model.
- Chapter 10, Vents, contains information about vent objects and how to add them to your **Airpak** model.
- Chapter 11, Openings, contains information about opening objects and how to add them to your **Airpak** model.
- Chapter 12, Person Objects, contains information about person objects and how to add them to your **Airpak** model.
- Chapter 13, Walls, contains information about wall objects and how to add them to your **Airpak** model.
- Chapter 14, Partitions, contains information about partition objects and how to add them to your **Airpak** model.
- Chapter 15, Sources, contains information about source objects and how to add them to your **Airpak** model.
- Chapter 16, Resistances, contains information about volumetric resistance objects and how to add them to your **Airpak** model.
- Chapter 17, Heat Exchangers, contains information about heat exchanger objects and how to add them to your **Airpak** model.
- Chapter 18, Hoods, contains information about hood objects and how to add them to your **Airpak** model.
- Chapter 19, Wires, contains information about wire objects and how to add them to your **Airpak** model.
- Chapter 20, Transient Simulations, contains information about solving problems involving time-dependent phenomena, including transient heat conduction and convection.
- Chapter 21, Species Transport Modeling, explains how to include the mixing and transport of species in your **Airpak** simulation.
- Chapter 22, Radiation Modeling, explains how to include radiative heat transfer in your **Airpak** simulation and model solar loading effects.
- Chapter 23, Optimization, contains information about solving design-optimization problems using **Airpak**.
- Chapter 24, Parameterizing the Model, describes how to use parameterization to determine the effect of various object sizes or other characteristics on the solution.

- Chapter 25, Using Macros, describes the predefined combinations of **Airpak** objects designed to fulfill specific functions in the model, and how you can use them.
- Chapter 26, Generating a Mesh, explains how to create a computational mesh for your **Airpak** model.
- Chapter 27, Calculating a Solution, describes how to compute a solution for your **Airpak** model.
- Chapter 28, Examining the Results, explains how to use the graphics tools in **Airpak** to examine your solution.
- Chapter 29, Generating Reports, describes how to obtain reports of flow rates, heat flux, and other solution data.
- Chapter 30, Variables for Postprocessing and Reporting, defines the flow variables that appear in the variable selection drop-down lists in the reporting and postprocessing panels.
- Chapter 31, Theory, describes the theory behind the physical models and numerical procedures in **Airpak**.

How To Use This Manual

Depending on your familiarity with computational fluid dynamics and **Airpak**, you can use this manual in a variety of ways.

For the Beginner

The suggested readings for the beginner are as follows:

- For an overview of **Airpak** modeling features, information on how to start up **Airpak**, or advice on how to plan your electronics cooling simulation, see Chapter 1. In this chapter you will also find a self-paced tutorial that illustrates how to solve a simple problem using **Airpak**.

You should be sure to try (or at least read through) this sample problem before working on any of the tutorials in the **Airpak** Tutorial Guide.

- To learn about the user interface, read Chapter 2.
- For information about the different files that **Airpak** reads and writes, see Chapter 3.
- To learn about importing IGES files and IFC files into **Airpak**, see Chapter 4.
- If you plan to use a unit system other than SI (British units, for example), see Chapter 5 for instructions.


- To learn how to define a project for your model, see Chapter 6.
- For information about defining your **Airpak** model, see Chapter 7.
- For information about available objects and how to add them to your **Airpak** model, see Chapters 8–19.
- For information about modeling the effect of time-dependent phenomena, see Chapter 20.
- For information about modeling species transport, see Chapter 21.
- To learn about including radiative heat transfer effects in your simulation, see Chapter 22.
- To learn how to solve constrained-design-optimization problems, see Chapter 23.
- For information about parameterizing your model, see Chapter 24.
- To learn how to use macros to define common combinations of **Airpak** objects, see Chapter 25.
- To learn how to generate a computational mesh for your model, see Chapter 26.
- To learn how to calculate a solution for your model or to modify parameters that control this calculation, see Chapter 27.
- To find out how to examine the results of your calculation using graphics and reporting tools, see Chapters 28 and 29.

For the Experienced User

If you are an experienced user who needs to look up specific information, there are two tools that allow you to use the **Airpak** User's Guide as a reference manual. The table of contents, as far as possible, lists topics that are discussed in a procedural order, enabling you to find material relating to a particular procedural step. There is also an index that allows you to access information about a specific subject, panel, button, menu item, or option.

Typographical Conventions Used In This Manual

Several typographical conventions are used in this manual's text to facilitate your learning process.



- An icon () at the beginning of a line marks an important note.
- Different type styles are used to indicate graphical user interface menu items and text inputs that you enter (e.g., **Open project** panel, enter the name `projectname`).
- A mini flow chart is used to indicate the menu selections that lead you to a specific panel. For example,

Model→**Generate mesh**


indicates that the **Generate mesh** option can be selected from the **Model** menu at the top of the **Airpak** main window.

The arrow points from a specific menu toward the item you should select from that menu. In this manual, mini flow charts usually precede a description of a panel or a screen illustration showing how to use the panel. They allow you to look up information about a panel and quickly determine how to access it without having to search the preceding material.

- A mini flow chart is also used to indicate the list tree selections that lead you to a specific panel or operation. For example,

 **Problem setup**→  **Basic parameters**

indicates that the **Basic parameters** item can be selected from the **Problem setup** node in the **Model manager** window

- Pictures of toolbar buttons are also used to indicate the button that will lead you to a specific panel. For example,  indicates that you will need to click on this button (in this case, to open the **Walls** panel) in the toolbar.

Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., \vec{a} , \vec{A}). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, \mathbf{I}).
- The operator ∇ , referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$$

∇ appears in several ways:

- The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p = \frac{\partial p}{\partial x}\vec{i} + \frac{\partial p}{\partial y}\vec{j} + \frac{\partial p}{\partial z}\vec{k}$$

- The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v}) = \left(\frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k} \right) (v_x\vec{i} + v_y\vec{j} + v_z\vec{k})$$

This tensor is usually written as

$$\begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix}$$

- The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}$$

- The operator $\nabla \cdot \nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}$$

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in Section 31.2.2, where convention dictates the use of Cartesian tensor notation. In this section, you will also find that some velocity vector components are written as u , v , and w instead of the conventional v with directional subscripts.

Mouse and Keyboard Conventions Used In This Manual

The default mouse buttons used to manipulate your model in the graphics window are described in Section 2.2.4. Note that you can change the default mouse controls in **Airpak** to suit your preferences (see Section 2.2.4). In this manual, however, descriptions of operations that use the mouse assume that you are using the default settings for the mouse controls. If you change the default mouse controls, you will need to use the mouse buttons you have specified, instead of the mouse buttons that the manual tells you to use.

The default keyboard key that is used in conjunction with the mouse buttons to move legends, titles, etc. in the graphics window is the <Ctrl> key. Note that you can change this key in **Airpak** to suit your preference (see Section 6.3). In this manual, however, descriptions of moving legends, titles, etc. assume that you are using the default setting (i.e., the <Ctrl> key). If you change the default setting, you will need to use the key you have specified, instead of the <Ctrl> key, when you move legends, titles, etc. in the graphics window.

When To Call Your Airpak Support Engineer

The Airpak support engineers can help you to plan your modeling projects and to overcome any difficulties you encounter while using Airpak. If you encounter difficulties we invite you to call your support engineer for assistance. However, there are a few things that we encourage you to do before calling:

- Read the section(s) of the manual containing information on the options you are trying to use.
- Recall the exact steps you were following that led up to and caused the problem.
- Write down the exact error message that appeared, if any.
- For particularly difficult problems, package up the project in which the problem occurred (see Section 3.6 for instructions) and send it to your support engineer. This is the best source that we can use to reproduce the problem and thereby help to identify the cause.

Contents

1	Getting Started	1-1
1.1	What is Airpak?	1-1
1.2	Program Structure	1-2
1.3	Program Capabilities	1-4
1.3.1	General	1-4
1.3.2	Model Building	1-4
1.3.3	Meshing	1-5
1.3.4	Materials	1-6
1.3.5	Physical Models	1-6
1.3.6	Boundary Conditions	1-7
1.3.7	Solver	1-7
1.3.8	Visualization	1-7
1.3.9	Reporting	1-8
1.3.10	Applications	1-8
1.4	Overview of Using Airpak	1-9
1.4.1	Planning Your Airpak Analysis	1-9
1.4.2	Problem Solving Steps	1-10
1.5	Starting Airpak	1-10
1.5.1	Starting Airpak on a UNIX System	1-11
1.5.2	Starting Airpak on a Windows System	1-11
1.5.3	Startup Screen	1-12
1.5.4	Startup Options for UNIX Systems	1-14
1.5.5	Environment Variables on UNIX Systems	1-15
1.6	Accessing the Airpak Manuals	1-17

1.6.1	Viewing the Manuals	1-18
1.6.2	Printing the Manuals	1-23
1.7	Sample Session	1-25
1.7.1	Problem Description	1-25
1.7.2	Outline of Procedure	1-26
1.7.3	Setting Up the Project	1-26
1.7.4	Resizing the Room	1-27
1.7.5	Setting Up the Basic Parameters	1-27
1.7.6	Adding Objects to the Room	1-31
1.7.7	Generating a Summary	1-43
1.7.8	Creating a Mesh	1-44
1.7.9	Changing the Meshing Priority	1-44
1.7.10	Checking the Flow Regime	1-47
1.7.11	Saving the Model	1-49
1.7.12	Calculating a Solution	1-49
1.7.13	Examining the Results	1-53
1.7.14	Generating Reports	1-62
1.7.15	Exiting From Airpak	1-65
1.7.16	Summary	1-66
2	User Interface	2-1
2.1	The Graphical User Interface	2-1
2.1.1	The Main Window	2-2
2.1.2	The Airpak Menus	2-2
2.1.3	The Airpak Toolbars	2-24
2.1.4	The Model manager Window	2-31
2.1.5	Graphics Windows	2-33
2.1.6	The Message Window	2-37
2.1.7	The Edit Window	2-38

2.1.8	File Selection Dialog Boxes	2-39
2.1.9	Control Panels	2-42
2.1.10	Accessing On-line Help	2-48
2.2	Using the Mouse	2-49
2.2.1	Controlling Panel Inputs	2-49
2.2.2	Using the Mouse in the Model manager Window	2-50
2.2.3	Using the Context Menus in the Model manager Window	2-50
2.2.4	Manipulating Graphics With the Mouse	2-59
2.3	Using the Keyboard	2-62
2.4	Quitting Airpak	2-64
3	Reading, Writing, and Managing Files	3-1
3.1	Overview of Files Written and Read by Airpak	3-1
3.2	Files Created by Airpak	3-3
3.2.1	Problem Setup Files	3-3
3.2.2	Mesh Files	3-3
3.2.3	Solver Files	3-3
3.2.4	Optimization Files	3-4
3.2.5	Postprocessing Files	3-4
3.3	Merging Model Data	3-5
3.3.1	Geometric Transformations	3-6
3.4	Saving a Project File	3-9
3.4.1	Recent Projects	3-10
3.5	Saving Image Files	3-10
3.5.1	Choosing the Image File Format	3-14
3.5.2	Specifying the Print Region	3-15
3.6	Packing and Unpacking Model Files	3-16
3.7	Cleaning up the Project Data	3-16

4	Importing and Exporting Model Files	4-1
4.1	Files That Can Be Imported Into Airpak	4-1
4.2	Importing IGES, STEP, and Tetin Surface Geometry Files Into Airpak .	4-2
4.2.1	Overview of Procedure for IGES, STEP, and Tetin File Import .	4-2
4.2.2	Reading an IGES, STEP, or Tetin File Into Airpak	4-3
4.2.3	Using Families	4-6
4.2.4	Visibility of CAD Geometry in the Graphics Window	4-17
4.3	Importing Other Files Into Airpak	4-20
4.3.1	General Procedure	4-20
4.3.2	IGES and DXF Files With Point and Line Geometry	4-21
4.3.3	DWG and DXF Files with Surface Geometry	4-23
4.3.4	IFC Files	4-23
4.3.5	CSV/Excel Files	4-25
4.4	Exporting Airpak Files	4-28
4.4.1	IGES, STEP, and Tetin Files	4-28
4.4.2	CSV/Excel Files	4-28
5	Unit Systems	5-1
5.1	Overview of Units in Airpak	5-1
5.2	Units for Meshing	5-2
5.3	Built-In Unit Systems in Airpak	5-3
5.4	Customizing Units	5-4
5.4.1	Viewing Current Units	5-4
5.4.2	Changing the Units for a Quantity	5-4
5.4.3	Defining a New Unit	5-7
5.4.4	Deleting a Unit	5-8
5.5	Units for Postprocessing	5-8

6	Defining a Project	6-1
6.1	Overview of Interface Components	6-1
6.1.1	The File Menu	6-1
6.1.2	The File commands Toolbar	6-3
6.1.3	The Model manager Window	6-4
6.2	Creating, Opening, Reloading, and Deleting a Project File	6-7
6.2.1	Creating a New Project	6-7
6.2.2	Opening an Existing Project	6-8
6.2.3	Reloading the Main Version of a Project	6-9
6.2.4	Deleting a Project	6-10
6.3	Configuring a Project	6-11
6.3.1	Display Options	6-12
6.3.2	Editing Options	6-14
6.3.3	Printing Options	6-15
6.3.4	Miscellaneous Options	6-17
6.3.5	Editing the Library Paths	6-17
6.3.6	Editing the Graphical Styles	6-20
6.3.7	Interactive Editing	6-23
6.3.8	Meshing Options	6-24
6.3.9	Solution Options	6-25
6.3.10	Postprocessing Options	6-25
6.3.11	Other Preferences and Settings	6-26
6.4	Specifying the Problem Parameters	6-26
6.4.1	Time Variation	6-29
6.4.2	Solution Variables	6-29
6.4.3	Flow Regime	6-32
6.4.4	Forced- or Natural-Convection Effects	6-36
6.4.5	Compass Orientation of Your Model	6-39
6.4.6	Ambient Values	6-39

6.4.7	Default Fluid, Solid, and Surface Materials	6-40
6.4.8	Initial Conditions	6-41
7	Building a Model	7-1
7.1	Overview	7-1
7.1.1	The Object creation Toolbar	7-1
7.1.2	The Object modification Toolbar	7-2
7.1.3	The Model Node in the Model manager Window	7-2
7.1.4	The Model Menu	7-4
7.2	Defining the Room	7-4
7.2.1	Resizing the Room	7-5
7.2.2	Repositioning the Room	7-9
7.2.3	Changing the Walls of the Room	7-12
7.2.4	Changing the Name of the Room	7-12
7.2.5	Modifying the Graphical Style of the Room	7-13
7.3	Configuring Objects Within the Room	7-13
7.3.1	Overview of the Object Panels and Object Edit Windows	7-14
7.3.2	Creating a New Object	7-19
7.3.3	Selecting and Deselecting an Object	7-19
7.3.4	Editing an Object	7-19
7.3.5	Deleting an Object	7-20
7.3.6	Resizing an Object	7-20
7.3.7	Repositioning an Object	7-21
7.3.8	Aligning an Object With Another Object in the Model	7-31
7.3.9	Copying an Object	7-38
7.4	Object Attributes	7-42
7.4.1	Description	7-43
7.4.2	Graphical Style	7-43
7.4.3	Position and Size	7-44

7.4.4	Geometry	7-45
7.4.5	Physical Characteristics	7-63
7.5	Adding Objects to the Model	7-64
7.6	Grouping Objects	7-65
7.6.1	Creating a Group	7-67
7.6.2	Renaming a Group	7-67
7.6.3	Changing the Graphical Style of a Group	7-68
7.6.4	Adding Objects to a Group	7-69
7.6.5	Removing Objects From a Group	7-70
7.6.6	Copying Groups	7-71
7.6.7	Moving a Group	7-72
7.6.8	Editing the Properties of Like Objects in a Group	7-72
7.6.9	Deleting a Group	7-72
7.6.10	Activating or Deactivating a Group	7-73
7.6.11	Using a Group to Create an Assembly	7-73
7.6.12	Saving a Group as a Project	7-73
7.7	Material Properties	7-74
7.7.1	Using the Materials Library and the Materials Panel	7-75
7.7.2	Editing an Existing Material	7-75
7.7.3	Viewing the Properties of a Material	7-81
7.7.4	Copying a Material	7-81
7.7.5	Creating a New Material	7-82
7.7.6	Saving Materials and Properties	7-83
7.7.7	Deleting a Material	7-84
7.7.8	Defining Properties Using Velocity-Dependent Functions	7-84
7.7.9	Defining Properties Using Temperature-Dependent Functions	7-85
7.8	Custom Assemblies	7-90
7.8.1	Creating and Adding an Assembly	7-90
7.8.2	Editing Properties of an Assembly	7-91

7.8.3	Assembly Viewing Options	7-96
7.8.4	Selecting an Assembly	7-96
7.8.5	Editing Objects in an Assembly	7-96
7.8.6	Copying an Assembly	7-96
7.8.7	Moving an Assembly	7-97
7.8.8	Saving an Assembly	7-97
7.8.9	Loading an Assembly	7-97
7.8.10	Merging an Assembly With Another Project	7-97
7.8.11	Deleting an Assembly	7-98
7.8.12	Expanding an Assembly Into Its Components	7-98
7.8.13	Summary Information for an Assembly	7-98
7.8.14	Total Volume of an Assembly	7-99
7.8.15	Total Area of an Assembly	7-99
7.9	Checking the Design of Your Model	7-99
7.9.1	Object and Material Summaries	7-99
7.9.2	Design Checks	7-99
8	Blocks	8-1
8.1	Geometry, Location, and Dimensions	8-1
8.2	Block Type	8-2
8.3	Surface Roughness	8-2
8.4	Physical and Thermal Specifications	8-2
8.5	Block-Combination Thermal Characteristics	8-3
8.5.1	Blocks with Coincident Surfaces	8-3
8.5.2	Blocks with Intersecting Volumes	8-4
8.5.3	A Block and an Intersecting Partition	8-7
8.5.4	Blocks Positioned on an External Wall	8-7
8.5.5	Cylinder, Polygon, Ellipsoid, or Elliptical Cylinder Blocks Positioned on a Prism Block	8-8
8.6	Adding a Block to Your Airpak Model	8-9

8.6.1	User Inputs for the Block Surface Specification	8-10
8.6.2	User Inputs for the Block Thermal Specification	8-14
9	Fans	9-1
9.1	Defining a Fan in Airpak	9-2
9.2	Geometry, Location, and Dimensions	9-3
9.2.1	Simple Fans	9-3
9.3	Flow Direction	9-4
9.4	Fans in Series	9-6
9.5	Fans in Parallel	9-6
9.6	Fans on Blocks	9-7
9.7	Specifying Swirl	9-8
9.7.1	Swirl Magnitude	9-8
9.7.2	Fan RPM	9-8
9.8	Fixed Flow	9-8
9.9	Fan Characteristic Curve	9-9
9.10	Adding a Fan to Your Airpak Model	9-11
9.10.1	Using the Fan curve Window to Specify the Curve for a Characteristic Curve Fan Type	9-14
9.10.2	Using the Curve specification Panel to Specify the Curve for a Characteristic Curve Fan Type	9-15
10	Vents	10-1
10.1	Overview	10-1
10.2	Planar Resistances	10-2
10.3	Geometry, Location, and Dimensions	10-3
10.4	Pressure Drop Calculations for Vents	10-3
10.5	Adding a Vent to Your Airpak Model	10-6
10.5.1	Using the Resistance curve Window to Specify the Curve for a Vent	10-13
10.5.2	Using the Curve specification Panel to Specify the Curve for a Vent	10-15

11	Openings	11-1
11.1	Geometry, Location, and Dimensions	11-2
11.2	Free Openings	11-2
11.3	Recirculation Openings	11-3
11.3.1	Recirculation Mass Flow Rate	11-4
11.3.2	Flow Direction for Recirculation Openings	11-4
11.3.3	Recirculation Opening Thermal Specifications	11-4
11.3.4	Recirculation Opening Species Filters	11-5
11.4	Adding an Opening to Your Airpak Model	11-6
11.4.1	User Inputs for a Free Opening	11-8
11.4.2	User Inputs for a Recirculation Opening	11-12
12	Person Objects	12-1
12.1	Location and Dimensions	12-2
12.2	Thermal Options	12-3
12.3	Adding a Person to Your Airpak Model	12-3
13	Walls	13-1
13.1	Geometry, Location, and Dimensions	13-2
13.1.1	Wall Thickness	13-2
13.2	Surface Roughness	13-3
13.3	Wall Velocity	13-3
13.4	Thermal Boundary Conditions	13-4
13.4.1	Specified Heat Flux	13-4
13.4.2	Specified Temperature	13-5
13.5	External Thermal Conditions	13-6
13.5.1	Convective Heat Transfer	13-6
13.5.2	Radiative Heat Transfer	13-7
13.6	Constructing Multifaceted Walls	13-9
13.7	Adding a Wall to Your Airpak Model	13-10

13.7.1	User Inputs for a Symmetry Wall	13-13
13.7.2	User Inputs for a Stationary or Moving Wall	13-13
14	Partitions	14-1
14.1	Defining a Partition in Airpak	14-2
14.2	Geometry, Location, and Dimensions	14-2
14.2.1	Partition Thickness	14-2
14.3	Thermal Model Type	14-3
14.4	Surface Roughness	14-4
14.5	Using Partitions in Combination with Other Objects	14-4
14.6	Adding a Partition to Your Airpak Model	14-4
14.6.1	User Inputs for the Thermal Model	14-7
14.6.2	User Inputs for the Low- and High-Side Properties of the Partition	14-11
15	Sources	15-1
15.1	Geometry, Location, and Dimensions	15-1
15.2	Thermal Options	15-2
15.3	Source Usage	15-2
15.4	Adding a Source to Your Airpak Model	15-3
15.4.1	User Inputs for Heat Source Parameters	15-5
16	Resistances	16-1
16.1	Geometry, Location, and Dimensions	16-2
16.2	Pressure Drop Calculation for a 3D Resistance	16-2
16.3	Adding a Resistance to Your Airpak Model	16-4
17	Heat Exchangers	17-1
17.1	Geometry, Location, and Dimensions	17-1
17.2	Modeling a Planar Heat Exchanger in Airpak	17-1
17.2.1	Modeling the Pressure Loss Through a Heat Exchanger	17-1
17.2.2	Modeling the Heat Transfer Through a Heat Exchanger	17-2

17.2.3	Calculating the Heat Transfer Coefficient	17-2
17.3	Adding a Heat Exchanger to Your Airpak Model	17-3
18	Hoods	18-1
18.1	Location and Dimensions	18-1
18.2	Flow Rate	18-3
18.3	Adding a Hood to Your Airpak Model	18-3
19	Wires	19-1
19.1	Adding a Wire to Your Airpak Model	19-1
20	Transient Simulations	20-1
20.1	User Inputs for Transient Simulations	20-1
20.2	Specifying Variables as a Function of Time	20-11
20.2.1	Displaying the Variation of Transient Parameters with Time . . .	20-13
20.2.2	Using the Time/value curve Window to Specify a Piecewise Linear Variation With Time	20-15
20.2.3	Using the Curve specification Panel to Specify a Piecewise Linear Variation With Time	20-17
20.3	Postprocessing for Transient Simulations	20-18
20.3.1	Examining Results at a Specified Time	20-18
20.3.2	Creating Time-Averaged Results	20-19
20.3.3	Creating an Animation	20-21
20.3.4	Generating a Report	20-22
20.3.5	Creating a History Plot	20-22
21	Species Transport Modeling	21-1
21.1	Overview of Modeling Species Transport	21-1
21.2	User Inputs for Species Transport Simulations	21-3
21.2.1	Using the Curve specification Panel to Specify a Spatial Boundary Profile	21-11
21.3	Postprocessing for Species Calculations	21-12

22 Radiation Modeling	22-1
22.1 Choosing a Radiation Model	22-1
22.2 Using the Surface-to-Surface Radiation Model	22-2
22.2.1 Radiation Modeling for Objects	22-2
22.3 User Inputs for Radiation Modeling	22-4
22.3.1 User Inputs for Specification of Radiation in Individual Object Panels	22-5
22.3.2 User Inputs for Specification of Radiation Using the Form factors Panel	22-8
22.4 Using the Discrete Ordinates Radiation Model	22-14
22.5 Modeling Solar Radiation Effects	22-14
22.5.1 User Inputs for the Solar Load Model	22-15
23 Optimization	23-1
23.1 When to Use Optimization	23-1
23.2 User Inputs for Optimization	23-2
24 Parameterizing the Model	24-1
24.1 Overview of Parameterization	24-1
24.2 Defining a Parameter in an Input Field	24-2
24.3 Defining Check Box Parameters	24-5
24.4 Defining Radio Button Parameters (Option Parameters)	24-8
24.5 Defining a Parameter (Design Variable) Using the Parameters and optimization Panel	24-9
24.6 Deleting Parameters	24-10
24.7 Defining Trials	24-11
24.7.1 Selecting Trials	24-13
24.8 Running Trials	24-13
24.8.1 Running a Single Trial	24-15
24.8.2 Running Multiple Trials	24-16
24.9 Function Reporting and Plotting	24-18

25 Using Macros	25-1
25.1 Boundary Condition Macros	25-1
25.1.1 Solar Flux Boundary Condition	25-1
25.1.2 Atmospheric Boundary Layers	25-4
25.2 Diffuser Macros	25-10
25.2.1 Diffuser Modeling Methods	25-10
25.2.2 Diffuser Types	25-12
25.2.3 Steps for Adding a Diffuser to Your Airpak Model	25-12
25.2.4 Geometry, Position, and Size	25-30
25.2.5 Specifying Airflow Performance Data	25-35
25.2.6 Specifying the Extrusion and Flow Directions	25-38
25.2.7 Additional Inputs for Specific Types of Diffusers	25-39
25.2.8 Specifying Supply-Air and Room-Air Conditions	25-41
25.2.9 Specifying the Modeling Method	25-42
25.3 Geometry Macros	25-45
25.3.1 Polygonal Ducts	25-46
25.3.2 Adding a Polygonal Duct to Your Airpak Model	25-46
25.3.3 Closed Box	25-48
25.3.4 1/4 Polygonal Cylinder	25-51
25.3.5 Cylinder Plate	25-53
25.3.6 Cylindrical Enclosure	25-55
25.3.7 Polygonal Circle	25-58
25.3.8 Polygonal Cylinder	25-60
25.3.9 Hemisphere	25-62
25.4 Object Rotation Macros	25-65
25.4.1 Rotating Individual Plates	25-65
25.4.2 Rotating Prismatic Blocks	25-67
25.4.3 Rotating Polygonal Blocks	25-68
25.4.4 Rotating Groups of Prismatic Blocks	25-70

26 Generating a Mesh	26-1
26.1 Overview	26-1
26.2 Mesh Quality and Type	26-3
26.2.1 Mesh Quality	26-3
26.2.2 Hexahedral, Tetrahedral and Hex-Dominant Meshes	26-4
26.3 Guidelines for Mesh Generation	26-4
26.3.1 Hexahedral Meshing Procedure	26-5
26.3.2 Tetrahedral Meshing Procedure	26-7
26.4 Creating a Minimum-Count Mesh	26-8
26.4.1 Creating a Minimum-Count Hexahedral Mesh	26-8
26.4.2 Creating a Minimum-Count Tetrahedral Mesh	26-10
26.5 Refining the Mesh Globally	26-11
26.5.1 Global Refinement for a Hexahedral Mesh	26-11
26.5.2 Global Refinement for a Tetrahedral Mesh	26-13
26.6 Refining the Mesh Locally	26-16
26.6.1 General Procedure	26-17
26.6.2 Definitions of Object-Specific Meshing Parameters	26-17
26.6.3 Defining Meshing Parameters for Multiple Objects	26-19
26.6.4 Meshing Parameters for Rooms	26-20
26.6.5 Meshing Parameters for Blocks	26-21
26.6.6 Meshing Parameters for Fans	26-24
26.6.7 Meshing Parameters for Vents	26-24
26.6.8 Meshing Parameters for Openings	26-26
26.6.9 Meshing Parameters for Person Objects	26-27
26.6.10 Meshing Parameters for Walls	26-27
26.6.11 Meshing Parameters for Partitions	26-28
26.6.12 Meshing Parameters for Sources	26-28
26.6.13 Meshing Parameters for Resistances	26-28
26.6.14 Meshing Parameters for Heat Exchangers	26-29

26.6.15	Meshing Parameters for Hoods	26-29
26.6.16	Meshing Parameters for Assemblies	26-29
26.7	Controlling the Meshing Order for Objects	26-30
26.8	Non-Conformal Meshing Procedures for Assemblies	26-31
26.9	Displaying the Mesh	26-32
26.9.1	Displaying the Mesh on Individual Objects	26-32
26.9.2	Displaying the Mesh on a Cross-Section of the Model	26-36
26.10	Checking the Mesh	26-39
26.10.1	Checking the Element Quality	26-39
26.10.2	Checking the Face Alignment	26-41
26.10.3	Checking the Element Volume	26-42
26.11	Exporting a Mesh	26-44
26.11.1	Exporting an I-DEAS Neutral File	26-45
26.11.2	Exporting an ANSYS Grid File	26-45
26.12	Loading an Existing Mesh	26-45
27	Calculating a Solution	27-1
27.1	Overview	27-2
27.2	General Procedure for Setting Up and Calculating a Solution	27-3
27.3	Choosing the Discretization Scheme	27-4
27.4	Setting Under-Relaxation Factors	27-5
27.5	Selecting the Multigrid Scheme	27-6
27.6	Selecting the Version of the Solver	27-7
27.7	Initializing the Solution	27-7
27.8	Monitoring the Solution	27-8
27.8.1	Defining Solution Monitors	27-8
27.8.2	Plotting Residuals	27-12
27.9	Defining Postprocessing Objects	27-13
27.10	Defining Reports	27-13

27.11 Setting the Solver Controls	27-14
27.11.1 Using the Solve Panel to Set the Solver Controls	27-14
27.11.2 Advanced Solution Control Options	27-18
27.11.3 Parallel Processing	27-21
27.11.4 Batch Processing of Airpak Projects on a Windows Machine . . .	27-26
27.11.5 Running the Solution Using the Remote Simulation Facility (RSF)	27-29
27.12 Performing Calculations	27-49
27.12.1 Starting the Calculation	27-49
27.12.2 The Solution residuals Graphics Display and Control Window . .	27-49
27.12.3 Changing the Solution Monitors During the Calculation	27-51
27.12.4 Ending the Calculation	27-52
27.12.5 Judging Convergence	27-52
27.13 Diagnostic Tools for Technical Support	27-52
28 Examining the Results	28-1
28.1 Overview: The Post Menu and Postprocessing Toolbar	28-1
28.1.1 The Post Menu	28-1
28.1.2 The Postprocessing Toolbar	28-4
28.2 Graphical Displays	28-5
28.2.1 Overview of Generating Graphical Displays	28-5
28.2.2 The Significance of Color in Graphical Displays	28-5
28.2.3 Managing Postprocessing Objects	28-6
28.2.4 Displaying Results on Object Faces	28-9
28.2.5 Displaying Results on Cross-Sections of the Model	28-12
28.2.6 Displaying Results on Isosurfaces	28-19
28.2.7 Displaying Results at a Point	28-26
28.2.8 Contour Attributes	28-31
28.2.9 Vector Attributes	28-35
28.2.10 Particle Trace Attributes	28-39

28.3	XY Plots	28-43
28.3.1	Variation Plots	28-43
28.3.2	Trials Plots	28-47
28.4	Selecting a Solution Set to be Examined	28-51
28.5	Zoom-In Modeling	28-52
29	Generating Reports	29-1
29.1	Overview: The Report Menu	29-1
29.2	Variables Available for Reporting	29-3
29.3	HTML Reports	29-5
29.4	Reviewing a Solution	29-8
29.5	Summary Reports	29-9
29.6	Point Reports	29-13
29.7	Full Reports	29-15
29.8	Fan Operating Points Report	29-19
29.9	ADPI Report	29-20
30	Variables for Postprocessing and Reporting	30-1
30.1	General Information about Variables	30-1
30.2	Definitions of Variables	30-2
30.2.1	Velocity-Related Quantities	30-2
30.2.2	Pressure-Related Quantities	30-2
30.2.3	Temperature-Related Quantities	30-3
30.2.4	Radiation-Related Quantities	30-4
30.2.5	Species-Transport-Related Quantities	30-5
30.2.6	Position-Related Quantities	30-5
30.2.7	Turbulence-Related Quantities	30-5
30.2.8	IAQ and Thermal Comfort Quantities	30-6

31 Theory	31-1
31.1 Governing Equations	31-1
31.1.1 The Mass Conservation Equation	31-1
31.1.2 Momentum Equations	31-2
31.1.3 Energy Conservation Equation	31-2
31.1.4 Species Transport Equations	31-3
31.2 Turbulence	31-4
31.2.1 Zero-Equation Turbulence Models	31-4
31.2.2 Advanced Turbulence Models	31-5
31.3 Buoyancy-Driven Flows and Natural Convection	31-21
31.3.1 The Boussinesq Model	31-21
31.3.2 Incompressible Ideal Gas Law	31-22
31.4 Radiation	31-23
31.4.1 Overview	31-23
31.4.2 Gray-Diffuse Radiation	31-23
31.4.3 Radiative Transfer Equation	31-23
31.4.4 The Surface-to-Surface Radiation Model	31-24
31.4.5 The Discrete Ordinates (DO) Radiation Model	31-26
31.5 Optimization	31-30
31.5.1 The Dynamic-Q Optimization method	31-30
31.5.2 The Dynamic-Trajectory (Leap-Frog) Optimization Method for Solv- ing the Subproblems	31-33
31.6 Solution Procedures	31-36
31.6.1 Overview of Numerical Scheme	31-36
31.6.2 Spatial Discretization	31-38
31.6.3 Time Discretization	31-44
31.6.4 Multigrid Method	31-45
31.6.5 Solution Residuals	31-53

This chapter provides an introduction to **Airpak**, an explanation of its structure and capabilities, an overview of using **Airpak**, and instructions for starting the **Airpak** application. A sample session is also included.

Information in this chapter is divided into the following sections:

- Section 1.1: What is **Airpak**?
- Section 1.2: Program Structure
- Section 1.3: Program Capabilities
- Section 1.4: Overview of Using **Airpak**
- Section 1.5: Starting **Airpak**
- Section 1.6: Accessing the **Airpak** Manuals
- Section 1.7: Sample Session

1.1 What is **Airpak**?

Airpak is an accurate, quick, and easy-to-use design tool that simplifies the application of state-of-the-art airflow modeling technology to the design and analysis of ventilation systems which are required to deliver indoor air quality (IAQ), thermal comfort, health and safety, air conditioning, and/or contamination control solutions. The ability to rapidly create and automatically mesh ventilation system problems is coupled with **FLUENT**'s fast, accurate, and well-proven unstructured solver engine. Moreover, postprocessing features essential to the ventilation industry give designers and engineers a tool that provides the most accurate solution possible in the shortest amount of time compared to other airflow modeling software tools.

Airpak uses the **FLUENT** computational fluid dynamics (CFD) solver engine for thermal and fluid-flow calculations. The solver engine provides complete mesh flexibility, and allows you to solve complex geometries using unstructured meshes. The multigrid and segregated solver algorithms provide robust and quick calculations.

1.2 Program Structure

Your Airpak package includes the following components:

- Airpak, the tool for modeling, meshing, and postprocessing
- FLUENT, the solver engine
- filters for importing model data from Initial Graphics Exchange Specification (IGES), AutoCAD (DXF and DWG), and Industrial Foundation Classes (IFC) files

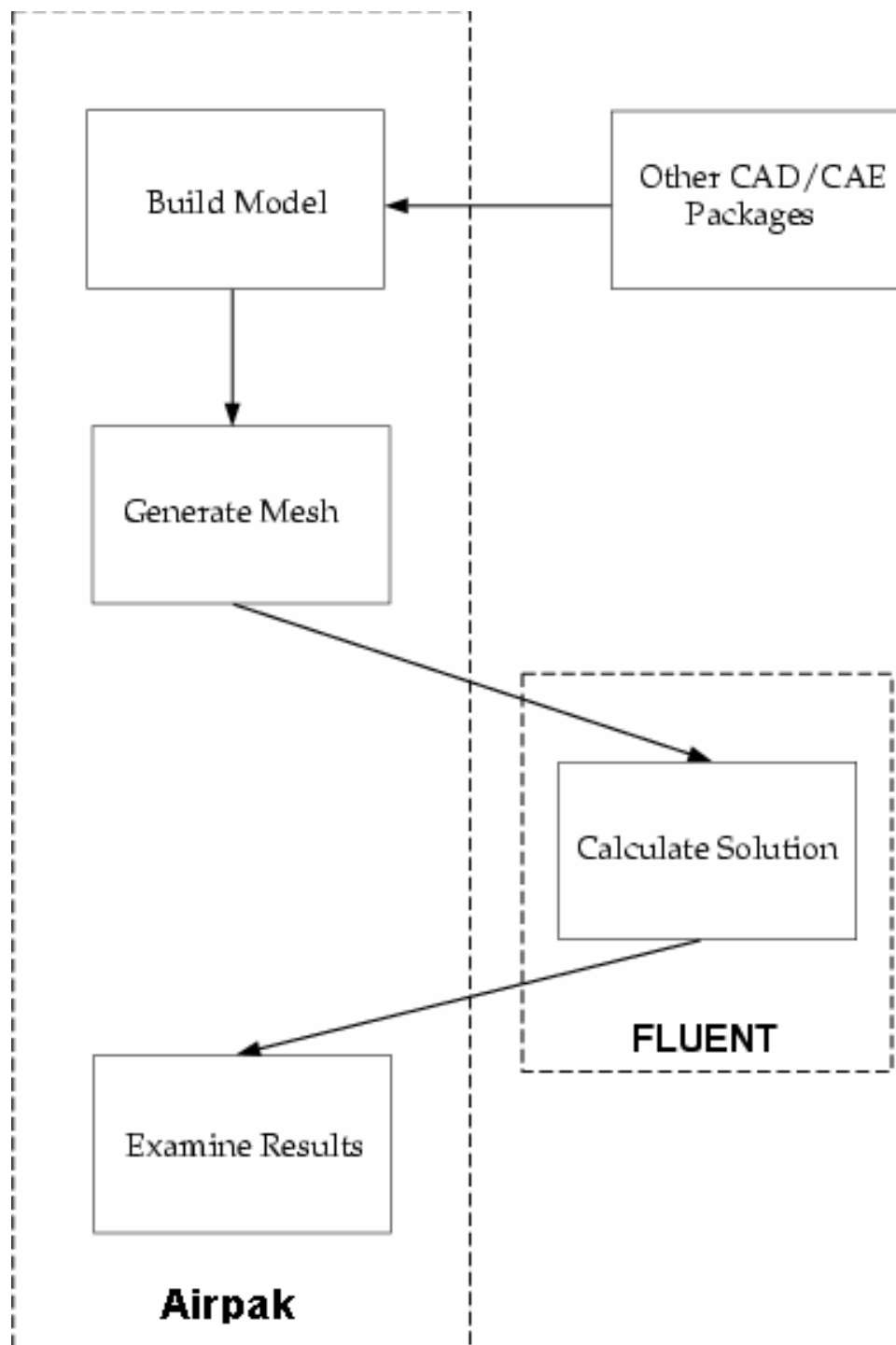


Figure 1.2.1: Basic Program Structure

Airpak is used to construct your model geometry and define your model. You can import model data from other CAD and CAE packages in this process. **Airpak** then creates a mesh for your model geometry, and passes the mesh and model definition to the solver for computational fluid dynamics simulation. The resulting data can then be postprocessed using **Airpak**, as shown in Figure 1.2.1.

1.3 Program Capabilities

All of the functions that are required to build an **Airpak** model, calculate a solution, and examine the results can be accessed through **Airpak**'s interactive menu-driven interface.

1.3.1 General

- mouse-driven interactive GUI controls
 - mouse or keyboard control of placement, movement, and sizing of objects
 - 3D mouse-based view manipulation
 - error and tolerance checking
- complete flexibility of unit systems
- geometry import using IGES, STEP, DXF, DWG, and IFC file formats
- library functions that allow you to store or retrieve groups of objects in an assemblies library
- on-line help and documentation
 - complete hypertext-based on-line documentation (including theory and tutorials)
- supported platforms
 - UNIX workstations
 - PCs running Windows NT 4.0, Windows 2000, or Windows XP

1.3.2 Model Building

- object-based model building with predefined objects
 - rooms
 - blocks
 - fans (with hubs)
 - person
 - openings

- vents
- partitions
- walls
- sources
- resistances
- heat exchangers
- hoods
- wires
- macros
 - boundary conditions (solar flux, atmospheric boundary layers, diffusers)
 - quick geometry/approximations (polygonal ducts, cylinders, and circles; closed boxes, cylindrical plates and enclosures; hemispheres)
 - object rotation (individual plates, prism blocks, and polygonal blocks; groups of prism blocks)
- 2D object shapes
 - rectangular
 - circular
 - inclined
 - polygon
- complex 3D object shapes
 - prisms
 - cylinders
 - ellipsoids
 - elliptical and concentric cylinders
 - prisms of polygonal and varying cross-section

1.3.3 Meshing

- automatic unstructured mesh generation
 - hexahedra, tetrahedra, pyramids, prisms, and mixed element mesh types
- meshing control
 - coarse mesh generation option for preliminary analysis
 - full mesh control
 - viewing tools for checking mesh quality
 - non-conformal meshing

1.3.4 Materials

- comprehensive material property database
- input for full anisotropic conductivity in solids
- temperature-dependent material properties

1.3.5 Physical Models

- laminar and turbulent flow models
- species transport
- ideal gas law
- steady-state and transient analysis
- forced, natural, and mixed convection heat transfer modes
- conduction in solids
- conjugate heat transfer between solid and fluid regions
- surface-to-surface radiation heat transfer model (with automatic view-factor calculation) and discrete ordinates radiation model
- solar load modeling
- volumetric resistances and sources for velocity and energy
- choice of mixing-length (zero-equation), indoor HVAC zero equation, two-equation (standard k - ϵ), RNG k - ϵ , enhanced two-equation (standard k - ϵ with enhanced wall treatment), or Spalart-Allmaras turbulence models
- contact resistance modeling
- non-isotropic volumetric flow resistance modeling, with non-isotropic resistance proportional to velocity (linear and/or quadratic)
- internal heat generation in volumetric flow resistances
- non-linear fan curves for realistic fan modeling
- lumped-parameter models for fans, resistances, and vents

1.3.6 Boundary Conditions

- wall and surface boundaries with options for specification of heat flux, temperature, species, convective heat transfer coefficient, radiation, and symmetry conditions
- openings and vents with options for specification of inlet/exit velocity, exit static pressure, inlet total pressure, inlet temperature, and species
- fans, with options for specified mass flow rate, fan performance curve, and angular specification of velocity direction
- recirculating boundary conditions for external heat exchanger simulation or species filters
- time-dependent and temperature-dependent sources
- time-varying ambient temperature inputs

1.3.7 Solver

For its solver engine, **Airpak** uses **FLUENT**, Fluent Inc.'s finite-volume solver. **Airpak**'s solver features include:

- segregated solution algorithm with a sophisticated multigrid solver to reduce computation time
- choice of first-order upwinding for initial calculations, or a higher-order scheme for improved accuracy

1.3.8 Visualization

- 3D modeling and dynamic viewing features
- visualization of velocity vectors, contours, particle traces, grid, cut planes, and isosurfaces
- point probes and XY plotting for data reporting
- contours of velocity components, speed, temperature, species mass fractions, relative humidity, pressure, heat flux, heat transfer coefficient, flow rate, turbulence parameters, thermal comfort parameters, and many more quantities
- velocity vectors color-coded by temperature, velocity magnitude, pressure, or other solved/derived quantities
- animation for viewing particle and dye traces

- animation of vectors and contours in transient analyses
- animation of plane cuts through the domain
- export of animations in AVI, MPEG, FLI, Flash, and animated GIF formats

1.3.9 Reporting

- writing to user-specified ASCII files of all solved quantities and derived quantities (heat flux, mass flow rate, heat transfer coefficient, etc.) on all objects, parts of objects, and user-specified regions of the domain
- time history of solution variables at any point in the model
- graphical monitoring of convergence history during the solution process
- report of operating point for fans that use a fan characteristic curve
- direct graphics output to printers and/or to user-specified files
 - color, gray-scale, or monochrome PostScript
 - PPM
 - TIFF
 - GIF
 - JPEG
 - VRML scripts
 - MPEG movies
 - AVI movies
 - FLI movies
 - animated GIF movies
 - Flash files

1.3.10 Applications

Airpak can be used to solve a wide variety of HVAC and contamination control applications, including, but not limited to, the following:

- commercial or residential building ventilation
- health care facilities
- telecommunications room ventilation

- clean rooms (pharmaceutical and semiconductor)
- industrial air conditioning
- industrial hygiene (health and safety)
- kitchen ventilation
- transportation comfort
- enclosed vehicular facilities
- engine test facilities
- external building flows
- architectural design

1.4 Overview of Using Airpak

Before you create your model in **Airpak**, you should plan the analysis for your model. When you have considered the issues discussed in Section 1.4.1, you can set up and solve your problem using the basic steps listed in Section 1.4.2.

1.4.1 Planning Your Airpak Analysis

When you are planning to solve a problem using **Airpak**, you should first give consideration to the following issues:

- **Defining the Modeling Goals:** What results are required? What level of accuracy is needed? The level of accuracy that is required will help you determine assumptions and approximations. How detailed should your problem setup be?
- **Choosing the Computational Model:** What are the boundary conditions?
- **Choosing the Physical Models:** What is the flow regime (laminar or turbulent) and fluid type? Is the flow steady or transient? What other physical models do you need to apply (e.g., gravity)?
- **Determining the Solution Procedure:** Can the problem be solved simply, using the default solver formulation and solution parameters? Can convergence be accelerated with a more judicious solution procedure? Will the problem fit within the memory constraints of your computer? How long will the problem take to converge on your computer?

Careful consideration of these issues before beginning your **Airpak** analysis will contribute significantly to the success of your modeling effort. When you are planning an analysis project, take advantage of the customer support provided to all **Airpak** users.

1.4.2 Problem Solving Steps

Once you have determined important features of the problem you want to solve using Airpak, follow the basic procedural steps outlined below.

1. Create a project file.
2. Specify the problem parameters.
3. Build the model.
4. Generate the mesh.
5. Calculate a solution.
6. Examine the results.
7. Generate summaries and reports.

Table 1.4.1 shows each problem solving step and the Airpak menu, window, or toolbar it is initiated from, as well as the chapter in this manual that describes the process.

Table 1.4.1: Problem Solving Steps in Airpak

Problem Solving Step	Interface Location	See...
1. Create project file	File menu	Chapter 6
2. Specify the problem parameters	Model manager window	Chapter 6
3. Build the model	Object toolbar	Chapters 7–19
4. Generate a mesh	Model menu	Chapter 26
5. Calculate a solution.	Solve menu	Chapter 27
6. Examine the results	Post menu or toolbar	Chapter 28
7. Generate summaries and reports.	Report menu	Chapter 29

1.5 Starting Airpak

The way you start Airpak will be different for UNIX and Windows systems, as described below. The installation process (described in the separate installation instructions for your computer type) is designed to ensure that the Airpak program is launched when you follow the appropriate instructions. If it is not, consult your computer systems manager or your Airpak support engineer.

Once you have installed Airpak on your computer system, you can start it as described below.

1.5.1 Starting Airpak on a UNIX System

For a UNIX system, start Airpak by typing `airpak` at the system prompt.

```
airpak
```

You can also start an Airpak application on a UNIX system using a startup command line option. For example, if you want to start Airpak and load a project that was previously created (e.g., `sample`), you can type

```
airpak sample
```

at the system prompt, and the project file named `sample` will be loaded into Airpak and displayed in the graphics window.

To start Airpak and load a packed project (e.g., `sample.tzr`) that has already been created, type the following command:

```
airpak sample.tzr
```

After startup, you will be prompted to specify the location for the unpacked project.

If the project you name is a new project, then an empty default room will appear in the graphics window. See Section 1.5.4 for details on startup command line options for UNIX systems.

1.5.2 Starting Airpak on a Windows System

For a Windows (Windows NT 4.0, Windows 2000, or Windows XP) system, there are several ways to start Airpak. Begin by setting environment variables. You can then start Airpak from the Start menu or from a desktop shortcut you create.

1. Set the environment variables for Airpak. Click on the Start button, select the Programs menu, select the **Fluent.Inc** menu, and then select the Set Environmental Variables utility that is found in the **Fluent.Inc** program group.

This program will set the **AIRPAK_ROOT** environment variables and adjust your command search path to find `airpak.exe` on your system.

2. Start Airpak.
 - (a) Start Airpak from the Start menu. Click on the Start button, select the Programs menu, select the **Fluent.Inc** menu, and then select the `airpak30` program item.

i Note that if the default “Fluent.Inc” program group name was changed when **Airpak** was installed, you will find the **airpak30** menu item in the program group with the new name that was assigned, rather than in the **Fluent.Inc** program group.

(b) Start **Airpak** from an MS-DOS Command Prompt window. There are three options:

- Type **airpak** at an MS-DOS command prompt.

airpak

- Start **Airpak** and load a project file that has already been created.

For example, you can type

airpak sample

at the prompt, and the project file named **sample** will be loaded into **Airpak** and displayed in the graphics window. If the project you name is a new project, then an empty default room will appear in the graphics window.

- Start **Airpak** and load a packed project that has already been created. For example, to start **Airpak** and unpack a project named **sample**, type the following command:

airpak.bat sample.tzr

After startup, you will be prompted to specify the location for the unpacked project.

(c) Start **Airpak** from a desktop shortcut.

First, create a desktop shortcut to

AIRPAK_ROOT\ntbin\ntx86\airpak.bat

where you must replace *AIRPAK_ROOT* by the full pathname of the directory where **Airpak** is installed on your computer system (see Section 1.5.5).

Then, double-click on the shortcut to start **Airpak**. See your Windows documentation for details on creating a desktop shortcut.

1.5.3 Startup Screen

When the application startup procedure is completed, **Airpak** displays the startup screen (shown in Figure 1.5.1), which consists of two components: the **Main** window and the **New/existing** panel.

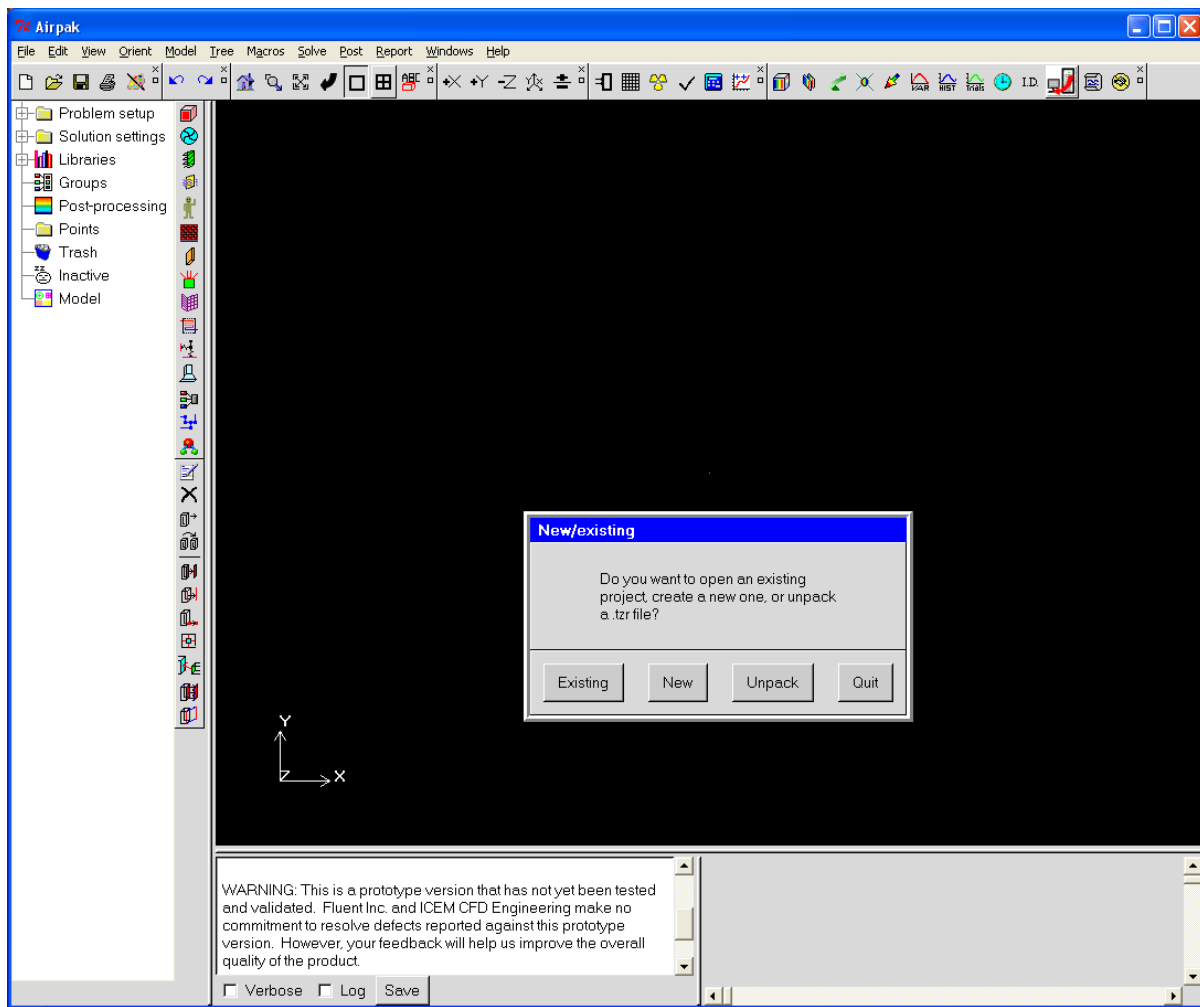


Figure 1.5.1: The Startup Screen

The Main Window

The **Main** window controls the execution of the **Airpak** program and contains sub-windows for navigating the project list tree (left), displaying messages (bottom left), editing general **Airpak** object parameters (bottom right), and displaying the model (center). You can resize any of these sub-windows within the **Main** window by holding down your left mouse button on any of the square boxes on the window borders and dragging your mouse in a direction allowed by the cursor. The **Main** window is discussed in Section 2.1.1.

The New/existing Panel

The New/existing panel is a temporary window that closes once you choose one of three options:

- To create a new project, click **New** in the New/existing panel, which will instruct Airpak to open the New project panel. In the New project panel, enter the name of the project in the **Project** field and click **Create**, or you can click **Cancel** and select **New project** in the **File** menu.
- To open an existing project, click **Existing** in the New/existing panel. In the Open project panel, you can use your left mouse button to select a project from the **Directory** list and click **Open**. To open a project that was recently edited, you can select the project name in the drop-down list to the right of **Recent projects** in the Open project panel and then click **Open**.
- To expand a compressed (or packed) file, click **Unpack** in the New/existing panel. In the File selection dialog box, you can use your left mouse button to select a **.tzz** file from the **Directory** list and click **Open**.



Selecting **Quit** from the New/existing panel will terminate your Airpak session.

For more details on how to create new projects or open existing projects see Section 6.2. See Section 2.1.8 for details on selecting a project using a file selection dialog box.

You can configure your graphical user interface for the current project you are running, or for all Airpak projects, using the **Options** node of the **Preferences** panel. See Section 6.3 for details on changing the configuration parameters.

1.5.4 Startup Options for UNIX Systems

Although Airpak can be started by simply entering **airpak** at the system command prompt, you can customize your Airpak startup using command line arguments. The general form of the Airpak command line is:

```
airpak -option value [-option value ... ] [projectname]
```

where **option** is the name of the option argument, and **value** is a value for a particular option. Items enclosed in square brackets are optional. (Do not type the square brackets.) Not all option arguments allow values to be specified. Arguments can be entered in any order on the command line, and are processed from left to right. Each command line argument is listed below, along with its functional description.

- **-x** specifies the X Windows graphics driver. By default (i.e., when you start **Airpak** without the **-x** option) **Airpak** will use the native graphics driver for each specific workstation platform. The native graphics driver typically takes advantage of the available graphics hardware, particularly for 3D graphics operations. The **-x** option is useful, for example, if you are accessing a 3D graphics workstation using an X terminal.
- **-xfast** enables a fast form of X Window graphics. This is accomplished by various shortcuts in the display of graphical information. The resulting graphics display will not be as aesthetically pleasing as the **-x** graphics, but it does result in faster performance when performing dynamic manipulation of the displayed model. This mode of operation is particularly useful when using an X terminal that is connected via the network to a workstation on which **Airpak** is running.
- **-unpack** allows you to restore files that were packed using the **Pack** process to their original state. You can compress project files into a single encoded file that is suitable for electronic file transfer to your technical support engineer using the **Pack** option in the **File** menu. To restore or “unpack” files, type

```
airpak -unpack filename.tzr
```

where **filename.tzr** is the name of the compressed file. See Section 3.6 for more details.

- **projectname** (if specified) must be the last argument in the command line argument list. It is the name of the **Airpak** project to be loaded on startup, and can be either a new project or an existing project. Specifying a **projectname** bypasses the **New/existing** panel.

1.5.5 Environment Variables on UNIX Systems

You can use environment variables to tailor the operation of **Airpak** to a particular environment. There are two types of environment variables: system environment variables and **Airpak** environment variables. System environment variables are used by **Airpak** at the system level, and are independent of the **Airpak** application. **Airpak** environment variables are specific to the execution of **Airpak**.

System Environment Variables

In most cases, the system environment variables shown in Table 1.5.1 will already have been set for your system when you log onto your computer. These environment variables must be set for your system before you can use **Airpak**.

For more information on environment variables, refer to documentation concerning shell commands for your computer system.

Table 1.5.1: System Environment Variables

Variable	Description
HOME	The path to your home directory where configuration files are located. (UNIX systems only)
TERM	The terminal type (e.g., xterm, hpterm). (UNIX systems only)
DISPLAY	The location of the display screen. This is used by the X Window system to determine which screen to display its output to (e.g., <code>unix:0.0</code>), and must be set for Airpak to operate. (UNIX systems only)
PATH	The list of directories to search for system-level commands.

You will need to include the `Fluent.Inc/bin` directory in the path. For example, if **Airpak 3.0.x** has been installed under `/usr/local`, there will be a `Fluent.Inc` directory present as `/usr/local/Fluent.Inc`. You would then need to set your `PATH` environment variable to include `/usr/local/Fluent.Inc/bin`.

Airpak Environment Variables

Airpak allows you to change certain aspects of its operation by setting **Airpak**-specific environment variables. There are two **Airpak**-specific environment variables, which are described in Table 1.5.2.

Table 1.5.2: Airpak-Specific Environment Variables

Variable	Description
AIRPAK_LIB_PATH	Allows you to specify the search path where Airpak loads the material library files or the files for customized macros. See Section 6.3.5 for details.
AIRPAK_LICENSE_FILE	Allows you to specify an alternate location for the Airpak license file. See the Airpak installation instructions for details.
AIRPAK_JOB_DIRECTORY	Allows you to specify a default location for unpacking files. See Section 3.6 for details.

1.6 Accessing the Airpak Manuals

As described in Section 2.1.2, Airpak's on-line help gives you access to the Airpak documentation through HTML files, which can be viewed with your standard web browser (e.g., Netscape Communicator). For printing, Adobe Acrobat PDF versions of the manuals are also provided. This section describes how to access the Airpak manuals outside of Airpak (i.e., not through the Airpak on-line help utility). See Section 2.1.2 for information about accessing the manuals through the on-line help.

You can access the manuals directly from the documentation CD or in the Airpak installation area. See the separate installation instructions for your platform type for information about installing the files from the documentation CD.

i The comments in this section about accessing files in the installation area assume that all files on the documentation CD have been installed. If the files you are looking for are not in the installation area, you will need to install them or access them directly from the CD.

The following sections provide information about accessing the manuals:

- Section 1.6.1: Viewing the Manuals
- Section 1.6.2: Printing the Manuals

1.6.1 Viewing the Manuals

To view the manuals, you will generally make use of the HTML files, either in the installation area or on the documentation CD. You will also need a web browser. If you do not have one, contact your support engineer and ask for the Netscape Communicator CD-ROM.

How to Access the HTML Files in the Installation Area

If the files on the documentation CD have been installed, you can view the HTML versions of the manuals by pointing your browser to

AIRPAK_ROOT/lib/help/index.htm

where you must replace *AIRPAK_ROOT* by the full pathname of the directory where *Airpak* is installed on your computer system (e.g., */usr/local/Fluent.Inc/airpak3.0.x*). See Section 1.5.5 for details.

If, for example, you are using Netscape Communicator as your browser, select the **File/Open Page...** menu item and click the **Choose File...** button to browse through your directories to find the file.

This will bring up the *Airpak* 3.0 documentation “home” page (Figure 1.6.1), from which you can select the HTML version of the particular *Airpak* manual you want to view.

How to Access the HTML Files on the CD

The procedure for viewing the manuals directly on the CD differs slightly for UNIX and Windows systems:

- For UNIX systems, you can view the manuals by inserting the CD into your CD-ROM drive and pointing your browser to the following file:

/cdrom/index.htm

where *cdrom* must be replaced by the name of your CD-ROM drive.

If, for example, you are using Netscape Communicator as your browser, select the **File/Open Page...** menu item and click the **Choose File...** button to browse through your directories to find the file.

- For Windows systems, you can view the manuals by inserting the CD into your CD-ROM drive and pointing your browser to the following file:

cdrom:\Index.htm

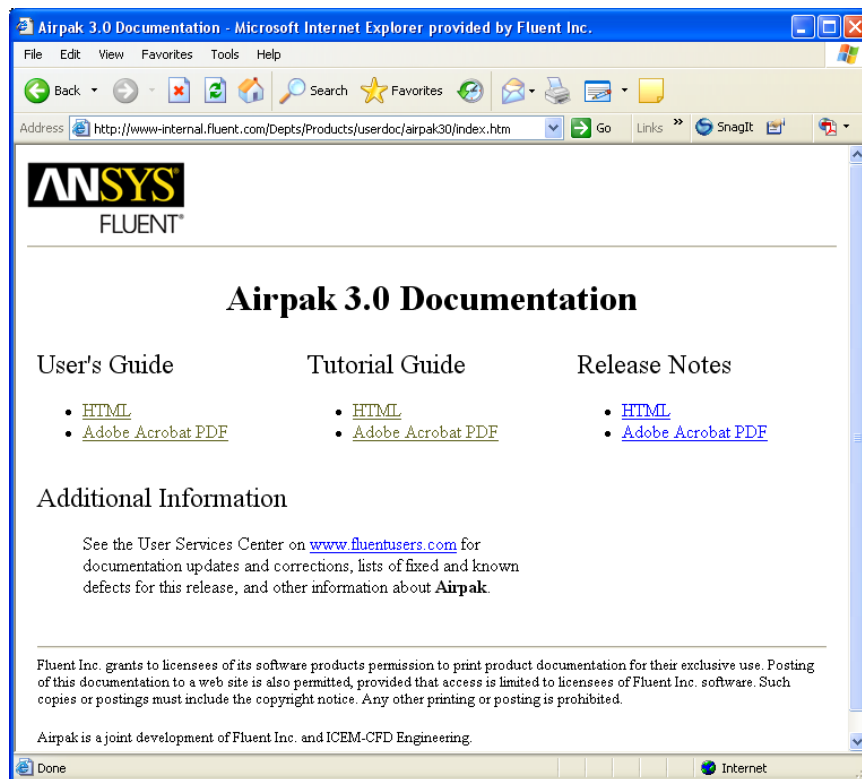


Figure 1.6.1: The Airpak 3.0 Documentation Home Page

where *cdrom* must be replaced by the name of your CD-ROM drive (e.g., E).

If, for example, you are using Netscape Communicator as your browser, select the File/Open Page... menu item and click the Choose File... button to select the CD-ROM drive (which will be named `Fluent_inc`, followed by the letter for the CD-ROM drive) and choose the appropriate file.

This will bring up the Airpak 3.0 documentation “home” page (Figure 1.6.1), from which you can select the HTML version of the particular Airpak manual you want to view.

Navigating the Manuals

When you are viewing a manual in your browser, a set of navigation buttons will appear at the upper right and lower right corners of the HTML page, as shown in Figure 1.6.2.

The navigation buttons are as follows:

Next will take you to the next page in the manual. Note that this is not the same function as the “Forward” button of your browser.

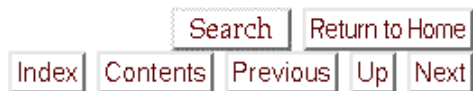


Figure 1.6.2: Navigation Buttons

Up will take you to the first page of the current manual division (chapter or section).

Previous will take you to the page just before the current one in the manual. Note that this is not the same function as the “Back” button of your browser.

Index will take you to the index for the manual.

Contents will take you to the table of contents for the manual.

Return to Home will take you to the Airpak documentation home page (Figure 1.6.1).

Search will display a full text search popup window (Figure 1.6.3).

Note that these buttons will not necessarily appear on all pages, and in some cases, they will appear, but be inactive; in such cases, they will be “grayed out”, as the **Previous** and **Up** buttons are in Figure 1.6.2.

Finding Information in the Manuals

Three tools are available to help you find the information you are looking for in a manual.

Index

The index gives an alphabetical list of keywords, each linked to relevant sections of the manual. You can access the index by clicking the **Index** button that appears at the top and bottom of the page. Note that the **Index** button will not appear if the manual does not have an index.

In a larger manual, the index will be split into a number of pages, each containing keywords starting with a particular letter; in this case, the **Index** button will take you to the “A” index page.

Table of Contents

The table of contents gives a list of the titles of the chapters, sections, and subsections of the manual in the order in which they appear. Each title is linked to the corresponding chapter or section. You can access the table of contents by clicking the **Contents** button that appears at the top and bottom of the page. Note that the **Contents** button will not appear if the manual does not have a table of contents.

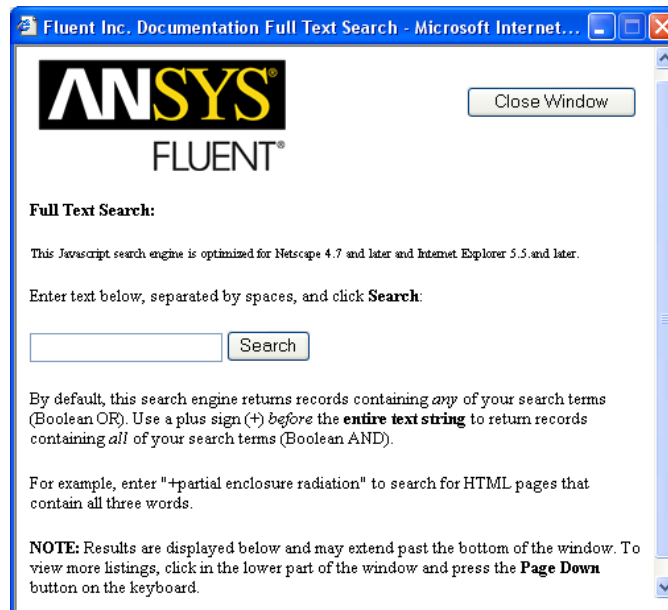


Figure 1.6.3: Full Text Search Engine

Browser Search


The full text search engine (Figure 1.6.3) allows you to search the User's Guide for either single or multiple keywords. You can access the full text search engine by clicking the **Search** button that appears at the top of the page. To display the popup window and use the search engine, Javascript should be enabled in your web browser. Note that the Javascript search engine has been optimized for Netscape 4.7 and later and Internet Explorer 5.5 and later.

To use the search engine, enter a specific term or keyword(s) in the text field and click the **Search** button. By default, the search engine will look for pages that contain *any* keyword input (Boolean OR). If you insert a "+" before the keyword input, then the search engine will look for pages that contain *all* of the keywords provided in the text field (Boolean AND). Search results appear in the lower half of the search engine popup window. If a scroll bar does not automatically appear when search results extend below the bottom edge of the window, you can make it be displayed by selecting the lower portion of the window and pressing the <Page Down> button on the keyboard. To clear the search text field, click the **Reset** button. To clear your previous search results from the window, click the **Clear Results** button.

You can also use the search capability provided by your browser to find words or expressions on a single page. For example, you can use the **Edit/Find in Page...** menu item in Netscape Communicator to search for the word “temperature” on a page of the manual.

Typographical Conventions

Throughout the manuals, mini flow charts are used to indicate the menu or list tree node selections that lead you to a specific command or panel. Words in green invoke menus, sub-menus, tree nodes, and node items and are connected by arrows that point from a specific menu or list tree node toward the item you should select from that menu or node. When appropriate, pictures of toolbar buttons are used to indicate the button that will lead you to a specific panel.

An icon () at the beginning of a line marks an important note.

Printing Portions of a Manual

Although you can print pages of a manual from your browser, a much higher-quality printout can be obtained by using the PDF files provided on your documentation CD. See Section 1.6.2 for details.

Modifying the Appearance of the Manuals

There are a few things that you might want to change about the way your browser displays the manuals in order to increase their usefulness.

Font Size

The absolute size of the text that you will see when viewing your **Airpak** documentation is dependent on a number of factors, including the resolution of your monitor screen. You can adjust the text size by changing the default font size in the preferences menu of your browser. In Netscape Communicator, for example, select the **Edit/Preferences...** menu item and then choose the **Appearance** category, where you will find the **Fonts** controls. Try several sizes to see the effect on the appearance of the manuals, and choose the one that is best for you.

Page Width

While reading a manual, you may find a figure that is wider than your browser’s window. As a result, some of the figure will be hidden from view. To see all of the figure, you can use the horizontal scroll bar at the bottom of your browser’s window, or increase your browser’s window size.

You might also want to adjust the window size to increase or decrease the page width to a comfortable reading width.

Tool Tips

When viewing the manuals with certain browsers, information about a figure will be displayed if you put your cursor over it. However, this information is not meaningful for most users, and you might find it somewhat distracting. On some browsers, you can disable the display of “Tool Tips” in the preferences menu.

1.6.2 Printing the Manuals

Adobe Acrobat PDF files are provided for printing all or part of the manuals.

About the PDF Files

The PDF files are appropriate for viewing and printing with Adobe Acrobat Reader, which is available for most UNIX and Windows systems. These files are distinguished by a `.pdf` suffix in their file names.

Note that you can select the paper size to which you are printing in Adobe Acrobat Reader by selecting the **File/Print Setup...** menu item and choosing the desired **Paper** size. If the page is too large to fit on your paper size, you can reduce it by selecting the **File/Print...** menu item and enabling the **Shrink to Fit** option.

If you do not have Adobe Acrobat Reader, you can download it (at no cost) from www.adobe.com. If you are not able to download files from the Internet, contact your support engineer and ask for the Adobe Acrobat Reader CD-ROM.

How to Access the PDF Files in the Installation Area

If the PDF files on the documentation CD have been installed, you can access them by pointing your browser to

`AIRPAK_ROOT/lib/help/index.htm`

where you must replace *AIRPAK_ROOT* by the full pathname of the directory where Airpak is installed on your computer system (see Section 1.5.5).

If, for example, you are using Netscape Communicator as your browser, select the **File/Open Page...** menu item and click the **Choose File...** button to browse through your directories to find the file.

This will bring up the Airpak 3.0 documentation “home” page (Figure 1.6.1), from which you can select the or PDF file(s) for the particular Airpak manual you want. For large manuals, PDF files are provided for the individual chapters in addition to a single PDF file for the entire manual.

How to Access the PDF Files on the CD

The procedure for accessing the PDF files directly on the CD differs slightly for UNIX and Windows systems:

- For UNIX systems, you can access the files by inserting the CD into your CD-ROM drive and pointing your browser to the following file:

`/cdrom/index.htm`

where *cdrom* must be replaced by the name of your CD-ROM drive.

If, for example, you are using Netscape Communicator as your browser, select the **File/Open Page...** menu item and click the **Choose File...** button to browse through your directories to find the file.

- For Windows systems, you can access the files by inserting the CD into your CD-ROM drive and pointing your browser to the following file:

`cdrom:\Index.htm`

where *cdrom* must be replaced by the name of your CD-ROM drive (e.g., **E**).

If, for example, you are using Netscape Communicator as your browser, select the **File/Open Page...** menu item and click the **Choose File...** button to select the CD-ROM drive (which will be named **Fluent_inc**, followed by the letter for the CD-ROM drive) and choose the appropriate file.

This will bring up the **Airpak 3.0** documentation “home” page (Figure 1.6.1), from which you can select the PDF file(s) for the particular **Airpak** manual you want. For large manuals, PDF files are provided for the individual chapters in addition to a single PDF file for the entire manual.

1.7 Sample Session

In the following demonstration, you will use **Airpak** to set up a problem, solve the problem, and postprocess the results. This is a basic introduction to the features of **Airpak**. Working through the examples in the tutorial guide will provide a more complete demonstration of the program's features.

1.7.1 Problem Description

The problem solved here is illustrated in Figure 1.7.1. It involves a room containing an air inlet opening, a vent, and walls that hold a constant temperature. The room is 4.57 m long, 2.74 m wide, and 2.74 m high. The opening measures 0.92 m \times 0.46 m and introduces a cold air jet into the room to ventilate it. The vent is 0.91 m \times 0.45 m. The interaction of inertial forces, buoyancy forces, and turbulent mixing is important in affecting the penetration and trajectory of the supply air.

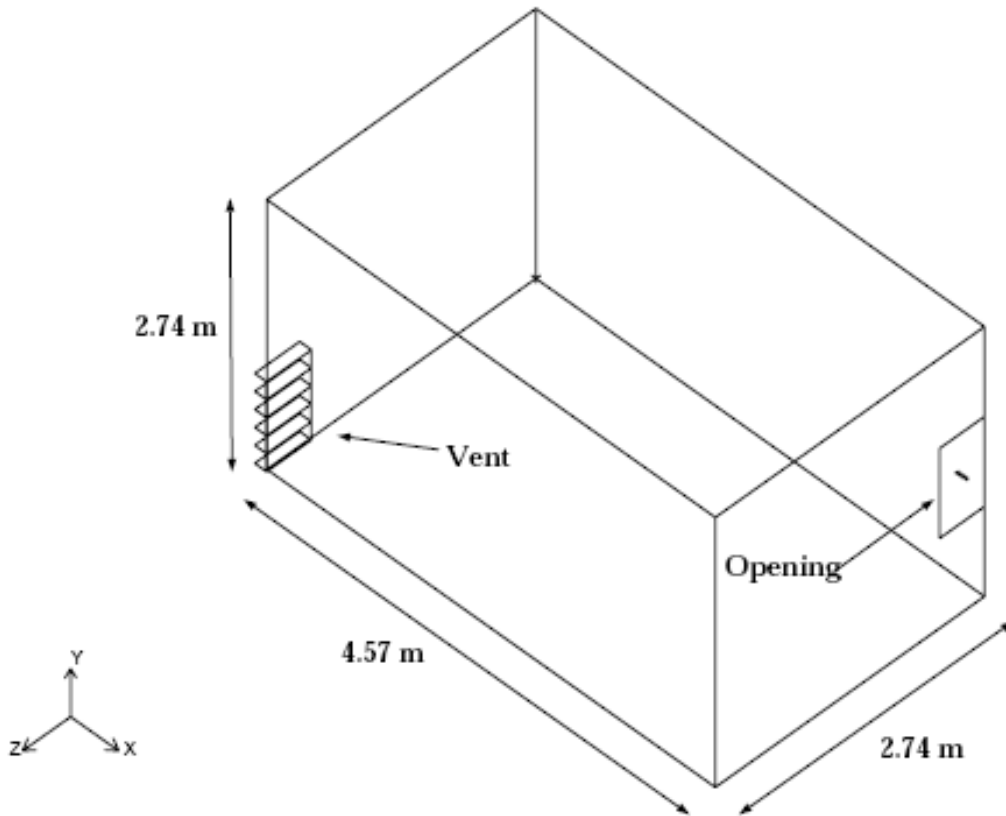


Figure 1.7.1: Problem Description

1.7.2 Outline of Procedure

The problem shown in Figure 1.7.1 is a case of steady ventilation flow through a room. Boundary temperatures and velocities are defined.

The steps you will follow in this sample session are reduced to the following:

- Create the room.
- Setup the basic parameters.
- Add the vent, opening, and wall modelling objects to the room.
- Generate a summary of the model.
- Create a mesh.
- Calculate a solution.
- Examine the results.

1.7.3 Setting Up the Project

Start **Airpak** as described in Section 1.5. The New/existing panel will be displayed, as shown in Figure 1.7.2. Click **New** to open the New project panel (Figure 1.7.3).

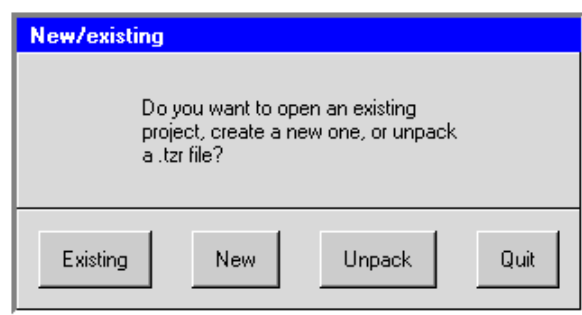


Figure 1.7.2: The New/existing Panel

Under **Project**, enter the name of your **Airpak** project (e.g., **sample**). Click **Create** to open the new project. **Airpak** will create a default room with the dimensions 10 m × 3 m × 10 m, and display the room in the graphics window.

You can rotate the room around a central point using the left mouse button, or you can translate it to any point on the screen using the middle mouse button. Zoom in and out from the room using the right mouse button. To restore the room to its default orientation, select **Home position** in the **Orient** menu.

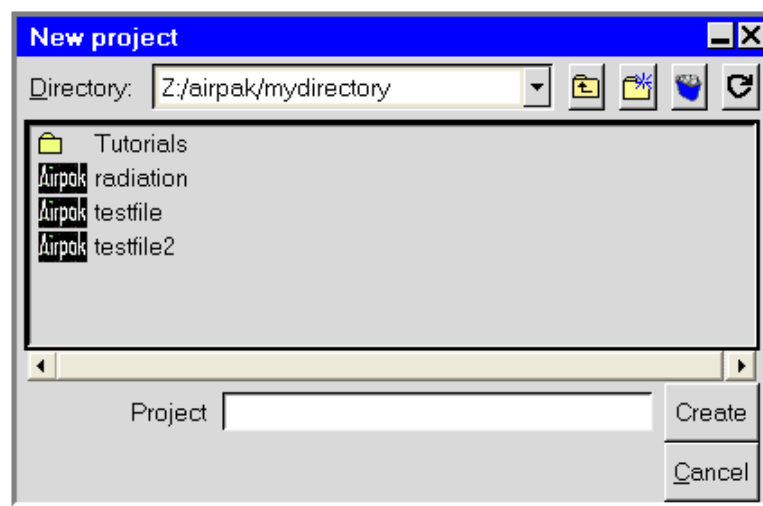


Figure 1.7.3: The New project Panel

1.7.4 Resizing the Room

To resize the room, ensure that the **Room** item in the **Model** node of the **Model manager** window is selected. Double click **Room** to display the **Room** edit panel.



In the **Geometry** tab, enter the coordinates shown in Table 1.7.1 into the **Room** edit panel. Click **Update** to resize the room. In the **Orient** menu, select **Isometric** view. This will display an isometric view of the room scaled to fit the graphics window.

Table 1.7.1: Coordinates for the Room

xS	0.0	xE	4.57
yS	0.0	yE	2.74
zS	0.0	zE	2.74

1.7.5 Setting Up the Basic Parameters

You will begin by defining the species within the room and setting ambient temperatures. These parameters are specified in the **Basic** parameters panel. To open the **Basic** parameters panel, select **Basic** parameters under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Basic parameters**

In the **General** setup tab of the **Basic** parameters panel, turn the **IAQ/Comfort** option **Off**,

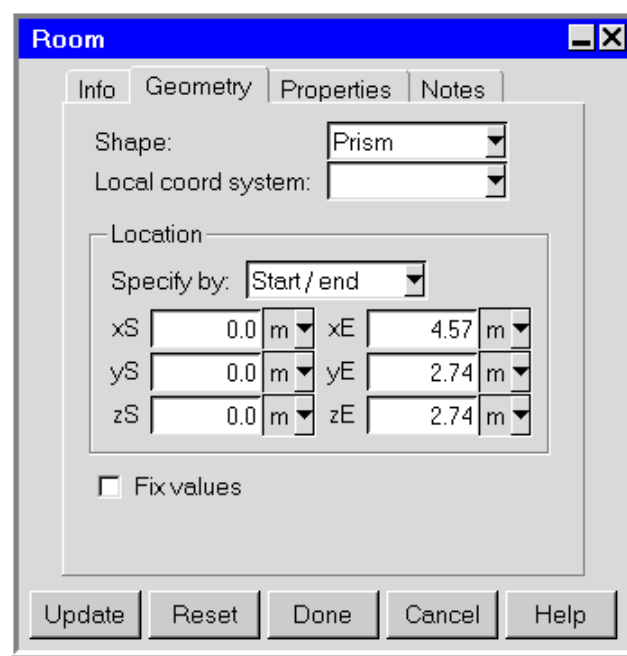


Figure 1.7.4: The Room Panel (Geometry Tab)

because you will not be considering air quality issues. Turn On Species and click the Edit button to open the Species definitions panel Figure 1.7.6).

Enter a value of 50 for the Initial concentration of h2o. Select RH from the menu next to the Initial concentration entry field for h2o and click Accept to close the panel.

In the Default values tab of the Basic Parameters panel, enter new Ambient values for the problem. Enter 20.56 for Temperature and 20.56 for Radiation temp. Click Accept to save the new settings.

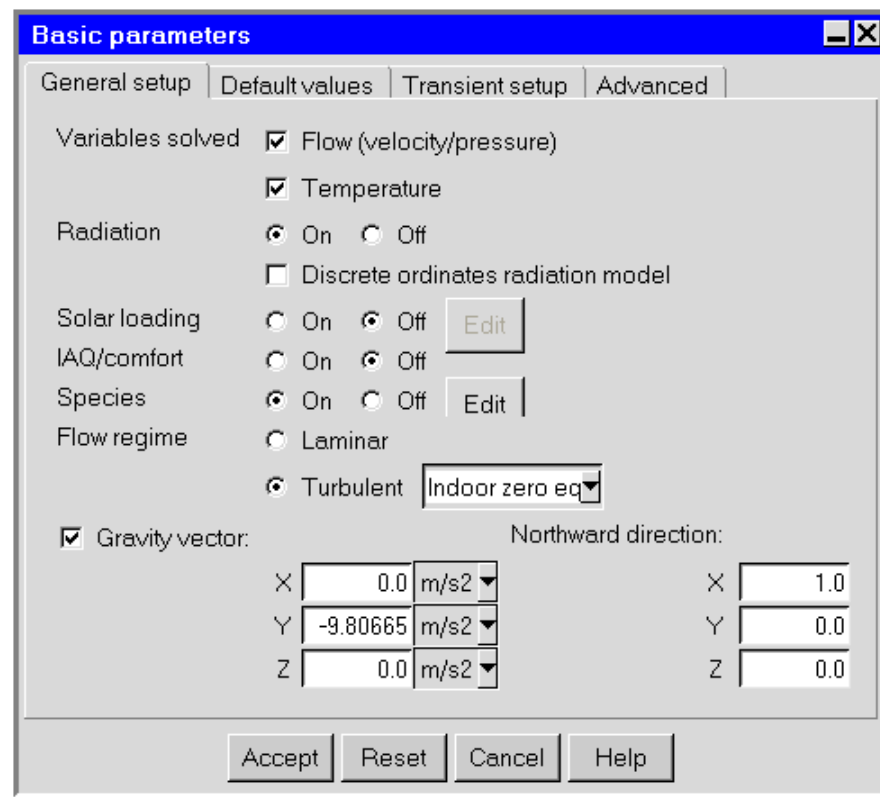


Figure 1.7.5: The Basic parameters Panel (General setup Tab)

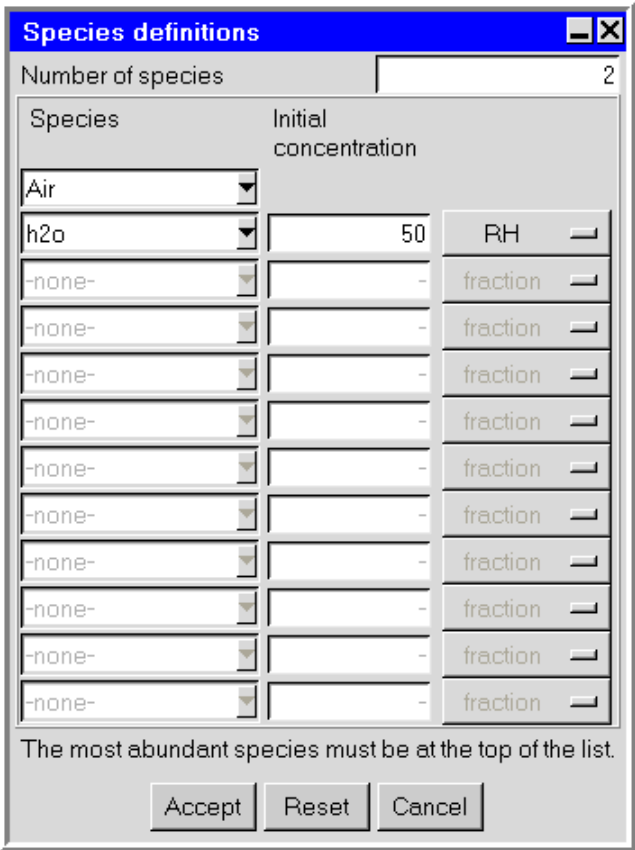




Figure 1.7.6: The Species definitions Panel

1.7.6 Adding Objects to the Room

You will now add objects to the room. The process required to add an object involves three steps:

1. Create a new object.
2. Specify the location of the object.
3. Specify the flow and thermal characteristics of the object.

Adding a Rectangular Vent

First, you will create a vent to the back wall of the room. Click the  button to create a new vent, and then click the  button to open the Vents panel (Figures 1.7.7 and 1.7.8).

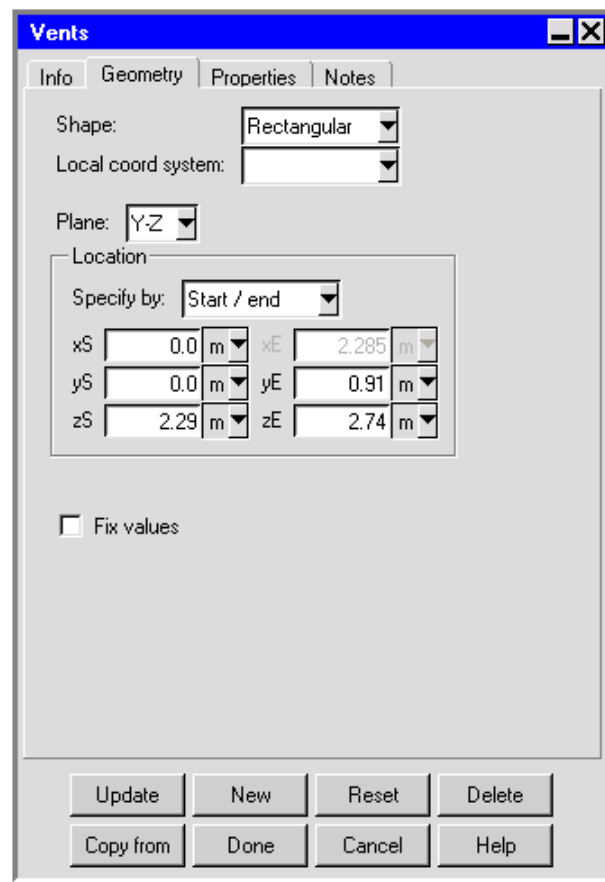


Figure 1.7.7: The Vents Panel (Geometry Tab)

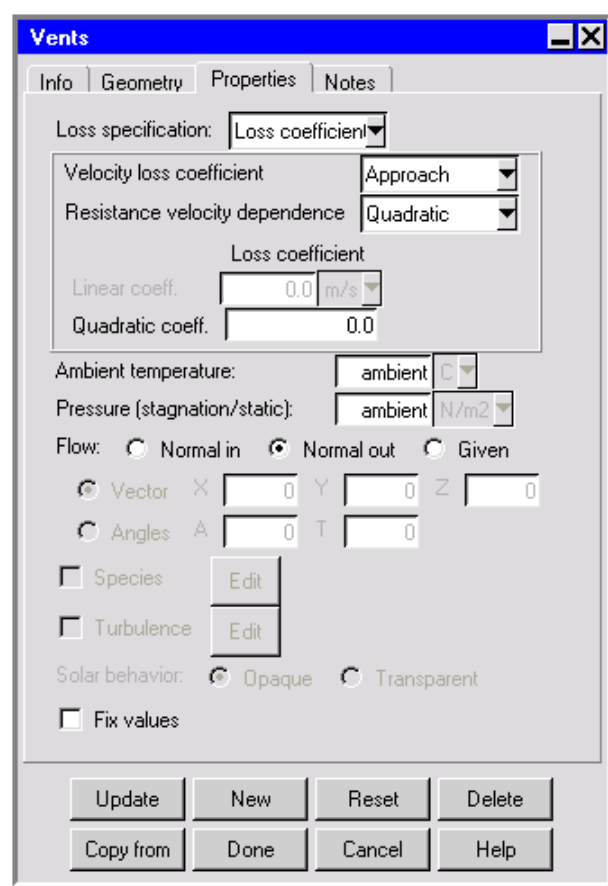


Figure 1.7.8: The Vents Panel (Properties Tab)

Click the **Geometry** tab. In the **Plane** drop-down list, select **Y-Z** to define the plane in which the vent lies. Under **Specify by**, enter the coordinates shown in Table 1.7.2 for the vent.

Table 1.7.2: Coordinates for the Vent

xS	0.0	xE	—
yS	0.0	yE	0.91
zS	2.29	zE	2.74



Click the **Properties** tab. Select **Approach** in the **Velocity loss coefficient** drop-down list. To specify that the vent is an outlet vent, keep the default option of **Normal out** for **Flow**. Click **Done** to accept all of the changes and close the panel.



Note that you can also change the shape and size of the vent using the mouse. Hold down the <Shift> key on the keyboard and use the right mouse button to move the vent's boundaries. To translate the vent in the room, hold down the <Shift> key and use the middle mouse button.

Adding an Opening

Next, you will add an opening on the front wall.

Click the  button to create a new opening, and then click on the  button to open the Openings panel (Figures 1.7.9 and 1.7.10).

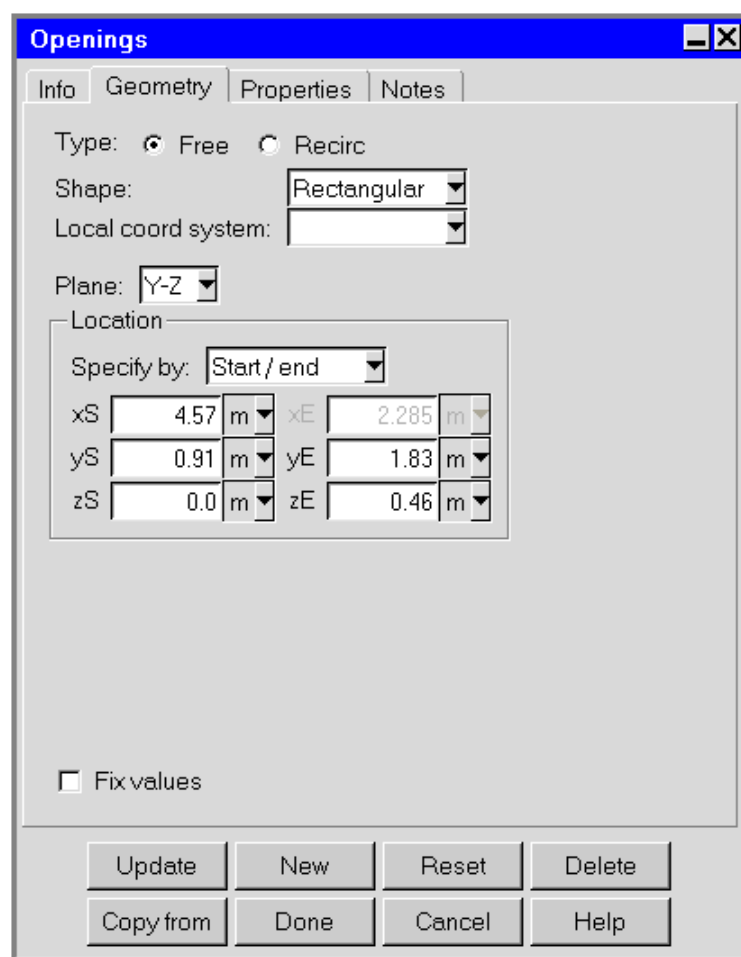


Figure 1.7.9: The Openings Panel (Geometry Tab)

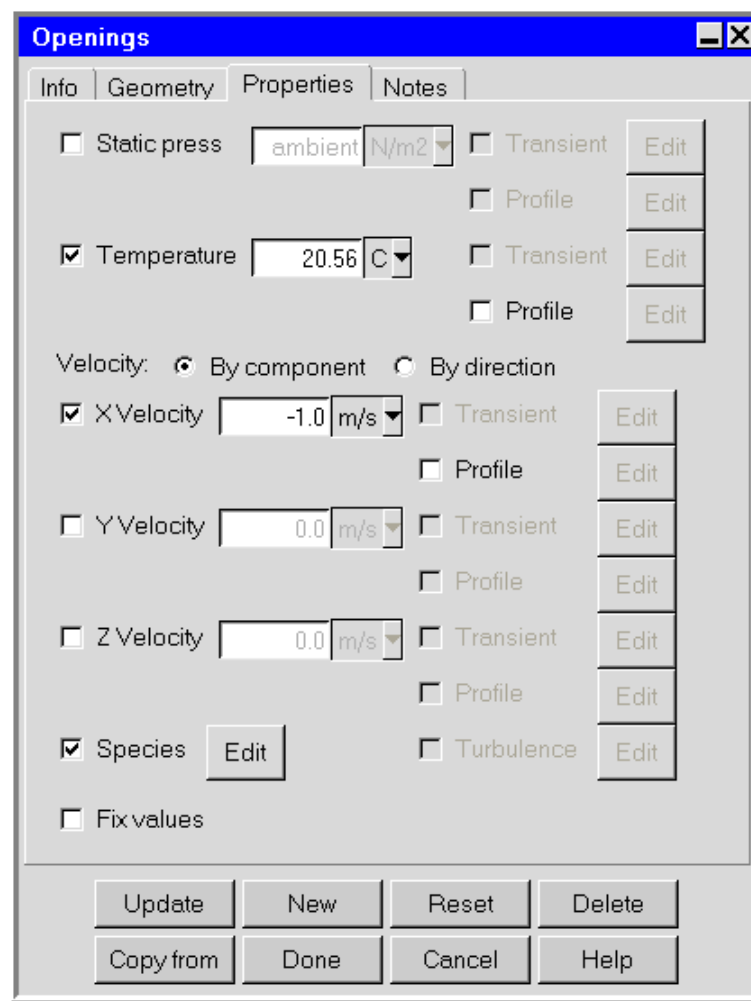


Figure 1.7.10: The Openings Panel (Properties Tab)

Click the **Geometry** tab (Figure 1.7.9). In the **Plane** drop-down list, select **Y-Z** to define the plane in which the opening lies. Enter the coordinates shown in Table 1.7.3.

Table 1.7.3: Coordinates for the Opening

xS	4.57	xE	—
yS	0.91	yE	1.83
zS	0.0	zE	0.46

Click the **Properties** tab (Figure 1.7.10). Select **Temperature** and enter a value of 20.56. Keep the default **Velocity** method of **By component**, select **X Velocity** and enter a value of -1.

Select **Species** and click **Edit**. This will open the **Species concentrations** panel (Figure 1.7.11). Enter a value of 50 for the **Concentration** of **h2o**. Select **RH** from the menu to the right of the **Concentration** entry field for **h2o**. Click **Done** in the **Species concentrations** panel to accept the changes. Click **Done** in the **Openings** panel to accept all changes and close the panel.

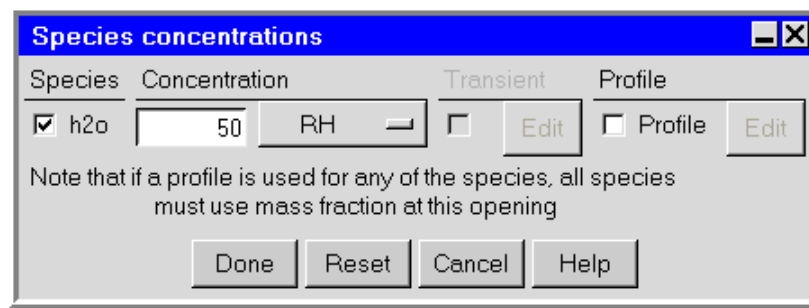




Figure 1.7.11: The Species concentrations Panel

Adding Walls

Finally, you will add walls to represent the boundaries of the room. For each pair of parallel walls, you will create one wall and then copy it to create the opposing wall.

Adding the Floor and Ceiling

To construct the wall representing the floor, click the  button to create a new wall, and then click the  button to open the **Walls** panel (Figures 1.7.12 and 1.7.13).

Click the **Geometry** tab. In the **Plane** drop-down list, select **X-Z** to define the plane in which the wall lies. Enter the coordinates shown in Table 1.7.4.

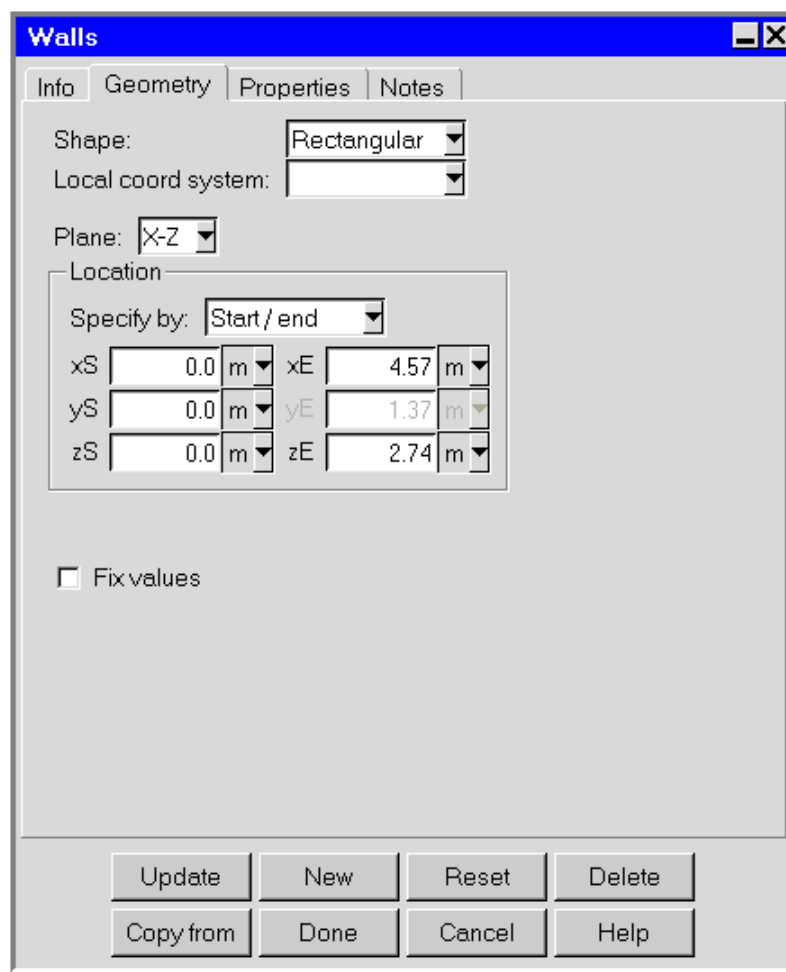


Figure 1.7.12: The Walls Panel (Geometry Tab)

Table 1.7.4: Coordinates for the Floor

xS	0.0	xE	4.57
yS	0.0	yE	—
zS	0.0	zE	2.74

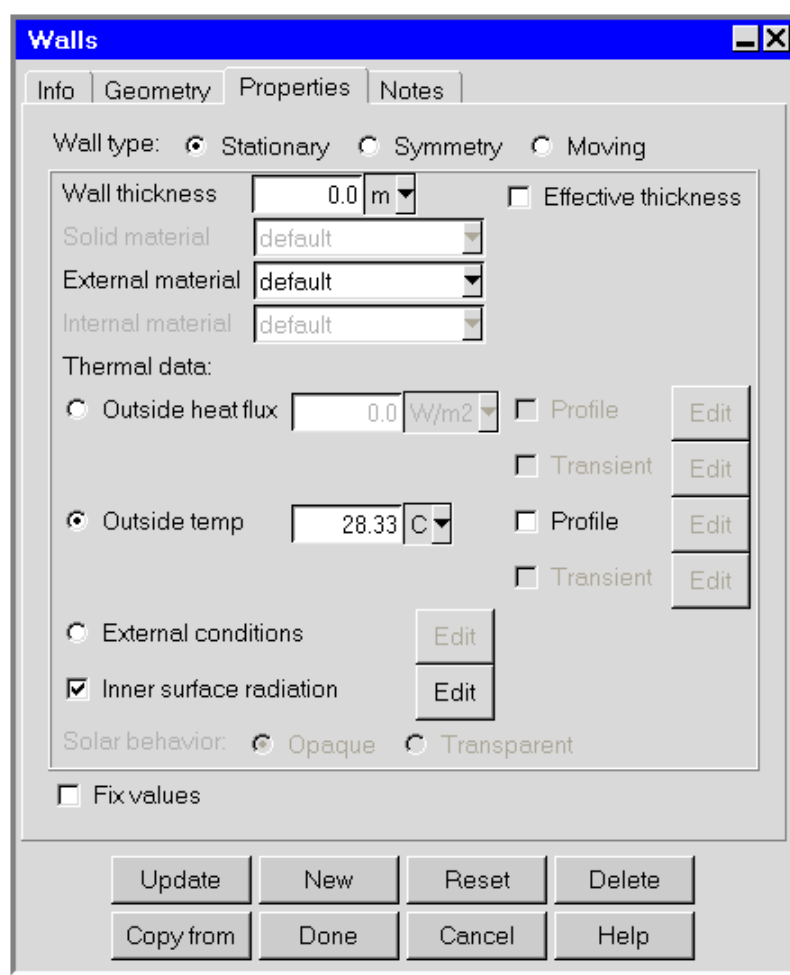


Figure 1.7.13: The Walls Panel (Properties Tab)

Click the **Properties** tab. Specify that the wall is at a constant temperature by selecting **Outside temp** under **Thermal data** and entering a value of 28.33.

Enable radiative heat transfer for the wall. Select **Inner surface radiation** and click **Edit** to open the **Radiation specification** panel (Figure 1.7.14). Select **All objects** and click **Accept**.

In the **Walls** panel, click the **Info** tab and change the default wall name to **floor** in the **Name** field. Click **Done** to accept all the changes and close the panel.

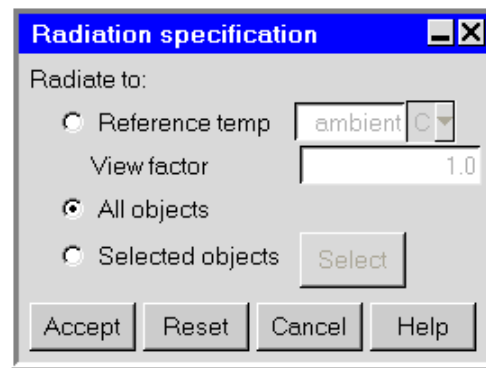





Figure 1.7.14: The Radiation specification Panel

Next, you will create the wall representing the ceiling by copying the first wall (**floor**) to the appropriate location. Select the **floor** object and click the  button to open the **Copy wall floor** panel (Figure 1.7.15). Alternatively, you can right-click the **floor** object item to open the panel.

Select **Translate** and specify a **Y** offset of 2.74. Click **Apply** to complete the process and display the copy (**floor.1**) in the **Model manager** window.

In the wall **Edit** panel for **floor.1**, change the default **Name** to **ceiling**. Click **Apply** to accept the changes.

Adding the Back and Front Walls

To construct the back wall, click the  button to create a new wall, and then click the  button to open the **Walls** panel.

In the **Geometry** tab, select **Y-Z** under **Plane** to define the plane in which the wall lies. Enter the coordinates shown in Table 1.7.5 for the wall.

Follow the procedure described above for the floor to specify an **Outside temp** of 28.33 and **Inner surface radiation** on **All objects**. Click **Done** in the **Walls** panel to accept the changes and close the panel. In the wall **Edit** panel, change the default **Name** to **wall-back**.

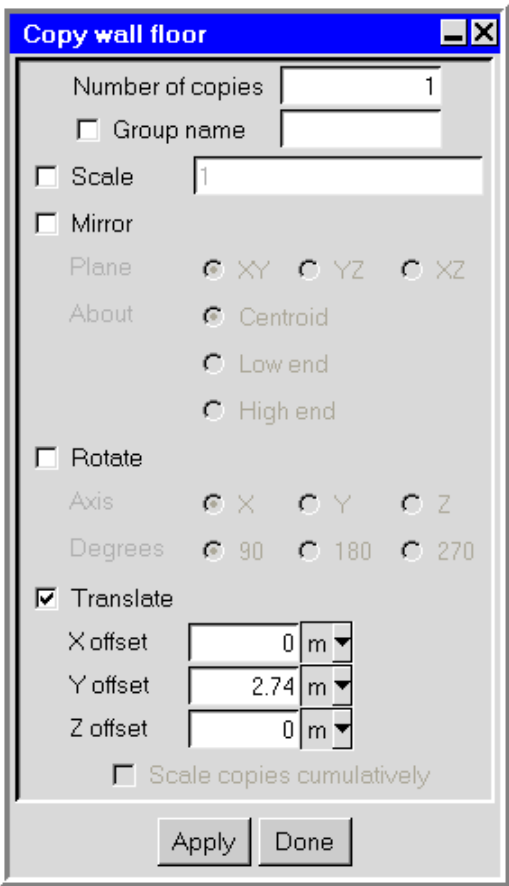




Figure 1.7.15: The Copy wall floor Panel

Table 1.7.5: Coordinates for the Back Wall

xS	0.0	xE	—
yS	0.0	yE	2.74
zS	0.0	zE	2.74

Next, follow the procedure described above for copying the floor to create the ceiling to copy the back wall to the front wall. Copy **wall-back** with an **X offset** of 4.57, a **Y offset** of 0, and a **Z offset** of 0. Click **Done** to accept the changes and close the panel. In the wall **Edit** panel, change **wall-back.1** to **wall-front** in the **Name** field. (Note that you will not be able to see the entire name in the **Name** field.)

Adding the Right and Left Walls

To construct the right wall, click the  button to create a new wall, and then click the  button to open the **Walls** panel.

In the **Geometry** tab, select **X-Y** and enter the coordinates shown in Table 1.7.3.

Table 1.7.6: Coordinates for the Right Wall

xS	0.0	xE	4.57
yS	0.0	yE	2.74
zS	0.0	zE	—

Follow the procedure described above for the floor to specify an **Outside temp** of 28.33 and **Inner surface radiation** on **All objects**. Click **Done** in the **Walls** panel to accept the changes and close the panel. In the wall **Edit** panel, change the default **Name** to **wall-right** and click **Accept**.

Next, follow the procedure described above for copying the floor to create the ceiling to copy the right wall to the left wall. Copy **wall-right** with an **X offset** of 0, a **Y offset** of 0, and a **Z offset** of 2.74. Click **Done** to accept the changes and close the panel. In the wall **Edit** panel, change **wall-right.1** to **wall-left** in the **Name** field and click **Accept**. The finished model should appear as shown in Figure 1.7.16.

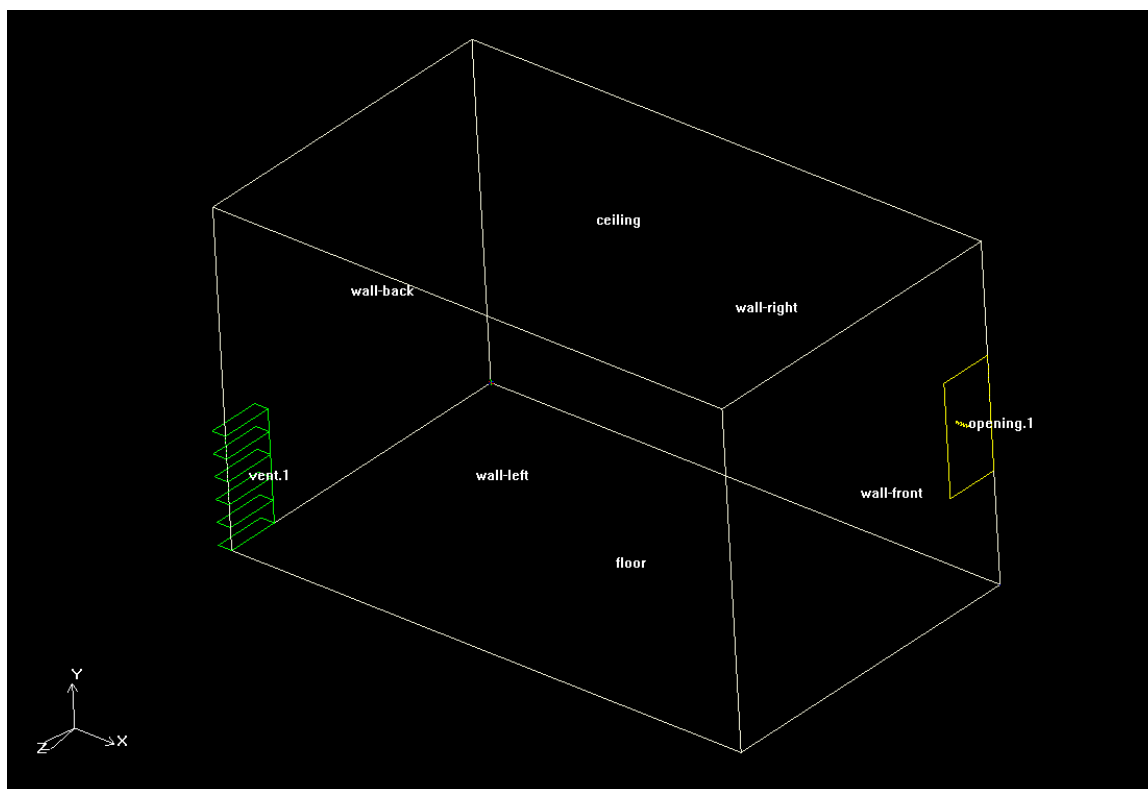


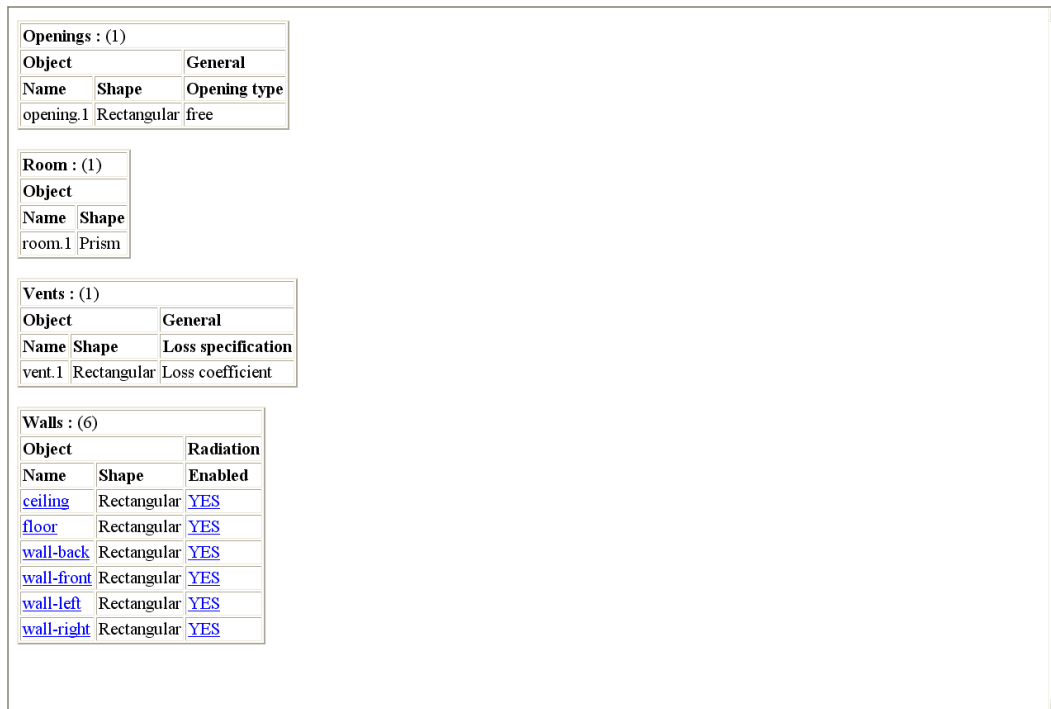
Figure 1.7.16: Room with Intake Opening, Exhaust Vent, and Walls

1.7.7 Generating a Summary

Once you have completed your model, you can display a summary of the model by selecting **Summary (HTML)** in the **View** menu.

View→**Summary (HTML)**

The HTML version of the summary will be displayed in your web browser. **Airpak** will automatically launch your web browser (Netscape or IE), as shown in Figure 1.7.17.



Openings : (1)		
Object		General
Name	Shape	Opening type
opening.1	Rectangular	free

Room : (1)	
Object	
Name	Shape
room.1	Prism

Vents : (1)		
Object		General
Name	Shape	Loss specification
vent.1	Rectangular	Loss coefficient

Walls : (6)		
Object		Radiation
Name	Shape	Enabled
ceiling	Rectangular	YES
floor	Rectangular	YES
wall-back	Rectangular	YES
wall-front	Rectangular	YES
wall-left	Rectangular	YES
wall-right	Rectangular	YES

Figure 1.7.17: Summary of the Model

1.7.8 Creating a Mesh

Once you are satisfied with your model, you can generate a mesh for it. First, you will create a default mesh, then you will display the mesh, and finally you will refine the mesh.

1.7.9 Changing the Meshing Priority

You will need to change the order in which the objects in the room are meshed so that room objects in contact with the walls will receive a higher priority than the walls in the meshing process. Click **Edit priorities** in the **Model** menu to open the **Object priority** panel (Figure 1.7.18).

Model → Edit priorities

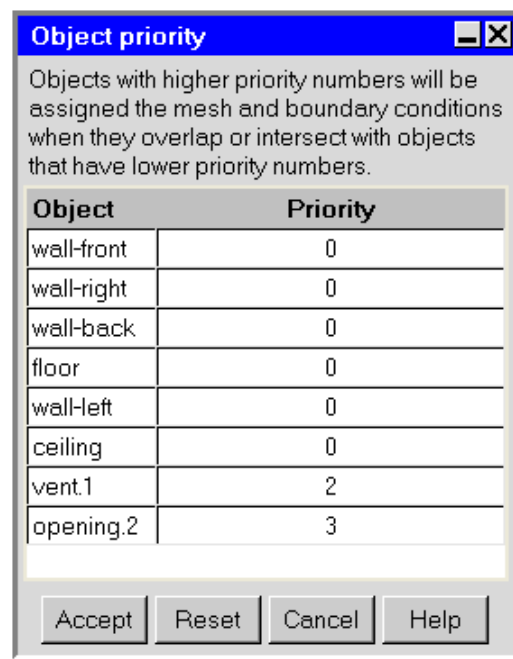


Figure 1.7.18: The Object priority Panel


Enter a value of 0 for floor, ceiling, wall-back, wall-front, wall-right, and wall-left.

Click **Accept** to modify the priorities and close the panel.

Creating a Coarse Mesh

You will now generate a coarse mesh for your model. To create a mesh, select **Generate mesh** in the **Model** menu.

Model—→Generate mesh

Airpak will open the Mesh control panel, as shown in Figure 1.7.19. Alternatively, you can click the  button to open the Mesh control panel.

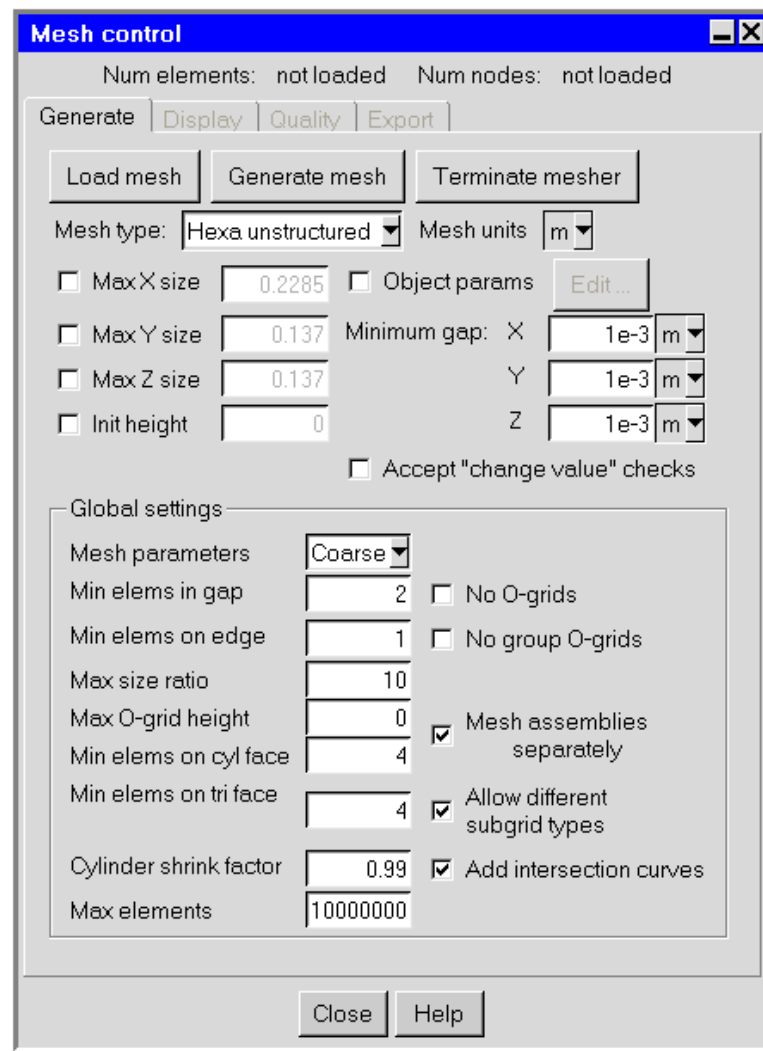


Figure 1.7.19: The Mesh control Panel

Turn off the Max X size, Max Y size, and Max Z size options. In the Mesh parameters drop-down list under Global settings, select Coarse.

Click **Generate mesh** to generate a mesh using the coarse mesh parameters to represent the geometry.

The **Message** window will display the results of the mesh generation procedure, including the number of elements or “hexas” (hexahedral brick elements), and information regarding the quality of the elements in the mesh. The number of mesh elements and the number of nodes in the mesh will also be displayed in the **Mesh control** panel. The **Message** window will also report the volumetric flow rate required to change the air in the room once per hour.

Displaying the Mesh

Airpak offers several options for displaying the mesh, including views on surfaces, within the entire room volume, and through a plane intersecting the room (a “plane-cut” view).

Viewing a Plane Cut Through the Mesh

To simplify the display of the mesh within the room, it is often useful to create a plane-cut view (i.e., a zero-thickness slice through the mesh).

Click the **Display** tab in the **Mesh control** panel and select the **Cut plane** option. Keep the default settings of (0, 0, 1.37) for (PX, PY, PZ) and (0, 0, 1) for (NX, NY, NZ) for the **Plane location**. These settings will result in a mesh display on an x - y plane passing through the point (0, 0, 1.37).

Display the mesh on the plane cut by selecting **Display mesh**. Move the slider bar below the **Plane location** coordinates to adjust the plane cut so that it is at the opening (Figure 1.7.20).

In general, when you view a mesh, there are a few things to look for:

In general:

- A “good” mesh should include a minimum of three nodes (four elements) in the channels between objects that are close to one another.
- The aspect ratio of the largest to smallest elements should not be less than 1 to 10 (0.1).
- Elements should be graded away from object surfaces (i.e., there should be no large jumps in size for elements near surfaces in the model).

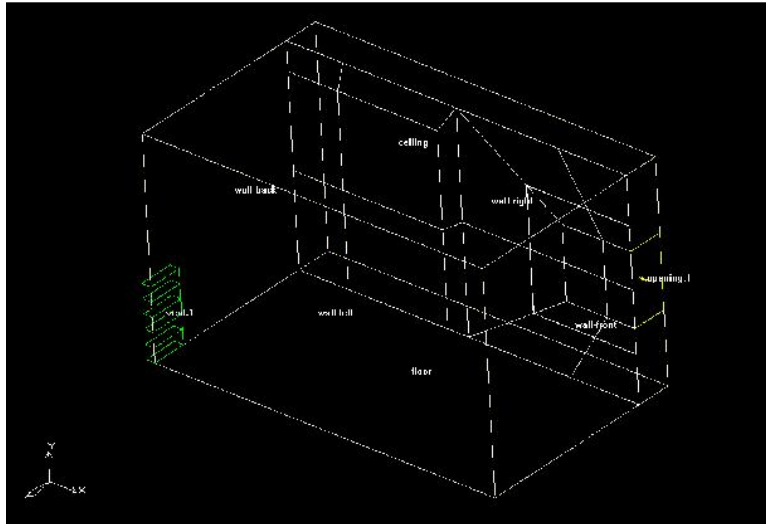


Figure 1.7.20: A Plane-Cut View of the Mesh

Refining the Mesh

For this example, the coarse mesh shown in Figure 1.7.20 is not sufficient to generate an accurate solution. Airpak provides tools to improve the mesh. You will change the global specification of the maximum element size to refine the mesh and improve the mesh quality.

To specify the maximum element size throughout the computational domain, click on the **Generate** tab in the **Mesh control** panel. Select the **Max X size**, **Max Y size**, and **Max Z size** options in the **Mesh control** panel, and enter values of 0.25, 0.25, and 0.25, respectively. Under **Global settings**, select **Normal** in the **Mesh parameters** drop-down list.

Click **Generate mesh** to generate the refined mesh. The refined mesh (Figure 1.7.21) will be displayed in the graphics window automatically because **Display mesh** is still selected in the **Display** section of the **Mesh control** panel. You can use the two square boxes to advance the plane cut and view the mesh throughout the model.

To turn off the mesh display, click the **Display** tab at the top of the **Mesh control** panel and deselect the **Display mesh** option. Then Close the **Mesh control** panel.

1.7.10 Checking the Flow Regime

Before starting the solver, you will first review estimates of the Reynolds and Peclet numbers to check that the proper flow regime is being modeled.

Check the values of the Reynolds and Peclet numbers by opening the **Solver setup** panel (Figure 1.7.22).

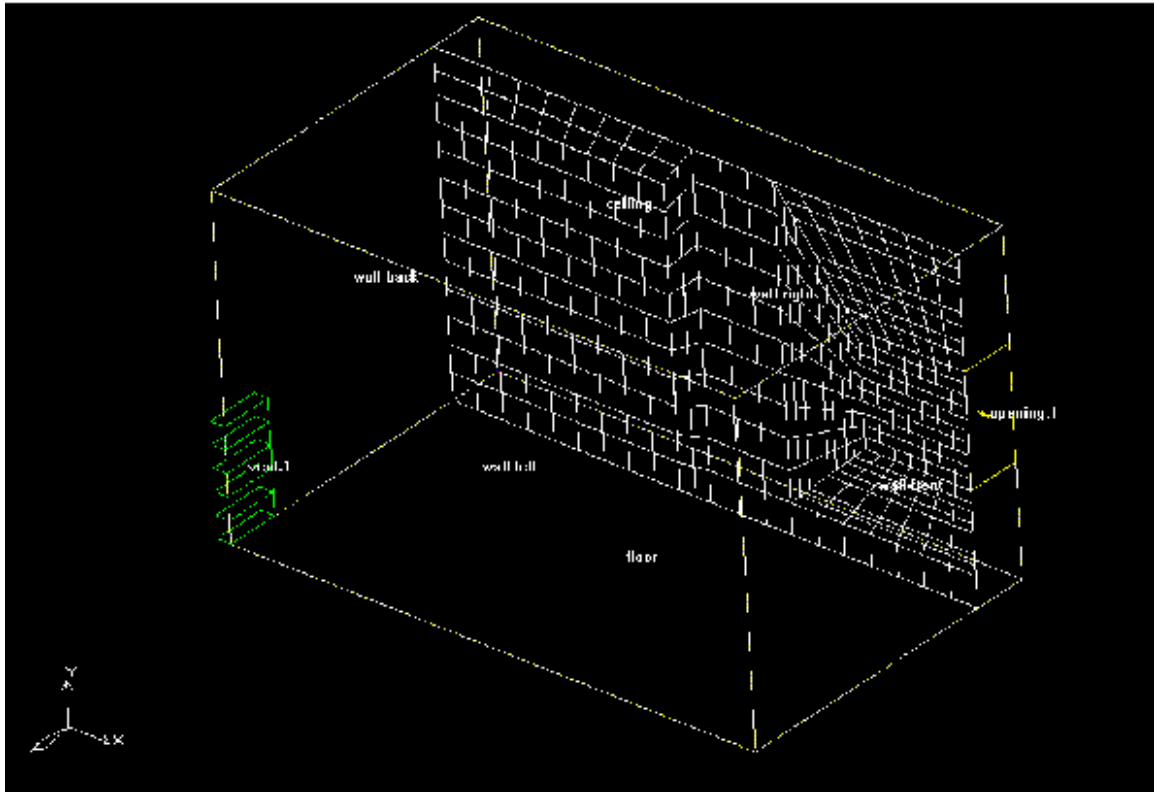


Figure 1.7.21: Plane-Cut View of the Refined Mesh

Solve—>Settings—>Basic

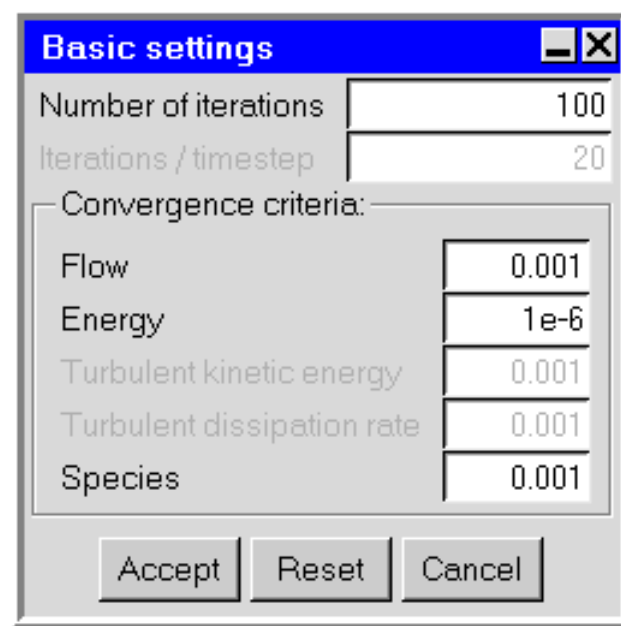


Figure 1.7.22: The Basic settings panel

Click the **Reset** button. Check the values printed to the **Message** window. The Reynolds and Peclet numbers are approximately 31,000 and 23,000, respectively, so the flow is turbulent. Since you are currently modeling turbulent flow, no changes are required. The **Message** window will also report that the initial air velocity has been reset to -10^{-4} times gravity. This modification improves the convergence of natural convection calculations.

Click **Accept** to save the new solver settings.

1.7.11 Saving the Model

You should save your model (including the mesh) before you calculate a solution for your model. You must save the project for your current session in order to continue your analysis in a future **Airpak** session. To save the project, select **Save project** in the **File** menu.



File—>Save project

1.7.12 Calculating a Solution

After the mesh has been generated and refined, **Airpak** is ready to solve the model.

Set the Solution Parameters

To review the default solution parameters, double click the **Basic settings** item under the **Solution settings** node of the **Model manager** window.

 **Solution settings** →  **Basic settings**

Airpak will open the **Basic settings** panel, as shown in Figure 1.7.22.

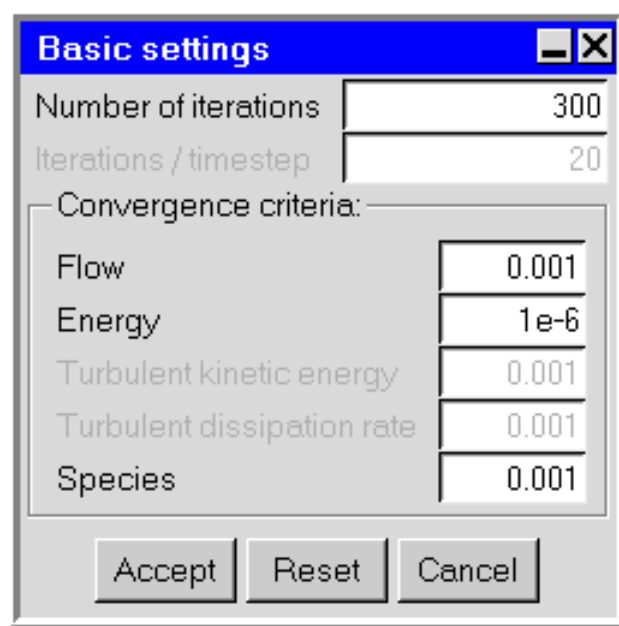


Figure 1.7.23: The Basic settings Panel

Increase the **Number of iterations** to 300 and keep the default values for the remaining settings (Figure 1.7.23). Click **Accept** to accept the solution parameters.

Calculating

You will use the **Solve** panel (Figure 1.7.24) to start the solver. To open the **Solve** panel, select **Run solution** in the **Solve** menu.

Solve → **Run solution**

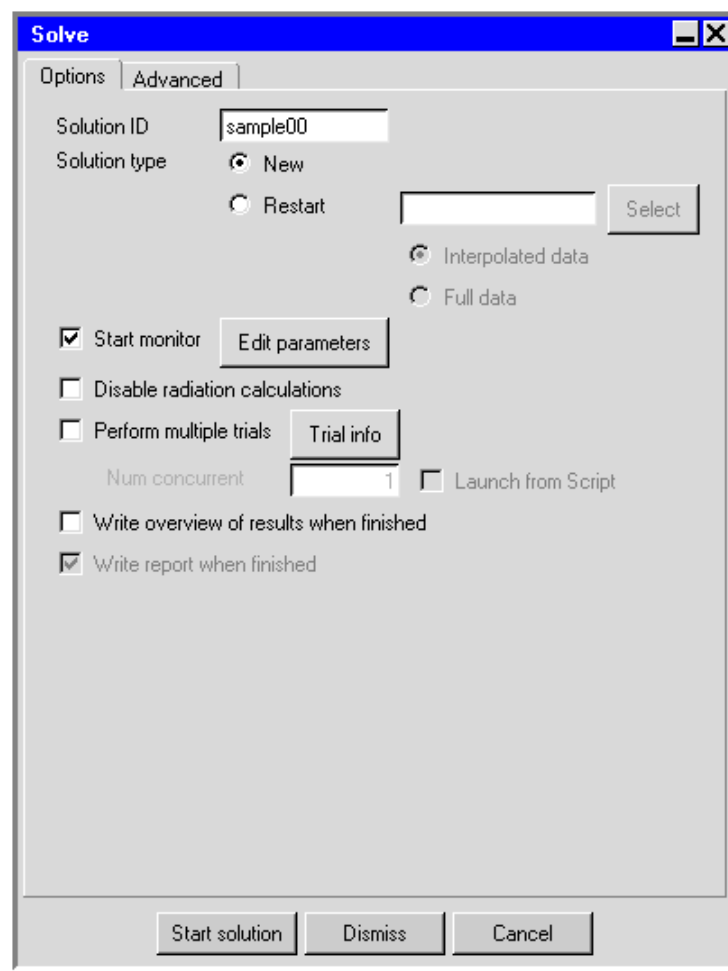


Figure 1.7.24: The Solve Panel (Options tab)

The default **Solution ID** consists of the project name and a sequential two-digit suffix (starting value = 00). To modify the **Solution ID**, you can type a new name in the **Solution ID** text entry box. In this example, you will keep the default ID.

Click **Start solution** to start the solver. The solver will start the calculation, and **Airpak** will open the **Solution residuals** graphics display and control window, where it will display the convergence history for the calculation.

The solution is converged to the default tolerances after a total of about 100 iterations. At this point, your residual plot will look something like Figure 1.7.25. Note that the exact number of iterations required for convergence may vary on different computers. Also the actual values of the residuals may differ slightly on different machines, so your plot may not look exactly the same as Figure 1.7.25.

Click **Done** to close the **Solution residuals** window.

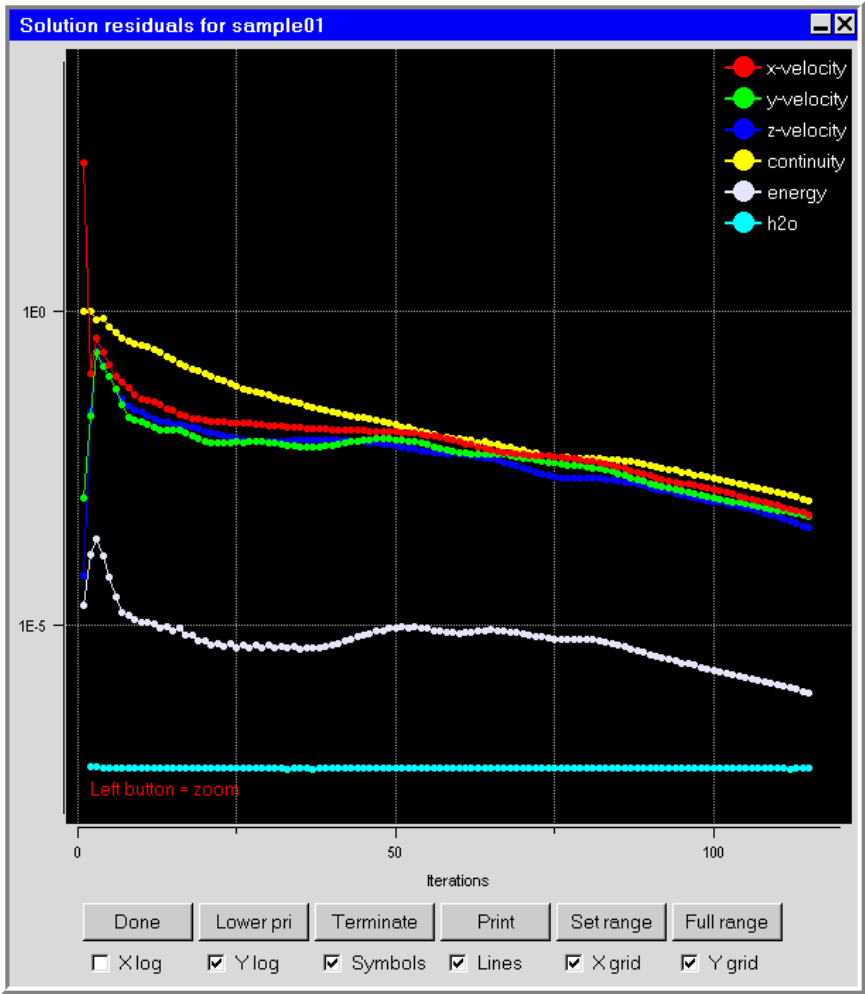


Figure 1.7.25: Residuals After Convergence

1.7.13 Examining the Results

Airpak provides a number of ways to view and examine the solution results, including:

- object-face views
- plane-cut views
- isosurface views
- point views
- variation plots

The following sections illustrate how to generate and display each view.

Object-Face Views

An object-face view allows you to examine the distribution of a solution variable on one or more faces of an object in the model. To generate an object-face view, you must select the object and specify both the variable to be displayed (e.g., temperature) and the attributes of the view (e.g., shading type).

You will use the **Object face** panel (Figure 1.7.26) to create a solid-band object-face view of the temperature distribution on the vent. To open the **Object face** panel, select **Object face** in the **Post** menu.

Post→**Object face**

To specify the vent as the object on which the results will be displayed, select **vent.1** in the **Object** drop-down list, and click **Accept**.

To specify the variable to be displayed and the attributes of the view, turn on the **Show Contours** option and click **Parameters**. This will open the **Object face contours** panel, shown in Figure 1.7.27.

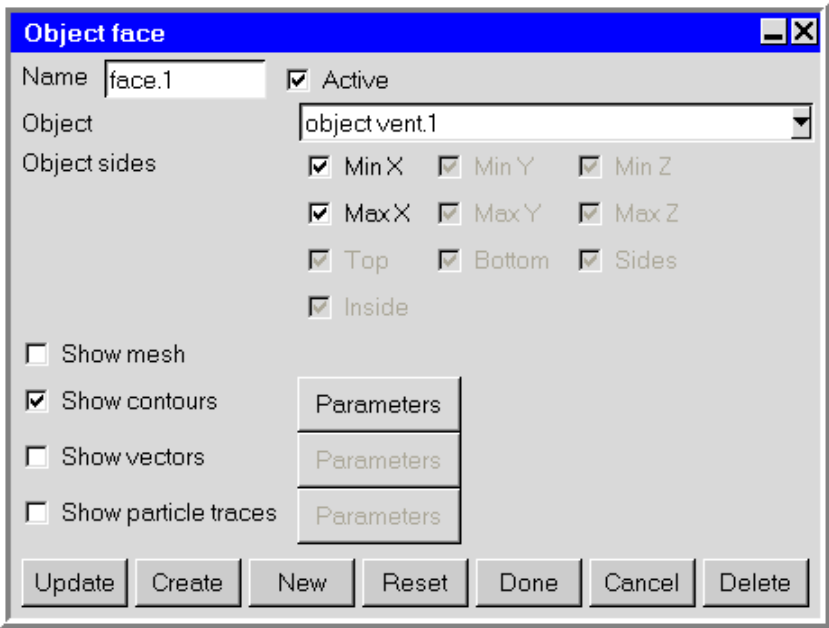


Figure 1.7.26: The Object face Panel

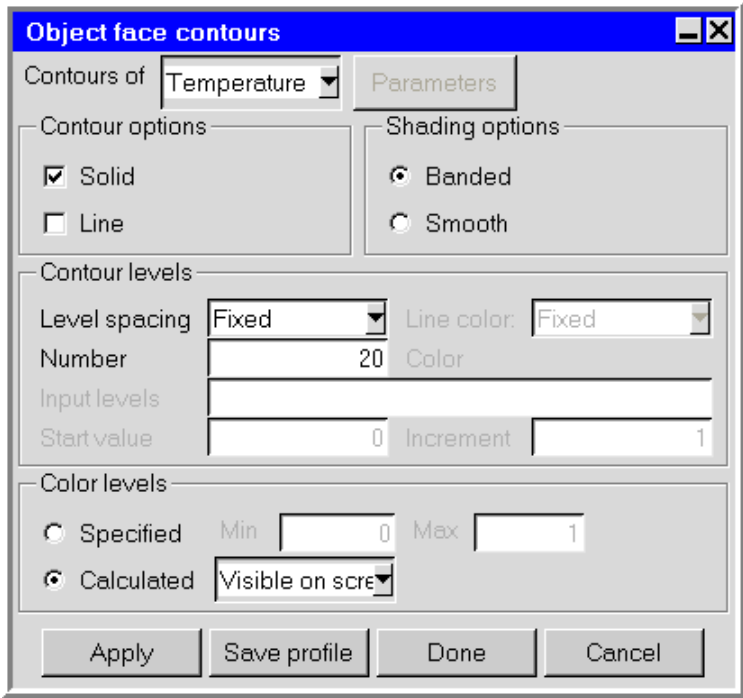


Figure 1.7.27: The Object face contours Panel

The default **Variable** is **Temperature**, so no change is needed. Keep the default selections of **Solid** under **Contour options** and **Banded** under **Shading options**. Keep the default option of **Fixed** for **Level spacing**. Under **Color levels**, choose **Visible on screen** from the **Calculated** drop-down list and click **Done** to accept the settings and close the **Object face contours** panel.

Click **Create** in the **Object face** panel to create and display the object face by loading the data. The solid-band temperature contours on the vent will be displayed in the graphics window. Select **Orient negative X** in the **Orient** menu and then use the **Zoom in** feature on the menu to get a close-up view of the contours on the vent (Figure 1.7.28).

In addition to the solid contour bands on the vent, **Airpak** displays a legend showing a vertical color-band spectrum and the associated temperatures in the graphics window. The spectrum includes the entire range of temperatures applicable to the solution.

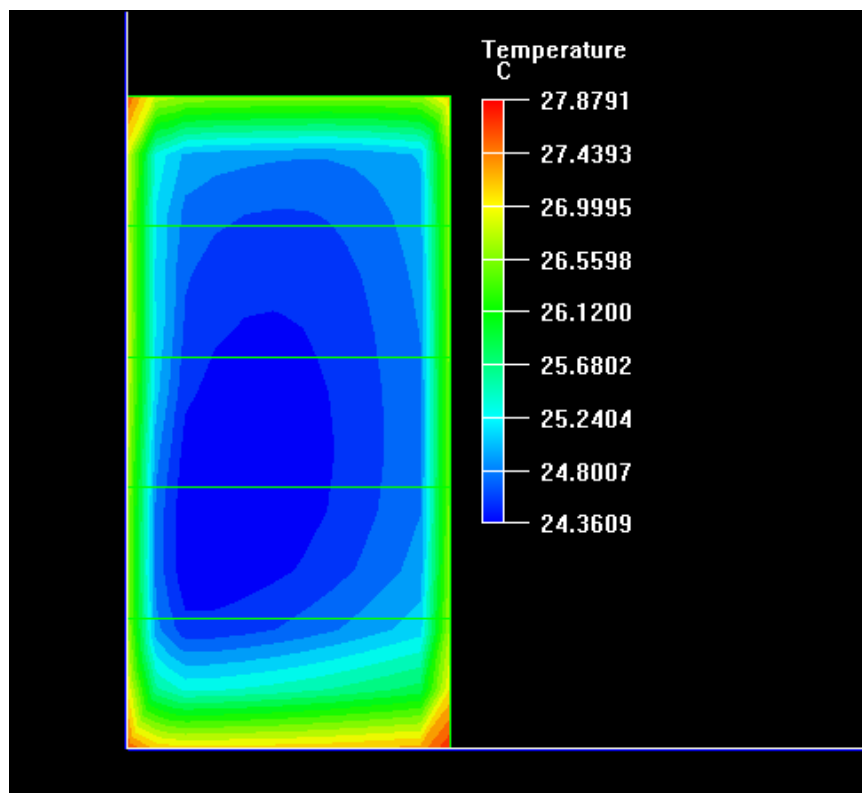


Figure 1.7.28: Contours of Temperature on the Vent



Before continuing, you should deactivate the line band contour plot so that it will not obstruct your view of subsequent postprocessing objects. In the **Object face** panel, deselect **Active** and click **Done**. If the **Object face** panel is not visible on your screen, double click the **face.1** item under the **Postprocessing** node of the

Model manager window to bring the Object face panel to the foreground. Once deactivated, the face.1 item will appear in the Inactive node of the Model manager window.

Plane-Cut Views

Plane-cut views allow you to observe the variation in a solution variable across the surface of a plane.

Select Home position in the Orient menu to return the room to its default orientation (front view).

You will use the Plane cut panel (Figure 1.7.29) to view the direction and magnitude of velocity across a horizontal plane near the middle of the room. To open the Plane cut panel, select Plane cut in the Post menu.

Post → Plane cut

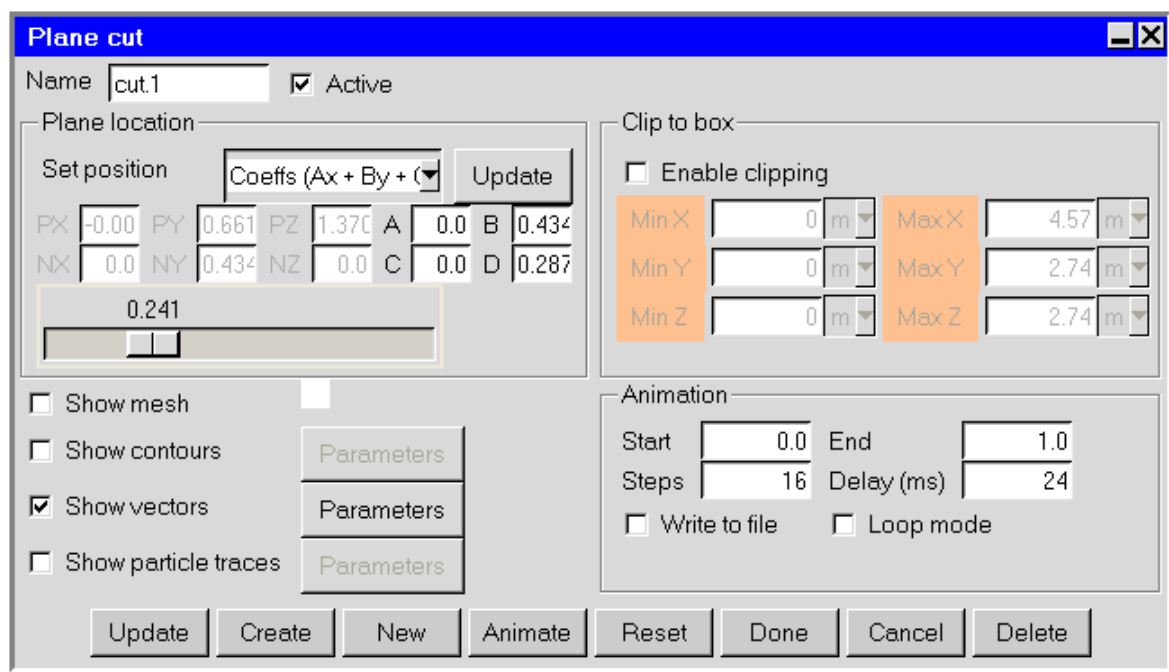


Figure 1.7.29: The Plane cut Panel

To specify a horizontal plane cut, select Horizontal - screen select in the Set position drop-down list under Plane location. To create a cut plane, click a point in the graphics window about 1/3 from the top of the vent using the left mouse button. Turn on the Show vectors option and click Done. Select Orient positive Y in the Orient menu to see a top-down view of the room. Then click on Scale to fit in the Orient menu to see the view of the room, as shown in Figure 1.7.30.

Note that depending on where you click in the graphics display, you will see different velocity vectors. If you wish to see Figure 1.7.30 exactly, refer to Figure 1.7.29 and enter the displayed coordinates for (PX, PY, PZ) and (NX, NY, NZ).

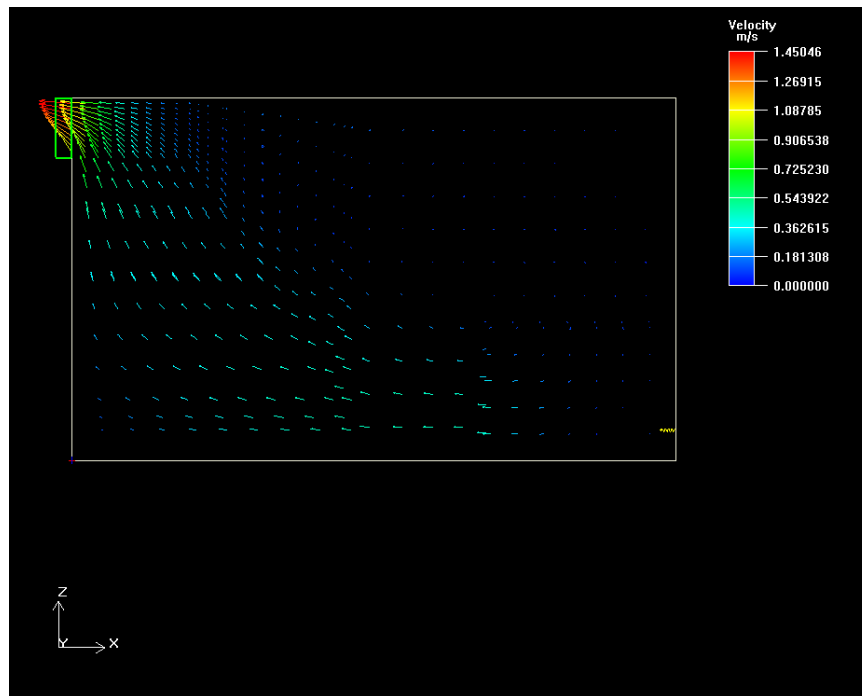


Figure 1.7.30: Velocity Vectors (Plane-Cut View)



To deactivate the plane-cut view, right click the **cut.1** item under the **Postprocessing** node of the **Model manager** window, deselect **Active** in the pull-down menu. Once deactivated, the **cut.1** item will appear in the **Inactive** node of the **Model manager** window.

Isosurface Views

Isosurface views display surfaces across which one of the primary or derived variables exhibits a single value. First, select **Isometric view** from the **Orient** menu to display the isometric view of the room.

Next, you will use the **Isosurface** panel (Figure 1.7.31) to view the surface across which all points have a temperature of 25°C. To open the **Isosurface** panel, select **Isosurface** in the **Post** menu.

Post → Isosurface

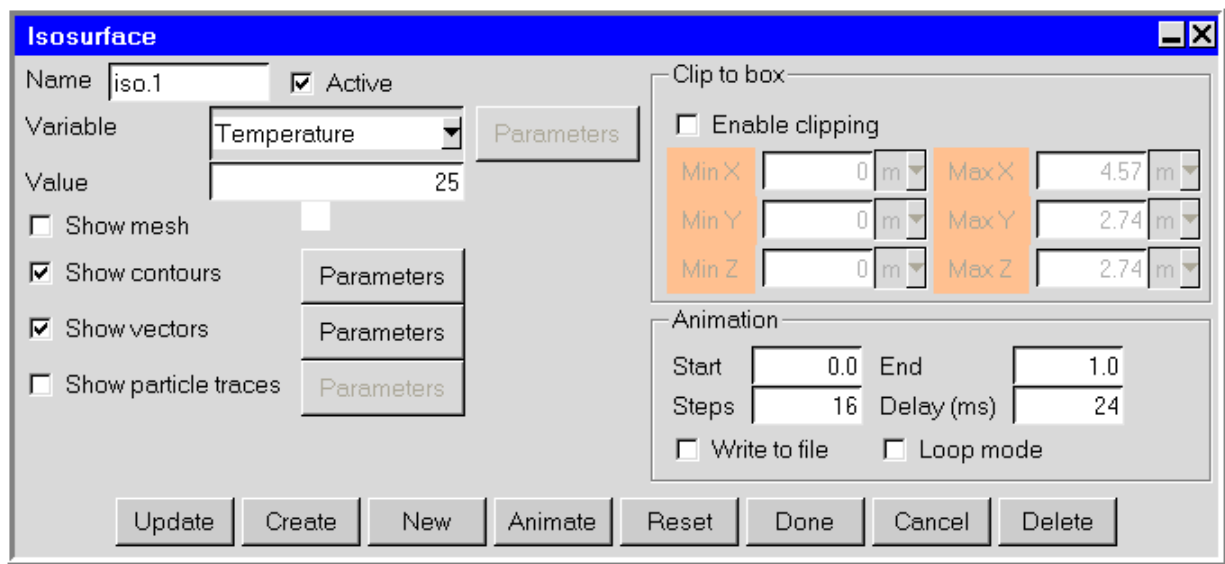


Figure 1.7.31: The Isosurface Panel

In the **Isosurface** panel, keep the default **Variable** of **Temperature**. Enter a value of 25 next to **Value** to specify an isosurface at 25°C. Select the **Show contours** and **Show vectors** options. Click **Done** to create the isosurface, an isometric view of which is shown in Figure 1.7.32.

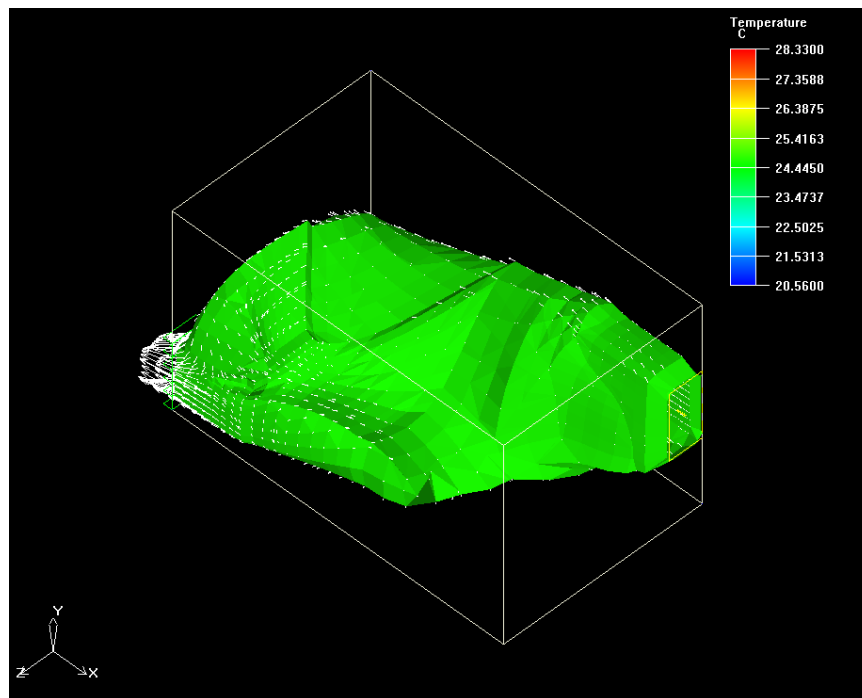


Figure 1.7.32: Isosurface of Temperature = 25 Degrees

i To deactivate the isosurface view, right click the `iso.1` item under the **Postprocessing** node in the **Model manager** window and deselect the **Active** option in the pull-down menu. Once deactivated, the `ice.1` item will appear in the **Inactive** node of the **Model manager** window.

Point Views

Point views allow you to probe the computational domain to sample the values of one of the primary or derived solution variables at any point.

You will use the **Point** panel (Figure 1.7.33) to probe for temperature. To open the **Point** panel, select **Point** in the **Post** menu.

Post → Point

Keep the default **Variable** of **Temperature**. In the **Position** drop-down list, enter `1 1 1` to create a starting point offset from the origin. Select **Leave trail** to enable tracing. Keep the default **Point size** of **4**. Click **Done** to create the point and close the panel.

Airpak draws the initial (blue) sample point at the x , y , and z coordinates specified (`1 1 1`) and displays the temperature at that point on the postprocessing **Edit** window next to **Value**, as shown in Figure 1.7.34.

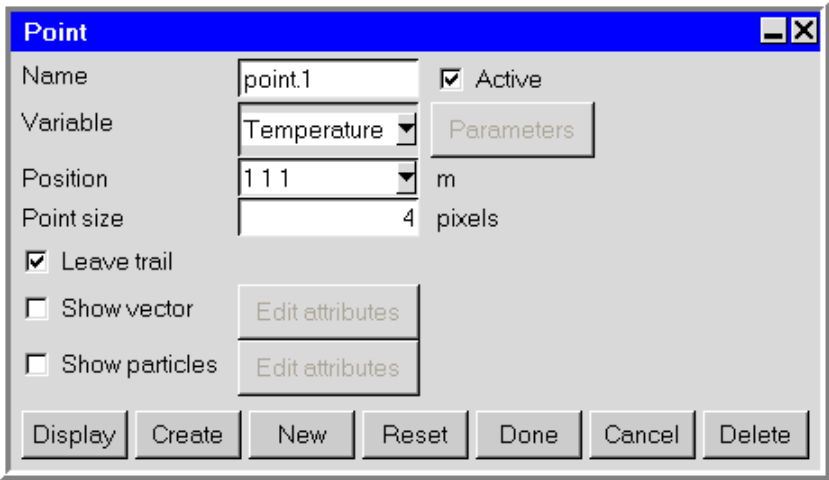


Figure 1.7.33: The Point Panel

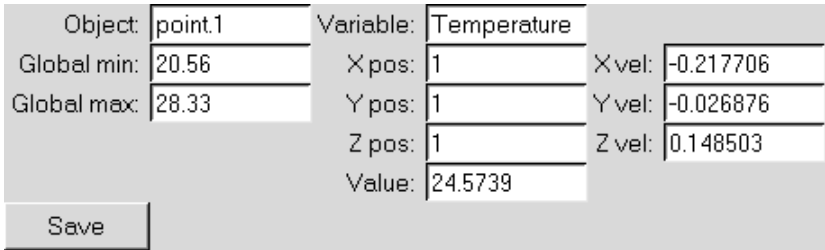


Figure 1.7.34: The Postprocessing Edit Window for a Point View

You can sample the temperature at a different location in the domain by moving the point. Select the point in the graphics window by holding down the <Shift> key on the keyboard and using the left mouse button to select the point. Drag the point to the new location using the left mouse button. As the point moves, it will leave behind a trace colored according to the temperature legend in the graphics window. The postprocessing **Edit** window displays the value of the temperature at the current location (**Value**), and the minimum (**Global min**) and maximum (**Global max**) values of temperature in the computational domain.



To deactivate the point trace before moving on, right click the **point.1** item under the **Postprocessing** node in the **Model manager** window, and deselect the **Active** option in the pull-down menu. Once deactivated, the **point.1** item will appear in the **Inactive** node of the **Model manager** window.

Variation Plots

Airpak variation plots allow you to view the variation of any one of the primary or derived solution variables along a line through the computational domain.

First, select **Home position** in the **Orient** menu to return the room to its default orientation (front view).

You will use the **Variation plot** panel (Figure 1.7.35) to view temperature along a horizontal line in the middle of the room. To open the **Variation plot** panel, select **Variation plot** in the **Post** menu.

Post → **Variation plot**

Keep the default **Variable** of **Temperature**. Specify a variation plot line normal to the screen at a point by clicking the **From point objects** button and entering the values 0.1, 1.0, 2.5 under **Point** and 0, 0, 1 under **Direction**. Click **Create** in the **Variation plot** panel to create a variation plot, as shown in Figure 1.7.36. Click **Dismiss** to close the panel.

Airpak draws the variation plot in a separate window (called the **Variation of Temperature** graphics display and control window in this example), and displays the line on which the variation plot is calculated in the graphics window. You can use the mouse to rotate the room and view the variation plot line from various angles. Click **Done** to close the **Variation of Temperature** window and remove the variation plot line from the graphics window. Note that you can also create a plot by clicking the **From screen** button in the **Variation plot** panel and then clicking a point in the graphics window.

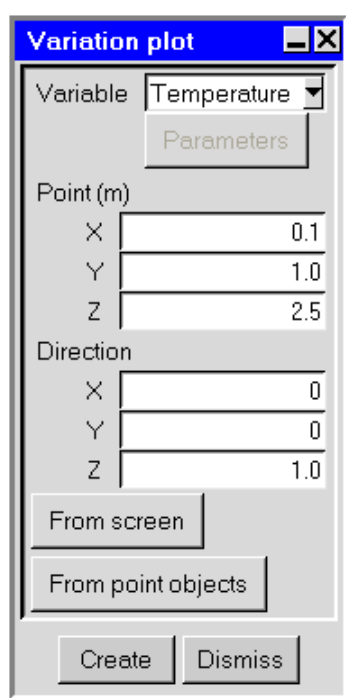


Figure 1.7.35: The Variation plot Panel

Saving Postprocessing Objects

You can save the objects (e.g., plane-cut views, isosurfaces) generated during the post-processing session to a file, so that you can reuse them the next time you view the model in Airpak. Select **Save post objects to file** in the **Post** menu.

Post → Save post objects to file

Click **Save** in the resulting **File selection** dialog box to save the file with the default name `post_objects`.



Saving postprocessing objects often requires a large amount of disk space.

1.7.14 Generating Reports

Airpak allows you to generate, view, and print reports detailing the results of the simulation. Reports regarding primary and derived solution variables such as temperature, flow rate, heat flux, and heat transfer coefficient can be generated either for individual objects or for collections of objects.

You will use the **Full report** panel (Figure 1.7.37) to generate an on-screen report of mass flow rate for the fan. To open the **Full report** panel, select **Full report** in the **Report** menu.

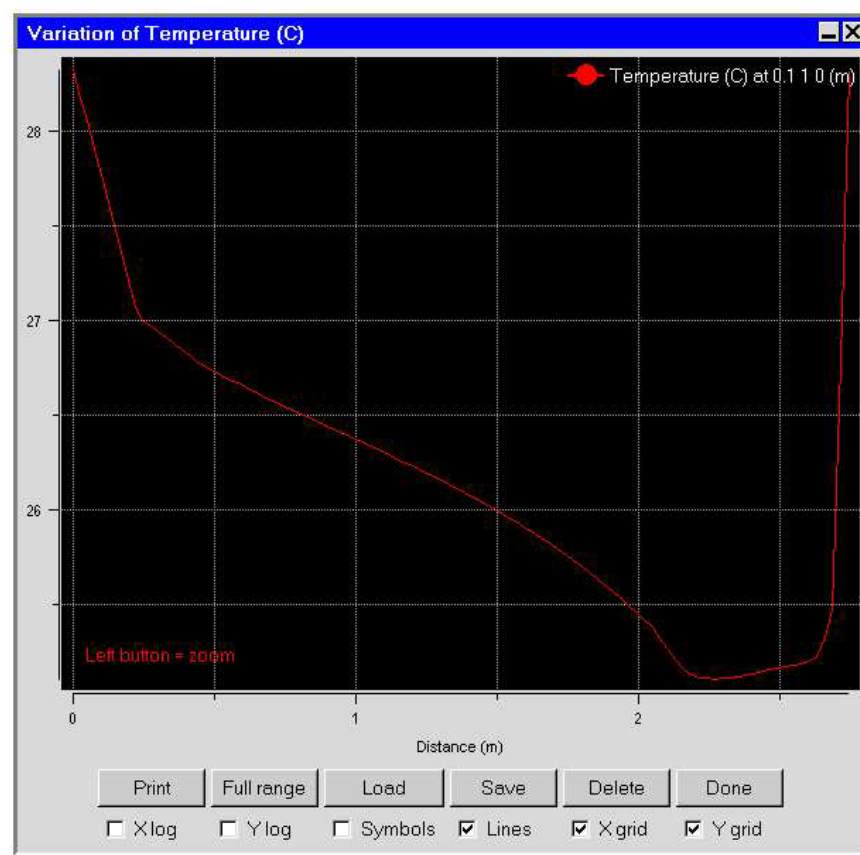


Figure 1.7.36: Variation of Temperature Plot

Report—>Full report

Figure 1.7.37: The Full report Panel

To specify mass flow rate as the variable to be reported, select **Mass flow** in the **Variable** drop-down list. Select the **Specific objects** option under **Report region** and select **vent.1** in the adjacent drop-down list, and click **Accept** to specify the fan as the object on which the results will be reported. Select the **Sides** option to create a report for the whole fan. Keep the **Write to window** option turned on to display the report in a window, and deselect the **Write to file** option. Click **Write** to generate the report. Airpak displays the report in a separate window (called the **Full report for Mass flow** window in this example), as shown in Figure 1.7.38. Airpak displays the surface area of the fan, and the average, maximum, and minimum values for the mass flow rate. Click **Done** to close the **Report on Mass flow** window.

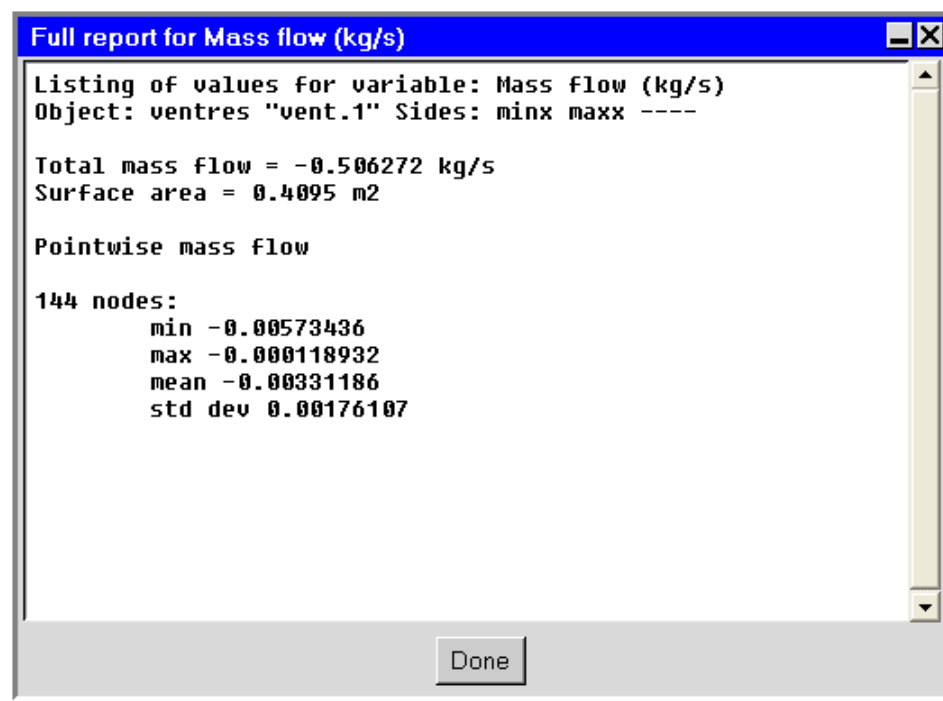


Figure 1.7.38: The Full report for Mass flow Window

1.7.15 Exiting From Airpak

When you are finished examining the results, you can end the **Airpak** session by clicking **Quit** in the **File** menu.

File→Quit

1.7.16 Summary

This example has been designed to show you how to use **Airpak** to solve a very simple problem. Example problems of increasing difficulty are solved in the **Airpak** Tutorial Guide, where the different modeling objects, physical models, and solution parameters that are available in **Airpak** are illustrated in greater detail.

The user interface for **Airpak** consists of a graphical interface with windows, menus, toolbars, and panels. An overview of the graphical interface, including information about the toolbars, menus, and panels is presented in this chapter. Details on using the mouse and keyboard and accessing the on-line help are also included.

- Section 2.1: The Graphical User Interface
- Section 2.2: Using the Mouse
- Section 2.3: Using the Keyboard
- Section 2.4: Quitting **Airpak**

2.1 The Graphical User Interface

Airpak's graphical user interface (GUI) consists of several major components: the menu bar, toolbars, control panels, the **Model manager** window, the **Message** window, and the graphics windows. When you use the GUI, you will be interacting with one of these components at all times within a single **Airpak** application window.

You interact with **Airpak** through the GUI by means of the mouse and the keyboard. To perform most operations in **Airpak**, you simply position the cursor of the mouse on the object or item you wish to act upon, and click the left mouse button.

You perform most tasks (e.g., file saving, object creation, object editing, etc.) using either the menu bar, the toolbars, or the **Model manager** window. Your work is displayed in the graphics window, where you can use the mouse to view various aspects of your model.

Information about the components of the GUI is presented in the following sections:

- Section 2.1.1: The Main Window
- Section 2.1.2: The **Airpak** Menus
- Section 2.1.3: The **Airpak** Toolbars
- Section 2.1.4: The **Model manager** Window
- Section 2.1.5: The Graphics Windows

- Section [2.1.6](#): The Message Window
- Section [2.1.7](#): The Edit Window
- Section [2.1.8](#): File Selection Dialog Boxes
- Section [2.1.9](#): Control Panels
- Section [2.1.10](#): Accessing On-line Help

2.1.1 The Main Window

When you start **Airpak**, the **Main** window (center) is displayed on the screen (Figure [2.1.1](#)). The **Main** window controls the execution of the **Airpak** program and has six primary components: the **Main Menu** bar (top), the **Model Display** window or graphics window (right), the **Model manager** window (left), the **Message** window (lower left), the **Edit** window (lower right), and several toolbars.

Resizing Airpak Windows

You can customize the appearance of the **Main** window by resizing any of the four major **Airpak** windows: the **Model manager** window; the **Message** window; the **Edit** window; and the **Model Display** graphics window.

To resize an **Airpak** window, click and hold down the left mouse button on the separator and drag the separator to the desired location. The slider bars for each **Airpak** window are located either to the lower right and/or the lower left of each window.

2.1.2 The Airpak Menus

The **Main Menu** bar (Figure [2.1.2](#)) contains twelve menus that are located along the top of the **Main** window. These menus (**File**, **Edit**, **View**, **Orient**, **Model**, **Tree**, **Macros**, **Solve**, **Post**, **Report**, **Windows**, and **Help**) are accessible at all times and allow you to access top-level **Airpak** functionality. When you select one of these menus in the **Main Menu** bar, a set of menu-specific options will be displayed. Some menu-specific options also have sub-options that you can choose from. In addition, note that each menu in the **Main Menu** bar has a keyboard shortcut so that the menu and its options can be accessed using the keyboard. For more information on using the keyboard in **Airpak**, see Section [2.3](#).

The File Menu

The **File** menu (Figure [2.1.3](#)) contains options for working with **Airpak** projects and project files. From this menu, you can open, merge, and save **Airpak** projects. In addition, you can import, export, compress, and decompress files relating to your **Airpak** model. There are also utilities designed to save or print your model geometries. A brief description

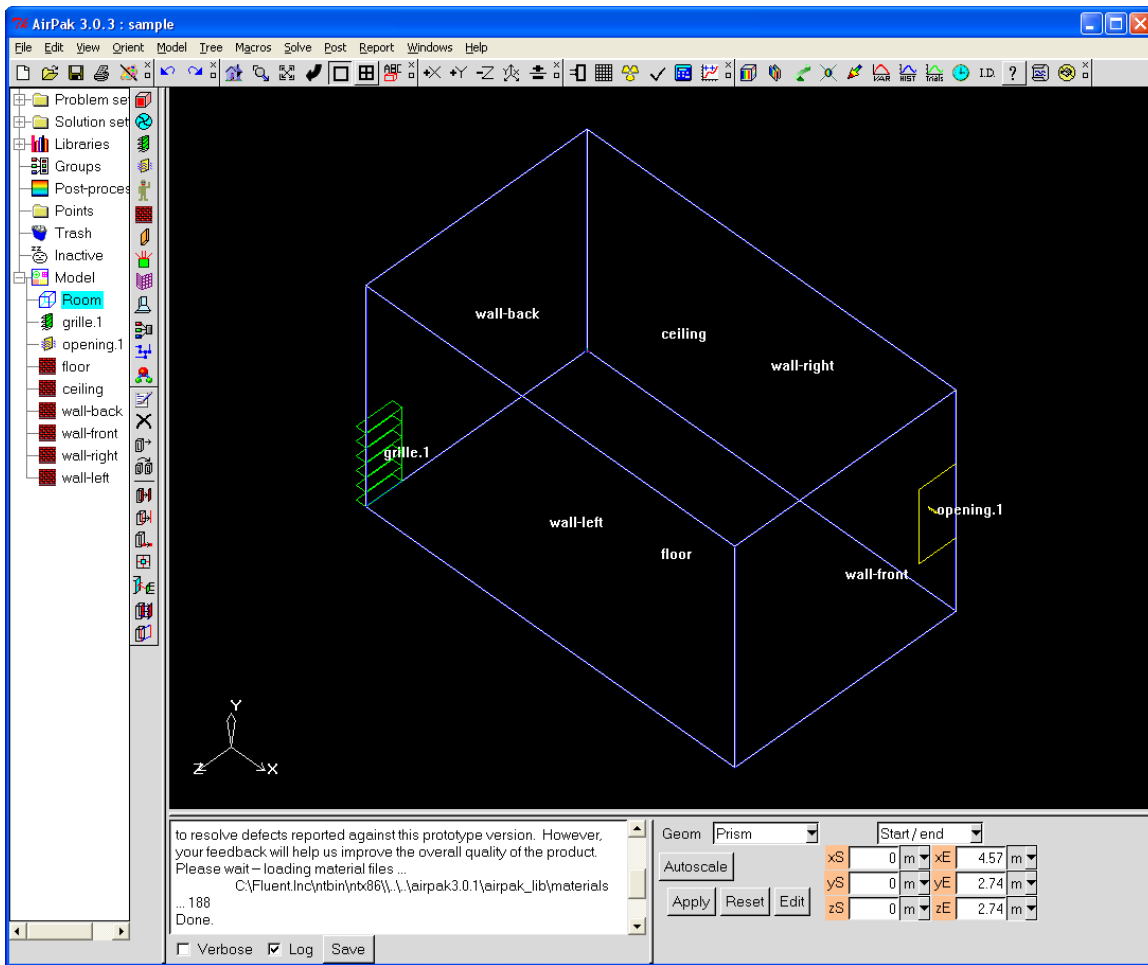


Figure 2.1.1: The Main Window

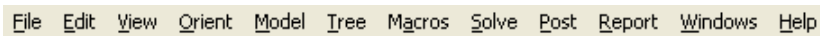


Figure 2.1.2: The Main Menu Bar

of the File menu options is provided below. See Chapter 3 for more information about reading, writing, and managing Airpak project files.

New project allows you to create a new Airpak project using the New project panel. Here, you can browse through your directory structure, create a new project directory, and enter a project name.

Open project allows you to open existing Airpak projects using the Open project panel. Here, you can browse through your directory structure, locate a project directory,

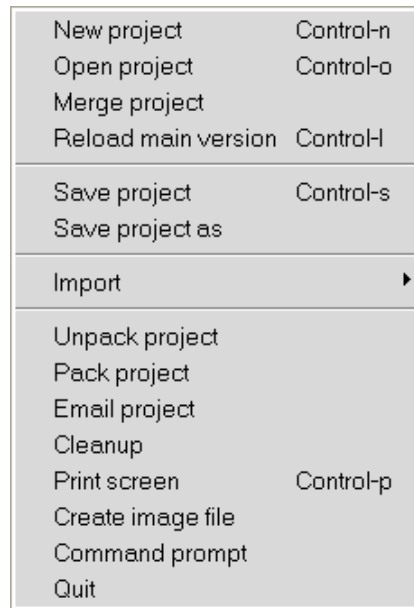


Figure 2.1.3: The File Menu

and either enter a project name, or specify an old project name from a list of recent projects. Additionally, you can specify a version name or number for the project.

Merge project allows you to merge an existing project into your current project using the **Merge project** panel.

Reload main version allows you to re-open the original version of the **Airpak** project when your project has multiple versions. See [Chapter 6](#) for more information.

Save project saves the current **Airpak** project.

Save project as allows you to save the current **Airpak** project under a different name using the **Save project** panel.

Import provides options to import IGES, DXF, and tetin file geometries into **Airpak**. You also can import DWG and IFC files, as well as comma separated values or spreadsheet format (CSV) using this option. See [Chapter 4](#) for more information about importing files.

Export allows you to export your work to IGES files as well as comma separated values or spreadsheet format (CSV/Excel) files. See [Chapter 4](#) for more information about exporting files.

Unpack project opens a File selection dialog that allows you to browse for and decompress **.tzz** files.

Pack project opens a File selection dialog that allows you to compact your project into a compressed `.taz` file.

Email project allows you to pack and email your project by specifying the Recipient email address, Subject for email, and Message text. See Section 27.13 for more information on emailing the project.

Cleanup allows you to clean up your project by removing or compressing data relating to mesh, post-processing, screen captures, summary output, reports, and scratch files using the Clean up project data panel.

Print screen allows you to print a PostScript image of the Airpak model that is displayed in the graphics window using the Print options panel. The inputs for the Print options panel are similar to those in the Graphics file options panel. See Section 3.5 for details.

Create image file opens a Save image dialog that allows you to save your model displayed in the graphics window to an image file. Supported file types include: GIF, JPEG, PPM, TIFF, VRML, and PS.

Shell window opens a separate window running an operating system shell. The window is initially in the subdirectory of the Airpak projects directory that contains all the files for the current projects. In this window you can issue commands to the operating system without exiting Airpak. Type `exit` in the window to close the window when you are finished using it. Note that on Windows machines, this menu item appears as Command prompt.

Quit exits the Airpak application. See Section 2.4 for details.

The Edit Menu

The Edit menu (Figure 2.1.4) contains options for editing your Airpak model. A description of the Edit menu options is provided below. See Chapter 7 for more information about editing objects in your Airpak model.

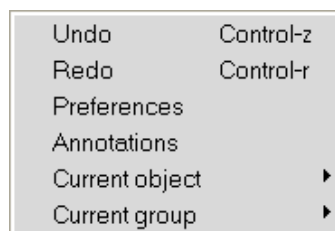


Figure 2.1.4: The Edit Menu

Undo allows you to undo the last model operation you performed. **Undo** can be used repeatedly to take you back to the first operation performed.

Redo allows you to redo one or more previously undone operations. This option applies only to operations undone by selecting the **Undo** option.

Preferences opens the **Preferences** panel, where you can configure the graphical user interface. See Section 6.3 for more information about setting preferences.

Annotations allows you to add annotations (e.g., labels and arrows) to the graphics window using the **Annotations** panel. See Section 2.1.5 for more information about annotations.

Current object allows you to perform various operations on the currently selected object in the model. Options include:

Modify applies any changes you have made to the object.

Reset resets settings for the current object to the values it had when the project was opened or to when the object settings were last updated.

Copy from opens the **Object selection** panel, where you can select and copy other objects in your model.

Move opens an object-specific **Move** panel, where you can scale, rotate, translate, or mirror an object.

Copy opens an object-specific **Copy** panel, where you can create a copy of an object and then scale, rotate, translate, or mirror the copied object.

Edit opens an object-specific **Edit** window, where you can set various object properties.

Delete removes the object from the model.

Active allows you to toggle the object's activity.

Remove from group allows you to remove the object from its group.

Current group allows you to perform various operations on the selected group of objects in the model. Options include:

Create creates a new group.

Add to group allows you to add objects to a group either by selecting a point or region on the screen or choosing an object name or pattern.

Remove from group allows you to remove objects from a group either by selecting a point or region on the screen or choosing an object name or pattern.

Rename allows you to rename a group.

Delete removes the selected group from the model.

Copy opens a group-specific **Copy** panel, where you can create a copy of a group and then scale, rotate, translate, or mirror the copied group.

Move opens a group-specific **Move** panel, where you can scale, rotate, translate, or mirror a group.

Edit opens a group-specific **Edit** window, where you can set various group properties.

Activate all activates all objects in the selected group.

Deactivate all deactivates all objects in the selected group.

Delete all removes all objects from the selected group and the **Airpak** model.

Create assembly creates an assembly from the selected group and adds the assembly to the **Airpak** model.

Copy params applies the parameters of the selected object to all objects of the same type in the selected group.

Save as project allows you to save the group as an **Airpak** project.

The View Menu

The **View** menu (Figure 2.1.5) contains options that control the appearance of the graphics window. A description of the **View** menu options is provided below.



Figure 2.1.5: The View Menu

Summary (HTML) allows you to display an HTML version of the summary of your model. To display the summary, select **Summary (HTML)** in the **View** menu. **Airpak** will automatically launch your web browser (e.g., Netscape, IE), as shown in Figure 1.7.17.

Location allows you to display the coordinates of a point in your model. To find the coordinates of a point, select **Location** in the **View** menu. Select the point in the graphics window using the left mouse button. **Airpak** will display the coordinates of the point you select in the graphics window and in the **Message** window. To exit from the **Location** mode, click with the right mouse button in the graphics window.

Distance allows you to calculate the distance between two points in your **Airpak** model. To find the distance between two points, select **Distance** in the **View** menu. Select the first point in the graphics window using the left mouse button. **Airpak** will display the coordinates of the point you select in the graphics window and in the **Message** window. Then select the second point in the graphics window, also using the left mouse button. **Airpak** will display the coordinates of the second point in the graphics window and in the **Message** window, calculate the distance between the two points, and display the distance in the **Message** window. To exit from the **Distance** mode, click with the right mouse button in the graphics window.

Angle allows you to calculate the angle created between two vectors in your **Airpak** model. To find the angle between two vectors, select **Angle** in the **View** menu. Select a vertex point in the graphics window using the left mouse button. Then select the end point of the first vector, also using the left mouse button. Then select the end point of the second vector, also using the left mouse button. **Airpak** will display the angle created by the two vectors in the **Message** window.

Bounding box allows you to determine the minimum and maximum coordinates for your model's bounding box. To find the minimum and maximum coordinates for the model's bounding box, select **Bounding box** in the **View** menu. **Airpak** will display the minimum and maximum x , y , and z coordinates for the model enclosure in the **Message** window.

Markers allows you to add or remove markers from the graphics window of your **Airpak** model.

Add allows you to add a marker to the graphics window of your **Airpak** model. To add a marker, select **Markers** then select **Add** in the **View** menu. This will open the **Add marker** panel (Figure 2.1.6).

To specify the position of the marker, you can enter the coordinates of the point (separated by spaces) next to **Position** or you can click on the **Select** button and then click on a location in the graphics window to select the point. To specify the text for the marker, enter the text in the **Text** box. Click **Accept** to add the marker to the graphics window.

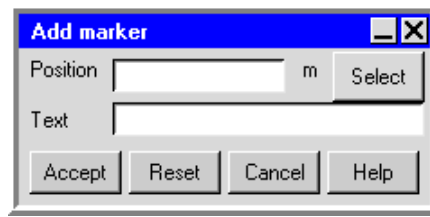


Figure 2.1.6: The Add marker Panel

Clear allows you to remove all of the markers from the graphics window.

Rubber bands allows you to add and remove rubber-banding rulers between two objects in the graphics window.

Add allows you to add a rubber-banding ruler between two points on two objects in the graphics window. To add a rubber band, select **Add** in the **View** menu. Select the point on the first object in the graphics window using the left mouse button. Then select the point on the second object in the graphics window, also using the left mouse button. **Airpak** will calculate the overall distance between the two points, and the distances in the x , y , and z coordinate directions. It will display this information in the graphics window next to an arrow drawn between the objects. If you move one of the objects, **Airpak** will update the display of the distances between the two objects.

Clear removes all of the rubber bands from the graphics window.

Set background allows you to set the background color of the graphics window. To set the background color, select **Set background** from the **View** menu and use the method or panel that is appropriate to your system.

Edit toolbars allows you to customize the appearance of **Airpak** by displaying or hiding any of the **Airpak** toolbars using the **Available toolbars** panel (Figure 2.1.7).

By default, all toolbars are visible. To hide toolbars, deselect the appropriate toolbar option and click **Accept**. Click **Reset** to display all previously hidden toolbars.

Shading contains options that control the rendering of your **Airpak** model. Options include:

Wireframe outlines the model's outer edges and those of its components. Neither the **Solid** nor the **Selected solid** option can be used with the **Wireframe** option.

Solid adds solid-tone shading to the visible surfaces of the model's internal components to give them a solid appearance. Neither the **Wireframe** nor the **Selected solid** option can be used when **Solid** is selected.

Selected solid adds solid-tone shading to the visible surfaces of the object currently selected in the object **Edit** window to give it a solid appearance. Neither the **Wireframe** nor the **Solid** option can be used when **Selected solid** is selected.

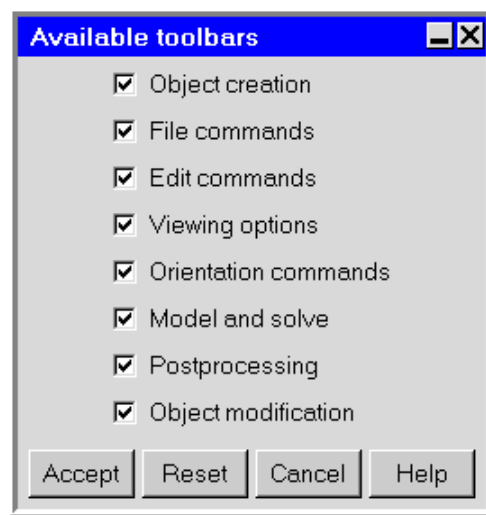


Figure 2.1.7: The Available toolbars Panel

Hidden line activates the hidden line removal algorithm, which makes objects that are drawn to look transparent now appear to be solid.

Display contains options that allows you to customize the appearance of the graphics window. Options include:

Object names displays object names next to objects in the graphics window. Options include:

Current assembly displays the names of the objects in the currently selected assembly.

None does not display any object names. This option is selected by default.

Selected displays the names of the currently selected objects.

Coord axes displays the coordinate axes reference in the lower left hand corner of the graphics window. This option is selected by default.

Visible grid displays the grid in the graphics window. The grid parameters must first be set in the **Interaction** section of the **Preferences** panel. See Section [6.3.7](#) for more information about the interactive editing.

Origin marker displays the origin of the graphics window. This option is selected by default. You can move the origin marker after it is displayed by holding down the <Control> key, clicking on the origin marker with the middle mouse button, and dragging it to a new location.

Rulers displays ruled coordinate axes from the origin of the graphics window.

Project title displays the project name and the **Airpak** version number in the graphics window. You can move the project title after it is displayed by

holding down the <Control> key, clicking on the project title with the middle mouse button, and dragging it to a new location.

Current date displays the current date in the graphics window. You can move the current date after it is displayed by holding down the <Control> key, clicking on the current date with the middle mouse button, and dragging it to a new location.

Construction lines displays construction lines from IGES models.

Construction points displays construction points from IGES models.

Mesh displays a mesh that has been loaded into a model.

Mouse position displays the position of the mouse in the lower left hand portion of the graphics window.

Depthcue adds a visual element of depth to enhance the model's 3D appearance. This effect is achieved by intensifying foreground lines and softening background lines. This option does not affect the X Window graphics driver.

Visible allows you to choose which objects of your **Airpak** model that you want displayed in the graphics window. You can toggle the visibility of the following items:

Room

Blocks

Fans

Vents

Openings

Persons

Walls

Partitions

Sources

Resistances

Heat exchangers

Hoods

Assemblies

Wires

To make any of these items become visible (or invisible) in the display, select (or deselect) the desired sub-option. Hidden objects appear in the **Model manager** window as grayed-out items under the **Inactive** node but are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest from view.

Action turns on objects that have motion associated with them, such as fans. This animation option uses considerable CPU time, and it is recommended that this option should not be left on, but used only as needed. The motion has no physical significance, and is intended only to aid in the recognition of model components.

Lights allows you to set the lighting parameters for viewing your Airpak model. To change the lighting parameters, select **Lights** in the **View** menu. This will open the **Lighting options** panel (Figure 2.1.8).

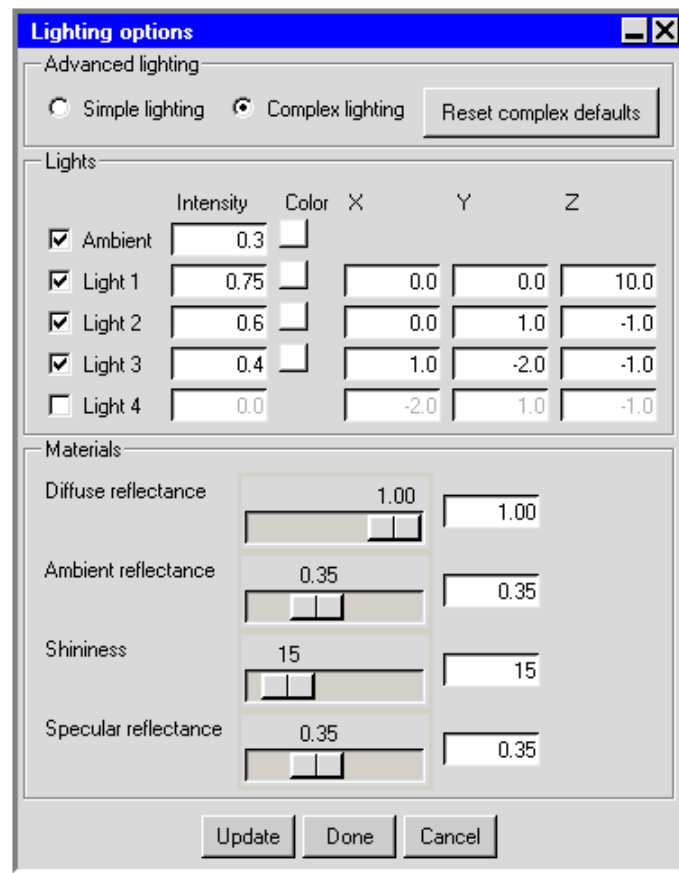


Figure 2.1.8: The Lighting options Panel

Under **Advanced lighting**, you can choose **Simple lighting** (the default) or **Complex lighting**. In both cases, the lighting direction is fixed relative to your view of the model. If **Complex lighting** is selected, you will have control over the **Intensity** and **Color** of the **Ambient** light, and the **Intensity**, **Color**, and direction of origin (**X**, **Y**, **Z**) of up to four additional lights. The **Intensity** may range from 0 to 1, and the **Color** can be specified as described in Section 6.3.6. You can enable or disable a particular light by toggling the check box next to its name.

For **Complex lighting**, you can also specify how the object surface materials will

respond to being lit. Under **Materials**, you can use the slider bars or directly specify values for the following parameters:

Diffuse reflectance is the ratio of the incident luminous flux reradiated in the visual spectrum by diffuse reflection. Diffuse reflectance plays the most important role in determining what you perceive the color of an object to be. It is affected by the color of the incident diffuse light and the angle of the incident light relative to the normal direction, with the highest intensity where the incident light falls perpendicular to the surface. The diffuse reflectance is not affected by the position of your viewpoint. Values may range from 0 to 1.

Ambient reflectance is the ratio of the incident luminous flux reradiated in the visual spectrum by ambient reflection. Ambient reflectance is most noticeable where an object receives no direct illumination. An object's total ambient reflectance is affected by the global ambient light and ambient light from individual light sources. Like diffuse reflectance, ambient reflectance is not affected by the position of your viewpoint. Values may range from 0 to 1.

Shininess controls the size and brightness of the highlight. Values may range from 0 to 128. The higher the value, the smaller and brighter (more focused) the highlight.

Specular reflectance is the ratio of the incident luminous flux reradiated in the visual spectrum by specular reflection. Specular reflection from an object produces highlights. Unlike diffuse and ambient reflection, the amount of specular reflection that you see does depend on the location of your viewpoint, and it is brightest along the direct angle of reflection. Values may range from 0 to 1.

To restore the default values for complex lighting, click **Restore complex defaults**.

The Orient Menu

The **Orient** menu (Figure 2.1.9) contains options that allow you to modify the direction from which you view your model in the graphics window. Besides selecting the view along the x , y , or z axis, you can zoom your model, scale it to fit exactly within the graphics window, or restore it to the default view along the negative z axis. A description of the **Orient** menu options is provided below.

Home position	h
Zoom in	z
Scale to fit	s
Orient positive X	
Orient negative X	Shift-x
Orient positive Y	Shift-y
Orient negative Y	
Orient positive Z	
Orient negative Z	Shift-z
Isometric view	Shift-i
Reverse orientation	Shift-r
Nearest axis	
Save user view	
Clear user views	

Figure 2.1.9: The Orient menu

Home position selects the default view of your model directed along the negative z axis.

Zoom in allows you to focus on any part of your model by opening and resizing a window around the desired area. After selecting this option, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the graphics window.

Scale to fit adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.

Orient positive X, Y, Z views the model toward the direction of the positive x , y , or z axis.

Orient negative X, Y, Z views the model toward the direction of the negative x , y , or z axis.

Isometric view views the model from the direction of the vector equidistant to all three axes.

Reverse orientation views the model along the current view vector but from the opposite direction (i.e., rotated 180 degrees).

Nearest axis orients the view to the nearest axis normal to the plane.

Save user view opens the **Query** panel (Figure 2.1.10) that prompts you for a view name and then saves the current view using your specified name. The new view name is attached to a listing of user views at the bottom of the **Orient** menu.

Clear user views removes the listing of user views from the bottom of the **Orient** menu.

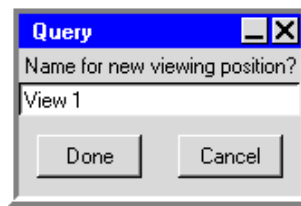


Figure 2.1.10: The Query panel

The Model Menu

The **Model** menu (Figure 2.1.11) contains options that allow you to generate a mesh, load non-native files, edit CAD data, and perform other model-related functions. A description of the **Model** menu options is provided below.

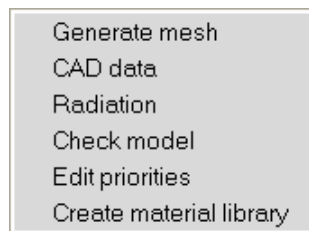


Figure 2.1.11: The Model Menu

Generate mesh opens the **Mesh control** panel, where you can provide settings to create a mesh for your **Airpak** model. See Chapter 26 for more information about meshes.

CAD data opens the **CAD data** panel that allows you to import into **Airpak** and edit geometry that was created using a commercial CAD program. See Chapter 4 for more information about importing files into **Airpak**.

Radiation opens the **Form factors** panel, where you can model the radiation for specific objects in your **Airpak** model. See Chapter 22 for more information about radiation models in **Airpak**.

Check model performs a check to test the model for problems in the design. See Section 7.9.2 for details.

Snap to grid opens the **Snap to grid** panel that allows you to snap a selected object in the graphics window to the grid. See Section 7.3.7 for details.

Edit priorities opens the **Object priorities** panel that allows you to prioritize the objects in your model. **Airpak** provides priorities based on object creation and uses the priorities when meshing the model. See Section 26.7 for details.

Create material library allows you to save a materials library for use with your Airpak model. See Section 7.7 for details.

The Tree Menu

The Tree menu (Figure 2.1.12) contains options that allow you to organize the tree structure of complicated projects in the Model manager window. See Section 2.1.4 for more information about the Model manager window. A description of the Tree menu options is provided below.

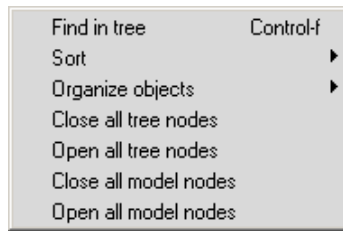


Figure 2.1.12: The Tree Menu

Find in tree opens the Find in tree panel (Figure 2.1.13) that allows you to search the Model manager window hierarchy for a specific object.

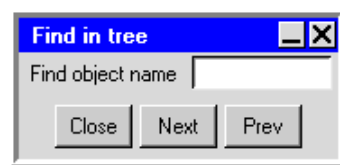


Figure 2.1.13: The Find in tree Panel

To locate a specifically named object, type the object name in the Find object name text field. Click the Next button to find the next occurrence of the name in the tree hierarchy. Each time a object name is found, its name is highlighted in the tree hierarchy. Click the Prev button to find the previously found object name.

You can type an object name that contains an asterisk or a question mark in place of characters or a character, respectively. For example, typing **fan*** will search the tree for all objects whose names start with **fan**; typing **vent?** will search the tree for all objects whose names consist of the word **vent** plus one character. Any object in the model whose name matches this text pattern will be highlighted when you click Next or Prev buttons in the Find in tree panel.

Sort allows you to sort the tree hierarchy. Options include:

Creation order sorts objects by their order of creation in the model (default).

Alphabetical sorts objects alphabetically by their names.

Meshing priority sorts objects by their meshing priority.

Organize objects allows you to customize the method in which objects in the tree hierarchy are organized. Options include:

Flat arranges objects in the order they are created.

Types groups objects in the tree hierarchy by object type.

Types / subtypes groups objects in the tree hierarchy by object type and subtype.

Types / subtypes / shapes groups objects in the tree hierarchy by object type, subtype, and shape.

Close all tree nodes allows you to simplify a complex tree hierarchy by closing all tree nodes in the **Model manager** window.

Open all tree nodes allows you to see all aspects of a **Airpak** model by opening all tree nodes in the **Model manager** window.

Close all model nodes automatically closes the **Model** node and all nodes underneath it in the **Model manager** window.

Open all model nodes automatically opens the **Model** node and all nodes underneath it in the **Model manager** window.

The Macros Menu

The **Macros** menu (Figure 2.1.14) contains options that allow you access defined **Airpak** macros. These macros are broken down into three types: boundary conditions, quick geometry/approximations, and object rotation. See Chapter 25 for more information about **Airpak** macros. A description of the **Macros** menu options is provided below.

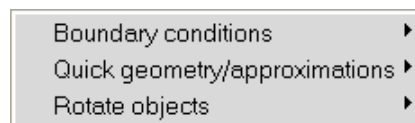


Figure 2.1.14: The **Macros** Menu

Boundary conditions contains available **Airpak** boundary condition macros. Options include:

Solar flux allows you to account for incident solar radiation based on the model geometry and pertinent solar information such as terrestrial location, date, and time. You can specify the resulting heat flux as a boundary condition.

Atmospheric boundary layer allows you to add an atmospheric boundary layer to your **Airpak** model. External wind flows by the wind direction, wind speed, height above ground, and local terrain parameter can be specified.

Diffuser provides several types of diffuser models that allow you to more accurately model the behavior of air inlet diffusers and predict the penetration and spread of air in a room.

Quick geometry/approximations contains available **Airpak** geometry approximation macros.

Options include:

Polygonal ducts allows you to include polygonal ducts in your **Airpak** model.

Closed box allows you to enclose spaces in your **Airpak** model and assign properties to the sides of the enclosure.

1/4 Polygonal cylinder allows you to create a 1/4 polygonal representation of a cylinder for blocks, sources, and resistances.

Cylinder plates allows you to create a cylinder by faceting the surface with thin conducting plates.

Cylindrical enclosure allows you to create a cylindrical enclosure by using four hollow polygonal blocks that change the shape of the room into a cylinder.

Polygonal circle allows you to create faceted 2D objects for fans, walls, vents, plates, openings, and sources.

Polygonal cylinder allows you to create a polygonal representation of a cylinder for blocks, sources, and resistances.

Hemisphere allows you to represent hemispherical objects in **Airpak** using solid cylindrical slices.

Rotate objects contains available **Airpak** macros for rotating objects. Options include:

Individual plates allows you to rotate a plate block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis.

Individual prism blocks allows you to rotate a prismatic block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis.

Individual polygonal blocks allows you to rotate a polygonal block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis.

Groups of prism blocks allows you to rotate a group of prismatic blocks through an arbitrary angle about a given coordinate axis.

The Solve Menu

The **Solve** menu (Figure 2.1.15) contains options that allow you to control the solution of your **Airpak** model. See Chapter 27 for more information. A description of the **Solve** menu options is provided below.

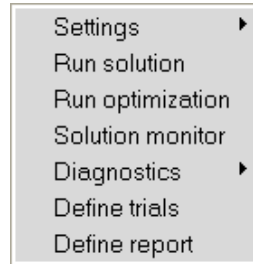


Figure 2.1.15: The **Solve** Menu

Settings allows you to set various solution parameters for your **Airpak** project. Options include:

- Basic** opens the **Basic settings** panel, where you can specify the number of iterations to be performed and convergence criteria **Airpak** should use before starting your CFD calculations. See Section 27.7 for details.

- Advanced** opens the **Advanced solver setup** panel, where you can specify the discretization scheme, under-relaxation factors, and the multigrid scheme. See Chapter 27 for details.

- Parallel** opens the **Parallel settings** panel, where you can specify the type of execution you wish to perform (e.g., serial (the default), parallel, or network parallel). See Section 27.11.3 for details.

Run solution opens the **Solve** panel, where you can set solution parameters for your **Airpak** model.

Run optimization opens the **Parameters and optimization** panel, where you can define parameters (design variables) and set the optimization process.

Solution monitor opens the **Solution monitors definition** panel, where you specify the variables to be monitored during the calculation.

Diagnostics allows you to edit the output files created after you have generated case files and solutions for your model. See Section 27.13 for details.

Define trials opens the **Parameters and trials** panel, where you can define trial calculations for your model. Each trial is based on a combination of parameter values defined in **Airpak**.

Define report opens the **Define summary report** panel, where you can specify a summary report for a variable on any or all objects in your **Airpak** model.

The Post Menu

The **Post** menu (Figure 28.1.1) contains options that allow your to access Airpak's post-processing objects. See Chapter 28 for more information. A description of the **Post** menu options is provided below.

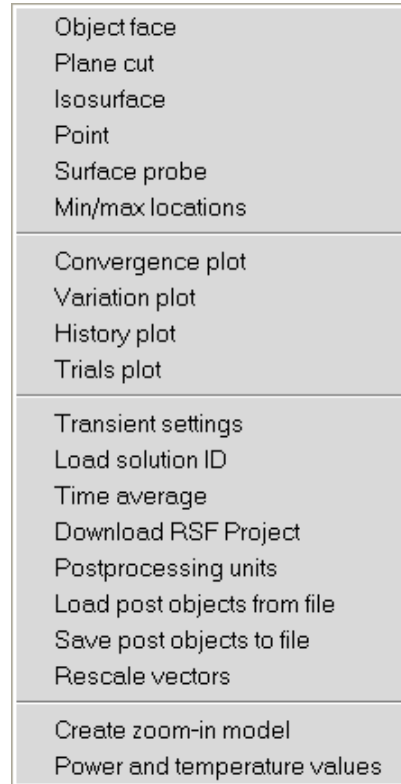


Figure 2.1.16: The Post Menu

Object face allows you to display results on object faces in the model.

Plane cut allows you to display results on cross-sections of the model.

Isosurface allows you to display results on defined isosurfaces in the model.

Point allows you to create points and display results at points in the model.

Surface probe allows you to display results at a point on a postprocessing object that has been created in the model.

Min/max locations allows you to display the location of the minimum and maximum values for postprocessed variables.

Convergence plot allows you to display the convergence history of the solution.

Variation plot allows you to plot a variable along a line through the model.

History plot allows you to plot solution variable histories over time.

Trials plot allows you to plot solution variables at specified points across multiple trials.

Transient settings opens the **Post-processing time** panel, where you can set parameters for transient simulations.

Load solution ID allows you to select a specific solution set to be examined.

Time average allows you to view the results of a time averaged (transient) problem as if it were steady-state.

Download RSF project opens the **Download RSF project** panel, where you can run an **Airpak** job using the Remote Simulation Facility (RSF) and download the results for postprocessing in **Airpak**.

Postprocessing units opens the **Postprocessing units** panel, where you can choose the units for different postprocessing variables.

Load post objects from file allows you to load postprocessing objects from a file.

Save post objects to file allows you to save postprocessing objects to a file.

Rescale vectors allows you to redisplay vectors drawn at their original sizes.

Create zoom-in model allows you to zoom in and define a region in your **Airpak** model and save that region as a separate **Airpak** project.

The Report Menu

The **Report** menu (Figure 2.1.17) contains options for generating output concerning the results of your **Airpak** model. See Chapter 29 for more information. A description of the **Report** menu options is provided below.

HTML report opens the **HTML report** panel, where you can customize your results and write out an HTML document that can be viewed in a web browser.

Solution overview allows you to view and create solution overview files.

View opens a **File selection** dialog box, where you can open a solution overview file (*.overview) in which summary data is stored for a particular solution.

Create opens a **Version selection** panel, where you can select a solution for which to create an overview file. See Section 29.4 for details.

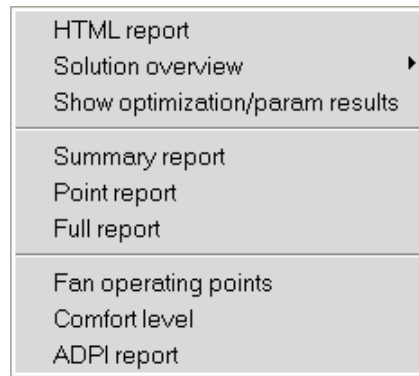


Figure 2.1.17: The Report Menu

Show optimization/param results opens an **Optimization run** panel, where you can view all the function values, design variables, and the running times for each optimization iteration, as well as the plots of the function values and design variables versus iteration number.

Summary report opens the **Define summary report** panel, where you can specify a summary report for a variable on any or all objects in your **Airpak** model.

Point report opens the **Define point report** panel, where you can create a report for a variable at any point in your **Airpak** model.

Full report allows you to customize the report of your results.

Fan operating points allows you to create a report of the fan operating points for fans using fan curves in your model.

Comfort level allows you to create a report of Indoor Air Quality (IAQ) and thermal comfort level results.

ADPI report allows you to create a report of the Air Diffusion Performance Index (ADPI) for a specified region in your model.

The Windows Menu

The **Windows** menu contains the names of **Airpak** panels when one or more of them are open. This feature is useful when you have many panels or toolbars open and you wish to quickly locate a specific toolbar or panel. An asterisk (*) to the left of a panel or toolbar name indicates that the panel or toolbar is currently hidden. You can show or hide toolbars using the **Available toolbars** panel through the **Edit toolbars** option under the **View** menu. See Section 2.1.2 for more information about showing and hiding toolbars using the **View** menu.

The Help Menu

The **Help** menu (Figure 2.1.18) contains options that allow you to access the on-line **Airpak** documentation, **Airpak** web sites, and also print a list of keyboard shortcuts available in **Airpak**. A description of the **Help** menu options is provided below.



Figure 2.1.18: The **Help** Menu

Help opens on-line **Airpak** documentation in a web browser.

Airpak on the Web opens the **Airpak** home page in a web browser.

User Services Center opens the **Airpak** User Services Center web page in a web browser.

List shortcuts prints the list of keyboard shortcuts for **Airpak** in the **Message** window.

2.1.3 The Airpak Toolbars

The **Airpak** graphical user interface (Figure 2.1.1) also includes eight toolbars located throughout the **Main** window. These toolbars (**File commands**, **Edit commands**, **Viewing options**, **Orientation commands**, **Model and solve**, **Postprocessing**, **Object creation**, and **Object modification**) provide shortcuts to performing common tasks in **Airpak**. By default, the toolbars are docked to the **Airpak** interface but can also be detached and treated as regular control panels. See Section 2.1.9 for more information about using detached toolbars.

The File commands Toolbar

The **File Commands** toolbar (Figure 2.1.19) contains options for working with **Airpak** projects and project files. A brief description of the **File commands** toolbar options is provided below. See Chapter 3 for more information about reading, writing, and managing files in **Airpak**.





New project () allows you to create a new **Airpak** project using the **New project** panel. Here, you can browse through your directory structure, create a new project directory, and enter a project name.




Figure 2.1.19: The File commands Toolbar

Open project () allows you to open existing **Airpak** projects using the **Open project** panel. Here, you can browse through your directory structure, locate a project directory, and either enter a project name, or specify an old project name from a list of recent projects. Additionally, you can specify a version name or number for the project.

Save project () saves the current **Airpak** project.

Print screen () allows you to print a PostScript image of the **Airpak** model that is displayed in the graphics window using the **Print options** panel. The inputs for the **Print options** panel are similar to those in the **Graphics file options** panel. See Section 3.5 for details.


Create image file () opens a **Save image** dialog that allows you to save your model displayed in the graphics window to an image file. Supported file types include: PPM, GIF, JPEG, TIFF, VRML, and PS.


The Edit commands Toolbar

The **Edit commands** toolbar (Figure 2.1.20) contains options that allow you to perform undo and redo operations in your **Airpak** model. A description of the **Edit commands** toolbar options is provided below. See Chapter 7 for more information about editing objects in **Airpak**.



Figure 2.1.20: The Edit commands Toolbar

Undo () allows you to undo the last model operation you performed. **Undo** can be used repeatedly to take you back to the first operation performed.


Redo () allows you to redo one or more previously undone operations. This option applies only to operations undone by selecting the **Undo** option.


The Viewing options Toolbar


The Viewing options toolbar (Figure 2.1.21) contains options that allow you to modify the way in which you view your model in the graphics window. A description of the Viewing options toolbar options is provided below.




Figure 2.1.21: The Viewing options Toolbar


Home position () selects the default view of your model directed along the negative z axis.


Zoom in () allows you to focus on any part of your model by opening and resizing a window around the desired area. After selecting this option, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the graphics window.

Scale to fit () adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.

Rotate about screen normal () rotates the current view by 90 degrees clockwise about the axis normal to the view.




One viewing window () displays a single graphics window.

Four viewing windows () displays four graphics windows, each with a different viewing perspective. By default, one view is isometric, another is of the x - y plane, another is of the x - z plane, and another is of the y - z plane.

Display object names () toggles the visibility of a model's object names in the graphics window.

The Orientation commands Toolbar

The Orientation commands toolbar (Figure 2.1.22) contains options that allow you to modify the direction from which you view your model in the graphics window. A description of the Orientation commands toolbar options is provided below.

Orient X, Y, Z (  ) views the model toward the direction of the positive x , y , or negative z axis.

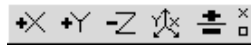




Figure 2.1.22: The Orientation commands Toolbar

Isometric view () views the model from the direction of the vector equidistant to all three axes.


Reverse orientation () views the model along the current view vector but from the opposite direction (i.e., rotated 180 degrees).


The Model and solve Toolbar


The Model and solve toolbar (Figure 2.1.23) contains options that allow you to generate a mesh, model radiation, check your model, and run a solution. A description of the Model and solve toolbar options is provided below.




Figure 2.1.23: The Model and solve Toolbar


Power and temperature limits () opens the Power and temperature limit setup panel, where you can review or change the power of objects, as well as specify the temperature limits.

Generate mesh () opens the Mesh control panel, where you can provide settings to create a mesh for your Airpak model. See Chapter 26 for more information about generating meshes.

Radiation () opens the Form factors panel, where you can model radiation for specific objects in your model. See Chapter 22 for more information about radiation models in Airpak.

Check model () performs a check to test the model for problems in the design. See Section 7.3.7 for details.

Run solution () opens the Solve panel, where you can set solution parameters for your Airpak model.

Run optimization () opens the Parameters and optimization panel, where you can define parameters (design variables) and set the optimization process.


The Postprocessing Toolbar


The Postprocessing toolbar (Figure 28.1.2) contains options that allow you to examine your results using Airpak's postprocessing objects. A description of the Postprocessing toolbar options is provided below. See Chapter 28 for more information about postprocessing.




Figure 2.1.24: The Postprocessing Toolbar


Object face () allows you to display results on object faces in the model.


Plane cut () allows you to display results on cross-sections of the model.


Isosurface () allows you to display results on defined isosurfaces in the model.


Point () allows you to display results at points in the model.

Surface probe () allows you to display results at a point on a surface in the model.


Variation plot () allows you to plot a variable along a line through the model.


History plot () allows you to plot solution variable histories over time.


Trials plot () allows you to plot trial solution variables.

Transient settings () opens the Post-processing time panel, where you can set parameters for transient simulations.

Load solution ID () allows you to select a specific solution set to be examined.

Time average () allows you to view the results of a time averaged (transient) problem as if it were steady-state.

Download RSF project () opens the Download RSF project panel, where you can run an Airpak job using the Remote Simulation Facility (RSF) and download the results for postprocessing in Airpak.


Summary report () opens the Define summary report panel, where you can specify a summary report for a variable on any or all objects in your Airpak model.


The Object creation Toolbar


The Object creation toolbar (Figure 7.1.1) contains options that allow you to add objects to your Airpak model. A description of the Object creation toolbar options is provided below. Unless otherwise noted, all objects are created in the center of the corresponding model room.





Figure 2.1.25: The Object creation Toolbar


Create blocks () allows you to create a block object. See Chapter 8 for details.

Create fans () allows you to create a fan object. See Chapter 9 for details.


Create vents () allows you to create a vent object. See Chapter 10 for details.

Create openings () allows you to create an opening object. See Chapter 11 for details.


Create persons () allows you to create a person object. See Chapter 12 for details.


Create walls () allows you to create a wall object. See Chapter 13 for details.

Create partitions () allows you to create a partition object. See Chapter 14 for details.


Create sources () allows you to create a source object. See Chapter 15 for details.


Create resistances () allows you to create a 3D resistance object. See Chapter 16 for details.

Create heat exchangers () allows you to create a heat exchanger object. See Chapter 17 for details.

Create hoods () allows you to create a hood object. See Chapter 18 for details.

Create assemblies () allows you to create an assembly object at the origin of the graphics window. See Chapter 7 for details.

Create wires () allows you to create a wire object. See Chapter 19 for details.

Create materials () allows you to create a materials node for the model in the Model manager window. See Chapter 7 for details.

The Object modification Toolbar

The Object modification toolbar (Figure 2.1.26) contains options that allow you to edit, delete, move, copy, or align an object in your Airpak model. A description of the Object modification toolbar options is provided below. See Chapter 7 for details about modifying objects in Airpak.

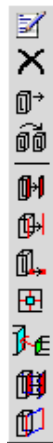



Figure 2.1.26: The Object modification Toolbar


Edit object () opens an object-specific Edit window, where you can set various object properties.


Delete object () removes the object from the model.

Move object () opens an object-specific **Move** panel, where you can scale, rotate, translate, or mirror an object.


Copy object () opens an object-specific **Copy** panel, where you can create a copy of an object and then scale, rotate, translate, or mirror the copied object.

Align and morph faces () aligns the faces of two objects.

Align and morph edges () aligns the edges of two objects.

Align and morph vertices () aligns the vertices of two objects.

Align object centers () aligns the centers of two objects.



Align face centers () aligns the centers of the faces of two objects.

Morph faces () matches the faces of two objects.

Morph edges () matches the edges of two objects.


2.1.4 The Model manager Window


The **Airpak Model manager** window (Figure 2.1.27) provides a localized area for defining your **Airpak** model and contains a project-specific listing of problem and solution parameters.

The **Model manager** window is presented in a tree-like structure with expandable and collapsible tree nodes that show or hide relevant tree items. To expand a tree node, use the left mouse button to click on the  icon on the left hand side of the tree. To collapse a tree node, click on the  icon.

You can edit and manage your **Airpak** project from within the **Model manager** window using the mouse. For example, you can select multiple objects, edit project parameters, add groups within groups, break apart assemblies, or edit objects, by clicking and dragging objects. In addition, the **Model manager** window includes a context menu, accessible by right-clicking the mouse, that allows you to easily manipulate your **Airpak** model. See Section 2.2.2 for more information on using the mouse in the **Model manager** window.

An **Airpak** project is organized in the **Model manager** window using six different categories:

Problem setup () allows you to set basic problem parameters, set the project title, and define local coordinate systems. Options include:

Basic parameters () opens the **Basic parameters** panel, where you can specify parameters for the current **Airpak** model. See Section 6.4 for details.

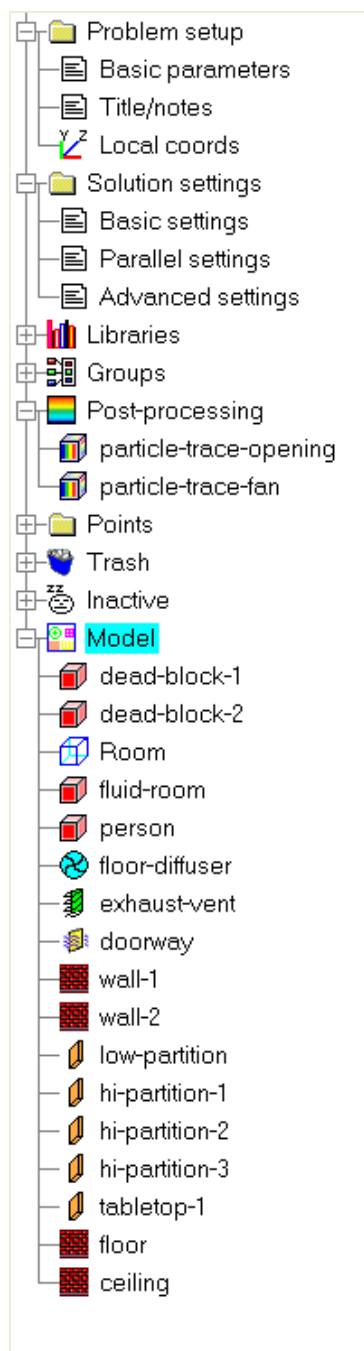






Figure 2.1.27: An Example of the Model manager Window


Title/notes () opens the **Title/notes** panel, where you can enter a title and notes for the current **Airpak** model. panel, where you can create local coordinate systems that can be used in your model other than the **Airpak** global coordinate system with an origin of (0, 0, 0). The origins of the local coordinate systems are specified with an offset from the origin of the global coordinate system. See Section 7.3.7 for details.


Solution settings () allows you to set **Airpak** solution parameters. Options include:


Basic settings () opens the **Basic settings** panel, where you can specify the number of iterations to be performed and convergence criteria **Airpak** should use before starting your CFD calculations. See Section 27.7 for details.


Parallel settings () opens the **Parallel settings** panel, where you can specify the type of execution you wish to perform (e.g., serial (the default), parallel, or network parallel). See Section 27.11.3 for details.


Advanced settings () opens the **Advanced solver setup** panel, where you can specify the discretization scheme, under-relaxation factors, and the multigrid scheme. See Chapter 27 for details.

Libraries () lists the libraries used in your **Airpak** project. By default, a **Main library** exists in your **Airpak** project that contains materials (fluids, solids, and surfaces), fan objects, and other complex objects. See Section 7.7 for details.

Groups () lists any groups of objects in the current **Airpak** project. See Section 7.6 for details about grouping objects.

Post-processing () lists any postprocessing objects in the current **Airpak** project. See Chapter 28 for details about postprocessing in **Airpak**.

Points () lists any point monitoring objects in the current **Airpak** project. See Section 27.8.1 for details about point monitors.

Trash () lists any objects that have been deleted from the **Airpak** model. Any items in the **Trash** node will only be available for the current **Airpak** session.

Inactive () lists any objects that have been made inactive in the **Airpak** model.

Model () lists all active objects and materials for the **Airpak** project.

2.1.5 Graphics Windows

Displaying graphics is an important aspect of the **Airpak** graphical user interface. There are two types of graphical displays in **Airpak**: a graphics window (or **Model Display** window) and a graphics display and control window.

The graphics window (or **Model Display** window) displays your **Airpak** model and takes up most of the **Main** window (Figure 2.1.1). It is the working space for building and manipulating your model. The graphics window contains only a graphical display of your model; it does not contain any control features.

At the lower left corner of the graphics window is a three-dimensional coordinate axes system, which indicates the current orientation of your model. The axis that is closest to your line of sight is displayed in a diamond shape. As you rotate your model, the axes rotate as well, and vice versa. You can manipulate objects in the graphics window using the mouse; see Section 2.2.4 for details.

The other type of graphics window you can encounter in **Airpak** is a graphics display and control window, which provides graphics display as well as control features (an example is shown in Figure 2.1.28). These types of windows open during specific model-building and simulation processes. For example, if you are displaying residuals while a solution is being calculated, the **Solution residuals** window will appear on the screen as shown in Figure 2.1.28. In addition to the graphics, a set of control features is provided, which are located at the bottom of the window. These control features are described in Section 27.12.2.

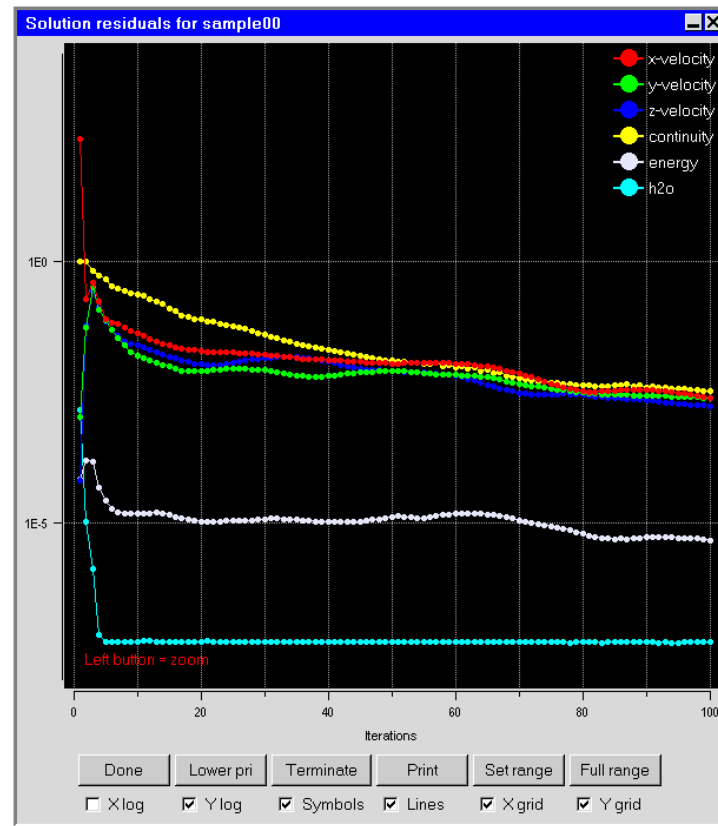


Figure 2.1.28: Example of a Graphics Window With Controls

Adding Annotations to the Graphics Window

You can add annotations (e.g., labels and arrows) to the graphics window using the Annotations panel. To open the Annotations panel, select the Annotations option in the Edit menu.

Edit → Annotations

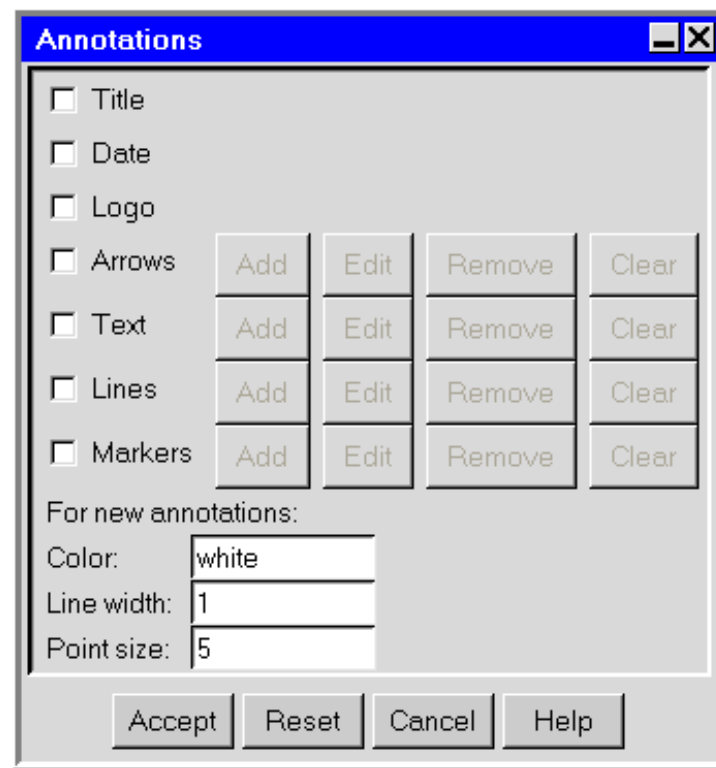


Figure 2.1.29: The Annotations Panel

The following options are available for annotations:

Title includes the job title in the graphics window.

Date includes the current date in the graphics window.

Logo includes the Airpak logo in the graphics window.

Arrows includes arrows in the graphics window. You can **Add** arrows, **Edit** arrows, **Remove** one arrow at a time, or **Clear** all arrows. Airpak displays messages in the graphics window regarding positioning and modifying arrows.

Text includes alphanumeric notations in the graphics window. You can **Add** text, **Edit** text, **Remove** one text annotation at a time, or **Clear** all text annotations. **Airpak** displays messages in the graphics window regarding positioning and modifying text annotations.

Lines includes lines in the graphics window. You can **Add** lines, **Edit** lines, **Remove** one line at a time, or **Clear** all lines. **Airpak** displays messages in the graphics window regarding positioning and modifying lines.

Markers includes markers in the graphics window. You can **Add** markers, **Edit** markers, **Remove** one marker at a time, or **Clear** all markers. **Airpak** displays messages in the graphics window regarding positioning and modifying markers.

For new annotations defines the **Color**, **Line width**, and **Point size** for new annotations. The defaults are **white** for **Color**, 1 for **Line width**, and 5 for **Point size**.

The annotations will remain in the graphics window for the current session until you turn them off in the **Annotations** panel. Annotations are saved when you save a job file. You can add annotations to your image files (see Section 3.5 for details about image files).

2.1.6 The Message Window

The **Message** window (Figure 2.1.30) is located below the graphics window. **Airpak** communicates with you through its **Message** window. It is used to display informative messages, such as those relating to meshing or solution procedures, as well as error messages and instructions. **Airpak** saves all information that is written to the **Message** window in memory. You can review this information at any time by using the scroll bar on the right-hand side of the **Message** window. To instruct **Airpak** to display more detailed messages related to the meshing and solution procedures, select the **Verbose** option in the **Message** window.

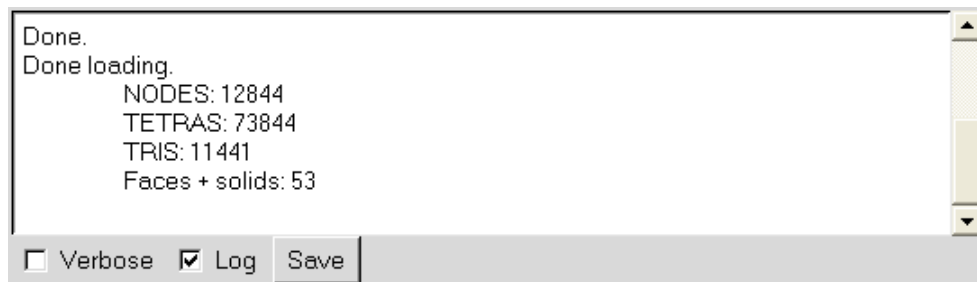


Figure 2.1.30: Sample **Message** Window

To write this information to a file, follow the steps below.

1. Click on the **Save** push button in the **Message** window. A **Save log** dialog box appears. The default name (**messages.01.txt**) for the saved information will appear in the **File name** field at the bottom of the **Save log** dialog box.
2. To specify a different file name, type it in the text field.
3. Click the **Save** button in the **Save log** dialog box to save the file. Click the **Cancel** button to cancel the procedure.

You can save the file multiple times in the same **Airpak** session.

You can also direct **Airpak** to log all information that is reported in the **Message** window to a log file. To do this, follow the steps below.

1. Select the **Log** option in the **Message** window (Figure 2.1.30). A **File selection** dialog box appears. The default name (**messages.log**) for the log file will appear in the **File name** field at the bottom of the **File selection** dialog box.
2. To specify a different filename, type it in the text field.
3. Click the **Open** button in the **File selection** dialog box to start the log file. Click the **Cancel** button to cancel the procedure.

2.1.7 The Edit Window

The **Edit** window is located in the lower right corner of the screen below the graphics window. This window displays geometric data and other general properties for a selected object. The layout of this window and the data displayed in this window change depending on the currently selected object in the graphics window. For example, when you are constructing a fan, the **Edit** window becomes the fan **Edit** window. An example of the **Edit** window is shown in Figure 2.1.31.

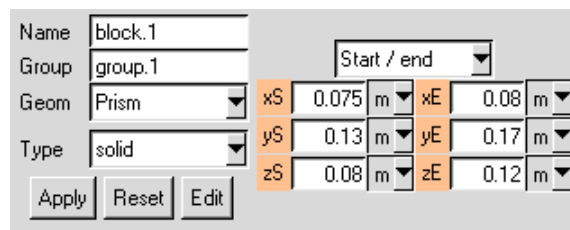


Figure 2.1.31: Sample Edit Window

2.1.8 File Selection Dialog Boxes

File selection dialog boxes enable you to choose a file for reading or writing. You can use them to look at your system directories and select a file.

Note that the appearance of the file selection dialog box will not always be the same. The version shown in Figure 2.1.32 will appear in many cases. If you are saving a project, the version shown in Figure 2.1.33 will appear. If you are opening a project, merging two projects, or loading an external assembly, a version similar to that shown in Figures 2.1.34 and 2.1.35 will appear.

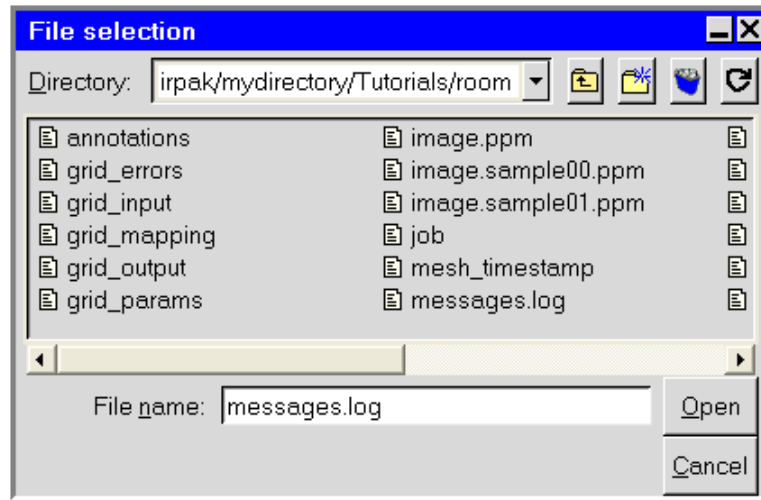



Figure 2.1.32: The File selection Dialog Box

The steps for file selection are as follows:

1. (Windows systems only) In the Drive drop-down list, select the drive on your system that contains the file.
2. Go to the appropriate directory. You can do this in three different ways:
 - Select or enter the desired directory in the Directory name field/drop-down list. You can enter the full pathname (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which Airpak was started. Be sure to include the final / character in the pathname. Note that you can also move one level up a directory tree using the  button.
 - Double-click on a directory, and then a subdirectory, etc., in the directories list until you reach the directory you want. Note that the directories list is always located under the Directory name field/drop-down list.

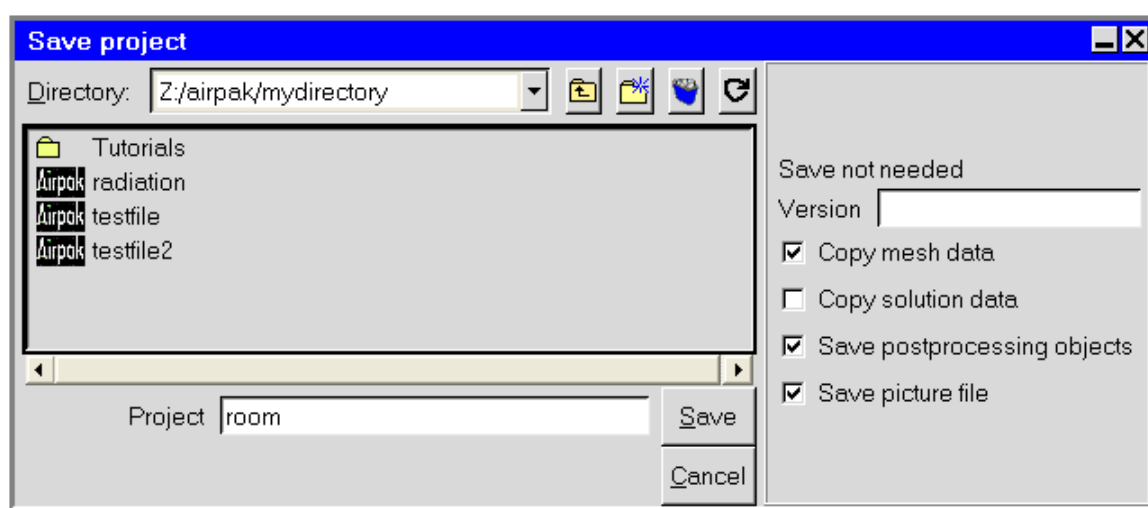


Figure 2.1.33: The Save project Panel Showing the File Selection Dialog

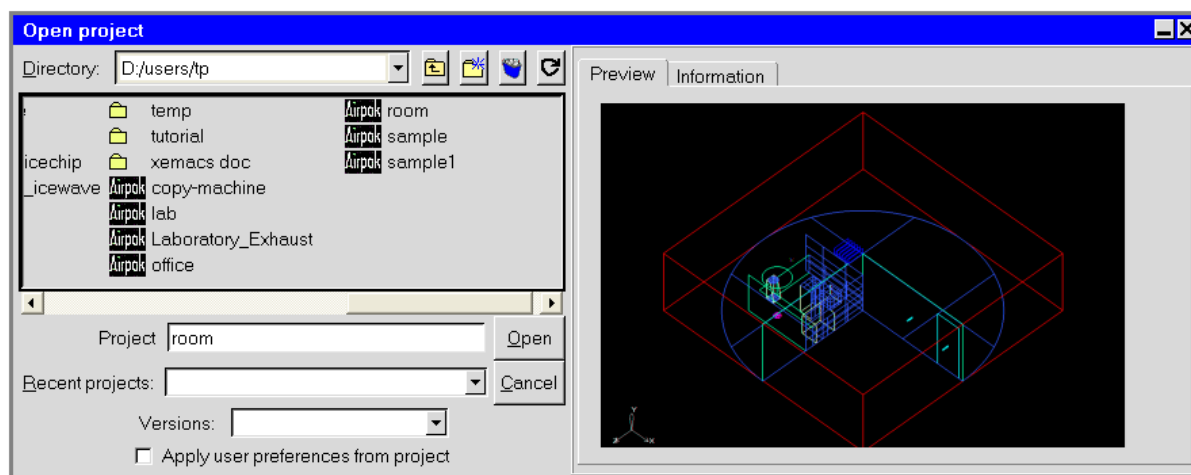


Figure 2.1.34: The Open project Panel Showing the File Selection Dialog and the Preview Tab

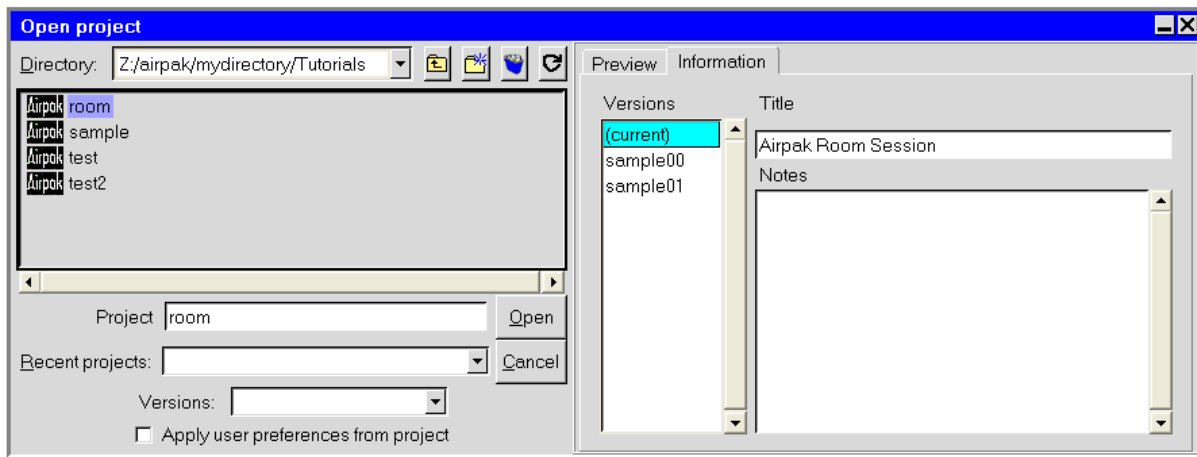




Figure 2.1.35: The **Open project** Panel Showing the File Selection Dialog and the Information Tab

- Most file selection dialogs contain a button that allow you to create a new directory () at the top of the panel. If you want to create a new directory, for example, to save a job file into a new directory, you can click the  button. **Airpak** will prompt you for the name of the new directory. Enter a name in the **New directory name** text entry box, and then click **Done**. **Airpak** will create a new directory with the specified name and then open the new directory.
3. Specify the file name, if necessary, by selecting it in the listing of files and directories, or by entering the name of the file in the **File name** text entry box. The name of this text entry box will change depending on the type of file you are selecting (e.g., **Project** in the **Save project** panel).
 4. (project files only) You can choose from a number of previously opened **Airpak** project files in the **Recent projects** drop-down list.
 5. (project files only) Select the version of the job file in the **Versions** drop-down list (e.g., in the **Open project** panel shown in Figure 2.1.34). This list displays the available versions for the project file selected. The version listing is also available in the **Information** tab of some file selection dialogs. For example, if you run a project with a project ID of **job01**, **Airpak** will save a model file called **job01.model1**. If, for example, you then change the material properties specified for a block in your model, and give this project an ID of **job02**, **Airpak** will save a model file called **job02.model1**. These files are all saved in the same directory. When you select the directory in the **Directory** list, all the different versions of the project in that directory will be displayed in the **Versions** drop-down list.

6. (project files only) To apply the settings you specified under **Options** in the **Pref-erences** panel (see Section 6.3) from the previous time you worked on the project, turn on the **Apply user preferences from project** option.
7. (project files only) **Title**, **Versions**, and **Notes** fields will appear in the **Information** tab of some of the file selection dialogs (e.g., Figure 2.1.35). In addition, the **Preview** tab (e.g., Figure 2.1.34) may contain a picture of the model if a picture has been saved. These items are described below.
 - **Title** displays the title of the job file selected. See Section 6.2.1 for information on adding a title to a job.
 - **Notes** displays any notes related to the job file selected. See Section 6.2.1 for information on adding notes to a job.
 - A picture of the geometry of the model will be displayed if you selected the **Save picture file** option in the **Save project** panel when you saved the project. See Section 3.4 for more information on the **Save picture file** option.
8. In some of the file selection dialogs there are other options that you can select, for example, on the right-hand side of the **Save project** panel (Figure 2.1.33). These options are described in the section related to the use of that particular panel, for example, the options in the **Save project** panel are described in Section 3.4.
9. Click on the **Save** button to read or write the specified file. Shortcuts for this step are as follows:
 - If your file appears in the listing of files and directories, double-click on it instead of just selecting it. This will automatically activate the **Open** button for opening files, the **Create** button for creating projects, or the **Save** button for saving files.
 - If you entered the name of the file in the **File name** text entry box, you can press the <Enter> key instead of clicking on the **Open** button for opening files, the **Create** button for creating projects, or the **Save** button for saving files.

2.1.9 Control Panels

Control panels, which are used to perform input tasks, are another major component of the GUI. They are displayed in a separate window, and are invoked by means of a higher-level function selection. Figure 2.1.36 shows an example of a control panel.

Working with a panel is similar to filling out a form. You provide input data to the panel's controls. Once you have finished entering data, you either apply the changes by "submitting" the form, or cancel the form. Clicking on the **Accept** push button accepts any changes you have made to the panel, and closes the panel. Clicking on the **Reset** button undoes all the changes you have made in the panel and restores all items in the

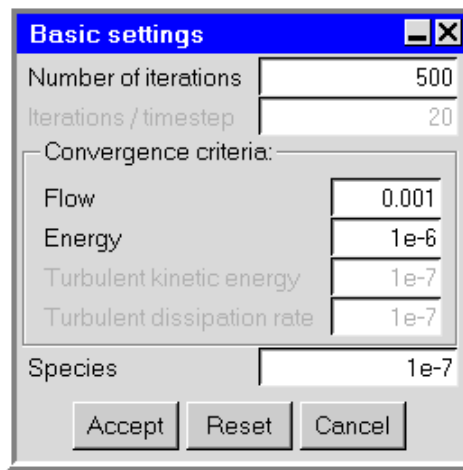


Figure 2.1.36: Example of a Control Panel

panel to their original states. **Cancel** closes the panel and ignores any changes made to the panel.

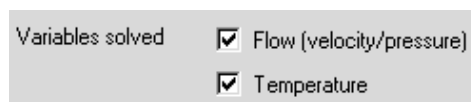
Each panel is unique, and uses a variety of input controls that are described below.

Push Button



A push button is a rectangular-shaped button that performs a function indicated by the button label. To activate a push button, place the mouse pointer over the push button and “click” the left mouse button. A “click” is one press and release of the mouse button. When push buttons are located on a menu bar, they usually cause a submenu to appear, or a panel to be displayed.

Check Box



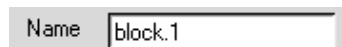
A check box is a square-shaped button that is used to turn on or off an item or action indicated by the check box label. Click the left mouse button on the check box to switch the state.

Radio Button



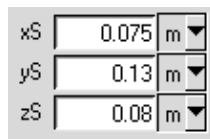
Radio buttons are diamond-shaped buttons that are located on a menu bar or panel. They are a set of mutually exclusive options that allow only one to be set in the “on” position at a time. When you click the left mouse button on a radio button, it will be turned on, and all others will be turned “off”.

Text Entry



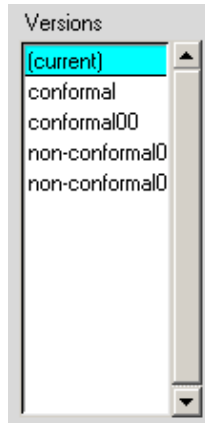
A text entry allows you to type text input. It will often have a label associated with it to indicate what the entry is for. Click the left mouse button on the text entry field to input text from the keyboard. If the text input overflows the field, you can scroll backward or forward in the field by pressing and holding down the middle mouse button. You can delete characters in the text entry field using the or <Back Space> key. In addition, you can double-click on the text entry with the left mouse button to highlight the entire field, and type the new entry.

Real Number Entry




A real number entry is similar to a text entry except it allows only real numbers to be entered (e.g., 10, -10.538, 50000.45, or 5.e-4).

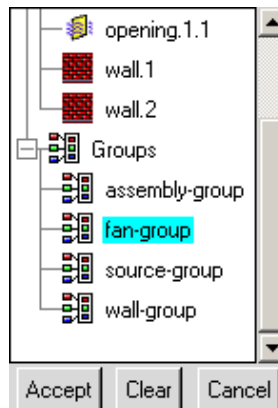
Single-Selection List



A single-selection list contains one or more items. Each item is printed on a separate line in the list. You can select an item (e.g., `ex3.300`) by placing the mouse pointer over the item line and clicking with the left mouse button. The selected item will become highlighted. Selecting another item will deselect the previously selected item in the list. If the list item overflows the window, you can scroll backward or forward in the field by clicking and holding down the middle mouse button. Single-selection lists are either visible in a panel, or hidden in the case of a drop-down list.

Drop-Down List

A drop-down list is a hidden single-selection list that shows only the current selection to save space. It will have a label associated with it to indicate what the list is for, and it is activated by clicking on the triangular button located next to the text field (). For example, in the **Object** drop-down list shown below, **All** is the current selection.

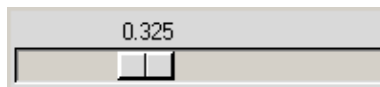


When you want to change the selection (e.g., from **assembly-group** to **fan-group**), follow the steps below:

1. Click on the button located next to the text field to display the list.
2. Place the mouse pointer over the new list item (e.g., **fan-group**). If the item is not visible, you can use the scroll bar to find it.
3. Click the left mouse button on the item to make the new selection. The list will close automatically, and the new selection will then be displayed.

If you want to abort the selection process while the list is displayed, you can move the pointer anywhere outside the list and click the left mouse button, or click **Cancel**.

Scale



The scale is used to select a value from a predefined range by moving a slider. The number shows the current value. To change the value, follow the steps below:

1. Place the pointer over the slider.
2. Press and hold down the left mouse button.
3. Move the pointer along the slider bar to change the value.
4. Release the left mouse button.

Tabs

Many of the **Airpak** panels (e.g., object editing panels) include tabbed regions that separate different categories of input fields. For instance, the **Walls** panel includes four tabs: **Info**, **Geometry**, **Properties**, and **Notes**, as seen in Figure 2.1.37. Here, you can access the appropriate category for the object, in this case, a wall. To display the contents of a particular tab, select the tab label with the left mouse button.

Many tabbed panels have the following types of tab categories:

Info displays general information about the object such as its name, its group, and other display properties.

Geometry displays geometric information such as its shape, its plane, and its coordinates.

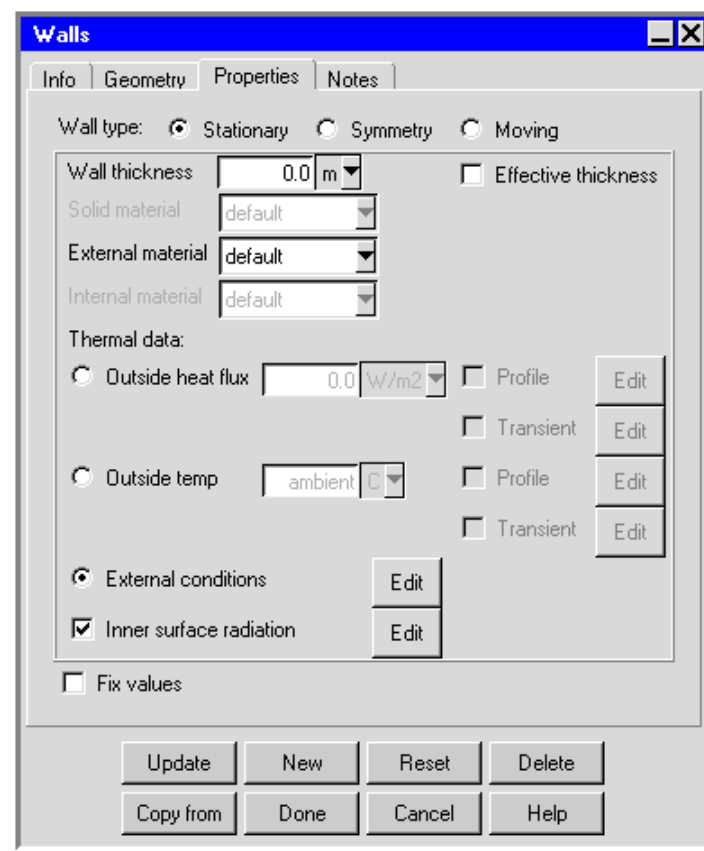



Figure 2.1.37: An Example of a Tabbed Panel


Properties provides access to the object's material and thermal properties.

Notes provides an area for you to leave notations about the object.

Note that some object and macro panels include different tabs that are based on the properties of the object.

Floating Toolbars

Most of the **Airpak** toolbars can be detached from the **Main** window and exist as *floating toolbars* that can be moved to any position in the window. Floating toolbars are identified by two small buttons in the upper right hand corner of the toolbar ().

You can detach a toolbar, by clicking on the  button. To move the detached toolbar, select the title bar and drag the toolbar to a new position in the **Main** window. To resize a detached toolbar, click and drag the edge of the toolbar to the desired position using the left mouse button.

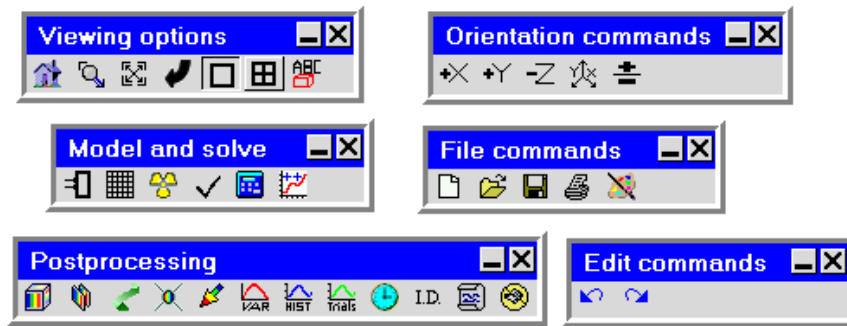





Figure 2.1.38: Floating Toolbars

To hide an attached toolbar entirely from view in the **Airpak** graphical interface, click on the  button.

You can re-attach a floating toolbar to the **Airpak** interface, by clicking on the  button in the upper right hand corner of the floating toolbar. To hide a floating toolbar entirely from view in the **Airpak** graphical interface, select the  icon in the upper right hand corner of the floating toolbar.

Whether attached or detached, you can retrieve a hidden toolbar using the **Windows** menu or selecting the **Edit toolbars** option in the **View** menu and using the **Available toolbars** panel.

View—>Edit toolbars

2.1.10 Accessing On-line Help

There are three types of help available in **Airpak**: on-line help, bubble help, and context-specific help. On-line help provides access to on-line versions of the **Airpak** manuals. Bubble help provides a brief explanation of the function performed by items in the **Main** window, the **Message** window, the **Edit** window, the **Model manager** window, the various toolbars, and control panels. Context-specific help provides specific information for a panel or window.

On-Line Help

To invoke the on-line help system, select the **Help** option in the **Help** menu. The HTML version of the **Airpak** User's Guide will be displayed in your web browser. **Airpak** will automatically launch your web browser. You can access information in the manual through the hyperlinks in the table of contents and index. To access the **Airpak** Tutorial Guide, click the **Return to Home** button at the top of the page and then select the HTML version

of the Tutorial Guide from the **Airpak** documentation home page. See Section 1.6 for information on using the on-line manuals.

Bubble Help

Bubble help (i.e., tool tips) is available for radio buttons, push buttons, and toggle buttons in the **Main** window, the **Message** window, the **Edit** window, the toolbars, and control panels. To use bubble help, hold the mouse pointer over an item for a few seconds. A bubble will appear giving a brief description of the function of the item. You can disable the bubble help, as described in Section 6.3.

Context-Specific Help

To use context-specific help, move the mouse pointer to any location within the panel you require help for, and press the **F1** key. **Airpak** will automatically launch your web browser, open the appropriate on-line document, and locate the heading or figure within the document that relates to the panel in question.

2.2 Using the Mouse

The mouse is used as the primary means of interacting with the graphical user interface (GUI) to access **Airpak**'s functionality. To take full advantage of the functionality available in **Airpak**, you will need a three-button mouse.

The mouse can be used to provide inputs to control panels, display control panels, access objects in the **Model manager** window, as well as manipulate objects in the graphics window.

2.2.1 Controlling Panel Inputs



The left mouse button is used to control panel inputs in the following ways:

- Executing selector button functions (e.g., push buttons, radio buttons, toggle buttons)
- Highlighting items in a list
- Enabling a text field for typing

In addition, the middle mouse button allows you to drag into view text entries or list items that overflow the field.

2.2.2 Using the Mouse in the Model manager Window

The left mouse button is used in the **Model manager** window in the following ways:

- Opening and closing tree nodes by clicking on the  or  icons to the left of the tree node name.
- Selecting node items by clicking on the item in the **Model manager** window. The item is highlighted when it is selected.
- Dragging and dropping items to other locations in the tree. To do this, hold down the left mouse button on an item (e.g., in the **Group**, **Model**, or **Materials** nodes), drag the item to another area of the tree, and drop the item into the tree by releasing the left mouse button.
- Selecting and operating on multiple items in the **Model manager** window by holding down the <Control> key while you select items. To select a succession of items (e.g., fan.1, fan.2,..., fan.10) select the first item (e.g., fan.1), hold down the <Shift> key and select the last item (e.g., fan.10). All items between the first item and the last item will become selected in the **Model manager** window.
- Double-clicking on certain tree items will open a control panel for additional input (e.g., to set project parameters and options, or edit object properties).

In addition, there are context menus available in the **Airpak Model manager** window, as described in the next section.

2.2.3 Using the Context Menus in the Model manager Window

Airpak includes a context menu that you can access by holding down the right mouse button on certain objects that are selected in the **Model manager** window. The context menu is useful in the **Model manager** window when you want to quickly perform common tasks on the objects in your model.

The Main library Node Context Menu

When the **Main library** node is selected, the context menu includes the following options:

Edit info opens the **Library name and info** panel, where you can view and edit the library name and information fields. See [Section 7.7](#) for details.

Paste from clipboard allows you to add objects or materials to the **Airpak** library that you have placed in the clipboard. See [page 2-59](#) for more information about the clipboard.

Refresh updates the library if any changes (additions or subtractions) have been made to the library repository.

The Groups Node Context Menus

When the **Groups** node is selected, the context menu includes the following options:

Create allows you to name a new group and then add the new group as an item under the **Groups** node.

When an individual group node is selected under the **Groups** node (e.g., **group.1**), the context menu includes the following options:

Paste from clipboard pastes the group from the clipboard to your model. See page 2-59 for more information about the clipboard.

Properties opens the **Group parameters** panel, where you can set color, line width, and shading properties for objects in the group.

Rename allows you to rename the group.

Delete moves the group to the **Trash** node.

Add allows you to add objects to a group by selecting a point or region on the screen or choosing an object name or pattern.

Remove allows you to remove objects from a group by selecting a point or region on the screen or choosing an object name or pattern.

Copy group opens the group **Copy** panel, where you can copy a group then scale, rotate, translate, or mirror the copied group.

Move group opens the group **Move** panel, where you can scale, rotate, translate, or mirror a group.

Edit objects opens the object **Edit** window, where you can edit object properties if all objects in the group are of the same type.

Visible toggles the display of the group in the graphics window.

Activate all activates all inactive objects in the group.

Deactivate all deactivates all active objects in the group.

Delete all deletes all objects in the group

Total volume prints the total volume of all objects from the group in the **Message** window.

Total area prints the total area of all objects from the group in the **Message** window.

Create assembly creates an assembly out of the group. See Section 7.8 for details.

Copy params applies the parameters of the selected object to all objects of the same type in the selected group.

Save as project allows you to save the selected group as a separate **Airpak** project.

See Section 7.6 for details about grouping objects.

When an individual item in a **Groups** node is selected (e.g., **block.1**), the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Paste from clipboard pastes the item from the clipboard to your model. See page 2-59 for more information about the clipboard.

Rename allows you to rename the item.

Copy object allows you to copy the selected object.

Move object allows you to move the selected object.

Edit object allows you to edit the properties of the item.

Delete moves the item to the **Trash** node.

Create assembly allows you to create an assembly from the selected item.

Create group allows you to create a group from the selected item.

Active allows you to toggle the item's activity.

Visible allows you to toggle the item's visibility. Hidden objects appear in the **Model manager** window as grayed out items and are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest.

Total volume prints the total volume of selected objects in the **Message** window.

Total area prints the total area of selected objects in the **Message** window.

Create monitor point allows you to create monitor points for selected objects. This will also copy the selected objects from the **Model** node into the **Points** node. See Section 27.8.1 for more information on object monitor points.

Edit mesh parameters opens up the **Per-object parameters** panel, where you can review and define meshing parameters specific to the selected object. See Section 26.6.1 for more information on defining object-specific meshing parameters.

Create object face allows you to create object faces for selected objects. Newly created object faces will show up under the **Post-processing** node. See Section 28.2.4 for more information on defining object faces.

Remove from group allows you to remove the object from the group. This option appears in the context menu only for selected objects under the **Groups** node.

See Chapter 7 for more information about editing objects and groups of objects.

The Post-processing Node Context Menu

When an item in the **Post-processing** node is selected, the context menu includes the following options:

Active allows you to toggle the postprocessing object's activity. If this option is turned off, the postprocessing object is placed under the **Inactive** node.

Edit allows you to edit the postprocessing object.

Delete moves the postprocessing object to the **Trash** node.

See Chapter 28 for more information about postprocessing in **Airpak**.

The Points Node Context Menus

When the **Points** node is selected, the context menu includes the following option:

Create at location displays the **Point** panel, where you can define the location of solution monitor points. See Section 27.8.1 for details.

Paste from clipboard allows you to add objects or materials to the **Airpak** library that you have placed in the clipboard. See page 2-59 for more information about the clipboard.

When an item in the **Points** node is selected, the context menu includes the following options (see Section 27.8.1):

Edit info displays the **Point** panel, where you can monitor the temperature, pressure, and velocity parameters at a central point within an object.

Move displays the **Move point** panel, where you can specify a new coordinate location for an existing monitor point.

Copy displays the **Copy point** panel, where you can specify parameters for copying an existing monitor point.

Delete deletes the monitor point.

The Trash Node Context Menus

When the **Trash** node is selected, the context menu includes the following options:

Paste from clipboard empties the contents of the clipboard into the **Trash** node. See page 2-59 for more information about the clipboard.

Empty trash deletes the contents of the **Trash** node completely from the **Airpak** project.

When an item in the **Trash** node is selected, the context menu includes the following option:

Undelete allows you to add the deleted object back to your model.

The Inactive Node Context Menu

When an item in the **Inactive** node is selected, the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Paste from clipboard pastes the item from the clipboard to your model. See page 2-59 for more information about the clipboard.

Rename allows you to rename the item.

Copy object allows you to copy the selected object.

Move object allows you to move the selected object.

Edit object allows you to edit the properties of the item.

Delete moves the item to the **Trash** node.

Create assembly allows you to create an assembly from the selected item.

Create group allows you to create a group from the selected item.

Active allows you to toggle the item's activity.

Visible allows you to toggle the item's visibility. Hidden objects appear in the **Model manager** window as grayed-out items and are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest.

Total volume prints the total volume of selected objects in the **Message** window.

Total area prints the total area of selected objects in the **Message** window.

Create monitor point allows you to create monitor points for selected objects. This will also copy the selected objects from the **Model** node into the **Points** node. See Section 27.8.1 for more information on object monitor points.

Edit mesh parameters opens up the **Per-object parameters** panel, where you can review and define meshing parameters specific to the selected object. See Section 26.6.1 for more information on defining object-specific meshing parameters.

Create object face allows you to create object faces for selected objects. Newly created object faces will show up under the **Post-processing** node. See Section 28.2.4 for more information on defining object faces.

Remove from group allows you to remove the object from the group. This option appears in the context menu only for selected objects under the **Groups** node.

The Model Node Context Menus

When the **Model** node is selected, the context menu includes the following options:

Paste from clipboard empties the contents of the clipboard into your model. See page 2-59 for more information about the clipboard.

Create allows you to add an **Airpak** object (e.g., block, fan, wire, etc.) to your model.

Merge project allows you to merge an existing project with your current project using the **Merge project** panel.

Load assembly allows you to load an assembly as another **Airpak** project using the **Load project** panel.

When an item in the **Model** node is selected (with the exception of the **Room**, the **Materials** node, and any assemblies), the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Paste from clipboard pastes the item from the clipboard to your model. See page 2-59 for more information about the clipboard.

Rename allows you to rename the item.

Copy object allows you to copy the selected object.

Move object allows you to move the selected object.

Edit object allows you to edit the properties of the item.

Delete moves the item to the **Trash** node.

Create assembly allows you to create an assembly from the selected item.

Create group allows you to create a group from the selected item.

Active allows you to toggle the item's activity.

Visible allows you to toggle the item's visibility. Hidden objects appear in the **Model manager** window as grayed out items and are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest.

Total volume prints the total volume of selected objects in the **Message** window.

Total area prints the total area of selected objects in the **Message** window.

Create monitor point allows you to create monitor points for selected objects. This will also copy the selected objects from the **Model** node into the **Points** node. See Section 27.8.1 for more information on object monitor points.

Edit mesh parameters opens up the **Per-object parameters** panel, where you can review and define meshing parameters specific to the selected object. See Section 26.6.1 for more information on defining object-specific meshing parameters.

Create object face allows you to create object faces for selected objects. Newly created object faces will show up under the **Post-processing** node. See Section 28.2.4 for more information on defining object faces.

Remove from group allows you to remove the object from the group. This option appears in the context menu only for selected objects under the **Groups** node.

See Chapter 7 for more information about adding objects and assemblies to your model, as well as about editing objects and groups of objects.

The Room Context Menu

When the Room in the **Model** node is selected, the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Paste from clipboard pastes the item from the clipboard to your model. See page 2-59 for more information about the clipboard.

Rename allows you to rename the item.

Move object allows you to move the selected object.

Edit object allows you to edit the properties of the item.

Visible allows you to toggle the item's visibility. Hidden objects appear in the **Model manager** window as grayed out items and are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest.

Total volume prints the total volume of the selected object in the **Message** window.

Total area prints the total area of the selected object in the **Message** window.

The Materials Node Context Menu

When an item in the **Materials** node is selected, the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Copy material copies the item and adds it to your **Airpak** model.

Rename allows you to rename the item.

Edit allows you to edit the properties of the item.

Delete moves the material item to the **Trash** node.

See Section 7.7 for more information about materials.

The Assembly Node Context Menu

When an assembly node is selected, the context menu includes the following options:

Add to clipboard copies the item to the clipboard. See page 2-59 for more information about the clipboard.

Paste from clipboard pastes the item from the clipboard to your model. See page 2-59 for more information about the clipboard.

Create allows you to create another **Airpak** object (e.g., block, fan, etc.) and adds it to the assembly.

Merge project allows you to merge an existing project with your current project using the **Merge project** panel. See Chapter 3 for more details.

Load assembly allows you to load an assembly as another **Airpak** project using the **Load project** panel.

Save as project saves the assembly as an **Airpak** project.

View separately allows you to move the assembly item up to the level of the **Model** node in the **Model manager** window.

Rename allows you to rename the assembly item.

Copy object allows you to copy the selected assembly object.

Move object allows you to move the selected assembly object.

Edit object allows you to edit the properties of the assembly item.

Delete moves the assembly item to the **Trash** node.

Create assembly allows you to create an assembly from the selected assembly item.

Active allows you to toggle the assembly item's activity.

Visible allows you to toggle the assembly item's visibility.

Open subtree opens the assembly tree structure in the **Model manager** window.

Close subtree closes the assembly tree structure in the **Model manager** window. Hidden objects appear in the **Model manager** window as grayed out items and are not visible in the graphics window. This allows you to view and edit portions of your model while hiding the rest.

Summary information opens the **Assembly contents** panel which lists the total number of objects in the assembly along with the number of objects of each individual object-type (see Section [7.8.13](#)).

Total volume prints the total volume of all objects from the assembly in the **Message** window.

Total area prints the total area of all objects from the assembly in the **Message** window.

See Section [7.8](#) for more information about assemblies.

Using the Clipboard

In the **Libraries**, **Groups**, **Inactive**, and **Model** nodes (including items within the **Materials** node or an **Assembly** node under the **Model** node) of the **Model manager** window, the context menu allows you to copy items to a temporary holding area called a clipboard. You can select objects from these nodes, copy them to the clipboard and paste them into other nodes of the **Model manager** window (**Libraries**, **Groups**, **Monitor points**, **Inactive**, **Trash**, and **Model** nodes). The clipboard is especially useful when you have a large **Airpak** model and you need to move one or more objects from one node of the **Model manager** window to another.

To use the clipboard, right-click on an appropriate item and select **Add to clipboard** from the resulting context menu. Next, right-click on the destination node in the **Model manager** window and select **Paste from clipboard**.

You can add more than one item to the clipboard by selecting multiple items in the **Model manager** window while holding down the <Control> key, displaying the context menu, and using the **Add to clipboard** option. When an item has been added to the clipboard, **Airpak** displays the number of clipboard items under the **Model manager** window.

2.2.4 Manipulating Graphics With the Mouse

You can modify the view of your **Airpak** model in the graphics window using the mouse. You can use the mouse buttons (left, right, and middle), either alone or in combination with a keystroke, to perform the following graphic manipulation functions:

- Rotating, translating, and zooming in on the entire model
- Adding, selecting, translating, and resizing individual objects
- Selecting and translating title, date, and axes
- Changing the spectrum on the color legend

You can change the default mouse controls in **Airpak** to suit your preferences using the **Mouse buttons** section of the **Preferences and settings** panel (Figure 2.2.1).

Edit—>Preferences

Rotating a Model

To rotate your model about a central point on the graphics display, position the cursor over the model, hold down the left mouse button (or the button that you specified as the **3D Rotate** button in the **Mouse buttons** section of the **Preferences** panel), and move the mouse in any direction. To rotate about an axis perpendicular to the screen, hold down the right mouse button (or the button that you specified as the **Scale/2D Rotate** button

in the **Mouse buttons** section of the **Preferences** panel) and move the mouse to the left and right.

Translating a Model

To translate your model to any point on the screen, position the cursor over the model, hold down the middle mouse button (or the button that you specified as the **Translate** button in the **Mouse buttons** section of the **Preferences** panel), and move the mouse to a new location.

Zooming In and Out

To zoom into your model, position the cursor over the model, hold down the right mouse button (or the button that you specified as the **Scale/2D Rotate** button in the **Mouse buttons** section of the **Preferences** panel), and move the mouse up (or in the direction that you specified as the **Zoom in** direction in the **Mouse buttons** section of the **Preferences** panel). To zoom out from your model, hold down the mouse button and move the mouse in the opposite direction.

Adding Objects to the Model

To add objects to your **Airpak** model using the mouse, hold down the left mouse button on one of the buttons in the **Object creation** toolbar (e.g., wall, fan, opening, etc.) and drag the cursor inside the graphics window. The new object will appear inside the room and you can continue to drag the object in the graphics window. Release the mouse button to set the object's position in the graphics window.

Selecting Objects Within a Model

To select an individual object in the graphics display, hold down the <Shift> key and use the left mouse button (or the button that you specified as the **Select/2D Rotate** button in the **Mouse buttons** section of the **Preferences** panel) to click on the object.

Translating Objects Within a Model

To translate an individual object in a model, hold down the <Shift> key and use the middle mouse button (or the button that you specified as the **Translate** button in the **Mouse buttons** section of the **Preferences** panel) to select the object and drag it to its new location.

You can also snap an object to the nearest edge or vertex of existing geometry while you are translating the object. See [Section 6.3](#) for details about enabling this feature.

Resizing Objects Within a Model

To resize an individual object in a model, hold down the <Shift> key, use the right mouse button (or the button specified as the **Scale/2D Rotate** button in the **Mouse buttons** section of the **Preferences** panel) to select the object, and then move the mouse to shrink or enlarge the object.

Moving the Display Identifiers

To move the title, date, coordinate axes, and color legend in the graphics display, hold down the <Control> key (or the key specified as the **Annotation edit** key in the **Editing** section of the **Preferences** panel, described in Section 6.3), use the middle mouse button to select an identifier, and then move the mouse to a new location.

Changing the Color Spectrum

To change the spectrum of color in the graphics display, position the cursor over the color spectrum, hold down the <Control> key (or the key specified as the **Annotation edit** key in the **Editing** section of the **Preferences** panel, described in Section 6.3), press and hold down the right mouse button, and drag the legend value lines up or down the spectrum.

Changing the Mouse Controls

You can change the default mouse controls in **Airpak** to suit your preferences using the **Mouse buttons** section of the **Preferences** panel.

Edit—>Preferences

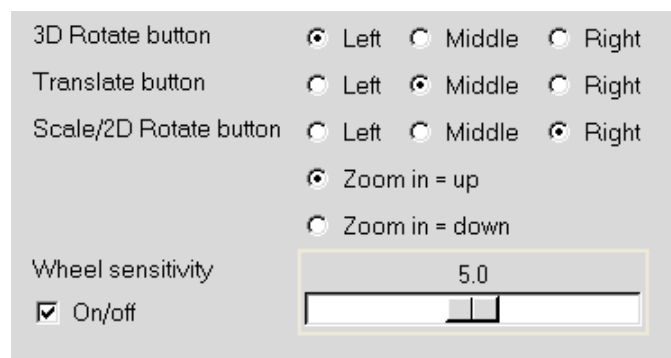


Figure 2.2.1: The **Mouse buttons** section of the **Preferences** panel

To change the default mouse control for a manipulation function, select the relevant button (Left, Middle, or Right). You can apply the changes either to the current project

by clicking **This project**, or to all **Airpak** projects by clicking **All projects** button. To close the Preferences panel without applying any changes, click **Cancel**.

i Note that, in this manual, descriptions of operations that use the mouse assume that you are using the default settings for the mouse controls. If you change the default mouse controls, you will need to use the mouse buttons you have specified, instead of the mouse buttons that the manual tells you to use.

Switching Between Modes

When you are using the mouse for a specific function in the graphics window other than manipulating graphics (described in Section 2.2.4), you can switch between the specific function and the graphics manipulation functions using the **F9** key on the keyboard. For example, if you are using the **Probe** option in the postprocessing objects **Edit** window to display the value of a variable at a point, and you want to rotate your model to see the value of the variable, press the **F9** key on the keyboard to switch the mode of the mouse from the **Probe** function to the graphics manipulation functions, and then use the mouse to rotate the display. When you have finished manipulating the graphics, press the **F9** key to switch the mode of the mouse back to the **Probe** function.

2.3 Using the Keyboard

You can also use the keyboard to modify the direction from which you view your model in the graphics window. The “hot keys” that are available in **Airpak** are listed below. You can view help for these keys in the **Message** window by selecting the **List shortcuts** option under the **Help** menu, or by typing **?** in the **Airpak** graphics window.

i Note that these hot keys are case-sensitive.

- **Control-a** toggles active objects
- **Control-c** copies and moves selected objects or groups
- **Control-e** edits an object or postprocessing object
- **Control-f** finds in tree
- **Control-l** opens the main version of the model
- **Control-m** opens/closes the **Model** subtree
- **Control-n** creates a new project
- **Control-o** opens an existing project

- **Control-p** prints the screen
- **Control-r** performs a redo of one or more previously undone operations
- **Control-s** saves the project
- **Control-t** opens/closes the currently-selected tree node
- **Control-v** toggles object visibility
- **Control-w** toggles between solid, selected solid, and wireframe shading of the model.
- **Control-x** moves selected objects
- **Control-z** performs an undo of the previous operation to the model
- **Delete** deletes the current object
- **F1** displays the main help page for **Airpak**
- **F5** sets the model's wireframe offset
- **F6** sets the model's wireframe offset to 0
- **F7** increments the model's wireframe offset. This allows the lines in your model to be drawn at a different depth than the solid colors. **Airpak** will move the lines forward toward you.
- **F8** decrements the model's wireframe offset. This allows the lines in your model to be drawn at a different depth than the solid colors. **Airpak** will move the lines away from you.
- **Shift-i** displays the isometric view of the model.
- **Shift-r** displays the reverse view of the model.
- **Shift-x** views the model toward the direction of the negative x axis.
- **Shift-y** views the model toward the direction of the positive y axis.
- **Shift-z** views the model toward the direction of the negative z axis.
- **Shift-?** prints the keyboard shortcuts in the **Message** window.
- **h** selects the default view of your model directed along the negative z axis.
- **s** scales the view to fit the graphics window.

- **z** allows you to focus on any part of your model by opening and resizing a window around the desired area. Position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the graphics window.

In addition, each menu in the **Main Menu** bar has a keyboard shortcut so that the menu and its options can be accessed using the keyboard. A combination of the <Alt> key and the underlined letter in the menu label will open the menu using the keyboard. You can then use the arrow keys on the keyboard to scroll through the menu's options and sub-options.

2.4 Quitting Airpak

You can exit **Airpak** by selecting **Quit** in the **File** menu.

File→Quit

If the present state of the project has not been written to a file, you will receive a warning message as shown in Figure 2.4.1.

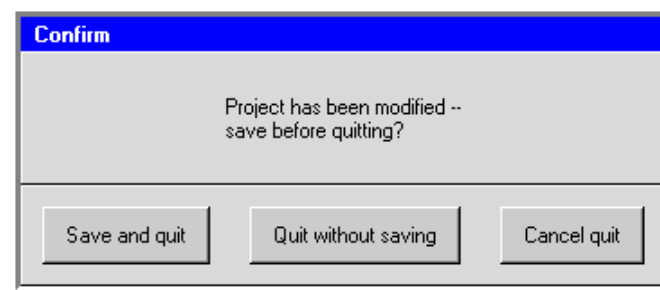


Figure 2.4.1: Warning Message Displayed When Quitting **Airpak** Before Saving Your Model

You can save the current project and quit **Airpak** by clicking the **Save and quit** button. You can quit **Airpak** without saving the current job by clicking the **Quit without saving** button. To cancel the quit, click the **Cancel quit** button.

Chapter 3. Reading, Writing, and Managing Files

This chapter describes the files that are read into and written out by **Airpak**, and how **Airpak** manages these files. Files that can be imported into **Airpak** from third-party packages (e.g., IGES files) are discussed in Chapter 4. Information in this chapter is divided into the following sections:

- Section 3.1: Overview of Files Written and Read by **Airpak**
- Section 3.2: Files Created by **Airpak**
- Section 3.3: Merging Model Data
- Section 3.4: Saving a Project File
- Section 3.5: Saving Image Files
- Section 3.6: Packing and Unpacking Model Files
- Section 3.7: Cleaning up the Project Data

3.1 Overview of Files Written and Read by **Airpak**

Table 3.1.1 lists the files that **Airpak** can read and/or write. You can use this table to get an overview of the files you may be using, to find out which codes write to a particular file, and to see where to look for more information on each file.

Table 3.1.1: Files Written or Read by Airpak

File Type	Created by	Used by	Default Suffix or Filename	See...
Model	Airpak	Airpak	model	Sec. 3.2.1
Problem	Airpak	Airpak	problem	Sec. 3.2.1
Job	Airpak	Airpak	job	Sec. 3.2.1
Mesh input	Airpak	mesher	grid_input	Sec. 3.2.2
Mesh output	mesher	Airpak	grid_output	Sec. 3.2.2
Case	Airpak	FLUENT	.cas	Sec. 3.2.3
Data	FLUENT	FLUENT	.dat and .fdat	Sec. 3.2.3
Residual	FLUENT	Airpak	.res	Sec. 3.2.3
Script	Airpak	Airpak	.SCRIPT or _scr.bat	Sec. 3.2.3
Solver input	Airpak	FLUENT	.uns_in	Sec. 3.2.3
Solver output	FLUENT	–	.uns_out	Sec. 3.2.3
Diagnostic	Airpak	–	.diag	Sec. 3.2.3
Optimization	Airpak	optimizer	.log, .dat, .tab .post, and .rpt	Sec. 3.2.4
Postprocessing	FLUENT	Airpak	.resd	Sec. 3.2.5
Log	Airpak	Airpak	.log	Sec. 2.1.6
Geometry	assorted	Airpak	.igs, .dxf, .eco, etc.	Chapt. 4
Image	Airpak	assorted	.gif, .jpg, .ppm, .tiff, .vrml, and .ps	Sec. 3.5
Packaged	Airpak	Airpak	.tzip	Sec. 3.6

3.2 Files Created by Airpak

Airpak creates files during the course of a simulation that are related to setting up the problem, generating a mesh, calculating a solution, and postprocessing the results. These files are described below.

3.2.1 Problem Setup Files

Airpak creates several files that relate to the setup of the simulation:

- The **model** file contains information related to the model: boundary conditions and geometry information.
- The **problem** file contains information on the settings for the problem: the under-relaxation factors, units for objects in the model, information on the color of objects, parameters for the mesh generator, units to be used for postprocessing, and information on default settings in the model.
- The **job** file has information on the project title and notes for the model.

Airpak saves each of these files for the current project when you save the project.

Airpak also saves different versions of the **model** and **problem** files for different version numbers of the same project. For example, if you want to run the same problem using different power settings for a component, then each solution can be saved using a different solution ID. The model and problem files for each solution will be saved using a different name, e.g., **projectname.model** and **projectname.problem**.

3.2.2 Mesh Files

The meshing procedure in Airpak creates two files that relate to the generation of the mesh for your simulation:

- The mesh input file (e.g., **grid.input**) contains the inputs for the mesh generator.
- The mesh output file (e.g., **grid.output**) contains the output from the mesh generator; i.e., the mesh file.

3.2.3 Solver Files

Airpak creates several files that are used by the solver to start the calculation:

- The case file (**projectname.cas**) contains all the information that is needed by Airpak to run the solver.

- The diagnostic file (`projectname.diag`) contains information about the correspondence between object names in the model file and object names in the case file.
- The solver input file (`projectname.uns.in`) is read by the solver to start the calculation.
- The script file (`projectname.SCRIPT` on UNIX systems, `projectname_scr.bat` on Windows systems) runs the solver executable, and can also be used to run the solver in batch mode.

Two files are created while the solver is running:

- The residual file (`projectname.res`) contains information about the convergence monitors.
- The solver output file (`projectname.uns_out`) contains information from the solver that is displayed on the screen during the calculation. Note that this file is written only on UNIX systems.

The solver saves two files when it has finished calculating: `projectname.dat` and `projectname.fdat`. These data files can be used to restart the solver (see Section 27.11.1).

3.2.4 Optimization Files

Airpak creates several files that are used during an optimization process:

- The log file (`optimization.log`) contains information about all the trials performed during the optimization run. It also contains design variable inputs and optimizer outputs.
- The input file (`in.dat`) is the input file that is read by the optimizer.
- The output file (`out.dat`) is the output file that is written out by the optimizer.
- The tab file (`optimization.tab`) contains the optimization data displayed in the Optimization run panel (see Chapter 23). This data is also used during plotting of optimization variables and functions.
- The postprocessing files (`optimization.post` and `optimization.rpt`) contain information required by the Airpak postprocessor to compute all the optimization function values during the optimization run.

3.2.5 Postprocessing Files

The solver creates a file (`projectname.resd`) that is used by Airpak for postprocessing.

3.3 Merging Model Data

Airpak allows you to merge an existing project into your current project. Both the existing project and the current project are defined with respect to a global (x,y,z) coordinate system. When the two projects have been merged, they will coexist in this global coordinate system. In many cases, this can result in objects from one project being overlaid on objects from another project. The Merge project panel includes options to perform geometric transformations on the existing project *before* it is merged into the current project.

To merge two projects, you can use the Merge project panel (Figure 3.3.1). To open the Merge project panel, select Merge project in the File menu.

File→Merge project

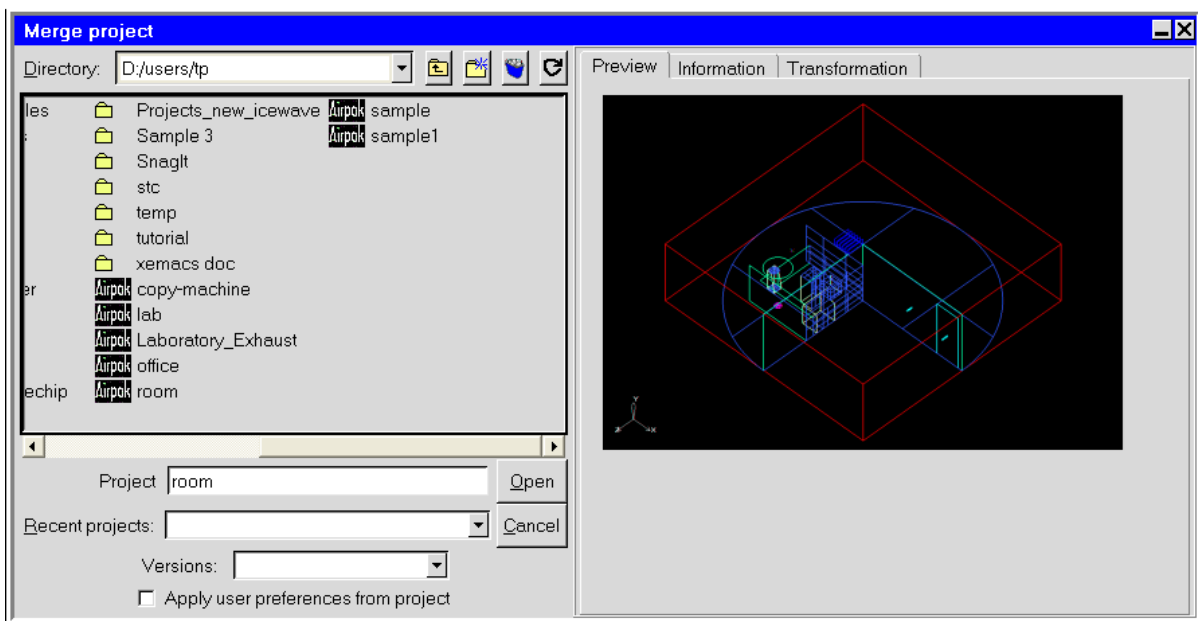


Figure 3.3.1: The Merge project Panel (Preview Tab)

1. Select the existing file to be merged with the current file. See Section 2.1.8 for details on selecting a file.
2. Click on the Transformation tab.
3. (optional) Enable the appropriate Geometric transformation options. See Section 3.3.1 for details.

4. In the **Group for merged objects** text entry box, enter a group name for all the objects in the existing project you are merging. The default name is **merge.n**, where **n** is a sequential integer starting with zero. Once a project has been merged, further **Copy** or **Move** transformations can be applied to this group using the **Group control** panel (see Section 7.6 for more details about groups). If the **Group for merged objects** field is left blank, a group will not be created.
5. If you want to apply the settings you specified under **Options** in the **Preferences** panel (see Chapter 6) for the current project to the project that is being merged, turn on the **Apply user preferences from project** option.
6. Click **Open** to merge the two projects (or click **Cancel** to close the panel without merging the projects).

Note that when a project is selected in the directory list, information about the title, available versions, and notes saved with the project is displayed under the **Information** tab of the **Merge** project panel.

You can also use the **Merge** project panel to delete projects (see Section 6.2.4), or to create a new directory (see Section 2.1.8).

3.3.1 Geometric Transformations

Airpak can transform the existing model data you are merging with your current model by using a combination of up to four geometric transformations: translation, rotation, mirroring, and scaling. To access the geometric transformations options, you will click on the **Transformation** tab in the **Merge** project panel. Only the transformations selected in the **Merge** project panel are performed on the merged geometry. If multiple geometric transformations are selected, **Airpak** applies them in the order in which they appear in the panel. For example, if both the **Rotate** and **Translate** options are selected, the imported geometry is rotated first and then translated. Note that not all combinations of transformations are commutative; i.e., the result is order-dependent, particularly if reflection is used.

To access the geometric transformations options, you will click on the **Transformation** tab in the **Merge** project panel (Figure 3.3.2).

Scaling Merged Model Data

To scale the existing model data you are merging with your current model, turn on the **Scale** option under **Geometric transformation**. Specify the scaling factor by entering a value in the **Scale** text entry box. The scaling factor must be a real number greater than zero. Values greater than 1 will increase the size, while values less than 1 will decrease the size. To scale the existing model data by different amounts in different directions, enter the scaling factors separated by spaces. For example, if you enter **1.5 2 3** in the

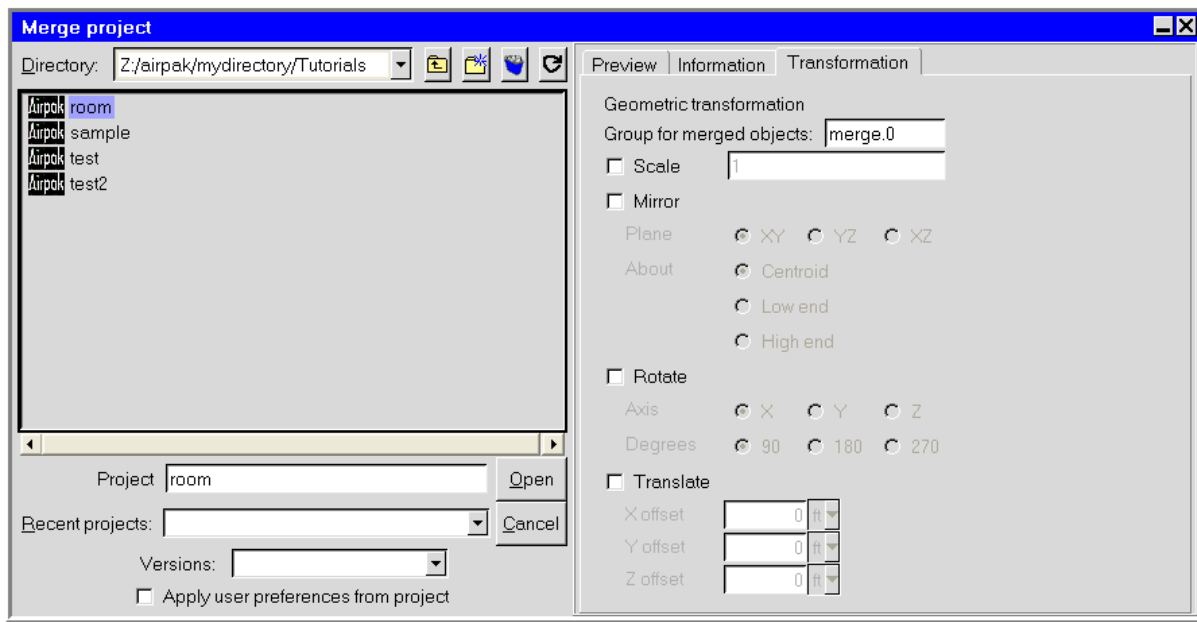


Figure 3.3.2: The Merge project Panel (Transformation Tab)

Scale text entry box, Airpak will scale the model data by 1.5 in the x direction, 2 in the y direction, and 3 in the z direction.

Mirroring Merged Model Data

To obtain the mirror image of the existing model data you are merging with your current model, turn on the **Mirror** option under **Geometric transformation**. You can specify the **Plane** across which to reflect the model data by selecting **XY**, **YZ**, or **XZ**. You can also specify the location about which the model is to be flipped by selecting **Centroid**, **Low end**, or **High end** next to **About** in the Merge project panel.

Rotating Merged Model Data


To rotate the existing model data you are merging with your current model, turn on the **Rotate** option under **Geometric transformation** in the Merge project panel. You can rotate the model data you are merging about any coordinate axis. Select **X**, **Y**, or **Z** next to **Axis**, and then select 90, 180, or 270 degrees of rotation.

Translating Merged Data

To translate the existing model data you are merging with your current model, turn on the **Translate** option under **Geometric transformation** in the Merge project panel. Define

the distance of the translation from the current origin by specifying an offset in each of the coordinate directions: **X offset**, **Y offset**, and **Z offset**. Note that all offsets are relative to the position of the existing project being merged.

3.4 Saving a Project File

To save the current project under its current name, click on the  button in the File commands toolbar or select **Save project** in the File menu. Airpak will save the project using the current name.

File→Save project

To save the current project under a different name, or for more options when saving the current project, select the **Save project as** option in the File menu. This opens the **Save project** panel (Figure 3.4.1).

File→Save project as

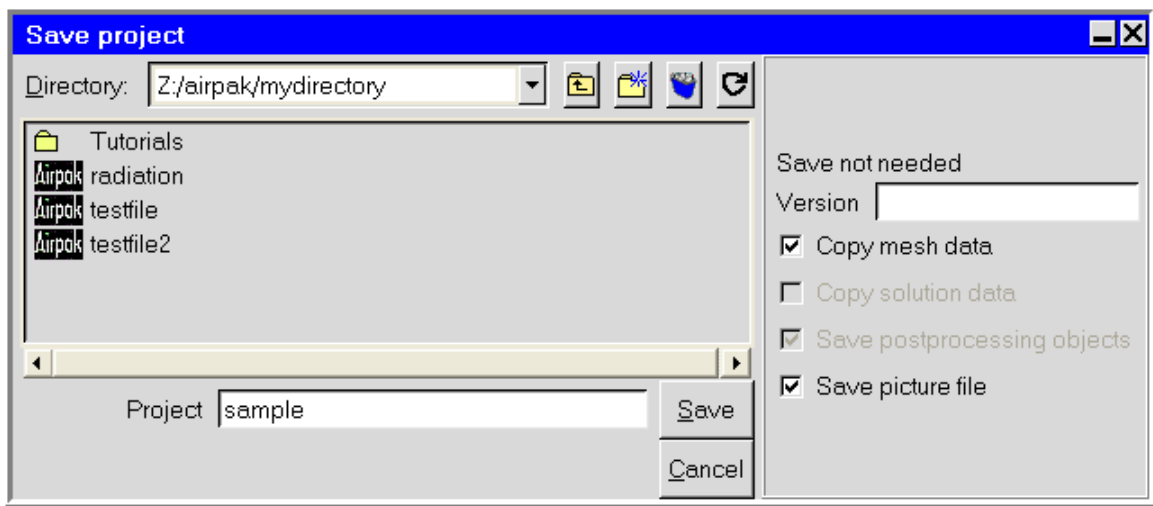


Figure 3.4.1: The Save project Panel

To save the current project, follow the steps below:

1. Specify the name of the project to be saved in the **Project** text entry box. You can choose the directory and filename in the **Directory** list. See Section 2.1.8 for more information on file selection. Alternatively, you can enter your own filename, which can be a full pathname to the file (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which Airpak was started. The filename can include any alphanumeric characters and most special characters. It cannot contain control characters, spaces, tabs, or the following characters:

\$] [} { / \ " * ?

2. Specify the **Version** for the project to be saved. See Section 2.1.8 for information on versions.
3. Select any other data to be saved with the project.
 - If you have created a mesh for the project, you can copy the mesh data to the new project by selecting the **Copy mesh data** option.
 - If you have solution data for the project, you can copy the solution data to the new project by selecting the **Copy solution data** option.
 - You can save postprocessing data with the project by selecting the **Save post-processing objects** option.
 - You can save a snapshot of the model as it currently appears in the graphics window by selecting the **Save picture file** option. The picture will be displayed when you select the project in the **Open project** panel (see Section 6.2.2) or the **Merge project** panel (see Section 3.3).
4. Click **Save** to save the current project (or click **Cancel** to close the panel without saving the current project).

You can also use the **Save project** panel to delete projects (see Section 6.2.4), or to create a new directory (see Section 2.1.8).

3.4.1 Recent Projects

Airpak keeps track of the most recent projects you saved, and allows you to select one from the **Recent projects** drop-down list in the **Open project** panel (Figure 3.4.2). You can open the **Open project** panel by choosing **Existing** in the **New/Existing** panel when starting Airpak, or at any time when you select **Open project** from the **File** menu.

3.5 Saving Image Files

Graphics window displays can be saved as image files in various formats, including TIFF, GIF, and PostScript. There may be slight differences, however, between images and the view displayed in the graphics window, since images are generated using the internal software renderer, while the graphics window may utilize specialized graphics hardware for optimum performance.

To set image parameters and save image files, you will use the **Save image** panel (Figure 3.5.1). To open the **Save image** panel, select **Create image file** in the **File** menu. This allows you to generate a file or a printed copy of your model as shown in the graphics window or in a selected region of the graphics window. Note that you can also add annotations to your image file (see Section 2.1.5).

The procedure for saving an image file is as follows:

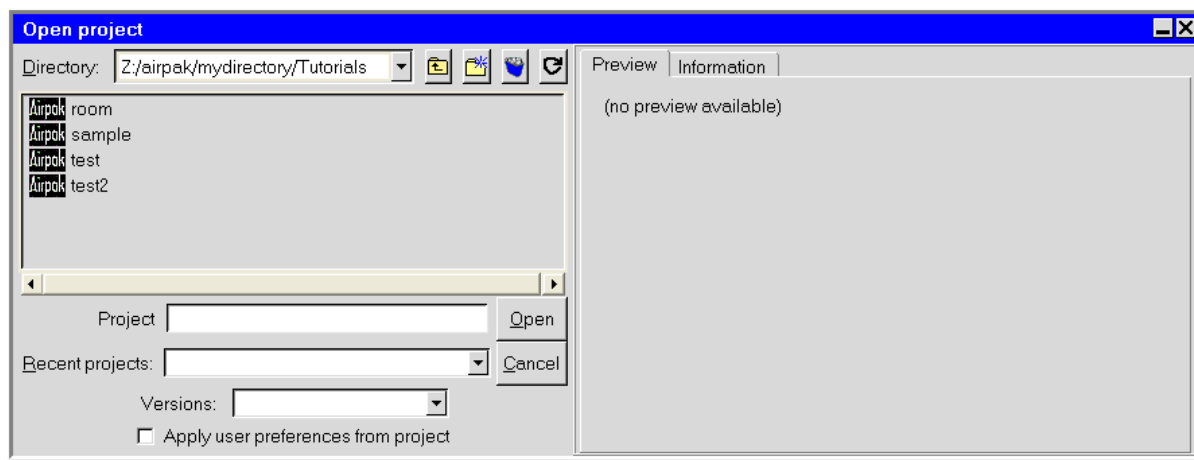


Figure 3.4.2: The Open project Panel

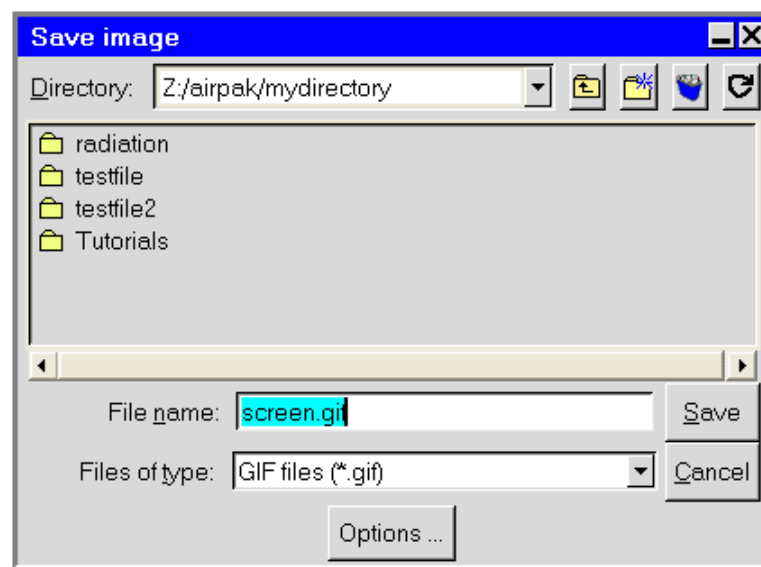


Figure 3.5.1: The Save image Panel

1. Specify the name for the image file to be saved. **Airpak** will assign a default prefix for the filename, which is shown in the **Files of type** drop-down list. You can enter your own filename, which can be a full pathname to the file (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started.
2. Select the image format. See Section 3.5.1 for details.
3. Click **Options...** to open the **Graphics file options** panel (Figure 3.5.2).

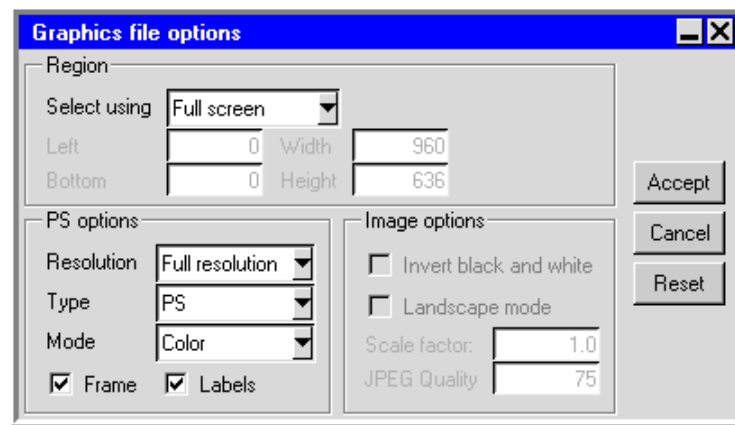


Figure 3.5.2: The Graphics file options Panel

- (a) Under **Region**, specify the desired selection method (**Full screen**, **Mouse selection**, or **Pixel location**) in the **Select using** drop-down list. If you select **Pixel location**, specify the region in the **Left**, **Bottom**, **Width** and **Height** fields. See Section 3.5.2 for more details.
- (b) Under **Image options**, use the **Invert black and white** option to control the foreground/background color. If this option is selected, the black and white colors of the graphics window being hardcopied will be swapped. This feature allows you to make images with a white background and black foreground, while the graphics window is displayed with a black background and white foreground.
- (c) If you are saving a PostScript file, set the appropriate **PS options**. See Section 3.5.1 for details.
- (d) Under **Image options**, use the **Landscape mode** option to specify the orientation of the image. If this option is turned on, the image is made in landscape mode; otherwise it is made in portrait mode.
- (e) Under **Image options**, specify a scale factor by entering a value in the **Scale factor** text entry box. The image in the image file will be scaled relative to its actual size in the graphics window. The scaling factor must be a real number

greater than zero. Values greater than 1 will increase the size, while values less than 1 will decrease the size.

- (f) Click accept to close the **Graphics file options** panel.
- 4. Click **Save** to save the image file (or click **Cancel** to close the **Save image** panel without saving the image file).

3.5.1 Choosing the Image File Format

To choose the image file format, select one of the following items in the Files of type drop-down list:

GIF files (*.gif) (Graphics Interchange Format) is a graphic image format.

JPEG files (*.jpg) (Joint Photographic Experts Group) is a graphic image format. You can define the JPEG Quality of a JPEG image under Image options in the Graphics file options panel (Figure 3.5.2). The maximum value of 100 will result in slightly reduced file compression, but there will be no loss of data when the image is decompressed. Lower values will result in more file compression, but some data will not be recovered when the image is decompressed. The default value of 75 should be acceptable for most cases.

PPM files (*.ppm) (Portable Pixmap) output is a common raster file format.

TIFF files (*.tiff) (Tagged Image File Format) is a common raster file format. The TIFF driver may not be available on all platforms.

VRML files (*.vrmf) (Virtual Reality Modeling Language) is a graphics interchange format that allows export of 3D geometrical entities that you can display in the Airpak graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window.

i Note that non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in Airpak, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Postscript files (*.ps) is a common vector file format. See below for details.

Setting Options for PostScript Files

Airpak provides several options for printing PostScript files in the Graphics file options panel (Figure 3.5.2). To enable the available options for making image PostScript files, select Postscript files (*.ps) from the Files of type drop-down list in the Save image panel (Figure 3.5.1) and click Options... This opens the Graphics file options panel.

To specify the PostScript options follow the procedure below:

1. Specify the **Resolution** by selecting one of the following items from the drop-down list:

Full resolution allows you to customize the PostScript image file using the options in steps 2–5. Image files saved using this option will have a white background instead of the black background displayed in the graphics window.

From screen will make a PostScript image file directly from what is displayed in the graphics window.

2. Specify the format in which the graphics window is stored in the output file by selecting **PS** or **EPS** from the **Type** drop-down list. EPS (Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device).
3. Specify the color mode. For a color-scale copy, select **Color** from the **Mode** drop-down list; for a gray-scale copy, select **Gray**; and for a black-and-white copy, select **Mono**. Note that most monochrome PostScript devices will render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray**.
4. Enable the **Frame** option, if desired. If this option is turned on, a frame will be included around the image in the image output.
5. Enable **Labels** for the image, if desired. The label, which consists of the project name, the machine name, and a date/time stamp, will appear at both the top and bottom of the page.
6. Click **Accept** to store the PostScript options.

3.5.2 Specifying the Print Region

There are three ways to define the region of the graphics window that should be printed to the file: you can pick the desired selection method in the **Graphics file options** panel, from the **Select using** drop-down list under **Region**.

Full screen instructs **Airpak** to use the whole graphics window as the print region.

Mouse selection allows you to select a region of the graphics window as the print region.

Airpak will ask you to define the region to be written to the file. Position the mouse pointer at a corner of the area to be included, hold down the left mouse button and

drag open a selection box to the desired size, and then release the mouse button. The selected area will be printed to the file. Note that **Airpak** will update the **Pixel location** values in the **Graphics file options** panel automatically when you define the region using the mouse.

Pixel location allows you to specify a region of the graphics window as the print region. Define the bottom left corner of the print region by specifying the **Left** and **Bottom** pixel locations. A value of 0 for both **Left** and **Bottom** specifies that the bottom left corner of the print region coincides with the bottom left corner of the graphics window. Specify the **Width** and **Height** of the region in pixels.

This option is useful if you want to capture different images of the same size and location in the graphics window. First, define the region you want to capture using the **Mouse selection** option. **Airpak** will display the pixel location values for this region in the **Graphics file options** panel. You can then use the **Pixel location** option to create further images of the same region and the same size.

3.6 Packing and Unpacking Model Files

To archive or pack up a project, select the **Pack project** option in the **File** menu.

File→Pack project

Airpak will open the **File selection** dialog box (see Section 2.1.8), in which you can specify the name of the packaged-up project. **Airpak** will combine all the relevant files for off-line (remote) diagnosis by your **Airpak** support engineer and write them to a compressed tar file with extension **.tzt**. The file is in a format suitable for transfer by electronic mail or other means to your support engineer.

To unarchive or unpack a packaged-up project, select the **Unpack project** option in the **File** menu.

File→Unpack project

Airpak will open the **File selection** dialog box, in which you can specify the name of the packaged-up project to be unpacked. **Airpak** will unpack the project and display the model in the graphics window. (You can also unpack a project using the **-unpack** command described in Section 1.5.4.)

3.7 Cleaning up the Project Data

A complete **Airpak** simulation can generate a significant amount of information. You can remove data associated with the current project from the project directory using the **Clean up project data** panel (Figure 3.7.1). You can also instruct **Airpak** to compress the files associated with a particular solution ID. This option will remove any files that are not required for further postprocessing of the solution and compress the remaining

files. When you postprocess the results associated with a compressed solution ID, Airpak automatically uncompresses the files.

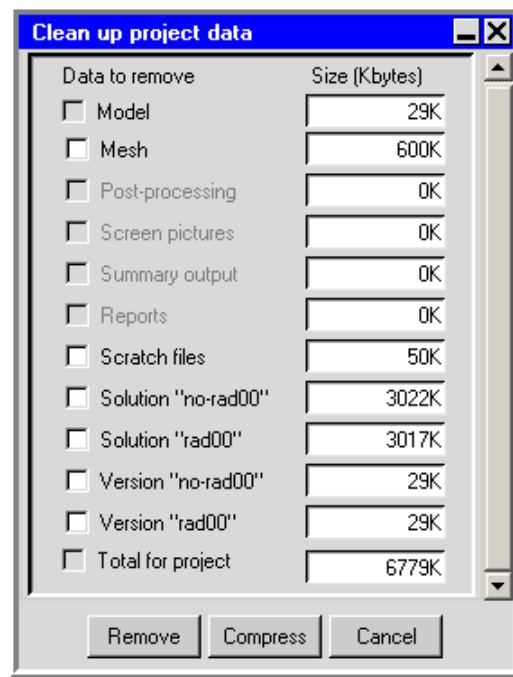


Figure 3.7.1: The Clean up project data Panel

The data associated with the current project are identified in the **Clean up project data** panel by the operation that created them (e.g., **Mesh**, **Post-processing**), and Airpak provides an estimate of the size of the data in kilobytes. The size of the total data for the current project, including the data for the model itself, is given at the bottom of the **Clean up project data** panel.

To clean up the project data, select **Cleanup** in the **File** menu.

File→**Cleanup**

This opens the **Clean up project data** panel shown in Figure 3.7.1. The following items are listed in the **Clean up project data** panel:

Model specifies the size of the data associated with the model itself. Model data cannot be removed, since it is not possible to recreate it from other data.

Mesh specifies the size of the data resulting from the mesh generation.

Post-processing specifies the size of the data resulting from postprocessing analysis.

Screen pictures specifies the size of the data resulting from requests to store copies of the graphics window.

Summary output specifies the size of the data resulting from generating a summary of the model objects.

Reports specifies the size of the data resulting from generating reports.

Scratch files specifies the size of any scratch files that were inadvertently left after a simulation.

Solution *projectname* specifies the size of the data files (i.e., **.cas**, **.dat**, **.resd**, etc.) resulting from running the solution. All solutions that exist for the current project are listed by solution ID.

Version *projectname* specifies the size of the version files (i.e., **job**, **model**, and **problem**) resulting from running the solution. All versions that exist for the current project are listed by solution ID. Version files cannot be compressed.

Total for project reports the total size of the files of all the data associated with the current project, including the model data.

To delete project data, turn on the options for the data that you want to delete, and click the **Remove** button at the bottom of the **Clean up project data** panel.

To compress project data, turn on the options for the data that you want to compress, and click the **Compress** button at the bottom of the **Clean up project data** panel.

Chapter 4. Importing and Exporting Model Files

You can import geometry that was created using a commercial CAD program into **Airpak** and also export **Airpak** files back into various other formats. There are several types of CAD file formats that are supported by **Airpak**. This chapter describes the CAD files that can be imported, the way in which each type of file can be imported, and how to export an **Airpak** project to other file formats.

Information in this chapter is divided into the following sections:

- Section 4.1: Files That Can Be Imported Into **Airpak**
- Section 4.2: Importing IGES, STEP, and Tetin Files Into **Airpak**
- Section 4.3: Importing Other Files Into **Airpak**
- Section 4.4: Exporting **Airpak** Files

4.1 Files That Can Be Imported Into **Airpak**

The following types of files can be imported into **Airpak**:

- International Graphics Exchange Specification (IGES) files containing point and line information (see Section 4.3)
- AutoCAD DXF files containing point and line information and DWG files containing curves and surfaces (see Section 4.3).
- IGES and STEP (STandard for the Exchange of Product model data) files containing surfaces and curves (see Section 4.2)
- Tetin files, a native ICEM-CFD file format (see Section 4.2) containing surfaces and curves.
- Comma separated values files (can be created or read in by spreadsheet programs like Excel) (see Section 4.3.5).
- Industrial Foundation Classes (IFC) 2x files (see Section 4.3)

All of these files can be imported using the **File**→**Import** menu. Tetin and IGES/STEP files can also be imported using the **CAD data** panel (Figure 4.2.1).

The procedure for importing each type of file is described in the sections specified above.

4.2 Importing IGES, STEP, and Tetin Surface Geometry Files Into Airpak

You can import IGES or STEP files that were created using a commercial CAD package (e.g., Pro/ENGINEER, I-deas), or tetin files (ICEM-CFD) into **Airpak** for use in heat transfer and fluid flow simulations. This allows you to use a CAD engine for your geometry design, and transfer your model to a CAE engine for design simulation and analysis. Using a CAD/CAE partnered system allows you to quickly and accurately design, prototype, and analyze mechanical system designs that may increase a product's quality and significantly reduce its time-to-market.

Airpak provides the capability to import an IGES, STEP, or tetin file, as well as utilities to simplify the geometry representation and allow the geometry to be represented as **Airpak** objects. The following shapes can be converted into **Airpak** objects: rectangles, prisms, circles, cylinders, polygons, and inclined rectangular planes.

The overall **Airpak** process for your ventilation system design involves the following steps:

1. Model ventilation design using a CAD product.
2. Save geometry in IGES, STEP, or tetin format.
3. Import IGES, STEP, or tetin file into **Airpak**.
4. Convert CAD geometry into **Airpak** objects.
5. Conduct ventilation design analysis using **Airpak**.

Note that this section discusses the import of IGES or STEP files that consist of surfaces, curves, points, or a combination of the three, and tetin files that consist of surfaces and curves. If your IGES file contains only point and line information, you can import your file more easily by selecting the **IGES points+lines** option in the **File**→**Import** menu (see Section 4.3).



Importing model geometry from IGES or STEP files is not a direct way to construct a model; it is a way of creating the geometry of objects you want in your model without specifying the dimensions from scratch.

4.2.1 Overview of Procedure for IGES, STEP, and Tetin File Import

The general procedure for importing an IGES, STEP, or tetin file into **Airpak** and cleaning up the geometry is as follows:

1. Read the IGES, STEP, or tetin file into **Airpak** (see Section 4.2.2).
2. Convert CAD geometry into **Airpak** objects (see Section 4.2.3).

The utilities you will use to read the IGES, STEP, or tetin file into Airpak and convert the CAD geometry into Airpak objects are located in the CAD data panel (Figure 4.2.1) and the CAD data operation options panel (Figure 4.2.2). To open the CAD data menu, select CAD data in the Model menu.

Model—>CAD data

Click Load, and select Load IGES/Step file or Load Tetin file from the pull-down menu.

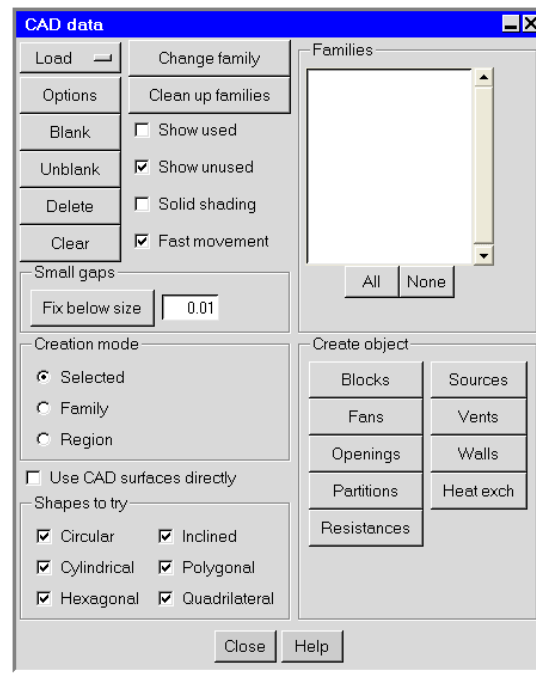


Figure 4.2.1: The CAD data Panel

Alternatively, you can import an IGES file containing surfaces and lower topology by selecting IGES/Step surfaces+curves in the File—>Import menu, or a tetin file by selecting Tetin surfaces+curves in the File—>Import menu,

To open the CAD data operation options panel, select Options in the CAD data panel.

4.2.2 Reading an IGES, STEP, or Tetin File Into Airpak

When you read an IGES or STEP file into Airpak, Airpak will first convert the IGES or STEP file into a tetin file (a native ICEM-CFD geometry format). The tetin file will then be read into Airpak.

To read an IGES or STEP file (or tetin file) into Airpak, follow the steps below.

1. Specify a minimum feature size for imported features and specify whether you want

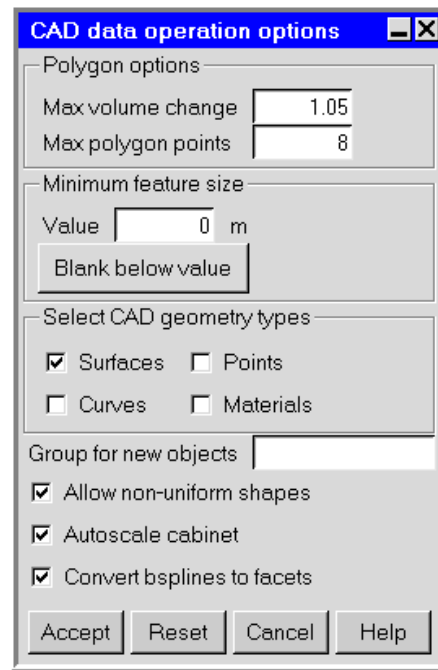


Figure 4.2.2: The CAD data operation options Panel

Airpak to scale the room after the CAD geometry has been imported. To open the CAD data operation options panel (Figure 4.2.2), click **Options** in the **CAD data** panel.

- (a) Specify the minimum size of the features that will be imported into Airpak in the **Value** text-entry field under **Minimum feature size**. The minimum feature size is a global tolerance that is used by Airpak to control the level of detail that is transferred from the IGES/STEP, or tetin file into Airpak. CAD geometry that is smaller than the specified minimum feature size will not be read into Airpak.

i Note that you can also remove CAD geometry that is smaller than a specified size *after* you have read the IGES/STEP or tetin file into Airpak, as described in Section 4.2.4.

- (b) To specify that Airpak should resize the room so that it is exactly the size required to fit the CAD geometry (and any Airpak objects in your model) when it has been imported, keep the default **Autoscale room** option turned on in the CAD data operation options panel (Figure 4.2.2).
- (c) Click **Accept** in the CAD data operation options panel.

2. Read the CAD geometry into Airpak. There are two ways to read an IGES, STEP or tetin file:

- Use the CAD data panel (Figure 4.2.1).
 - (a) Click **Load** and select one of the following options from the pull-down menu: **Load IGES/Step file** or **Load tetin file**.
 - (b) Select the IGES, STEP, or tetin file in the resulting **File selection** dialog box. See Section 2.1.8 for information on the **File selection** dialog box. The following options are available:
 - To load surface data from the IGES/STEP or tetin file, select **Load surfaces**.
 - To load curve data from the IGES/STEP or tetin file, select **Load curves**.
 - To load point data from the IGES/STEP or tetin file, select **Load points**.
 - (IGES or STEP import only) To copy the IGES or STEP file into the current project directory, select **Copy IGES file to project directory**.
 - (tetin import only) To load material data from the tetin file, select **Load material**. This will cause material points to be loaded from the tetin file, if any are present. These points are used to define the interior of an object if it has been meshed with the tetrahedral mesher as a CAD object instead of being converted to an **Airpak** object.
 - (tetin import only) To specify the triangulation tolerance, enter a value for the **Spline discretization**. It is recommended that you keep the default value. If you have a small tetin file that uses a very large amount of memory, you can increase the **Spline discretization** value slightly to fix the problem.
 - To load a combination of surface, curve, and point information, select and/or deselect the appropriate options in the **File selection** dialog box.
 - (c) Click **Accept** to read the IGES or tetin file into **Airpak**.
- Use the **File→Import** menu.

File→Import→IGES/Step surfaces+curves,

File→Import→Tetin surfaces+curves,

 - (a) Enter the name of the IGES, STEP, or tetin file (e.g., **file.igs**, **file.stp**) in the **File name** field in the resulting **File selection** or **Select tetin file** panel. You can enter your own filename, which can be a full pathname to the file (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started. Alternatively, you can choose a filename from those available in the **Directory** list. See Section 2.1.8 for more information on the **File selection** dialog box.

- (b) (IGES/STEP files only) To import files with a combination of surface, curve, and point data, select and/or deselect the appropriate **Load surfaces**, **Load curves**, and **Load points** options, respectively, in the **File selection** dialog box. To copy the IGES or STEP file into the current project directory, select **Copy IGES file to project directory**.
- (c) Click **Open** to read the IGES, STEP, or tetin file into **Airpak**.

3. To read another IGES, STEP or tetin file into **Airpak**, repeat the steps above.

An example of an imported CAD geometry is shown in Figure 4.2.3.

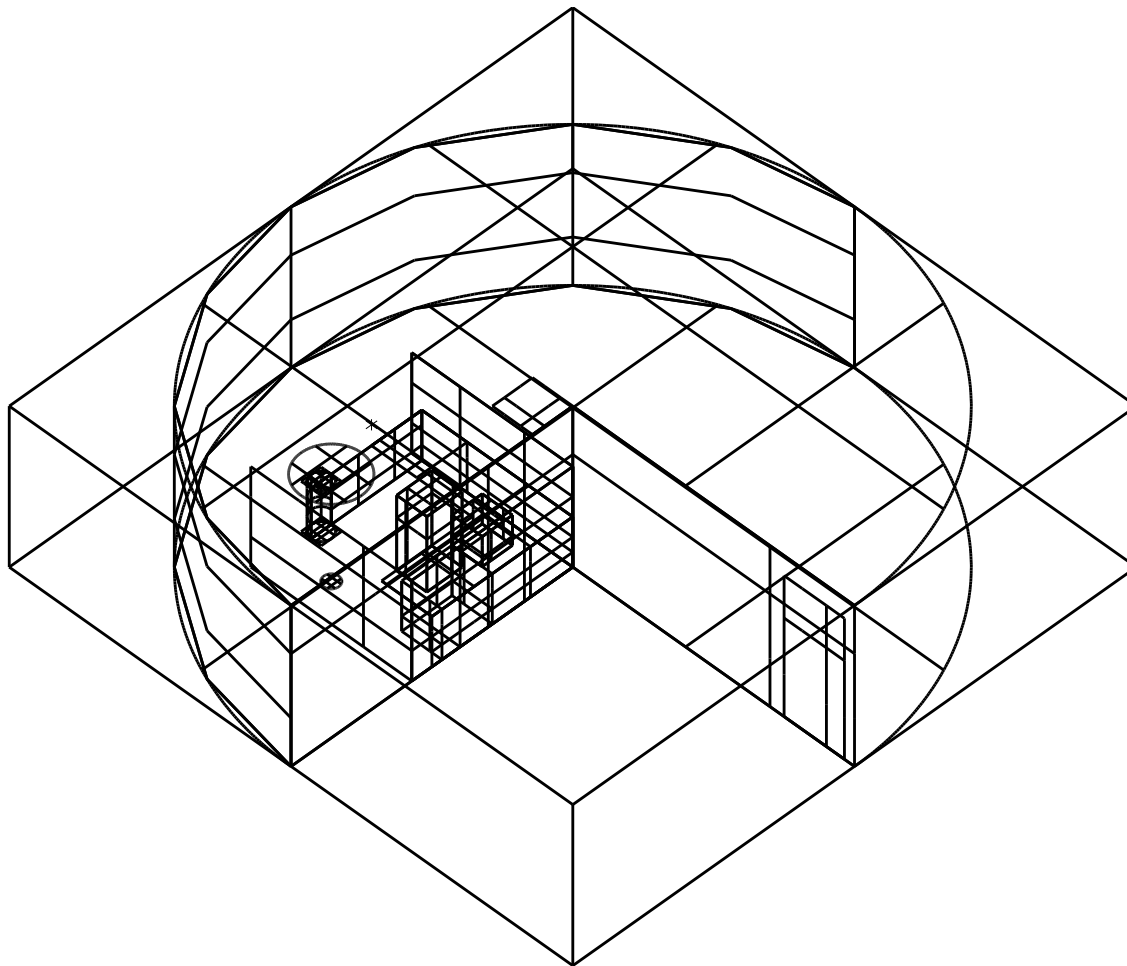


Figure 4.2.3: Example of an Imported CAD Geometry

4.2.3 Using Families

A family is a group of CAD objects. **Airpak** will import families as part of the IGES, STEP or tetin import process if the imported file already contains them. In **Airpak**,

you can move CAD objects from one family to another, create new families, and delete unused ones.

Airpak displays different families using different colors in the graphics window. You can temporarily remove a family from the graphics window as described in Section 4.2.4.

Moving CAD Geometry From One Family to Another Family

To move CAD geometry from one family to another, follow the steps below.



Note that the information below assumes that you are using the default mouse controls in Airpak. If you have changed the default mouse controls (using the **Mouse buttons** section of the **Preferences** panel, described in Section 2.2.4), you will use different mouse buttons than the ones described below to perform these operations. Airpak will inform you which mouse buttons should be used for which operations in a message at the bottom of the graphics window.

1. Click **Change family** in the CAD data panel (Figure 4.2.1).
2. Select the CAD geometry in the graphics window by clicking on it using the left mouse button or by defining a rectangular box on the screen. To define a box, position the mouse pointer at a corner of the area where the objects to be included are located, hold down the left mouse button and drag open a selection box to enclose the objects to be included, and then release the mouse button. The objects within the bounded area will be selected.

To deselect the last CAD object that you selected, click the right mouse button in the graphics window. This operation can be used repeatedly to take you back to the first selection that you made, and then to cancel the current operation.

To deselect one particular CAD object that you have selected, hold down the <Shift> key on the keyboard and click on the object using the left mouse button.

The following options are also available, when appropriate, when selecting CAD geometry:

- To select all the CAD geometry in a particular family, place the mouse pointer over the graphics window and press the **f** key on the keyboard. **Airpak** will open the **Select family** panel (Figure 4.2.4). Select the name of the family and click **Accept**. To select all family names, click **All**. To deselect all selected family names, click **None**.



Figure 4.2.4: The Select family Panel

- To instruct **Airpak** to select all the CAD objects that are *partially* inside the region you define, place the mouse pointer over the graphics window and press the **m** key on the keyboard before you draw a box to enclose the geometry to be selected. To instruct **Airpak** to select all the CAD objects that are *entirely* inside the region you define, press the **m** key on the keyboard again before you draw a box to enclose the geometry to be selected. By default, only CAD geometry entirely inside the region you define will be selected.
- To select all the CAD geometry within a polygon region that you define in the graphics window, place the mouse pointer over the graphics window and press the **p** key on the keyboard. The mouse pointer will change shape. Use the left mouse button to create points in the graphics window to define the corners of the polygon region. Click the middle mouse button when you have finished defining the polygon region. **Airpak** will select all the CAD geometry enclosed in the polygon region. Under **Shapes to try**, select the shapes that you want **Airpak** to try to fit to the CAD geometry.



Note that restricting the shapes that you want **Airpak** to try to fit to the CAD geometry will speed up the conversion process and ensure that **Airpak** does not use a different type of shape than the one you require (e.g., **Airpak** could use a polygon shape to create a rectangle).

- Alternatively, if you want to create CAD shaped **Airpak** objects, you can select the option **Use CAD surfaces directly**.
- Click **Options**, and specify any relevant options in the **CAD data operation** options panel.
- Under **Create object** in the **CAD data** panel, click the button for the object into which you want to convert the CAD geometry.

i Note that if you select the option **Use CAD surfaces directly** you can only create blocks and partitions with the geometry type **CAD**.

Converting Selected CAD Geometry Into Airpak Objects

To convert selected CAD geometry into an Airpak object, follow the steps below.

i Note that the information below assumes that you are using the default mouse controls in Airpak. If you have changed the default mouse controls (using the **Mouse buttons** section of the **Preferences** panel, described in Section 2.2.4), you will use different mouse buttons than the ones described below to perform these operations. Airpak will inform you which mouse buttons should be used for which operations in a message at the bottom of the graphics window.

1. Specify options related to the conversion of the CAD geometry in the **CAD data** panel (Figure 4.2.1).

Model→CAD data

- (a) Under **Creation mode**, select **Selected**.
- (b) Under **Shapes to try**, select the shapes that you want Airpak to try to fit to the CAD geometry. For example, if you are converting the surfaces shown in bold in Figure 4.2.5 into a fan, select **circ** (circular) in the **CAD data** panel and deselect all other shapes under **Shapes to try**.

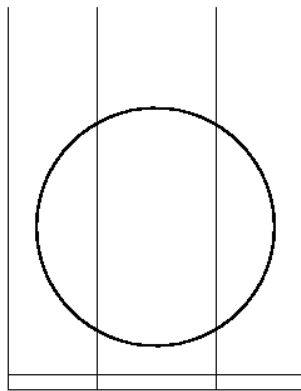


Figure 4.2.5: Surfaces That Define a Circle

- (c) Click **Options**, and specify any relevant options in the **CAD data operation options** panel. These options are described on page 4-16.

- (d) Under **Create object**, select the object into which you want to convert the CAD geometry.
2. Select the CAD geometry in the graphics window. See Section 4.2.3 for details on selecting and deselecting CAD geometry.
3. When you have selected all the CAD geometry to be converted to the **Airpak** object, click the middle mouse button in the graphics window. **Airpak** will convert the selected CAD geometry into an **Airpak** object (the type of object you selected in the **CAD data** panel).
4. To convert more selected CAD geometry into **Airpak** objects, repeat the steps above. When you have finished converting selected CAD geometry into **Airpak** objects, click the middle mouse button in the graphics window to exit the selection mode.

Converting Families of CAD Objects Into Airpak Objects

If you select a group of families in **Airpak**, **Airpak** can convert each family in the group into an **Airpak** object. For example, if the five shapes shown in bold in Figure 4.2.6 represent five different families, you can select all of these families at once and **Airpak** can convert each family into an **Airpak** object.

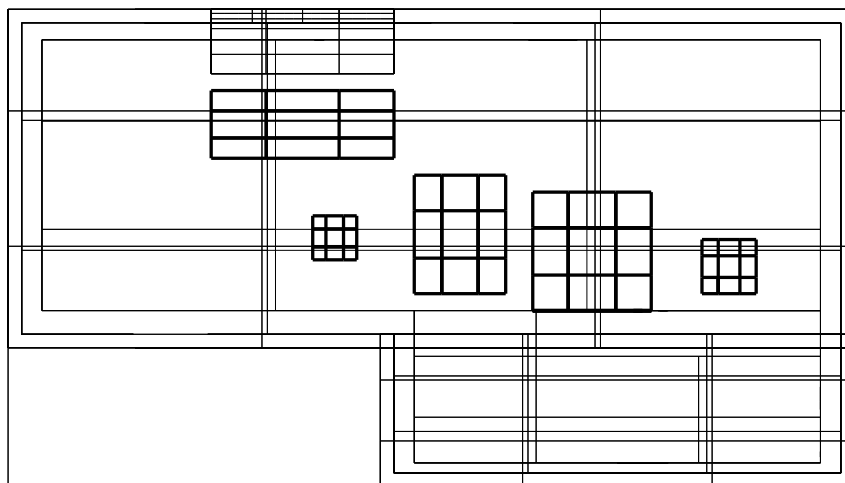


Figure 4.2.6: Five Families of CAD Objects

To convert families of CAD objects into **Airpak** objects, follow the steps below.



Note that the information below assumes that you are using the default mouse controls in **Airpak**. If you have changed the default mouse controls (using the **Mouse buttons** section of the **Preferences** panel, described in Section 2.2.4), you will use

different mouse buttons than the ones described below to perform these operations. **Airpak** will inform you which mouse buttons should be used for which operations in a message at the bottom of the graphics window.

1. Specify options related to the conversion of the CAD geometry in the **CAD data** panel (Figure 4.2.1).

Model→**CAD data**

- (a) Under **Creation mode**, select **Family**.
 - (b) Under **Shapes to try**, select the shapes that you want **Airpak** to try to fit to the families. For example, if the five shapes shown in bold in Figure 4.2.6 are to be converted into five prism-shaped blocks, select **hexa** (prism) in the **CAD data** panel and deselect all other shapes under **Shapes to try**.
 - (c) Click **Options**, and specify any relevant options in the **CAD data operation options** panel. These options are described on page 4-16.
 - (d) Under **Create object**, select the object into which you want to convert the CAD geometry.
2. Select the CAD geometry in the graphics window. See Section 4.2.3 for details on selecting and deselecting CAD geometry.
 3. When you have selected all the CAD geometry to be converted to **Airpak** objects, click the middle mouse button in the graphics window. **Airpak** will convert the selected CAD geometry into **Airpak** objects, creating one object for each family that you selected (the type of object you selected in the **CAD data** panel).
 4. To convert more families of CAD objects into **Airpak** objects, repeat the steps above. When you have finished converting families of CAD objects into **Airpak** objects, click the middle mouse button in the graphics window to exit the selection mode.

Converting Regions of the Model Into Airpak Objects

When converting CAD geometry into **Airpak** objects, you can define a region in your model and then divide the region into segments. **Airpak** can then convert all the CAD geometry within each segment into an **Airpak** object. For example, you could select a CAD object that is L-shaped, as shown in Figure 4.2.7a. You could then divide the L-shaped geometry into two regions, as shown in Figure 4.2.8b. **Airpak** can split the CAD objects into two parts along the dividing line and then convert the CAD geometry in each region into an **Airpak** object (i.e., **Airpak** will create two rectangular **Airpak** objects for the example shown in Figure 4.2.7.)

To convert CAD objects within a specified region into several **Airpak** objects, follow the steps below.

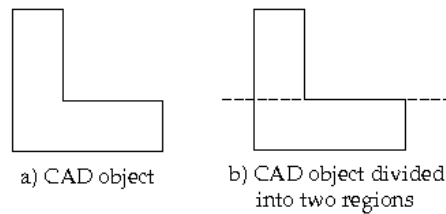


Figure 4.2.7: Converting an L-Shaped CAD Object



Note that the information below assumes that you are using the default mouse controls in **Airpak**. If you have changed the default mouse controls (using the **Mouse buttons** section of the **Preferences** panel, described in Section 2.2.4), you will use different mouse buttons than the ones described below to perform these operations. **Airpak** will inform you which mouse buttons should be used for which operations in a message at the bottom of the graphics window.

1. Specify options related to the conversion of the CAD geometry in the **CAD data** panel (Figure 4.2.1).

Model → CAD data

- (a) Under **Creation mode**, select **Region**.
 - (b) Under **Shapes to try**, select the shapes that you want **Airpak** to try to fit to the CAD geometry. For example, if you are converting the surfaces shown in Figure 4.2.7 into rectangular partitions, select **quad** (rectangle) in the **CAD data operation options** panel and deselect all other shapes under **Shapes to try**.
 - (c) Click **Options**, and specify any relevant options in the **CAD data operation options** panel. These options are described on page 4-16.
 - (d) Under **Create object**, select the object into which you want to convert the CAD geometry.
2. Select the CAD geometry in the graphics window. See Section 4.2.3 for details on selecting and deselecting CAD geometry.
 3. When you have selected all the CAD surfaces to be converted, click the middle mouse button in the graphics window. **Airpak** will request that you divide up the region you have selected into segments. **Airpak** will create **Airpak** objects for the selected CAD geometry in the regions of the model.

To divide the CAD geometry into regions, hold the mouse pointer over the place in the graphics window where you want to create a dividing line and press the **v** key

on the keyboard to draw a vertical dividing line, or press the **h** key to produce a horizontal dividing line. To create horizontal and vertical dividing lines using the mouse buttons, position the pointer where you want the start of the line to be, hold down the left mouse button, drag the pointer to the desired location, and release the left mouse button. An example of a dividing line is shown in Figure 4.2.8 where the CAD geometry is divided into two regions.

To remove the last dividing line that you created, click the right mouse button in the graphics window. This operation can be used repeatedly to take you back to the first dividing line that you created.

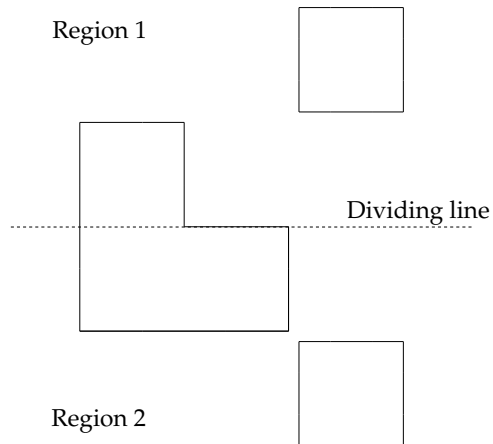


Figure 4.2.8: Three CAD Objects Divided into Two Regions

4. When you have divided the CAD geometry into the appropriate regions, click the middle mouse button in the graphics window. **Airpak** will open the **Multiple regions** panel (Figure 4.2.9), which contains the following options:

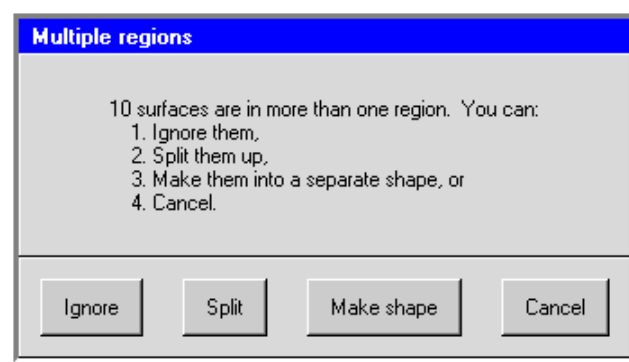


Figure 4.2.9: The Multiple regions Panel

Ignore instructs **Airpak** to ignore any CAD objects that are in more than one region and convert into **Airpak** objects (the type of object you selected in the **Model** menu) any CAD objects that are completely in the selected region. For example, the two square CAD objects in Figure 4.2.8 will be converted into two square **Airpak** objects and the L-shaped CAD object will be ignored.

Split instructs **Airpak** to split any CAD objects that are in more than one region along the specified dividing line and then convert the CAD geometry in each region into an **Airpak** object (the type of object you selected in the **Model** menu). For example, the L-shaped CAD object in Figure 4.2.10a will be split along the dividing line, and then all the CAD objects in region 1 will be converted into one **Airpak** object and all the CAD objects in region 2 will be converted into another **Airpak** object, as shown in Figure 4.2.10b.

Make shape instructs **Airpak** to convert any CAD objects that are completely in one region into **Airpak** objects, and convert any CAD objects that are in more than one region into **Airpak** objects. For example, the two square CAD objects in Figure 4.2.11a will be converted into two square **Airpak** objects, and the L-shaped object will be converted into a large square **Airpak** object, as shown in Figure 4.2.11b.

Cancel instructs **Airpak** to exit the selection mode.

5. To convert more regions of CAD geometry into **Airpak** objects, repeat the steps above. When you have finished converting regions of CAD geometry into **Airpak** objects, click the middle mouse button in the graphics window to exit the selection mode.

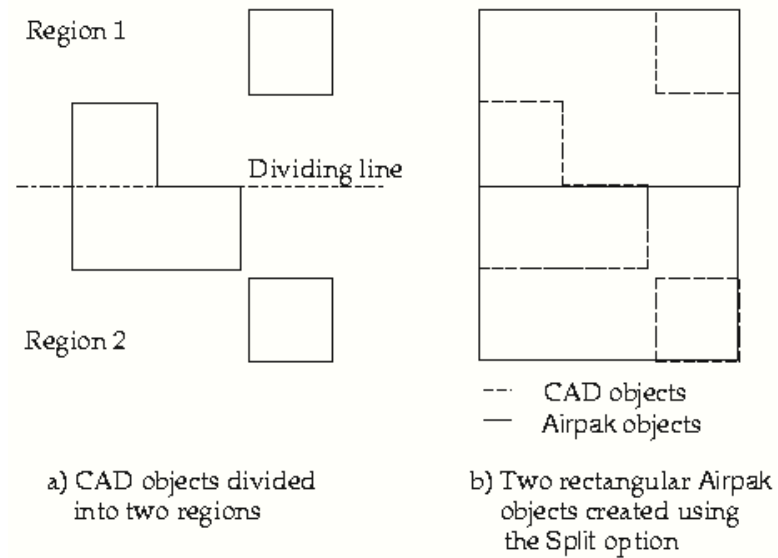


Figure 4.2.10: Three CAD Objects Converted Into Two Airpak Objects Using the Split Option

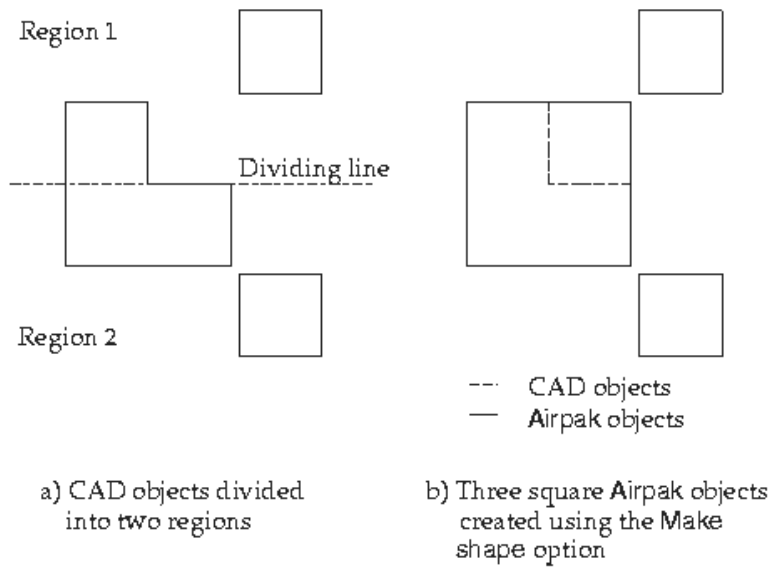


Figure 4.2.11: Three CAD Objects Converted Into Three Airpak Objects Using the Make shape Option

CAD Import Options

The following options are available in the **CAD data operation options** panel (Figure 4.2.2).

Polygon options contains options for creating polygon objects from imported CAD geometry.

Max volume change allows you to specify how much the volume of a polygon or a non-uniform object can expand in order to simplify the object (e.g., by removing polygon vertices that are almost collinear).

Max polygon points specifies the maximum number of points Airpak can use when it creates a polygon.

Minimum feature size contains options for setting the minimum size of object features created from imported CAD geometry.

Value specifies the minimum size of features to be imported into Airpak from the IGES, STEP or tetin file.

Blank below value specifies that Airpak should blank (i.e., not display) features smaller than the specified **Minimum feature size**(see above and Section 4.2.4).

Select CAD geometry types contains options for selecting which parts of the imported CAD geometry should be used to create Airpak objects.

Surfaces specifies that Airpak should use the selected surfaces to create an Airpak object.

Curves specifies that Airpak should use the selected curves to create an Airpak object.

Points specifies that Airpak should use the selected points to create an Airpak object.

Materials specifies that Airpak should use the material points in a tetin file (if present). These points are used to define the interior of an object if it has been meshed with the tetrahedral mesher as a CAD object instead of being converted to an Airpak object.

Group for new objects allows you to specify the name of a group to which Airpak will add new Airpak objects.

Allow non-uniform shapes allows Airpak to create non-uniform shapes when it tries to find the best shape to fit to the selected CAD geometry.

Autoscale room specifies that Airpak should scale the room to be the exact size of the CAD geometry in the IGES or tetin file.

Convert bsplines to facets specifies Airpak to take the CAD surfaces that come in via tetin/step/iges import and convert to triangular facets internally, and save them in the model file. These facets are used for objects that have CAD shapes.

4.2.4 Visibility of CAD Geometry in the Graphics Window

There are several ways to display CAD geometry in the graphics window or remove the CAD geometry from the graphics window, either temporarily or permanently. You can also shade any CAD geometry that is visible in the graphics window. These options are described below.

Changing the Visibility of a Family of CAD Objects

To change the visibility of a family of CAD objects, open the **CAD data** panel (Figure 4.2.1). In the **Families** list, click on the name of a family to select or deselect it. A family is selected if the region around it in the **Families** list is colored; it will be visible in the graphics window. A family is deselected if the region around it in the **Families** list is not colored; it will not be visible in the graphics window. Click **All** to select all families; click **None** to deselect all families. The visibility of a family will be updated in the graphics window as soon as you select or deselect the name of the family in the **Families** list.

This option can be useful if you are working with a complicated geometry and you want to unclutter the display. The family will be removed from the graphics window but not from your model.

Displaying Used and Unused CAD Geometry

Used CAD geometry is IGES or STEP geometry that has been used to create **Airpak** objects; unused CAD geometry is IGES or STEP geometry that has not been used to create **Airpak** objects.

To display unused CAD geometry in the graphics window, turn on the **Show unused** option in the **CAD data** panel (Figure 4.2.1). To remove unused CAD geometry from the graphics window, turn off the **Show unused** option.

To display used CAD geometry in the graphics window, turn on the **Show used** option in the **CAD data** panel. To remove used CAD geometry from the graphics window, turn off the **Show used** option.

The default setting in **Airpak** is that unused CAD geometry is displayed in the graphics window and used CAD geometry is removed from the graphics window.

Blanking and Unblanking CAD Surfaces

You can temporarily remove or “blank” selected CAD surfaces from the graphics window. To blank selected surfaces, follow the steps below.



Note that the information below assumes that you are using the default mouse controls in **Airpak**. If you have changed the default mouse controls (using the **Mouse**

buttons section of the **Preferences** panel, described in Section 2.2.4), you will use different mouse buttons than the ones described below to perform these operations. **Airpak** will inform you which mouse buttons should be used for which operations in a message at the bottom of the graphics window.

Model→CAD data

1. Click **Blank** in the **CAD data** panel (Figure 4.2.1).
2. Select the CAD surfaces to be blanked in the graphics window. See Section 4.2.3 for details on selecting and deselecting CAD geometry.
3. When you have selected all the CAD geometry to be blanked, click the middle mouse button in the graphics window. **Airpak** will blank the selected CAD geometry.
4. To blank more CAD geometry, repeat the steps above. When you have finished blanking CAD geometry, click the middle mouse button in the graphics window to exit the **Blank surfaces** mode.

To unblank all blanked surfaces (i.e., to redisplay all CAD surfaces that you temporarily removed from the graphics window), click **Unblank** in the **CAD data** panel.

Model→CAD data

You can also blank features that are smaller than a specified size in your **Airpak** model. To blank small features, specify the minimum size of the features that will be imported into **Airpak** in the **Value** text-entry field under **Minimum feature size** in the **CAD data operation options** panel (Figure 4.2.2) and click the **Blank below value** button. **Airpak** will blank features smaller than the specified minimum feature size.

To unblank small features that you blanked using the method above, reduce the value of the **Minimum feature size** in the **CAD data operation options** panel and click the **Blank below value** button.



Note that you cannot use **Unblank** in the **CAD data** panel to unblank surfaces that were blanked using the **Minimum feature size** in the **CAD data operation options** panel.

Deleting All Remaining CAD Surfaces

To delete all remaining CAD surfaces from your model, click **Clear** in the **CAD data** panel (Figure 4.2.1).

Model→CAD data

Airpak will display a warning message asking if you want to permanently remove all CAD data from your **Airpak** model (Figure 4.2.12). Click **Yes** to remove all remaining CAD surfaces.

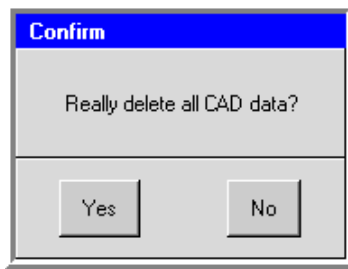


Figure 4.2.12: Warning Message Before All CAD Geometry is Deleted

Shading CAD Surfaces

To shade all of the remaining CAD surfaces in the graphics window, turn on the **Solid shading** option in the **CAD data** panel (Figure 4.2.1).

Model → CAD data

To display all of the remaining CAD surfaces in wireframe format, turn off the **Solid shading** option in the **CAD import** menu.

Manipulating CAD Geometry With the Mouse

You can modify the view of your CAD geometry in the graphics window using the mouse as described in Section 2.2.4. To enable faster movement of your model in the graphics window if your model is very large, turn on the **Fast movement** option in the **CAD data** panel (Figure 4.2.1).

Model → CAD data

Airpak will remove the curves in your model from the graphics window while you move the model and then redisplay the curves when you stop moving the model.

Fixing Small Gaps in the CAD Geometry

To eliminate gaps in the CAD geometry below a specified size, enter a value under **Small gaps** in the **CAD data** panel (Figure 4.2.1) and click **Fix** below size.

Model → CAD data

Airpak will merge existing curves and surfaces, create points by intersecting curves, and split existing surfaces as necessary to fix the specified gaps.

4.3 Importing Other Files Into Airpak

This section describes how to import IGES and AutoCAD DXF files containing point and line inform, DWG and DXF Files containing surface and curve geometry, IFC files, and comma separated values (CSV/Excel) files into **Airpak**.

The general procedure for importing these files is described in Section 4.3.1. Specific information on importing IGES and DXF files with point and line geometry is provided in Section 4.3.2 and on importing DWG and DXF files with surface geometry in Section 4.3.3.



Importing model geometry from IGES and DXF files is not a direct way to construct a model; it is a way of creating the geometry of objects you want in your model without specifying the dimensions from scratch.

4.3.1 General Procedure

To import model geometry into **Airpak**, follow the procedure below. Note that when model geometry from any of the following file formats is imported into **Airpak**, the points between line segments are displayed in blue.

In the File→Import menu.

File→Import

1. Select the type of file you want to import into **Airpak**.
 - For an IGES file that contains point and line information, select **IGES points+lines**.
 - For a DXF file that contains point and line information, select **DXF points+lines**.
 - For an DWG (or DXF) file that contains surface and curve information, select **DWG surfaces+curves**.
 - For a comma separated ASCII file, select **CSV/Excel**.
 - For an IFC file, select **IFC file**.
2. Enter the name of the import file (e.g., **file.igs**) in the **File name** field in the dialog box. You can enter your own filename, which can be a full pathname to the file (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started. Alternatively, you can choose a filename by selecting the file in the directory list in the dialog box.
3. (IFC files only) Enter a new name for the **Airpak** job that you want to contain the imported data in the **Job name** field. If you are importing an IFC file and you leave the **Job name** field blank, **Airpak** will assign a job name derived from the IFC file name.

4. (IGES and DXF point and line geometry files only) Keep the default option of **Scale room to fit objects**. This sizes the room automatically to the dimensions required to contain the imported data.
5. (DWG surface and curve geometry files only) To import a combination of surface, curve, and point data, select and/or deselect the appropriate **Load surfaces**, **Load curves**, and **Load points** options, respectively, in the **File selection** dialog box. To specify the triangulation tolerance, enter a value for the **Spline discretization**. It is recommended that you keep the default value. If you have a small file that uses a very large amount of memory, you can increase the **Spline discretization** value slightly to fix the problem. To load material data from the DWG file, select **Load material**. This will cause material points to be loaded from the file, if any are present. These points are used to define the interior of an object if it has been meshed with the tetrahedral mesher as a CAD object instead of being converted to an Airpak object.
6. Click **Open** to import the model data.

4.3.2 IGES and DXF Files With Point and Line Geometry

When Airpak imports geometry from an IGES or DXF file as described in Section 4.3.1, only point and line geometry are imported; all other geometric entities are ignored. In addition, the imported geometry has no physical characteristics associated with it (i.e., no thermal parameters, material properties, etc.). Consequently, when you import geometry from these file formats into Airpak, you must assign physical characteristics to each object.

When the model geometry is imported into Airpak, the lines that describe the imported geometry are displayed and the ends of each line have blue dots associated with them. Any point imported from the IGES or DXF file also has a blue dot on it.

Once you have imported model data from a CAD source (see above Section 4.3.1), you can use it in your Airpak model by following the procedure below.

First you create an object in Airpak, then you snap the Airpak object to the corresponding imported geometry. Finally you stretch the Airpak object to fit the geometry.

1. Create an object (e.g., a block) in Airpak.
2. Hold down the <Shift> key on the keyboard and use the right mouse button to select a portion (either an edge or corner) of the Airpak object. It will become highlighted in red when selected.
3. Use the <Shift> key and the right mouse button to “snap” the selected portion of the Airpak object to the corresponding edge or corner of the imported geometry. You can either drag the edge/corner of the Airpak object to the corresponding

edge/corner of the imported geometry, or simply click the edge/corner of the imported geometry. The **Message** window will report that the object is snapped. The **Airpak** object is now “anchored” to the imported model geometry.

4. Stretch the **Airpak** object until it fits the imported geometry by holdingdown the **<Shift>** key and using the right mouse button to drag the anchored edge or corner to extend the length or width of the corresponding imported geometry. You only need to fit two red dots to two blue dots or as few as two edges to achieve total coincidence in all three dimensions. The resulting object is now highlighted in the **Edit** panel. You can rename the object, assign it to a group, or perform any function you would perform on any other **Airpak** object, because now it *is* an **Airpak** object.

If you want to convert a number of objects from a complex imported model geometry, you can use the **Visible** option in the **Options** menu to remove the object types from the model after you convert them in order to reduce the number of objects on the screen as you continue to work.

Note that the CAD geometry is not saved when you save a job file; it is available only during the session in which it is imported.

4.3.3 DWG and DXF Files with Surface Geometry

Airpak can import AutoCAD DWG files that contain surface information. You can then create Airpak objects from selected surfaces in a manner similar to that of an IGES file. See Section 4.2 for details about IGES file translation.

To import a DWG or DXF file with surfaces, curves, points, or a combination of the three:

- In the File→Import menu, select DWG surfaces+curves.
File→Import→DWG surfaces+curves
- Enter the File name in the File selection panel.
- Select Load surfaces, Load curves, Load points or a combination of the three.
- Select Load material from the DWG, ACIS or tetin file. This will cause material points to be loaded from the DWG, ACIS, or tetin file, if any are present. These points are used to define the interior of an object if it has been meshed with the tetrahedral mesher as a CAD object instead of being converted to an Airpak object.
- Leave the default Spline discretization at 0.05 to specify the triangulation tolerance. If you have a small DWG, ACIS, or tetin file that uses a very large amount of memory, you can increase the Spline discretization value slightly to fix the problem.
- Click Open to import the file into Airpak.

4.3.4 IFC Files

Airpak can import Industrial Foundation Classes (IFC) 2x files that have been exported from several CAD packages and other IFC write-compliant software. The International Alliance for Interoperability (IAI) has developed this language for exchanging data used in construction and facilities management industries between software tools. IFC is an open standard supported by many commercial CAD packages.

You can import several basic types of building objects from an IFC file into Airpak directly as Airpak objects, which significantly decreases the time required to build a model and also reduces the possibility of errors during geometry recreation. For example, this capability allows you to import architecture/engineering/construction (AEC) model data created in other CAD programs into Airpak as walls, doors, windows, or spaces.

You can obtain information about IFC from www.iai.org.

To import an IFC file in Airpak:

- In the File→Import menu, select IFC file.
File→Import→IFC file

- Enter the File name in the Select IFC file panel and click Open.

See Section 4.3.1 for details.

Table 4.3.1 lists examples of IFC geometries and the corresponding **Airpak** objects that will be created upon importing the IFC file. See Chapter 7 for more information about specific **Airpak** objects.

Table 4.3.1: Airpak Objects Created From IFC Geometry

IFC Geometry	Airpak Object
doors	openings (rectangular, inclined, or polygonal)
openings	openings (rectangular or inclined)
windows	partitions (rectangular or inclined)
walls	partitions (rectangular, inclined, or polygonal)
floorslabs	partitions (prism or polygonal)
spaces	fluid blocks (prism or polygonal)
columns	hollow blocks (prism, cylindrical, or polygonal)

4.3.5 CSV/Excel Files

Airpak allows you to import geometry and power for simple objects (including blocks, walls, vents, 2D fans, resistances, openings, and sources) in the form of comma separated values (CSV) files. Complex objects are not supported for this operation. Material properties and certain boundary conditions are also not supported for this type of import.

An example of this file format is given below. For details about how to export a CSV/Excel file, see Section 4.4.2.

```
# Partitions Rectangular Object
name,high_emis_on,high_reftemp,high_rtype,low_rtype,partition_type,
all_conduct_thin_thickness,fl_material,use_contact_res,eff_thick,low_enabled,
low_material,all_conduct_thin_material,all_conduct_thin_on,sol_material,
high_material,high_enabled,contact_res,te,eff_thick_on
partition.1,0,20.0,reftemp,reftemp,conduct_thin,0.0,default,0,0.01,0,default,
default,0,default,default,0,0.0,0.0,0

# Partitions Rectangular
name,xs,ys,zs,xe,ye,ze,xd,yd,zd,volume_flag,split_flag,partition_flag,diff_flag,
plane,iradius,thickness,numcopies,copyspace,xoff,yoff,zoff
partition.1,0.4,0.4,0.5,0.6,0.6,0.5,0.2,0.2,0,0,0,1,0,2,0,0,1,0,0,0,0

# Fans Circular Object
name
fan.1

# Fans Circular
name,xc,yc,zc,split_flag,plane,radius,iradius,xoff,yoff,zoff
fan.1,0.5,0.5,0.5,16,2,0.1,0,0,0,0

# Blocks Polygon Object
name
block.1

# Resistances Prism Object
name
resistance.1

# Blocks Polygon
name,volume_flag,split_flag,changes,nverts,plane,height,xoff,yoff,zoff,
vert1,tvert1,vert2,tvert2,vert3,tvert3
block.1,1,0,0,3,0,0.2,0,0,0,0.8 0 0.717157,,0.8 0.282843 0.717157,,0.8 0 1,
```

```
# Resistances Prism
name,xs,ys,zs,xe,ye,ze,xd,yd,zd,volume_flag,diff_flag,xoff,yoff,zoff
resistance.1,0.8,0.241689,0.0363489,1,0.441689,0.236349,0.2,0.2,0.2,1,0,0,0,0
```

Typically, the flags you would use are `name`, `xs`, `ys`, `zs`, `xe`, `ye`, `ze`, `xd`, `yd`, `zd`. In many cases (e.g., for 2D objects, inclined shapes, or cylindrical shapes), plane information is also useful. This is represented by an integer value of 0, 1, or 2, where 0 maps to `yz`, 1 maps to `xz` and 2 to `xy`. In general:

- `xs`, `ys`, `zs` represent the starting coordinates
- `xe`, `ye`, `ze` represent the ending coordinates
- `xd`, `yd`, `zd` represent the lengths in each direction.

For cylindrical or circular objects, the relevant dimensions become `xc`, `yc`, `zc`, `radius`, `height`.

If you import the file from **Airpak** into Excel and use the comma as a separator, then each column will get imported correctly.

To import a file from Excel into **Airpak**, it is recommended that you group all object types and shape types together. For example, if you wish to create a number of cuboidal blocks, you might create the following input (where the commas separate columns):

```
# Blocks Prism
name,xs,ys,zs,xe,ye,ze,xd,yd,zd,volume_flag,diff_flag,xoff,yoff,zoff
block.1,0.4,0.4,0.4,0.6,0.6,0.6,0.2,0.2,0.2,1,0,0,0,0
block.2,0.3,0.3,0.3,0.6,0.6,0.6,0.2,0.2,0.2,1,0,0,0,0
block.3,0.4,0.4,0.4,0.6,0.6,0.6,0.2,0.2,0.2,1,0,0,0,0
```

Alternatively, you could input the same syntax into a text editor and then import the text file into **Airpak**.

To add cylindrical blocks, for example, append the following to the previous list:

```
# Blocks Cylinder
name,xc,yc,zc,xc2,yc2,zc2,volume_flag,changes,plane,radius,iradius,
radius2,iradius2,height,start_angle,end_angle,xoff,yoff,zoff

block.4,0.4,0.5,0.5,0.6,0.5,0.5,1,0,0,0.112838,0,0.112838,0,0.2,0,0,0,0,0
block.5,0.4,0.5,0.5,0.6,0.5,0.5,1,0,0,0.112838,0,0.112838,0,0.2,0,0,0,0,0
block.7,0.4,0.5,0.5,0.6,0.5,0.5,1,0,0,0.112838,0,0.112838,0,0.2,0,0,0,0,0
```

You can proceed this way to generate as many object types and shapes as you wish.

Note that not all of the columns are necessary. It is also not necessary to create the header section with the labels as shown above (i.e., name, xs, ys, etc.). In the following example, you will import three prism (cuboid) blocks. The starting coordinates are expressed by the first three values and the dimensions are expressed by the last three values.

```
# Blocks Prism
block.1,0.4,0.4,0.4,0.2,0.2,0.2
block.2,0.3,0.3,0.3,0.2,0.2,0.2
block.3,0.4,0.4,0.4,0.2,0.2,0.2
```

When you import this syntax into **Airpak** using the CSV/Excel option in the File→Import menu, **Airpak** will ask you for inputs of units, choice of separator, and type of object by which to represent the input geometry. When **Airpak** encounters a line that begins with a #, it will pause and ask for the next set of choices. For example, if you import the following syntax

```
# Blocks Prism
block.1,0.4,0.4,0.4,0.2,0.2,0.2
block.2,0.3,0.3,0.3,0.2,0.2,0.2
block.3,0.4,0.4,0.4,0.2,0.2,0.2

# Openings Rectangular
opening.1,0.4,0.4,0.5,0.6,0.6,0.5
opening.2,0.2,0.2,0.3,0.6,0.6,0.3
```

Airpak will stop after the first set of blocks and ask you how you want to import the next set. This process will continue until **Airpak** reaches the end of the file.



Note that if you do not have any lines starting with a #, your only choice will be to import all of the objects as a single type/shape.

The same procedure can be used to import power values for a set of objects.

To import power values for a set of rectangular sources you can use the following syntax.

```
# Sources Rectangular Object
name,temp_total,temp_total_units
M-1,0.708813428,W
M-2,1.895909642,W
M-3,2.316375884,W
```

i Note that for the power import, the object geometry needs to be imported/created prior to the power values. If the object geometry is not created prior to the power import, the objects will be created using the default geometry.

4.4 Exporting Airpak Files

The following sections detail how to export Airpak projects in various other formats.

4.4.1 IGES, STEP, and Tetin Files

To export an IGES or STEP (or tetin) file from Airpak, select the appropriate item in the File→Export menu and then specify a name with the appropriate extension in the File selection panel.

File→Export→IGES, File→Export→Step, File→Export→Tetin

4.4.2 CSV/Excel Files

To export an object or a list of objects in CSV/Excel file format, select the appropriate objects in the Model manager window, and then select CSV/Excel in the File→Export menu. Airpak will open the Save object panel (Figure 4.4.1), where you can specify the details of the object that you want to export.

File→Export→CSV/Excel

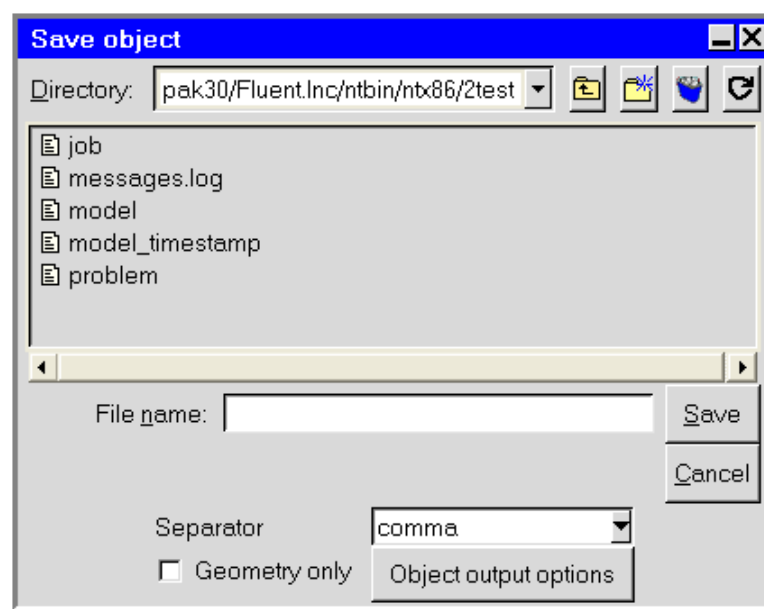


Figure 4.4.1: The Save object Panel for CSV/Excel Files

After you have entered a **File name**, you will need to specify the type of **Separator** that will be used to separate the data in the output file (i.e., **tab**, **space**, **comma**, or **semi-colon**).

If the object geometries are all that you want to export, you should turn on the **Geometry only** option. However, if you also want to export other model data (e.g., radiation, heat transfer, flow direction information), leave the **Geometry only** option turned off and instead click **Object output options**. Airpak will then open a series of panels (e.g., Figure 4.4.2) where you can select the data to be exported.

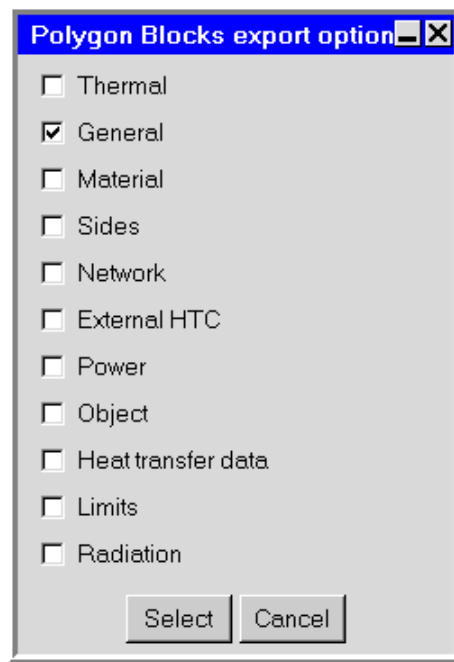


Figure 4.4.2: An Example of a CSV/Excel Export Options Panel

In each of the export options panels, turn on the options for the data to be exported for the various objects. When you are done with one object, click **Select** in the panel to proceed to the next panel. When you are finished with all of the objects, click **Save** in the **Save object** panel to complete the export.

i Object geometries will not be written unless **General** is turned on in each of the export options panels.

This chapter describes the units used in **Airpak** and how you can control them. Information is organized into the following sections:

- Section 5.1: Overview of Units in **Airpak**
- Section 5.2: Units for Meshing
- Section 5.3: Built-In Unit Systems in **Airpak**
- Section 5.4: Customizing Units
- Section 5.5: Units for Postprocessing

5.1 Overview of Units in **Airpak**

Airpak allows you to work in any unit system, including mixed units. Thus, for example, you can work in Imperial (British) units with heat input in Watts, or you can work in SI units with length defined in inches. This is accomplished by providing **Airpak** with a correct set of conversion factors between the units you want to use and the standard SI unit system that is used internally. **Airpak** uses these conversion factors for input and output, internally storing all parameters and performing all calculations in SI units.


Units can be altered part-way through a problem setup and/or after you have completed your calculation. If you have input some parameters in SI units and then you switch to Imperial, all of your previous inputs (and the default prompts) are converted to the new unit system. If you have completed a simulation in SI units but you would like to report the results in any other units, you can alter the unit system and **Airpak** will convert all of the problem data to the new unit system when results are displayed. As noted above, all problem inputs and results are stored in SI units internally. This means that the parameters stored in the project files are in SI units. **Airpak** simply converts these values to your unit system at the interface level.



Note that you must specify all inputs in SI units in the **Transient temperature** panel.

5.2 Units for Meshing

The units used for meshing will be the same as the default units specified for length (meters) in Airpak. You can change the units for meshing in the **Mesh Control** panel.

Open the Mesh control panel (Figure 5.2.1) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.

Model → Generate mesh

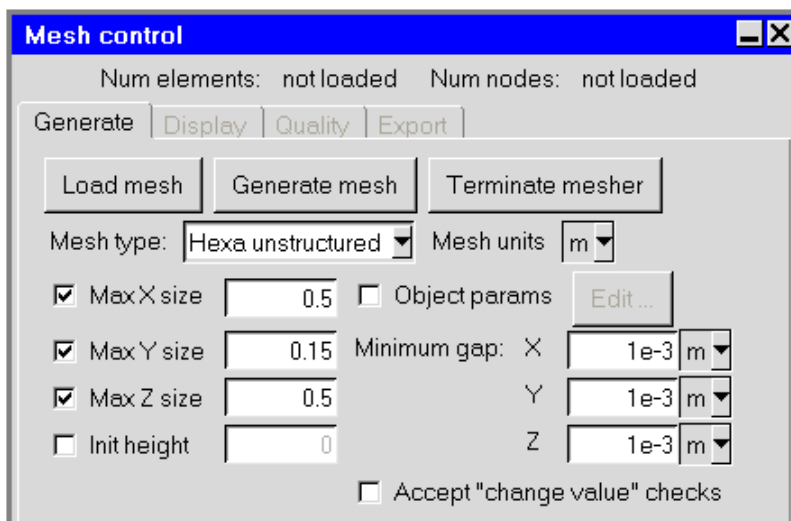


Figure 5.2.1: Portion of the Mesh control Panel (Generate Tab)

Click on the **Generate** tab to access the controls for mesh generation. Select the units you would like to change to for a particular mesh from the drop-down list next to **Mesh units** (Figure 5.2.2). Click **Done** in the **Mesh Control** panel to change the meshing units and close the panel.



Figure 5.2.2: The Mesh units drop-down list

5.3 Built-In Unit Systems in Airpak

Airpak provides two built-in unit systems: Imperial and SI. You can convert all units from one system to another in the **Units** section of the **Preferences** panel.

Edit → Preferences

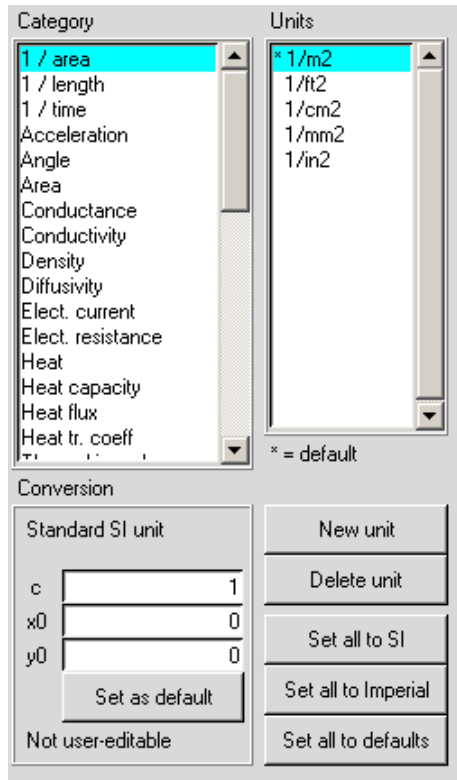


Figure 5.3.1: The Units Section of the Preferences Panel

To choose the Imperial standard for all units, click on the **Set all to Imperial** button; to select the International System of units (SI) standard for all units, click on the **Set all to SI** button. Clicking on one of these buttons will immediately change the unit system. You can then click **Cancel** to close the **Preferences** panel if you are not interested in customizing any units.

Changing the unit system in the **Preferences** panel causes all current and future inputs that have units to be based on the newly selected unit system.

5.4 Customizing Units

If you would like a mixed unit system, or any unit system different from the default SI system supplied by Airpak, you can use the **Units** section of the **Preferences** panel (Figure 5.3.1) to select an available unit or specify your own unit name and conversion factor for each quantity.

5.4.1 Viewing Current Units

Before customizing units for one or more quantities, you may want to view the current units. To view the built-in definitions for a particular category, select the category in the **Category** list in the **Units** section of the **Preferences** panel (Figure 5.3.1). The built-in definitions will be displayed in the **Units** list. For example, to view the built-in definitions for length, select **Length** in the **Category** list. The built-in definitions for length are displayed in the **Units** list: m, ft, cm, mm, microns, in, mil, and Cu-oz/ft².

5.4.2 Changing the Units for a Quantity

Airpak will allow you to modify the units for individual quantities. This is useful for problems in which you want to use one of the built-in SI unit systems, but you want to change the units for one quantity (or for a few). If, for example, you want to use SI units for your problem, but the dimensions of the geometry are given in inches, you can select the SI unit system and then change the unit of length from meters to inches.

Changing the Default Unit

The default unit for a particular **Category** is marked with an asterisk (*) in the **Units** list. To change the default units for a particular quantity, follow these steps:

1. Select the quantity in the **Category** list (they are arranged in alphabetical order) in the **Units** section of the **Preferences** panel (Figure 5.3.1).
2. Choose a new unit from those that are available in the **Units** list.
3. Click **Set as default** under **Conversion**.
4. If you wish to change the units for another quantity, repeat the steps above.
5. Apply the changes to the unit system either to the current project or to this and all future projects. To apply changes to the unit system to the current project only, click **This project** in the **Preferences** panel. To apply changes to the unit system to the current project and all future Airpak projects, click **All projects** in the **Unit definitions** panel.

For the example cited above, you would choose **Length** in the **Category** list, and then select **in (inches)** in the **Units** list. Airpak displays the equation for the conversion between meters and inches:

$$\text{in} = c * (\text{m} + x_0) + y_0 \quad (5.4-1)$$

When the **Conversion** factors $c=39.37008$, $x_0=0$, and $y_0=0$ are substituted into the above equation, it becomes

$$\text{inches} = 39.37008 \times \text{meters} \quad (5.4-2)$$

You would then click **Set as default** to make inches the default unit of length for your model (see Figure 5.3.1).

You should substitute a length in meters into Equation 5.4-2 to calculate the length in inches. For example, to convert a length of 10 m into inches:

$$\begin{aligned} \text{length in inches} &= (39.37008 \times 10) \text{ inches} \\ &= 393.7008 \text{ inches} \end{aligned}$$

i It is not possible to edit the **Conversion** factors for units that are predefined in Airpak; these units are marked as **Not user-editable** under **Conversion** (see Figure 5.3.1).

Changing the default unit for a **Category** in the **Units** section of the **Preferences** panel causes all future inputs that have units in that **Category** to be based on the newly selected default unit. For example, if you change the unit of length as described in the example above, and then you create a new block, the units defined for the block that relate to length will be in inches (see Figure 5.4.1).

Specify by: Start / end	
xS	3.543 in
yS	1.575 in
zS	3.543 in
xE	4.331 in
yE	2.362 in
zE	4.331 in

Figure 5.4.1: Units Defined for Individual Inputs for a Block

Note that changing the default unit for a **Category** does not change the units for any previous inputs. Previous inputs still use the old units, so you do not need to make any changes to them.

Changing the Unit for an Individual Input

You can change the unit for an individual input by following the steps below:

1. Click on the unit definition to the right of the text field to display the list of available units.
2. Place the mouse pointer over the new list item.
3. Click the left mouse button on the item to make the new selection. The list will close automatically and the new selection will then be displayed.

If you want to abort the selection process while the list is displayed, you can move the pointer anywhere outside the list and click the left mouse button.

For example, if you wanted to change the unit for **xE** in Figure 5.4.1 from inches to meters, you would left-click on **in** to the right of the **xE** text entry field. You would then select **m** from the list of available units. The unit for **xE** would be changed to meters and the units for the other parameters would remain unchanged.

The Fix values Option

You can also change the units in which a quantity is displayed, without changing its value, using the **Fix values** option. This option is available in the **Geometry** tab of the **Room** panel, in the **Geometry** and **Properties** tabs of the **Object** panels, and in the **Properties** tab of the **Materials** panels. The **Fix values** option is off by default in the **Room** and the **Object** panels.

If, for example, you specify a block of size $0.2\text{ m} \times 0.2\text{ m} \times 0.2\text{ m}$, and then you change the units for the dimensions of the block from meters to centimeters, the block will be defined with dimensions of $0.2\text{ cm} \times 0.2\text{ cm} \times 0.2\text{ cm}$. (Internally **Airpak** represents this as $0.002\text{ m} \times 0.002\text{ m} \times 0.002\text{ m}$.)

If the **Fix values** option is selected in the **Blocks** panel, and you change the units for the dimensions of the block from meters to centimeters, the values for the dimensions of the block will change from 0.2 m to 20 cm in the **Blocks** panel (the internal value stays the same, 0.2 m).

The **Fix values** option is on by default in the **Materials** panel. A database of materials is provided with **Airpak** (see Section 7.7), and the units for these materials are defined as SI units by default. When the **Fix values** option is on in the **Materials** panel, you can change the units for a quantity without changing the internal value for that quantity. For example, the default value and units for the density of air are 1 kg/m^3 . If you change the units from kg/m^3 to lb/ft^3 for density, the value of density for air will be displayed as 0.0624 lb/ft^3 in the **Materials** panel, and the internal value is still 1 kg/m^3 .

If the **Fix values** option is not selected in the **Materials** panel and the units are changed for a quantity, the internal value of the quantity will be changed. In the above example of changing the units from kg/m^3 to lb/ft^3 for density, the value of density for air will be displayed as $1 \text{ lb}/\text{ft}^3$ in the **Materials** panel, and the internal value will be changed to $16.02 \text{ kg}/\text{m}^3$.

5.4.3 Defining a New Unit

To create a new unit to be used for a particular quantity, follow the procedure below:

1. In the **Units** section of the **Preferences** panel (Figure 5.3.1), select the quantity in the **Category** list.
2. Click the **New unit** button to open the **New unit name** panel (Figure 5.4.2).

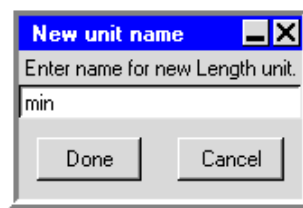


Figure 5.4.2: The New unit name Panel

3. Enter the name of your new unit in the text entry box and click **Done**. The new unit will appear in the **Units** list in the **Units** section of the **Preferences** panel.
4. Select the new unit in the **Units** list, and enter the conversion factors (c , $x0$, and $y0$) under **Conversion**.
5. Click on **Set as default** under **Conversion** if you want the new unit to be the default unit for that **Category**.

For example, if you want to use minutes as the unit of time, select **Time** in the **Category** list in the **Units** section of the **Preferences** panel and click on the **New** button. In the resulting **New unit name** panel, enter **min** in the text entry box and click **Done**. The new unit **min** will appear in the **Units** list in the **Units** section of the **Preferences** panel. Enter 0.016667 (which is equal to $1/60$) for c under **Conversion** in the **Unit definitions** panel.

Determining the Conversion Factor, c

The conversion factor c you specify (under **Conversion** in the **Units** section of the **Preferences** panel) tells **Airpak** the number to multiply by to obtain your customized unit

value from the SI unit value. Thus the conversion factor c should have the form custom units/SI units. For example, if you want the unit of length to be inches, you should input a conversion factor c of 39.37008 inches/meter. If you want the unit of speed to be feet/min, you can determine the conversion factor c by using the following equation:

$$x \frac{\text{ft}}{\text{min}} = y \frac{\text{m}}{\text{s}} \times \frac{60 \text{ s}}{\text{min}} \times \frac{\text{ft}}{0.3048 \text{ m}} \quad (5.4-3)$$

You should input a conversion factor c of 196.85, which is equal to $60/0.3048$.

5.4.4 Deleting a Unit

To delete a unit, follow the procedure below:

1. In the **Units** section of the **Preferences** panel (Figure 5.3.1), select the quantity in the **Category** list.
2. Select a unit that you want to delete from the **Units** list.
3. Click the **Delete unit** button and select **Accept** in the **Confirm** panel.

5.5 Units for Postprocessing

You can choose the units for postprocessing for different variables using the **Postprocessing units** panel (Figure 5.5.1). There are several ways to open the **Postprocessing units** panel:

- Select **Postprocessing units** in the **Post** menu.

Post → Postprocessing units

- Select **Solution monitor** in the **Solve** menu.

Solve → Solution monitor

This opens the **Solution monitor definition** panel. Next, click on the **Output units** button at the bottom of the **Solution monitor definition** panel.

- Select **Define report** in the **Solve** menu.

Solve → Define report

This opens the **Define summary report** panel. Next, click on the **Units** button in the **Define summary report** panel.

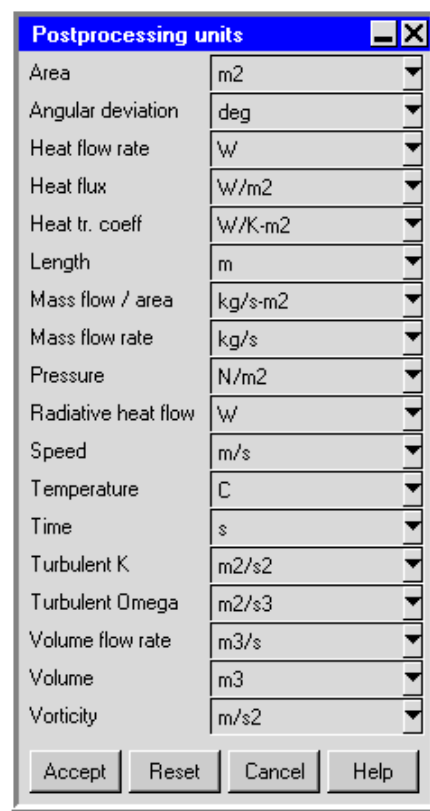


Figure 5.5.1: The Postprocessing units Panel

- Select Convergence plot in the Post menu.

Post → Convergence plot

This opens the Solution monitor definition panel. Next, click on the Output units button at the bottom of the Solution monitor definition panel.

- Click on Summary report in the Report menu.

Report → Summary report

This opens the Define summary report panel. Next, click on the Edit units button in the Define summary report panel.

- Select Point report in the Report menu.

Report → Point report

This opens the Define point report panel. Next, click on the Edit units button in the Define point report panel.

The Postprocessing units panel shows the units defined for the different variables available for postprocessing. You can change the unit for a particular variable by selecting a new

unit from the unit definition drop-down list to the right of the variable, as described in Section [5.4.2](#).

Once you have planned your **Airpak** analysis and have identified important features of the problem you want to solve (see Section 1.4), you are ready to begin the first step in the **Airpak** problem solving process: defining a project. All of the functions that are needed to define a project are found in the **File** menu and toolbar and the **Model manager** window. Consequently, this chapter will begin with an overview of the **File** menu and its functions, followed by a discussion of the relevant portions of the **Model manager** window. Once you have defined a project, you can then move on to building your model (Chapter 7).

The information in this chapter is divided into the following sections:

- Section 6.1: Overview of Interface Components
- Section 6.2: Creating, Opening, Reloading, and Deleting a Project File
- Section 6.3: Configuring a Project
- Section 6.4: Specifying the Problem Parameters

6.1 Overview of Interface Components

6.1.1 The File Menu

The **File** menu (Figure 6.1.1) contains options for working with **Airpak** projects and project files. From this menu, you can open, merge, and save **Airpak** projects. In addition, you can import, export, compress, and decompress files relating to your **Airpak** model. There are also utilities designed to save or print your model geometries. A brief description of the **File** menu options is provided below. See Chapter 3 for more information about reading, writing, and managing **Airpak** project files.

New project allows you to create a new **Airpak** project using the **New project** panel. Here, you can browse through your directory structure, create a new project directory, and enter a project name.

Open project allows you to open existing **Airpak** projects using the **Open project** panel. Here, you can browse through your directory structure, locate a project directory, and either enter a project name, or specify an old project name from a list of recent projects. Additionally, you can specify a version name or number for the project.

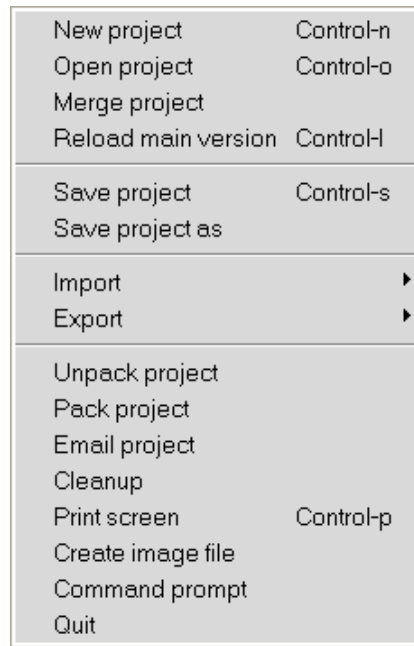


Figure 6.1.1: The File Menu

Merge project allows you to merge an existing project into your current project using the Merge project panel.

Reload main version allows you to re-open the original version of the Airpak project when your project has multiple versions.

Save project saves the current Airpak project.

Save project as allows you to save the current Airpak project under a different name using the Save project panel.

Import provides options to import IGES, DXF, and tetin file geometries into Airpak. You also can import IFC files, as well as comma separated values or spreadsheet format (CSV) using this option. See Chapter 4 for more information about importing files.

Export allows you to export your work as comma separated values or spreadsheet format (CSV) or as an IGES, STEP, or tetin file. See Chapter 4 for more information about exporting files.

Unpack project opens a File selection dialog that allows you to browse for and decompress .tzt files. See Section 3.6 for details.

Pack project opens a File selection dialog that allows you to compact your project into a compressed .tzt file. See Section 3.6 for details.

Email project allows you to pack and email your project by specifying the **Recipient** email address, **Subject** for email, and **Message** text. See Section 27.13 for more information on emailing the project.

Cleanup allows you to clean up your project by removing or compressing data relating to mesh, post-processing, screen captures, summary output, reports, message logs, and scratch files using the **Clean up project data** panel.

Print screen allows you to print a PostScript image of the **Airpak** model that is displayed in the graphics window using the **Print options** panel. The inputs for the **Print options** panel are similar to those in the **Graphics file options** panel. See Section 3.5 for details.

Create image file opens a **Save image** dialog that allows you to save your model displayed in the graphics window to an image file. Supported file types include: GIF, JPEG, PPM, VRML, TIFF, and PS.

Shell window opens a separate window running an operating system shell. The window is initially in the subdirectory of the **Airpak** projects directory that contains all the files for the current projects. In this window you can issue commands to the operating system without exiting **Airpak**. Type **exit** in the window to close the window when you are finished using it. (Note that on Windows systems, this menu item is called **Command prompt**.)


Quit exits the **Airpak** application.


6.1.2 The File commands Toolbar


The **File Commands** toolbar (Figure 6.1.2) contains options for working with **Airpak** projects and project files. A brief description of the **File commands** toolbar options is provided below. See Chapter 3 for more information about reading, writing, and managing files in **Airpak**.





Figure 6.1.2: The File commands Toolbar

New project () allows you to create a new **Airpak** project using the **New project** panel. Here, you can browse through your directory structure, create a new project directory, and enter a project name.

Open project () allows you to open existing **Airpak** projects using the **Open project** panel. Here, you can browse through your directory structure, locate a project directory, and either enter a project name, or specify an old project name from a list of recent projects. Additionally, you can specify a version name or number for the project.



Save project () saves the current **Airpak** project.

Print screen () allows you to print a PostScript image of the **Airpak** model that is displayed in the graphics window using the **Print options** panel. The inputs for the **Print options** panel are similar to those in the **Graphics file options** panel. See Section 3.5 for details.

Create image file () opens a **Save image** dialog that allows you to save your model displayed in the graphics window to an image file. Supported file types include: PPM, GIF, JPEG, VRML, TIFF, and PS.


6.1.3 The Model manager Window


The **Airpak Model manager** window (Figure 6.1.3) provides a localized area for defining your **Airpak** model and contains a project-specific listing of problem and solution parameters.


The **Model manager** window is presented in a tree-like structure with expandable and collapsible tree nodes that show or hide relevant tree items. To expand a tree node, use the left mouse button to click on the  icon on the left hand side of the tree. To collapse a tree node, click on the  icon.

You can edit and manage your **Airpak** project from within the **Model manager** window using the mouse. For example, by clicking on or dragging individual items, you can edit objects, select multiple objects, edit project parameters, add groups within groups, and break apart assemblies. In addition, the **Model manager** window includes a context menu, accessible by right-clicking the mouse, that allows you to easily manipulate your **Airpak** model. See Section 2.2.2 for more information on using the mouse in the **Model manager** window.

An **Airpak** project is organized in the **Model manager** window using six different categories:

 **Problem setup** allows you to set basic problem parameters, set the project title, and define local coordinate systems. Options include:

 **Basic parameters** opens the **Basic parameters** panel where you can specify parameters for the current **Airpak** model. See Section 6.4 for details.

 **Title/notes** opens the **Title/notes** panel where you can enter a title and notes for the current **Airpak** model.

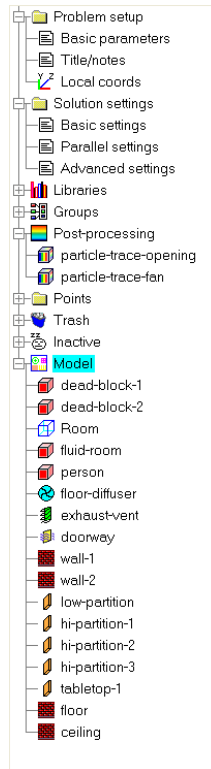


Figure 6.1.3: An Example of the Model manager Window



Local coords opens the **Local coord systems** panel where you can create local coordinate systems that can be used in your model other than the **Airpak** global coordinate system with an origin of (0, 0, 0). The origins of the local coordinate systems are specified with an offset from the origin of the global coordinate system. See Section 7.3.7 for details.



Solution settings allows you to set **Airpak** solution parameters. Options include:



Basic settings opens the **Basic settings** panel where you can specify the number of iterations to be performed and convergence criteria **Airpak** should use before starting your CFD calculations. See Section 27.7 for details.



Parallel settings opens the **Parallel settings** panel where you can specify the type of parallel execution you wish to perform. Options include serial (the default), parallel, or network parallel. See Section 27.11.3 for details.



Advanced settings opens the **Advanced solver setup** panel where you can specify the discretization scheme, under-relaxation factors, and the multigrid scheme. See Chapter 27 for details.



Libraries lists the libraries used in your **Airpak** project. By default, a **Main library** exists in your **Airpak** project that contains materials (fluids, solids, and surfaces), fan objects, and packages. See Section 7.7 for details.



Groups lists any groups of objects in the current **Airpak** project. See Section 7.6 for details about grouping objects.



Post-processing lists any postprocessing objects in the current **Airpak** project. See Chapter 28 for details about postprocessing in **Airpak**.



Points lists any point monitoring objects in the current **Airpak** project. See Section 27.8.1 for details about point monitors.



Trash lists any objects that have been deleted from the **Airpak** model.




Inactive lists any objects that have been made inactive in the **Airpak** model.



Model lists all active objects and materials for the **Airpak** project.

6.2 Creating, Opening, Reloading, and Deleting a Project File

6.2.1 Creating a New Project

The first step in the process of defining a project is to create a new project file. To do this, select **New project** in the **File** menu or click on the  button in the **File** commands toolbar.

File→New project

Airpak will open the **New project** file selection dialog box (Figure 6.2.1). Here, you can select a project name, choose a location for the project, then click **Create** to create the new project. For more information on file selection dialog boxes, see Section 2.1.8.

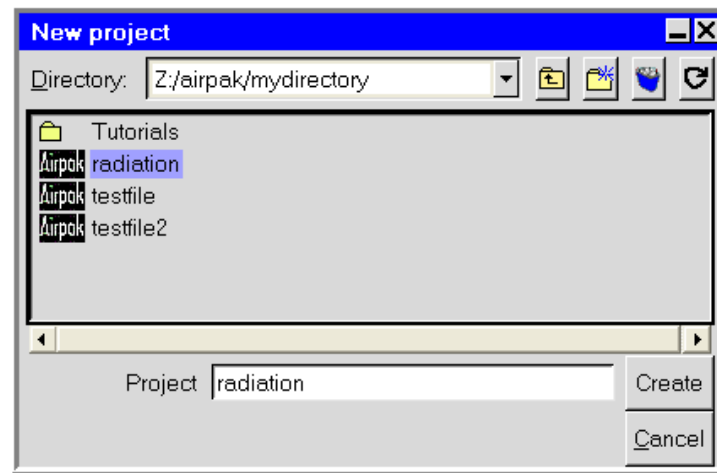


Figure 6.2.1: The New project File Selection Dialog Box

The **Message** window will report

```
Creating new project  
Done loading.
```

In addition, a new directory for your project (e.g., **project2**) will be created, and the default room will be displayed in the graphics window.

If you are creating a new project and already have a project open that has not been written to a file, **Airpak** will display a warning message as shown in Figure 6.2.2.

To save the current project before opening the new project, click the **Save** button. **Airpak** will open the **Save project** panel, where you can specify the name of the file. (See Section 3.4 for more details on saving a project file.) To start the new project without saving the current project, click the **Don't save** button. Alternatively, to cancel opening the new project, click the **Cancel switch** button.

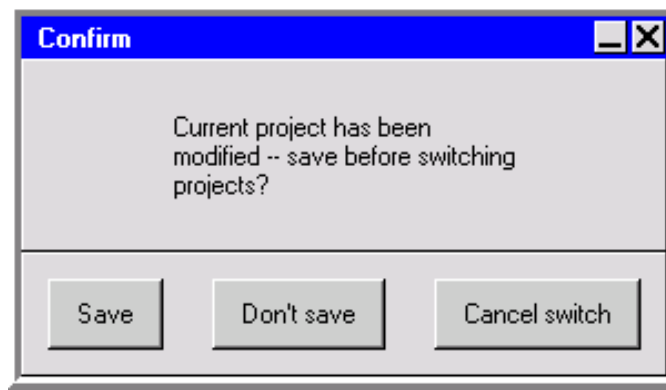




Figure 6.2.2: Warning Message Displayed When Opening a New Project Before Saving the Current Project

Title and Notes


You can enter a title and some notes for your new **Airpak** project in the **Title/notes** panel (Figure 6.2.3). To open the **Title/notes** panel, double-click on the **Title/notes** item under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Title/notes**

To enter a title and notes for your project:

1. Enter a title in the **Title** text entry box. The title can be longer and more descriptive than the project name. The title can be displayed in the graphics window by toggling **Project title** in the **Display** drop-down list in the **View** menu (see Section 2.1.2).
2. Enter notes for the project in the **Notes** text entry box. There is no restriction on the number and type of text characters you can use in your project's title and notes.
3. Click **Accept**.

6.2.2 Opening an Existing Project

You will use the **Open project** panel (Figure 6.2.4) to open an existing project. To do this, select **Open project** in the **File** menu or click on the  button in the **File commands** toolbar.

File → **Open project**

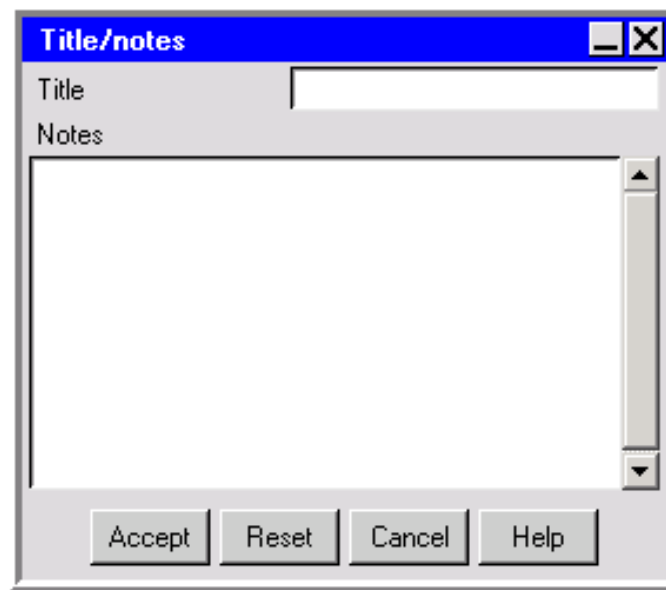


Figure 6.2.3: The Title/notes Panel

Select a project (e.g., `project1`) in the **Open project** panel (see Section 2.1.8 for information on selecting a file). If you want to apply the settings you specified under **Options** in the **Preferences** panel (see Section 6.3) from the previous time you worked on the project, turn on the **Apply user preferences from project** option.

Click **Open** to display the graphics for the selected model in the graphics window.

You can also use the **Open project** panel to delete projects (see Section 6.2.4), or to create a new directory (see Section 2.1.8).

6.2.3 Reloading the Main Version of a Project

When your project has multiple versions and solutions, and you wish to make a change to the original model, you can quickly revert to the main version of your **Airpak** project. To do this, select **Reload main version** in the **File** menu.

File→**Reload main version**

While **Airpak** allows you to have multiple versions of the same model, any changes you make to a version of a model will not be saved to the main version. **Airpak** allows you to quickly reload the main version of a model so that changes can be made.

Each project and its corresponding versions are stored in the project directory. The main version of the model can be identified by the `model`, `job`, and `project` files. Subsequent versions of the project can be identified with a version tag (i.e. `rad01.problem`, `rad01.job`, `rad01.model`, etc.).

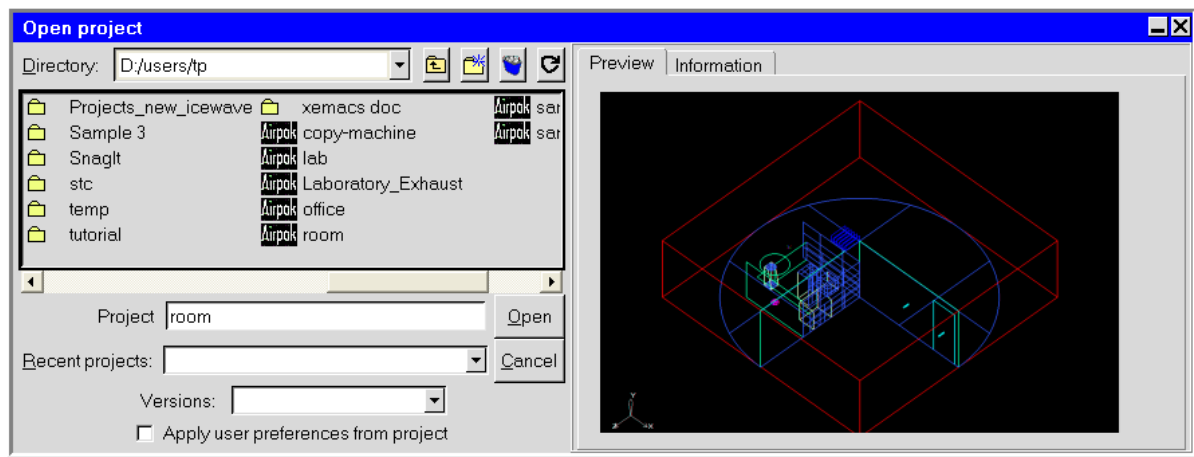




Figure 6.2.4: The Open project Panel

6.2.4 Deleting a Project

You can use the Open project panel (Figure 6.2.4) to delete an existing project. To do this, select Open project in the File menu or click on the  button in the File commands toolbar.

File→Open project

To delete an existing project, select the project (e.g., **project1**) in the Open project panel (Figure 6.2.4) and then click on the  button. When you delete a project, the project sub-directory (e.g., **/project1**) and all the files in it that were created for the project are deleted. A dialog box will ask you to confirm the deletion.

You can also delete projects in the Merge project panel (see Section 3.3) and the Save project panel (see Section 3.4).

6.3 Configuring a Project

You can configure your graphical user interface for the current project you are running, or for all Airpak projects, using the Options node of the Preferences panel (Figure 6.3.1).

Edit → Preferences

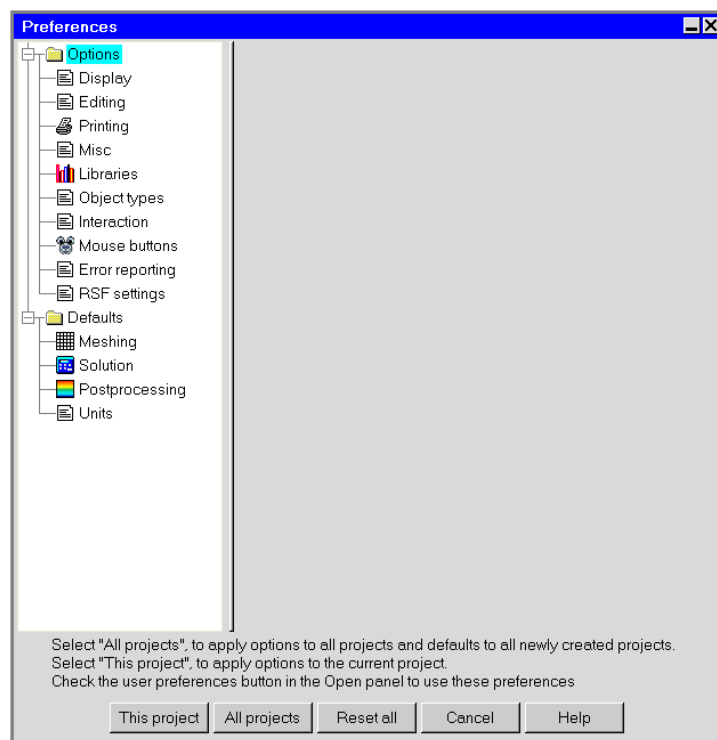


Figure 6.3.1: The Preferences Panel

When you click on one of the item nodes under the Options node, the Preferences panel (Figures 6.3.2–6.3.5) will change to allow you to define many configuration settings for your Airpak project.

The settings that are specified when any of the items under **Options** are selected in the **Preferences** panel are stored in a file named `.airpak_config`, which is located in your home directory. You should not modify the `.airpak_config` file directly; instead, make your desired configuration changes in the **Preferences** panel. The settings specified under **Defaults** apply directly to the **Airpak** model, and are not related to how the model is displayed or how you interact with the model. These are saved in `.airpak_defaults`.

The default settings for the **Preferences** panel are appropriate for many applications. To reset the panel to the default settings, click the **Reset all** button at the bottom of the panel (Figure 6.3.1). You can make changes to the **Preferences** panel, and apply the changes either to the current project by clicking **This project**, or to all **Airpak** projects by clicking on the **All projects** button. To close the panel without applying any changes, click **Cancel**.

A description of each item under the **Options** node in the **Preferences** panel is provided below. For other items in this panel, please see the appropriate section.




If you want to load project-specific **Options** that were saved during a previous **Airpak** session, you will need to turn on the **Apply user preferences from project** option in the **Open project** panel the next time you load the project. See Section 6.2.2 for more information about opening a project.

6.3.1 Display Options

To set display options for your model, select the **Display** item under the **Options** node in the **Preferences** panel.



Options →  Display

Color legend data format specifies the format of the labels that define the color divisions in the color legend for a postprocessing object (see Section 28.2.2 for details about the color legend and spectrum). The following data formats are available in **Airpak**:

exponential displays real values with a mantissa and exponent (e.g., 1.0e-02).

You can define the number of digits in the fractional part of the mantissa in the **Color legend precision** field.

float displays real values with an integral and fractional part (e.g., 1.0000). You can set the number of digits in the fractional part by changing the value of the **Color legend precision**.

general displays real values with either float or exponential form, depending on the size of the number and the defined **Numerical display precision**.

Color legend data format

☐ exponential ☐ float ☒ general

Numerical display precision

Color objects gray: ☐ None

☒ Unselected assemblies

☐ All

Display object names: ☐ None

☐ Selected object

☒ Current assembly

Screen up direction: ☒ Y ☐ Z

Display scale: X

 Y

 Z

Figure 6.3.2: The Display Section of the Preferences Panel


Numerical display precision defines the number of fractional digits displayed in the labels for the color legend.

Color objects gray contains options for changing the color of objects in the graphics window to gray.

None specifies that no objects are to be colored gray.

Unselected assemblies specifies that only assemblies that are not selected should be colored gray.

All specifies that all objects are to be colored gray.

Display object names contains options for displaying the names of **Airpak** objects in the graphics window. You can select from the following options, or you can click the  button in the **Viewing options** toolbar to cycle through them.

None specifies that no object names are displayed.

Selected object specifies that only the name of the selected object is displayed.

Current assembly specifies that the names of all objects in the current assembly are displayed. If no assemblies are defined, the names of all objects under the **Model** node are displayed.



Screen up direction allows you to choose the direction of the vertical axis to be either **Y** or **Z**.

Display scale allows you to specify the scaling for the display of your project in the graphics window. For example, if you have a long and thin model ($x=10$ m, $y=0.1$ m, $z=0.1$ m), you may want to view the model so that all directions are on the same scale in the graphics window. For the above example, you would enter 0.01 for **X**, 1 for **Y**, and 1 for **Z**.

Background color allows you to specify the background color of the graphics window by opening a **Select the new background color** window (or a similar panel, depending on the platform of your machine). Since this window is not a part of the **Airpak** application, the procedure for changing the color will vary by platform.

6.3.2 Editing Options

To set editing options for your model, select the **Editing** item under the **Options** node in the **Preferences** panel.

 **Options** →  **Editing**

Default dimensions allows you to specify whether you want **Start/end** or **Start/length** as the default selection for dimensions in the **Room** panel, the **Object** panels, and some of the **Macros** panels.



Figure 6.3.3: The Editing Section of the Preferences Panel

Annotation edit key specifies which keyboard key is used in conjunction with the mouse buttons to move legends, titles, etc. in the graphics window. You can select **Control** (for the <Control> key), **Shift** (for the <Shift> key), or **Meta** (for the <Alt> key).

i Note that, in this manual, descriptions of operations that use the **Annotation edit key** assume that you are using the default setting (i.e., **Control**). If you change the default setting, you will need to use the key you have specified, instead of the <Control> key, when you move legends, titles, etc. in the graphics window.

Fix values contains options for fixing the values of quantities in the various object panels. See Chapter 5 for details.



All specifies that the values of all quantities will be fixed. This will cause the **Fix values** option to be unavailable in the **Object** and **Room** panels.

None specifies that no quantities will be fixed. This will also cause the **Fix values** option to be unavailable in the **Object** and **Room** panels.

Per-object specifies that the **Fix values** option can be toggled on or off for each individual object. This is the default selection.

6.3.3 Printing Options

To set printing options for your model, select the **Printing** item under the **Options** node in the Preferences panel.

 **Options** →  **Printing**

Print command for text files (UNIX systems only) designates the command used to print text files. The symbols %p and %f designate the printer name and filename, respectively. When the command is issued, **Airpak** replaces %p with the name of your default printer (specified by the LPDEST environment variable). You can also

Print command for text files	<code>lp -d%p %f</code>
Print command for PS files	<code>lp -d%p %f</code>

Figure 6.3.4: The Printing Section of the Preferences Panel

specify the printer name explicitly (e.g., `-dmyprinter`). **Airpak** will replace `%f` with the name you specify for the file. The default command is

```
lp -d%p %f
```

The symbol `%t` can be used to denote a temporary file whose name is uniquely generated by **Airpak**.

This command is not used by Windows systems. The Windows **Print** menu can be used instead.

Print command for PS files (UNIX systems only) is the command used to print PostScript (PS) files. The default command is the same as the **Print command for text files**, described above.

This command is not used by Windows systems. The Windows **Print** menu can be used instead.

6.3.4 Miscellaneous Options

To set other miscellaneous options for your model, select the **Misc** item under the **Options** node in the **Preferences** panel.



 **Options** →  **Misc**





Figure 6.3.5: The Misc Section of the Preferences Panel

Xterm options (UNIX systems only) allows you to specify options for the text window that is opened using the **Shell window** item in the **File** menu. See Section 2.1.2 for more details on opening a text window.

Bubble help delay allows you to specify the delay time before the bubble help appears when you hold your mouse pointer over an item in the GUI. To disable the bubble help, specify a **Bubble help delay** of 0.

6.3.5 Editing the Library Paths

The **Libraries** section of the **Preferences** panel (Figure 6.3.6) allows you to change the path settings to include libraries of macros and materials so that **Airpak** can find them.

 **Options** →  **Libraries**

The path to the default **Airpak** library is displayed in the **Location** text field. By default, this library contains information about the materials (see Section 7.7) and macros (see Chapter 25) that are predefined in **Airpak**.

If you have materials or macros that are not predefined in **Airpak**, but you want to include them in your model, you can add new libraries to the list. When you start **Airpak**, it will load the material and macro information from the libraries specified in the list. See Section 7.7.6 for information about creating a user-defined **materials** file.

The following operations can be performed in the **Libraries** section of the **Preferences** panel:

- To create a new library, click **New library**. Enter a name for the library in the **Library name** text field and then enter the path in the **Location** text field or click **Browse** to search for a specific directory.

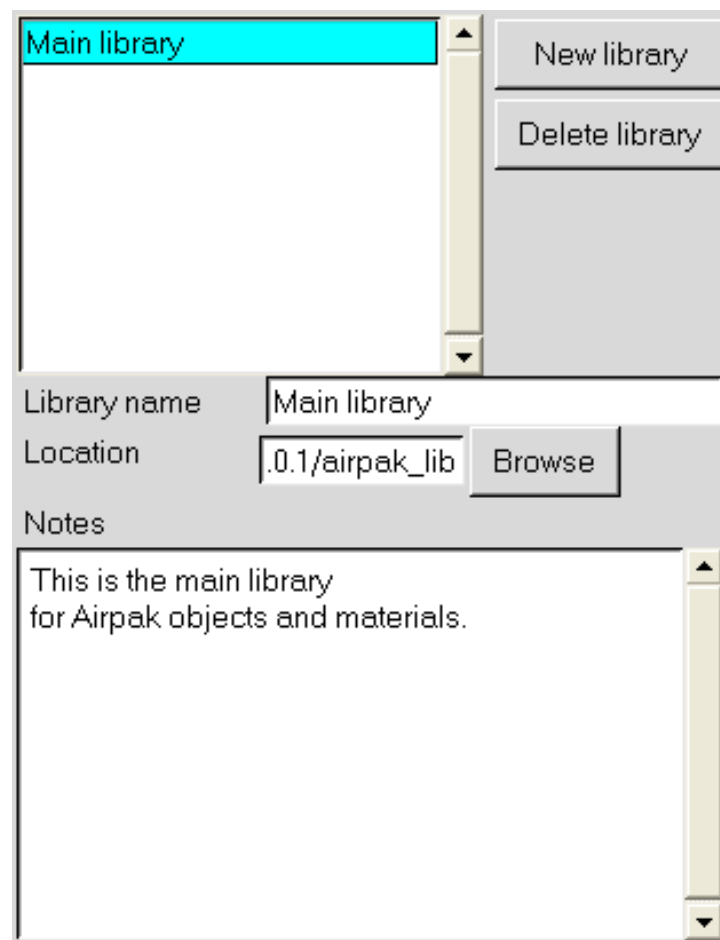


Figure 6.3.6: The Libraries Section of the Preferences Panel

- To modify an existing library path, select the library in the list of libraries and then edit the path in the **Location** text field, or click **Browse** to search for a specific directory.
- To delete a library, select the library name in the list of libraries and click **Delete library**.

You can make changes to the **Libraries** section of the **Preferences** panel, and apply the changes either to the current project by clicking **This project**, or to all **Airpak** projects by clicking **All projects**. Click **Cancel** to close the **Preferences** panel without saving the changes. Once you have created a new library, a new node appears under the **Libraries** node in the **Model manager** window.

Changing the Name of a Library

To change the name of any user-defined library, right-click on the specified library node and select **Edit info** in the resulting pull-down menu. **Airpak** will open the **Library name and info** panel (Figure 6.3.7), where you can change the **Name** or add any descriptive information in the large text entry field.

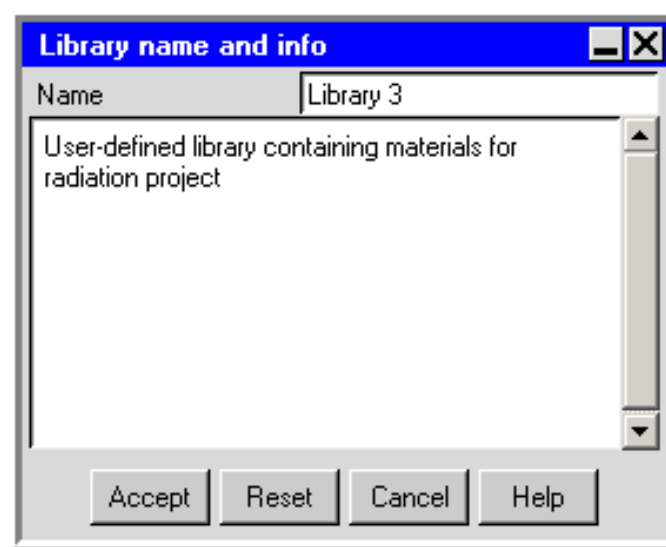


Figure 6.3.7: The Library name and info Panel



Adding Items to a Library

To add an item to a new library, select the item in the **Model manager** window and drag the item into the new library node. If the item you drag into the library is a material, a **Materials** node will appear under the new library node containing the new material

item in its appropriate category (Solid, Fluid, Surface, etc.). If the item you drag into the library is an object or assembly, Airpak will open a **Save project** panel. You will need to save the object or assembly as a separate project to make it available in the new library. See Section 3.4 for details about using the **Save project** panel.

6.3.6 Editing the Graphical Styles

The **Object types** section of the **Preferences** panel (Figure 6.3.9) allows you to customize the color, line width, shading, decoration, and font type of individual objects associated with your Airpak model, as they are displayed in the graphics window. For example, you may want to change the shading of a rack of PCBs to a different type to better distinguish it from other objects in your graphics window. The types of objects that you can customize in the **Graphical styles** panel are listed in Table 6.3.1. A description of each option in the **Object types** section of the **Preferences** panel follows.

 Options →  **Object types**

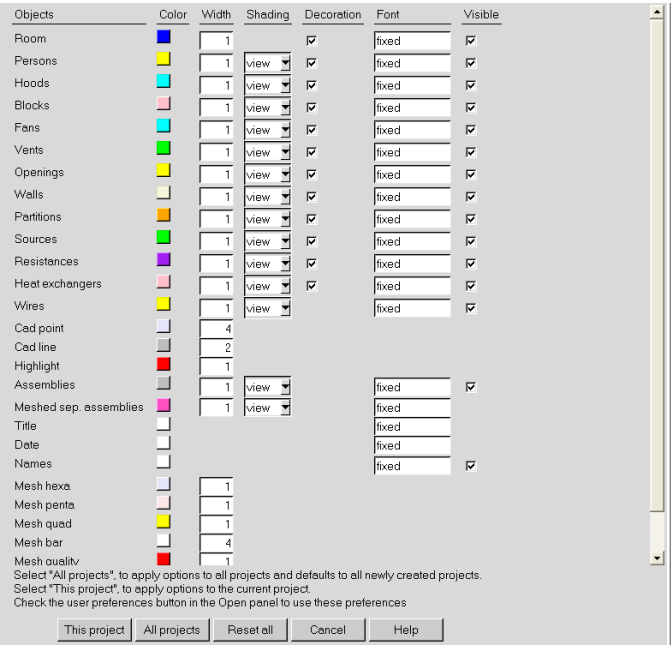


Figure 6.3.8: The **Object types** Section of the **Preferences** Panel

Color displays the color currently associated with the corresponding object type. When you click on this option, a color palette menu opens. You can replace the default color for an object by selecting the new color in the color palette menu. To select a different color, click on the icon in the lower right corner of the palette menu. Select a new color using the method or panel that is appropriate to your system.

Table 6.3.1: Object Choices for Editing Graphical Styles

Object		Description
Room		Airpak room
Persons	Hoods	Airpak modeling objects
Blocks	Fans	
Openings	Walls	
Partitions	Sources	
Resistances	Vents	
Wires	Assemblies	
Heat exchangers		
Cad point		Reference points and lines associated with model geometry that have been imported from IGES or DXF files.
Cad line		
Highlight		The color to which a model object changes when it is selected or becomes highlighted.
Assemblies		Any object, including a group, that you have created for reuse, and is stored either internally in the model or in a separate user-defined location.
Meshed sep. assemblies		Any assembly that is meshed separately from the rest of the model.
Title		The title and date of the model that can be displayed in the graphics window.
Date		
Names		The names of model objects that can be optionally displayed with the object.
Mesh hexa		The elements that comprise the mesh for volumes and surfaces.
Mesh penta		
Mesh quad		
Mesh bar		
Mesh quality		
Mesh selected		
Mesh adjacent		

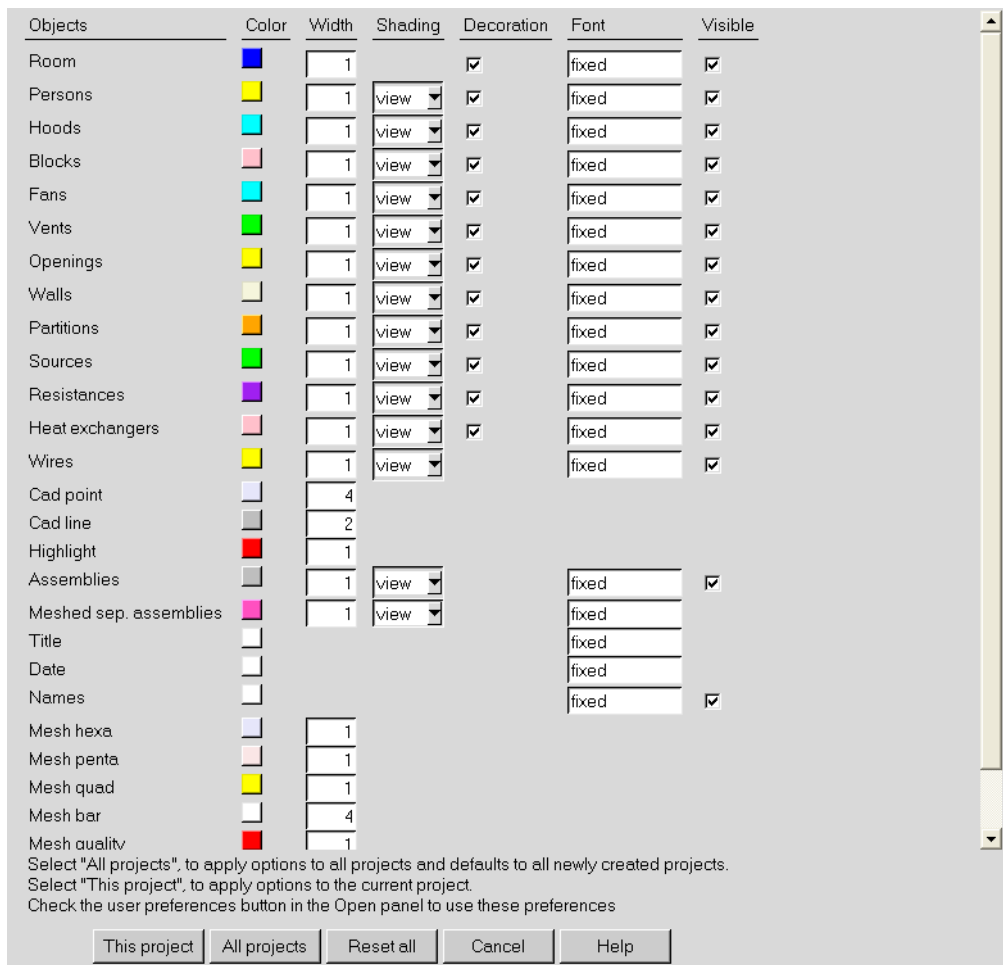


Figure 6.3.9: The Object types Section of the Preferences Panel

Width specifies the width of the line used to display the object, which is drawn in wireframe format.

Shading specifies the type of shading to be applied to an object when it is displayed. To change the default shading type, click on the square button to the right of the text field. Select a shading type from the resulting drop-down list: **view**, **wire**, **flat**, or **gouraud**. Note that when you select the **view** option, the type of shading that will be applied to the object is taken from the **Shading** drop-down list in the **View** menu (see Section 2.1.2).

Decoration is a toggle button that adds graphical detail (blades, deflectors, etc.) to fans, vents, openings, sources, resistances, and heat sinks. By default, decorations are turned on.



Font specifies the font used for text that is associated with the object.

Visible toggles whether the specified type of object will be visible in the graphics window.

6.3.7 Interactive Editing

The Interaction section of the Preferences panel (Figure 6.3.10) allows you to perform snapping when repositioning an object in the graphics window. Snapping can be done by:

- using a *grid-snap* technique to position the room or an object at specified discrete distances along each axis.
- using an *object-snap* technique to position an object using a vertex, line, or plane of another object.

 Options →  Interaction

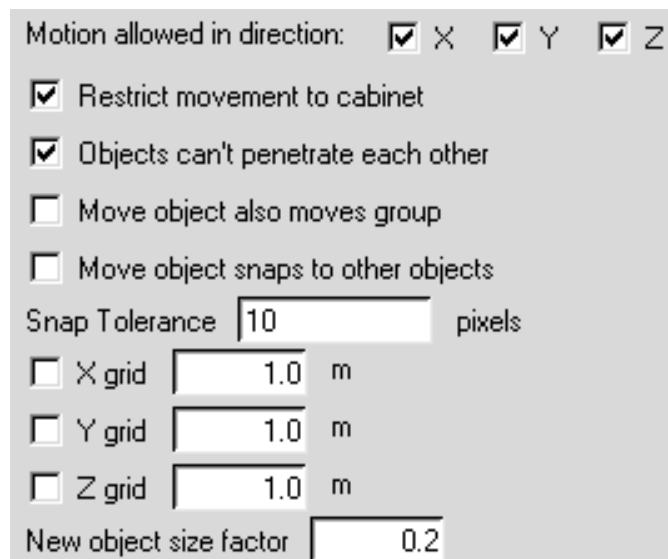


Figure 6.3.10: The Interaction Section of the Preferences Panel

You can set the global size of the grid independently along each axis. To set the global size of the grid for this project only, follow the steps below.

1. Select X grid, Y grid, and/or Z grid in the Interaction section of the Preferences panel.
2. Enter values for X grid, Y grid, and/or Z grid in the relevant text entry fields.
3. Click This project.



You can set the object snap length by specifying a value (in pixels) for the **Snap Tolerance**. When this value has been set, and the model is oriented in one of the **X**, **Y**, or **Z** views, a dragged object will automatically snap into alignment with a second object of similar shape when it comes within the specified number of pixels of a vertex, line, or plane of the second object.

See Section 7.3.7 for more information about moving objects and setting object interaction parameters.

To set the global size of the grid or the object snap length for all **Airpak** projects, follow the procedure above, but click **All projects** (instead of **This project**) in the **Interaction** section of the **Preferences** panel (Figure 6.3.10). **Airpak** sets the values for the grid-snap distances to 1 and the object snap length to 10 by default, and uses the default length units for your model. If none of the snap options are selected, the movement of the room or the object using the mouse will be continuous.

6.3.8 Meshing Options

To set meshing options for your model, select the **Meshing** item under the **Defaults** node in the **Preferences** panel (Figure 6.3.11).

 Defaults →  Meshing

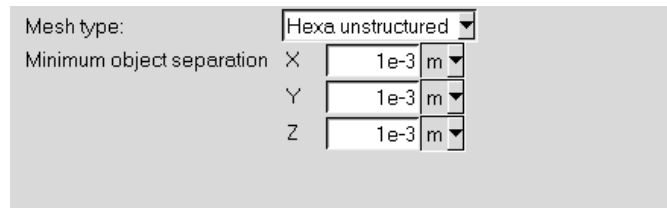




Figure 6.3.11: The Meshing Section of the Preferences Panel

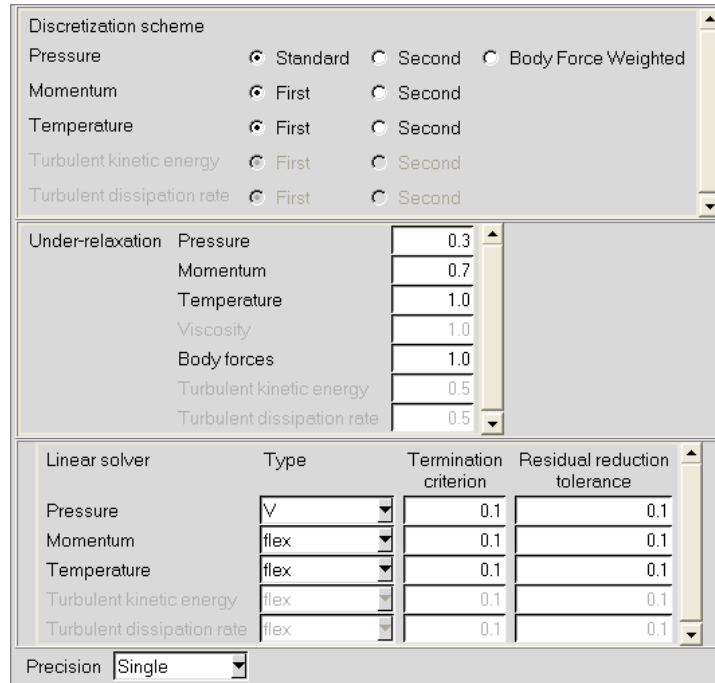
Mesh type is used to specify the type of mesh to be used in your **Airpak** project. By default, **Airpak** uses the **Hexa unstructured** option which denotes an unstructured hexahedral mesh. The other mesh types you can choose from are **Hexa cartesian** and **Tetra** (i.e. tetrahedral).

Minimum object separation is used to specify the minimum distance separating objects in your model in the *x*, *y*, and *z* coordinate directions. This distance may be expressed in any valid number format (e.g., 0.001, 1e-3, 0.1e-2). This specification is used by **Airpak** whenever the distance between two objects is less than this value, but greater than the model's zero tolerance.

6.3.9 Solution Options

To set advanced solution options for your model, select the **Solution** item under the **Defaults** node in the Preferences panel (Figure 6.3.12).

 Defaults →  Solution



Discretization scheme			
Pressure	<input checked="" type="radio"/> Standard	<input type="radio"/> Second	<input type="radio"/> Body Force Weighted
Momentum	<input checked="" type="radio"/> First	<input type="radio"/> Second	
Temperature	<input checked="" type="radio"/> First	<input type="radio"/> Second	
Turbulent kinetic energy	<input checked="" type="radio"/> First	<input type="radio"/> Second	
Turbulent dissipation rate	<input checked="" type="radio"/> First	<input type="radio"/> Second	

Under-relaxation		
Pressure		0.3
Momentum		0.7
Temperature		1.0
Viscosity		1.0
Body forces		1.0
Turbulent kinetic energy		0.5
Turbulent dissipation rate		0.5

Linear solver	Type	Termination criterion	Residual reduction tolerance
Pressure	V	0.1	0.1
Momentum	flex	0.1	0.1
Temperature	flex	0.1	0.1
Turbulent kinetic energy	flex	0.1	0.1
Turbulent dissipation rate	flex	0.1	0.1



Precision:

Figure 6.3.12: The Solution Section of the Preferences Panel

For details on advanced solver setup options see Sections 27.3 - 27.6.

6.3.10 Postprocessing Options

To set postprocessing options for your model, select the **Postprocessing** item under the **Options** node in the Preferences panel (Figure 6.3.13).

 Defaults →  Postprocessing

Post-processing tolerance is used during postprocessing operations for a number of tasks. For example, it is used to determine whether a point lies on a plane or an isosurface and to snap plane cuts to adjacent nodes. This tolerance, which is a dimensionless fraction of the cell size, may be expressed in any valid number format (e.g., 0.001, 1e-3, 0.1e-2).

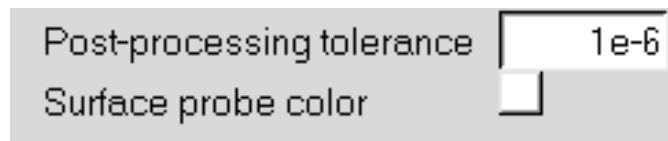





Figure 6.3.13: The Postprocessing Section of the Preferences Panel

Surface probe color allows you to change the color of the probed point and text. To change the color, click the colored rectangular next to **Surface probe color**. This will open a color palette menu. To select a different color, click on the icon in the lower right corner of the palette menu.

6.3.11 Other Preferences and Settings

When you click on other items in the Preferences panel, you will be able to edit additional preferences as follows:

-  **Units** allows you to modify the default unit definitions and conversion factors. See Chapter 5 for details.
-  **Mouse buttons** allows you to change the default mouse controls in Airpak. See Section 2.2.4 for details.
-  **Error reporting** allows you to pre-fill the Error report panel with the following information necessary to email a report: Your name, Your email address, HTTP proxy host (if any), and HTTP proxy port, as shown in Figure 6.3.14.

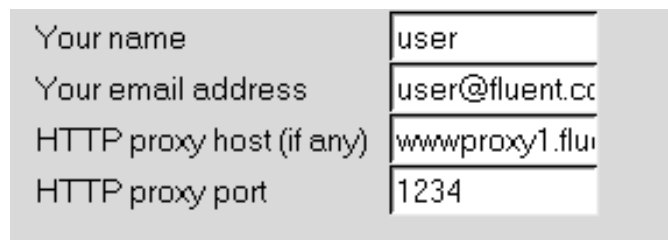


Figure 6.3.14: The Error reporting Section of the Preferences Panel

6.4 Specifying the Problem Parameters



You can specify parameters for the current model in Airpak using the **Basic parameters** panel (Figures 6.4.1, 6.4.2, 6.4.3, and 6.4.4). The types of parameters you can use to describe your model include the following:

- time variation
- solution variables
- species transport
- IAQ/comfort
- flow regime
- gravity
- ambient values
- default fluid, solid, and surface materials
- initial conditions

The default settings for the **Basic parameters** panel are as follows:

- steady-state
- solution of flow (velocity and pressure), temperature, and surface-to-surface radiation solution variables
- no species transport
- IAQ/thermal comfort postprocessing calculations activated
- turbulent flow
- natural convection included, compass setting with north in x direction
- ambient temperature of 20°C, ambient pressure of 0 N/m², ambient radiation temperature of 20°C
- fluid is air, solid is brick, and surface is non-metallic paint.
- ambient temperature and no flow for initial conditions

To open the **Basic parameters** panel (Figures 6.4.1, 6.4.2, 6.4.3, and 6.4.4), double-click on the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Basic parameters**

There are four push buttons at the bottom of the **Basic parameters** panel. To accept any changes you have made to the panel and then close the panel, click **Accept**. To undo all the changes you have made in the panel and restore all items in the panel to their original states when the panel was opened, click on the **Reset** button. To close the panel and ignore any changes made to it, click **Cancel**. To access the on-line documentation, click **Help**.

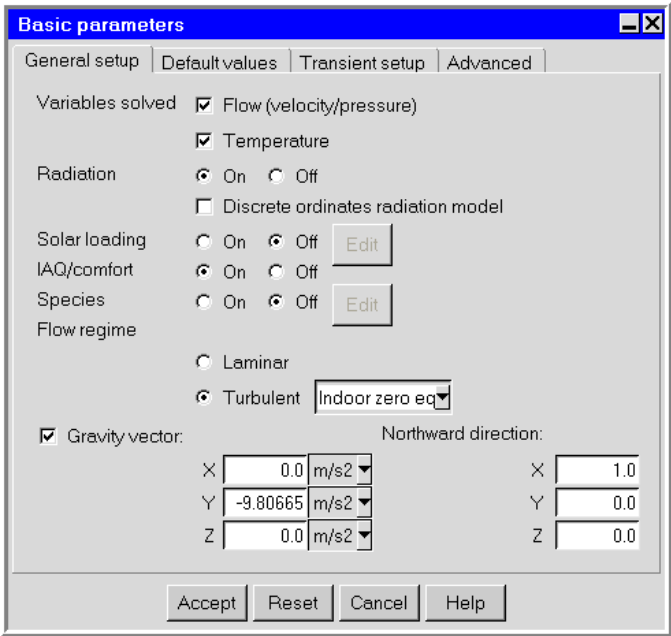


Figure 6.4.1: The Basic parameters Panel (General setup Tab)

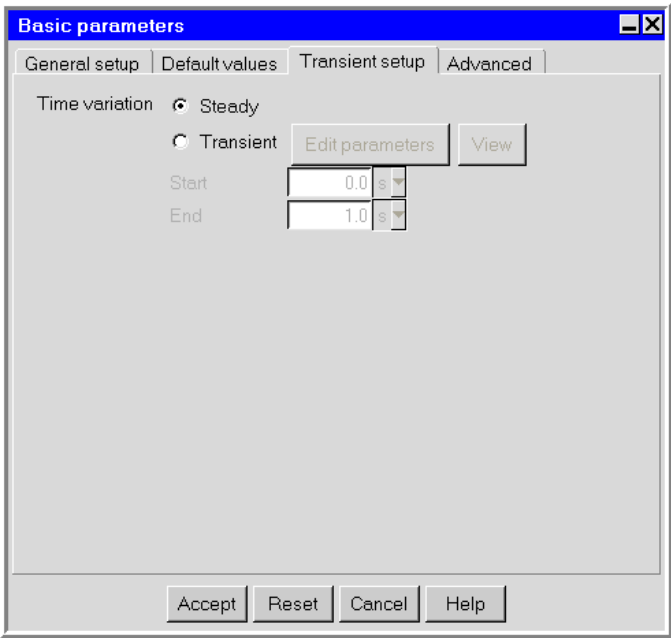


Figure 6.4.2: The Basic parameters Panel (Transient setup Tab)

6.4.1 Time Variation

Airpak allows you to solve two types of flow problems:

- steady-state
- transient

The default setting for time variation in the **Basic parameters** panel (**Transient setup** tab) is steady-state (Figure 6.4.2). The procedure for defining a transient simulation is described in Chapter 20.

6.4.2 Solution Variables

Airpak allows you to choose the variables you want it to solve in your simulation. Four options are presented in the **Basic parameters** panel (**General setup** tab): **Flow (velocity/pressure)**, **Temperature**, **Radiation**, and **Species**. (Figure 6.4.1). These options are discussed below.

Flow, Temperature, and Species Variables

Airpak allows you to solve the problem for any of the following combinations of flow, temperature, and/or species variables:

- flow (velocity and pressure fields) only
- flow and temperature distributions (velocity, pressure, and temperature fields)
- temperature distribution only
- species transport

In the case of temperature distribution only, the flow solution from a previous simulation (for the same problem with an identical mesh) can be used to supply a velocity field for the thermal simulation. This is useful when the dominant mechanism for heat transfer by the fluid is forced convection, rather than natural convection (see Section 6.4.4 for more details on forced and natural convection). In this case, the solution of the energy equation does not affect the flow solution, so these systems can be solved independently rather than as a coupled simulation. The flow-only simulation can be performed first, followed by multiple thermal simulations, if required.



You *must* include either flow variables or temperature variables in your simulation, or both.

To select the variables to be solved, follow one of the procedures below:

- To solve the problem for flow, select **Flow (velocity/pressure)** under **Variables solved** in the **General setup** tab of the **Basic parameters** panel (Figure 6.4.1). You must also specify the **Flow regime** in the **General setup** tab of the **Basic parameters** panel to be **Laminar** or **Turbulent** (see Section 6.4.3 for more details on selecting a flow type).
- To solve the energy equation for the temperature distribution together with the flow equations, select both **Flow (velocity/pressure)** and **Temperature** under **Variables solved** in the **General setup** tab of the **Basic parameters** panel.
- You can also solve first for the flow, and then for the temperature. This approach is valid only if the solution of the energy equation does not affect the flow solution, which is the case if forced convection is the dominant heat transfer mechanism and the **Gravity vector** is not enabled in the **Basic parameters** panel (**General setup** tab). To solve only for temperature using a previous solution, specify the ID for the previous solution next to **Restart** in the **Solve** panel and select **Full data** under **Restart**. To open the **Solve** panel, select **Run solution** in the **Solve** menu. See Section 27.11.1 for details on restarting a calculation using a previous solution. The previous solution will be used to access the flow field for use in solving the energy equation. Note that solving only for temperature using a velocity field of zero is equivalent to solving only for heat conduction.
- To solve the problem for species, select **On** for **Species** in the **General setup** tab of the **Basic parameters** panel (Figure 6.4.1). The procedure for defining a species transport calculation is described in Chapter 21.

Airpak also includes the capability to predict indoor air quality (IAQ) parameters, such as the mean age of air, and thermal comfort parameters including the mean radiant temperature, predicted mean vote (PMV), and predicted percentage dissatisfied (PPD). By default, Airpak performs a supplementary calculation immediately after the solution of the flow, temperature, and species equations to compute the mean age of air and the mean radiant temperature. If you do not want to calculate the mean age of air and the mean radiant temperature for your simulation, select **Off** next to **IAQ/Comfort** in the **Problem setup** panel. See Section 30.2.8 for details on calculating PMV and PPD.

Radiation Variables

You can choose whether to solve the problem for radiation. If you want to solve for radiation, select **On** for **Radiation** in the **General setup** tab of the **Basic parameters** panel (Figure 6.4.1). If you do not want to solve for radiation, select **Off**. The surface-to-surface radiation model is used for modeling radiation by default. If you want to use the discrete ordinates (DO) radiation model instead, select the **Discrete ordinates radiation model** in the **Basic parameters** panel.

See Chapter 22 for information on radiation modeling.

Postprocessing for Solution Variables

Airpak provides postprocessing options for displaying, plotting, and reporting the solution variables. The following variables are contained in the **Variable** and **Value** drop-down lists that appear in the postprocessing and reporting panels. See Chapter 30 for their definitions.

Velocity-related quantities that can be reported are as follows:

- UX
- UY
- UZ
- Speed
- Velocity
- Mass flow
- Volume flow
- Angular deviation

Pressure-related quantities that can be reported are as follows:

- Pressure

Temperature-related quantities that can be reported are as follows:

- Temperature
- Heat flux
- Heat flow
- Heat tr. coeff

Species-related quantities that can be reported are as follows:

- species (mass), the mass fraction of a species
- species (mole), the mole fraction of a species

IAQ/comfort-related quantities that can be reported are as follows:

- Relative humidity
- Mean age of air
- Radiation temp, the mean radiant temperature
- PPD
- PMV

Radiation-related quantities that can be reported are as follows:

- Radiative heat flow

6.4.3 Flow Regime

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum and energy, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

Laminar flow is smooth, regular, deterministic, and steady (unless you define a transient simulation). Turbulent flow is random, chaotic, non-deterministic, and essentially unsteady due to statistical fluctuations. For laminar flow, **Airpak** solves the classical Navier-Stokes and energy conservation equations. For turbulent flow, **Airpak** solves the Reynolds-averaged forms of these equations, which, in effect, smooth out (time-average) the stochastic fluctuations. See Section 31.2 for more details on these equations.

Laminar Flow

In laminar flow, fluid mixing and heat transfer take place on a molecular level. The molecular (or dynamic) viscosity and the thermal conductivity are the quantities that measure the amount of mixing and heat transfer.

Turbulent Flow

In turbulent flow the degree of fluid mixing and heat transfer is much greater than in laminar flow, and takes place on a global, or macroscopic, level rather than on a molecular level. The amount of fluid mixing is measured by an effective viscosity, which is the sum of the dynamic viscosity and the turbulent eddy viscosity. The latter is not a measurable quantity since it depends on the details of the flow. The eddy viscosity is part of the turbulence model and is calculated by **Airpak**. The eddy viscosity is typically 100–1000 times greater than the measured molecular viscosity.

Similarly, for turbulent flow, the amount of heat transfer is measured by an effective thermal conductivity, which is the sum of the fluid's thermal conductivity and a turbulent conductivity.

Airpak provides a mixing-length zero-equation turbulence model, an indoor zero-equation turbulence model, a two-equation turbulence model (the standard k - ϵ model), the RNG k - ϵ turbulence model, and the Spalart-Allmaras turbulence model. In most ventilation design cases, the indoor zero-equation model will sufficiently account for the effects of turbulence. See Section 31.2 for details on the turbulence models available in **Airpak**.

Specifying the Flow Regime

The nature of the flow regime (laminar or turbulent) is indicated by the values of certain dimensionless groups, such as the Reynolds and Rayleigh numbers. The Reynolds number is generally the appropriate measure for forced convection, while the Rayleigh number is generally appropriate for natural convection. The flow is laminar when these numbers are relatively small and turbulent when they are large. See Section 6.4.4 for more details on forced and natural convection and Rayleigh and Reynolds numbers.

To specify the flow regime for your **Airpak** model, follow the steps below:

1. Select **Laminar** or **Turbulent** under **Flow regime** in the **General setup** tab of the **Basic parameters** panel (Figure 6.4.1) to specify a laminar or turbulent flow.
2. If you select **Turbulent**, activate the turbulence model to be used in the simulation by selecting **Zero equation**, **Indoor zero equation**, **Two equation (standard k - ϵ model)**, **RNG**, or **Spalart-Allmaras** in the drop-down list.

Determining Turbulence Parameters for Objects in Your Airpak Model

If you use one of the k - ϵ models (standard k - ϵ or RNG k - ϵ) to model turbulent flow in your **Airpak** model, you can specify parameters related to transported scalar quantities for fans, vents, and free openings. There are two ways to specify the turbulence parameters for an object: you can specify the turbulent kinetic energy (k) and dissipation rate (ϵ); or you can specify the turbulence intensity (I) and length scale (ℓ). This section provides guidelines for the most appropriate way of determining the turbulence parameters.

Specification of Turbulence Quantities

In most situations, it is appropriate to specify a uniform value of the turbulence quantity at the boundary where inflow occurs. Examples are fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown.

In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be used to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can “swamp” the boundary layers.

You can specify the turbulence quantities in terms of turbulent kinetic energy (k) and turbulent dissipation rate (ϵ). Alternatively, you can specify the turbulence quantities in terms of more convenient quantities such as turbulence intensity (I) and turbulence length scale (ℓ). These quantities are discussed further below.

Turbulence Intensity

The turbulence intensity, I , is defined as the ratio of the root-mean-square of the velocity fluctuations, u' , to the mean flow velocity, u_{avg} .

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, you will have a good estimate of the turbulence intensity at the fan, vent, or free opening from external, measured data.

For internal flows, the turbulence intensity at the fan, vent, or free opening is dependent on the upstream history of the flow. If the flow upstream is under-developed and undisturbed, you can use a low turbulence intensity. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully-developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows:

$$I \equiv \frac{u'}{u_{\text{avg}}} = 0.16(Re_{D_H})^{-1/8} \quad (6.4-1)$$

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

Turbulence Length Scale

The turbulence length scale, ℓ , is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows.

In fully-developed duct flows, ℓ is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between ℓ and the physical size of the duct is

$$\ell = 0.07L \quad (6.4-2)$$

where L is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe. In a channel of non-circular cross-section, you can base L on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of Equation 6.4-2, which relates a physical dimension (L) to the turbulence length scale (ℓ), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

For wall-bounded flows in which the fan, vent, or free opening involves a turbulent boundary layer, you should use the turbulence intensity and length scale method to define the turbulence parameters. You should use the boundary-layer thickness, δ_{99} , to compute the turbulence length scale, ℓ , from $\ell = 0.4\delta_{99}$.

Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as I , L , or μ_t/μ , you must typically resort to an empirical relation. Several useful relations, most of which are used within **Airpak**, are presented below.

Estimating Turbulent Kinetic Energy from Turbulence Intensity

The relationship between the turbulent kinetic energy, k , and turbulence intensity, I , is

$$k = \frac{3}{2}(u_{\text{avg}}I)^2 \quad (6.4-3)$$

where u_{avg} is the mean flow velocity.

This relationship is used in **Airpak** whenever the turbulence intensity and length scale method is used instead of specifying explicit values for the turbulent kinetic energy (k) and turbulent dissipation rate (ϵ).

Estimating Turbulent Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ , you can determine ϵ from the relationship

$$\epsilon = C_{\mu}^{3/4} \frac{k^{3/2}}{\ell} \quad (6.4-4)$$

where C_{μ} is an empirical constant specified in the turbulence model (approximately 0.09). The determination of ℓ is discussed above.

This relationship is used in **Airpak** whenever you use the intensity and length scale method instead of specifying explicit values for the turbulent kinetic energy (k) and turbulent dissipation rate (ϵ).

Postprocessing for Turbulent Flows

Airpak provides postprocessing options for displaying, plotting, and reporting the solution variables. You can generate graphical plots or reports of the following quantities:

- TKE (two-equation and RNG turbulence models only)
- Epsilon (two-equation and RNG turbulence models only)
- Viscosity ratio

These variables are contained in the **Variable** and **Value** drop-down lists that appear in the postprocessing and reporting panels. See Chapter 30 for their definitions.

6.4.4 Forced- or Natural-Convection Effects

Natural (or free) convection arises when the air density varies due to temperature differences.

The motion of fluid in an enclosure has a significant effect on the temperature distribution in the enclosure by convecting heat from one area to another. Forced convection occurs when a device such as a fan is pushing air past a heated object and convecting heat from the object as a result of its motion. In some applications, both forced and free convection (i.e., mixed convection) play a role in determining the overall temperature distribution. In general, forced-convection effects greatly dominate natural-convection effects when fans are present. Both forced-convection and natural-convection flows can be modeled by **Airpak**.

For most ventilation flows having relatively low air velocities, natural-convection effects are important and it is essential to include the effects of gravity. When gravity is present, either the Boussinesq approximation or the ideal gas law is used (see Section 31.3 for

more details about these models). The Boussinesq model is the default approach for single-fluid problems; however, the ideal gas law should be used for problems in which density differences are greater than a few percent. For mixtures containing two or more species, the ideal gas law approach is used.

Including Gravity Effects

The effects of gravity are included in your **Airpak** simulation by default. To set the gravitational acceleration in each Cartesian coordinate direction, enter the appropriate values in the X, Y, and Z fields in the **General setup** tab of the **Basic parameters** panel. Note that the default gravitational acceleration in **Airpak** is -9.80665 m/s^2 in the *y* direction. To ignore the gravitational effects in your calculation, turn off the **Gravity vector** option in the **Basic parameters** panel.

Including Temperature-Dependent Density Effects

Airpak provides two options for the definition of a temperature-dependent fluid density. The default option is the Boussinesq model, which should be used for natural-convection problems involving small changes to temperature (see Section 31.3.1).

The second option for a single fluid problem is the ideal gas law (see Section 31.3.2), which should be used when density differences are greater than a few percent. The ideal gas law should not be used to calculate time-dependent natural convection in closed domains, because this could result in the violation of the principle of the conservation of mass for a closed system. Note that the ideal gas law is used by default for problems containing a mixture of fluids.

To use the ideal gas law to define a temperature-dependent fluid density, follow the steps below.

1. Click the **Advanced** tab in the **Basic parameters** panel (Figure 6.4.3).
2. Select the **Ideal gas law** option.
3. Set the operating pressure (**Operating pressure**).



The input of the operating pressure is of great importance when you are computing density with the ideal gas law. You should use a value that is representative of the mean flow pressure. The operating pressure is set to 101325 Pa by default, which is the atmospheric pressure at sea level. The operating pressure will decrease with increasing altitude.

4. Select **Operating density** and set the operating density, if required. By default, **Airpak** will compute the operating density by averaging over all elements. In some cases, you may obtain better results if you explicitly specify the operating density instead

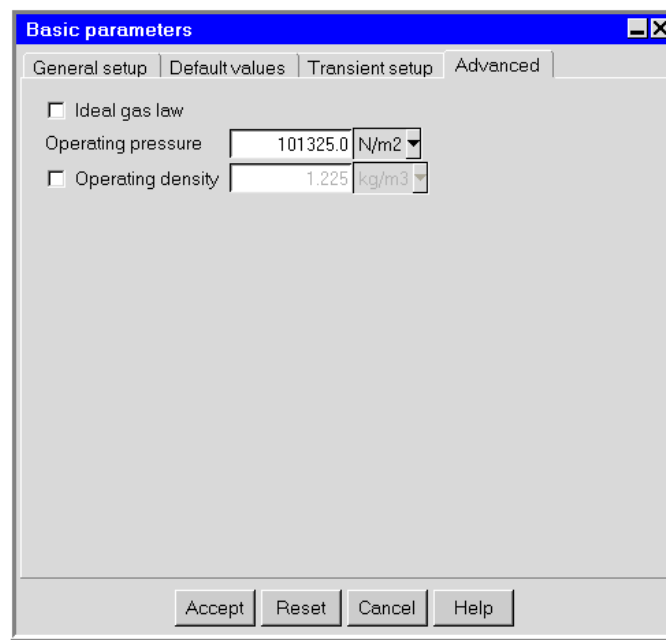


Figure 6.4.3: The Basic parameters Panel (Advanced Tab)

of having **Airpak** compute it for you. For example, if you are solving a natural-convection problem with a pressure boundary, it is important to understand that the pressure you are specifying is p'_s in Equation 31.3-7. Although you will know the actual pressure p_s , you will need to know the operating density in order to determine p'_s from p_s . Therefore, you should explicitly specify the operating density rather than use the computed average. The specified value should, however, be representative of the average value.

In some cases, the specification of an operating density will improve convergence behavior, rather than the actual results. For such cases, use the approximate bulk density value as the operating density, and be sure that the value you choose is appropriate for the characteristic temperature in the domain.

i If you use the ideal gas law and you have created a new fluid material or copied a fluid material, make sure that you specify the correct molecular weight for the new or copied material.

Determining the Flow Regime

Prior to solving the model, **Airpak** will determine whether the flow will be dominated by forced or natural convection. For problems dominated by forced convection, **Airpak** computes the Reynolds number (Re) and the Peclet number (Pe), both of which are

dimensionless. For flows dominated by natural convection (i.e., buoyancy-driven flows), **Airpak** computes the Rayleigh number (Ra) and the Prandtl number (Pr), which are also dimensionless.

The Reynolds number measures the relative importance of inertial forces and viscous forces. When it is large, inertial forces dominate, boundary layers form, and the flow may become turbulent. The Peclet number is similar to the Reynolds number and measures the relative importance of advection to diffusion for the transport of heat. For most flows simulated by **Airpak**, both the Reynolds and Peclet numbers are large.

The Prandtl number measures the relative magnitude of molecular diffusion to thermal diffusion. The Rayleigh number is a measure of the importance of the buoyancy effects.

If the Reynolds number is greater than 2000 or the Rayleigh number is greater than 5×10^7 , then selecting the **Turbulent** option in the **General setup** tab of the **Basic parameters** panel is recommended (see Section 6.4.3 for more details on specifying the flow type).

To review the estimates of the Reynolds and Peclet numbers or the Prandtl and Rayleigh numbers, open the **Basic Settings** panel by double-clicking on the **Basic settings** item under the **Solution settings** node in the **Model manager** window. Click **Reset** in the **Basic settings** panel. **Airpak** recomputes the solver setup defaults based on the physical characteristics of the model as defined, and displays estimates of the Reynolds and Peclet numbers or the Prandtl and Rayleigh numbers in the **Message** window.

6.4.5 Compass Orientation of Your Model

If you are using the atmospheric boundary layer macro (see Section 25.1.2 for details), it is important to specify the vector that defines the northward direction for your model. To specify this vector, enter the appropriate values in the X, Y, and Z fields under **Northward direction** in the **Problem setup** panel (Figure 6.4.1). Note that the default northward direction in **Airpak** is the x direction.

6.4.6 Ambient Values

Ambient values reflect the conditions surrounding the outside of the room. You can specify ambient values for pressure and radiation temperature by entering values in the **Gauge Pressure** and **Radiation temp** text entry boxes under **Ambient** in the **Default values** tab of the **Basic parameters** panel (Figure 6.4.4). Note that the ambient value for pressure must be the gauge pressure. You can also enter an ambient value for **Temperature**, and you can define this temperature to vary as a function of time for a transient simulation. The **Temperature** specified in the **Basic parameters** panel is used as s_0 in the transient equations (see Chapter 20 for information on transient simulations).

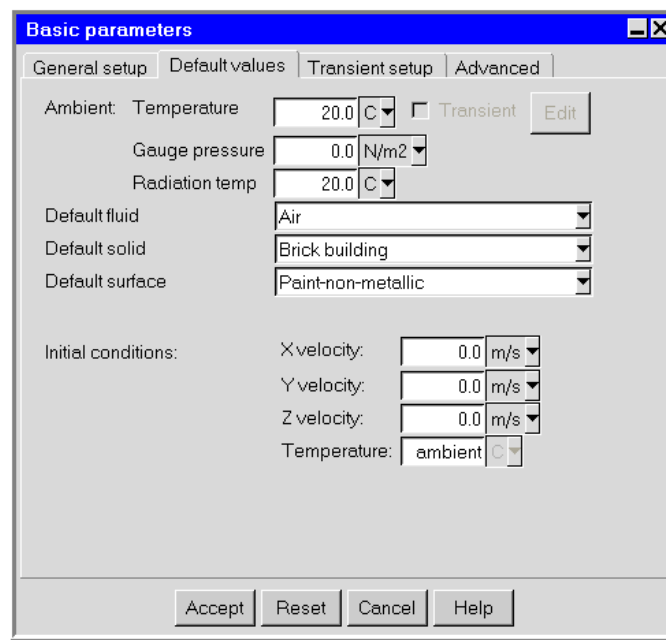



Figure 6.4.4: The Basic parameters Panel (Default values Tab)

6.4.7 Default Fluid, Solid, and Surface Materials

Airpak allows you to specify a default material for fluids, solids, and surfaces in the Default values tab of the Basic parameters panel (Figure 6.4.4). The default fluid in Airpak is Air, the default solid is Brick Building(brick), and the default surface is Paint-non-metallic (non-metallic paint).

Changing the Default Material

To change a default material, follow the steps below:

1. Click on the  button located next to the relevant text field to display the list of available materials.
2. Place the mouse pointer over the new list item (e.g., Al-Pure under Metal/alloys in the Default solid drop-down list). If the item is not visible, you can use the scroll bar.
3. Click the left mouse button on the item to make the new selection. The list will close automatically, and the new selection will then be displayed.

If you want to abort the selection process while the list is displayed, click **Cancel** at the bottom of the list.

Changing the Properties of a Material

The properties of the materials in the Default fluid, Default solid, and Default surface lists can be modified using the **Materials** panel. To open the **Materials** panel, select the **Edit definition** option from the materials drop-down list. For more information on material properties, see Section 7.7.

Specifying the Material for Individual Objects

Any modeling objects that require the specification of a fluid, a solid, or a surface material will be defined with the relevant default material specified in the **Basic parameters** panel (**Default values** tab), by default. For example, if you create a solid block in your model, the material specified for the solid block will be shown as **default** in the **Blocks** panel, which is the default solid material in the **Basic parameters** panel (i.e., **Al-Extruded**). You can change the selection of the fluid, solid, or surface material for individual objects by selecting the new material in the drop-down list in the panel related to that object. For example, to change the solid material for a solid block from **default** (**Brick Building**) to **Al-Pure**, follow the steps below:

1. Open the list of available materials for **Solid material** in the **Blocks** panel.
2. Scroll down the list and under **Metals/alloys** select **Al-Pure** in the materials drop-down list.

You can view the properties of the currently selected material, edit the definition of the material, and create a new material using the **Materials** panel, as described in Section 7.7.

6.4.8 Initial Conditions

Airpak allows you to set initial conditions for the fluid in your model. If you are performing a steady-state analysis, the initial conditions are the initial guess for the various solution fields used by the solution procedure. If you are performing a transient simulation, the initial conditions are the physical initial state of the fluid.

You can specify an initial **X velocity**, **Y velocity**, **Z velocity**, and **Temperature** for all objects in the model by entering values under **Initial conditions** in the **Default values** tab of the **Basic parameters** panel (Figure 6.4.4).

Once you have created a new project file (or opened an existing project) using the File menu (see Chapter 6), you are ready to build your **Airpak** model. The default room will be displayed in the graphics window when you create a new project. You can resize the room if desired, and then add objects to the room using the **Object creation** toolbar. This chapter begins with an overview of the **Model** menu and the **Object creation** toolbar, and describes information related to building your **Airpak** model. Once you have built your model, you will go on to mesh it, as described in Chapter 26.

The information in this chapter is divided into the following sections:

- Section 7.1: Overview
- Section 7.2: Defining the Room
- Section 7.3: Configuring Objects Within the Room
- Section 7.4: Object Attributes
- Section 7.5: Adding Objects to the Model
- Section 7.6: Grouping Objects
- Section 7.7: Material Properties
- Section 7.8: Custom Assemblies
- Section 7.9: Checking the Design of Your Model

7.1 Overview

In creating your **Airpak** model, you will make use of several parts of the graphical user interface: the **Object creation** toolbar, the **Object modification** toolbar, the **Model** node in the **Model manager** window, and the **Model** menu.

7.1.1 The Object creation Toolbar

The **Object creation** toolbar (Figure 7.1.1) contains buttons that allow you to add objects to your **Airpak** model and specify material properties for those objects. See Chapters 8–19 for information about adding specific objects to your **Airpak** model. See Section 7.7 for information about specifying object material properties.



Figure 7.1.1: The Object creation Toolbar

7.1.2 The Object modification Toolbar

The Object modification toolbar (Figure 7.1.2) is used to make changes to objects in your Airpak model and contains buttons that allow you to edit, move, copy, delete, and align objects that you have created within the room. See Section 7.3 for information about modifying object configuration within the room. See Section 7.4 for information about modifying individual object attributes.

7.1.3 The Model Node in the Model manager Window

The Model node in the Model manager window (Figure 7.1.3) is used for many of the same functions as the Object creation and Object modification toolbars. Right-clicking on the Model node and its corresponding items allows you to create, edit, move, copy, and delete objects within your Airpak model. See Section 7.3 for information about modifying object configuration within the room. See Section 7.4 for information about modifying individual object attributes.



Figure 7.1.2: The Object modification Toolbar

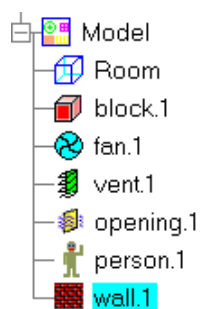


Figure 7.1.3: The Model Node in the Model manager Window

7.1.4 The Model Menu

The **Model** menu contains options that you will typically use once you have finished adding **Airpak** objects to your model. These options include functions related to generating a mesh (**Model**→**Generate mesh**) and specifying the order in which objects are meshed (**Model**→**Edit priorities**). These functions are described in Chapter 26. There are also options related to importing geometry from third-party CAD software (**Model**→**CAD data**) and radiation modeling (**Model**→**Radiation**) that are described in Chapters 4 and 22, respectively.

7.2 Defining the Room

When you start a new project, **Airpak** automatically creates a 3D rectangular room with the dimension $1\text{ m} \times 1\text{ m} \times 1\text{ m}$ and displays the room in the graphics window. The default view of the room is in the direction of the negative z axis. The sides of the room represent the physical boundary of the model and no object (except for external walls with non-zero thickness) can extend outside the room.

The **Edit** window in the lower right corner of the screen will become the room **Edit** window (Figure 7.2.1).

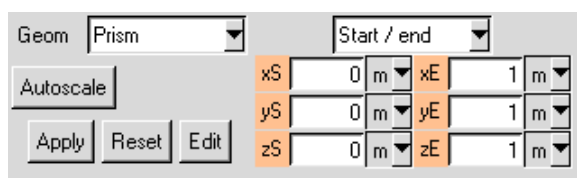


Figure 7.2.1: The Room Edit Window

The options for modifying the default room are as follows:

- resizing the room
- repositioning the room
- changing the walls of the room
- changing the description of the room
- changing the graphical style of the room

Each option is described in detail below. Click the **Apply** button to modify the room to reflect any changes you have made to the text entry fields in the room **Edit** window. Click the **Reset** button to undo all the changes you have made to the text entry fields in the panel and to restore all text entry fields in the panel to their original states. Click the **Edit** button to open the **Room** panel (Figures 7.2.2–7.2.5).



7.2.1 Resizing the Room

You can resize the room in several different ways:

- Specify the dimensions of the room in the room **Edit** window (Figure 7.2.1). You can specify the starting and ending points of the room by selecting **Start/end** in the drop-down list at the top of the window and entering the starting point of the room (x_S , y_S , z_S) and the ending point of the room (x_E , y_E , z_E). After you type a number into a text entry field (or type numbers in several fields), you must click **Apply** or press the <Enter> key on the keyboard to update the model and display the updated model in the graphics window. If you do not click **Accept** or press the <Enter> key, **Airpak** will not update the model.

Alternatively, you can specify the starting point of the room and the length of the sides of the room by selecting **Start/length** in the drop-down list and entering the starting point (x_S , y_S , z_S) and the lengths of the sides (x_L , y_L , and z_L) of the room.

- Specify the dimensions of the room in the **Room** panel (Figures 7.2.2–7.2.5). To open this panel, double-click on the **Room** item under the **Model** node in the **Model** manager window.

 Model →  Room

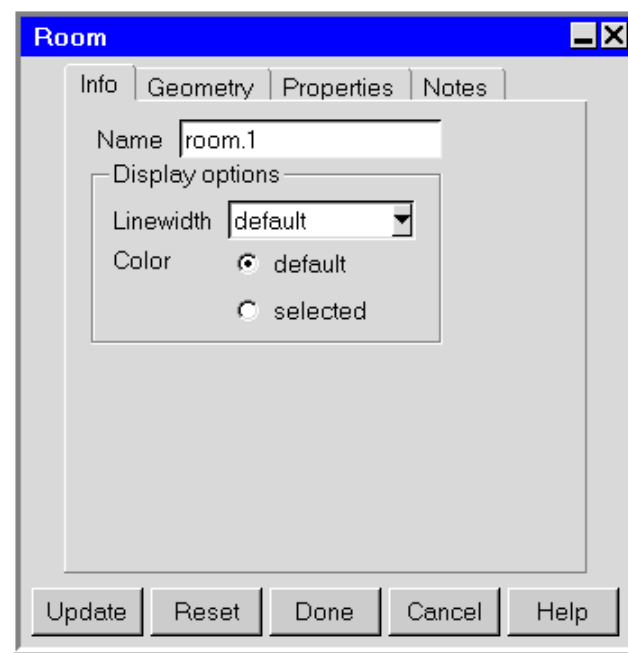


Figure 7.2.2: The Room Panel (Info Tab)

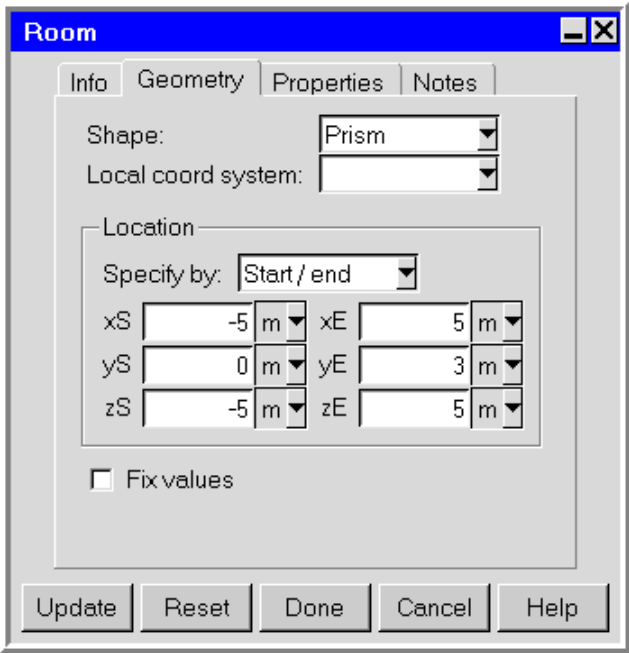


Figure 7.2.3: The Room Panel (Geometry Tab)

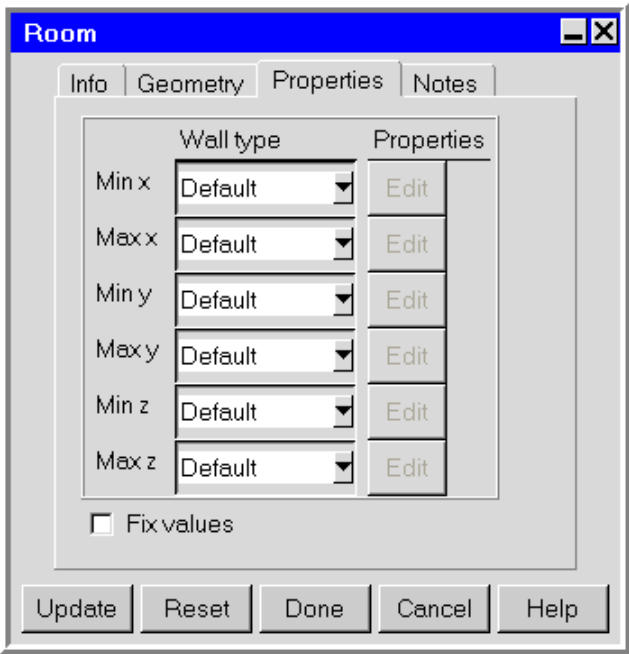


Figure 7.2.4: The Room Panel (Properties Tab)

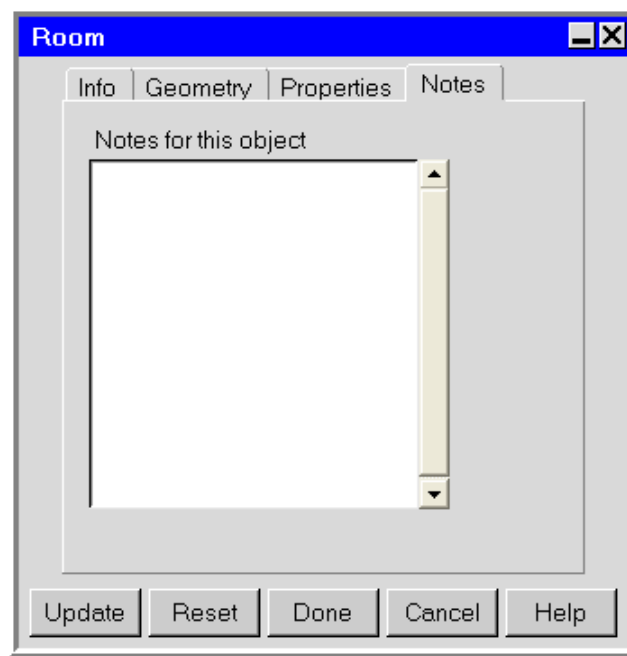


Figure 7.2.5: The Room Panel (Notes Tab)


You can specify the starting and ending points of the room by clicking on the **Geometry** tab in the **Room** panel, selecting **Start/end** from the **Specify by** drop-down list, and entering the starting point of the room (x_S , y_S , z_S) and the ending point of the room (x_E , y_E , z_E). Alternatively, you can specify the starting point of the room and the length of the sides of the room by selecting **Start/length** from the **Specify by** drop-down list and entering the starting point (x_S , y_S , z_S) and the lengths of the sides (x_L , y_L , and z_L) of the room.

- Hold down the <Shift> key on the keyboard, use the right mouse button to click on the room, and then move the mouse to shrink or enlarge the room.
- In the object **Edit** window (Figure 7.2.1), click on the coordinate (displayed in orange) that you want to change. Click the left mouse button on a point in the graphics window that is inside the boundary of the current room. The room will be contracted in the chosen coordinate direction. For example, if you click on y_E (value = 1.0 m) and then click on a point in the graphics window with $y=0.7$ m, then the room height will be contracted such that y_E is now 0.7 m. The x and z coordinates will remain constant.



This feature cannot be used to extend the room beyond the boundaries that were in place prior to resizing.

- Scale the room using the **Move all objects in model** panel (Figure 7.2.6). To open

this panel, right-click on **Room** under the **Model** node in the **Model manager** window and select **Move object** from the pull-down menu. Alternatively, you can select the room and click on the  button in the **Object modification** toolbar.

 Model →  Room → Move object

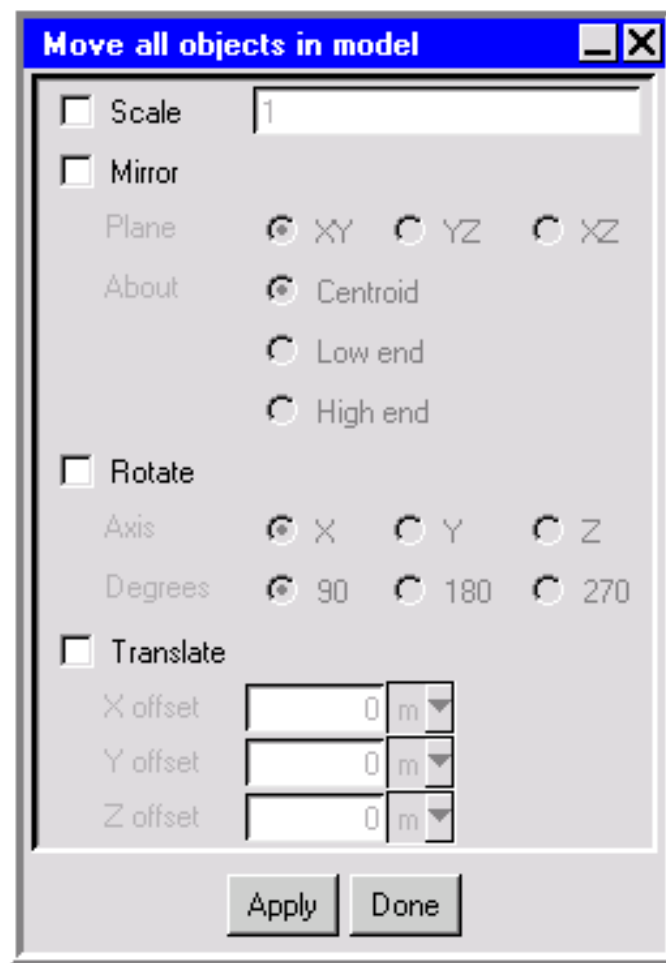




Figure 7.2.6: The Move all objects in model Panel

To scale the room, turn on the **Scale** option in the **Move all objects in model** panel. Specify the scaling factor by entering a value in the **Scale** text entry box. The scaling factor must be a real number greater than zero. Values greater than 1 will increase the size, while values less than 1 will decrease the size. To scale the room by different amounts in different directions, enter the scaling factors separated by spaces. For example, if you enter 1.5 2 3 in the **Scale** text entry box, **Airpak** will scale the room by a factor of 1.5 in the *x* direction, 2 in the *y* direction, and 3 in the *z* direction. Click **Apply** to scale the room. Click **Done** to close the **Move all objects in model** panel.

 If your model contains objects inside the room, these objects will be scaled also.



- Click on **Autoscale** in the room **Edit** window (Figure 7.2.1) to resize the room so that it is exactly the size required to fit all the objects in the model. This option can be used at any stage during the model-building process.

 Airpak does *not* automatically resize the room to fit the modeling objects. You need to click **Autoscale** to resize it.

7.2.2 Repositioning the Room

You can reposition the room in several different ways:

- Select or specify a local coordinate system for the room in the **Room** panel (Figure 7.2.3).

 Model →  Room

By default, the global coordinate system is used, which has an origin of (0, 0, 0). To use a local coordinate system for the room, select a local coordinate system from the **Local coord system** drop-down list. You can also create a new local coordinate system, edit an existing local coordinate system, and view the definition of a local coordinate system, as described in Section 7.3.7.

- You can reposition the room along the x , y , and z axes by selecting **Start/end** in the drop-down list in the room **Edit** window (Figure 7.2.1) or in the **Specify by** drop-down list in the **Room** panel (Figure 7.2.3) and modifying the starting point (x_S , y_S , z_S) and ending point (x_E , y_E , z_E) of the room. Alternatively, you can modify the starting point (x_S , y_S , z_S) and the lengths of the sides (x_L , y_L , and z_L) of the room if you select **Start/length** in the **Specify by** drop-down list in the **Room** panel or in the drop-down list in the room **Edit** window.
- Hold down the <Shift> key on the keyboard, use the middle mouse button to click on the room, and then drag the room to its new location. Options available for moving the room interactively using the mouse are available in the **Interaction** section of the **Preferences** panel (Figure 7.2.7) and are described below.

Edit → Preferences →  Interaction

X, Y, Z allow you to select which combination of the three axes the room can be translated along. For example, if you want to translate the room only along the x direction, select X and deselect Y and Z next to **Motion allowed in direction**.

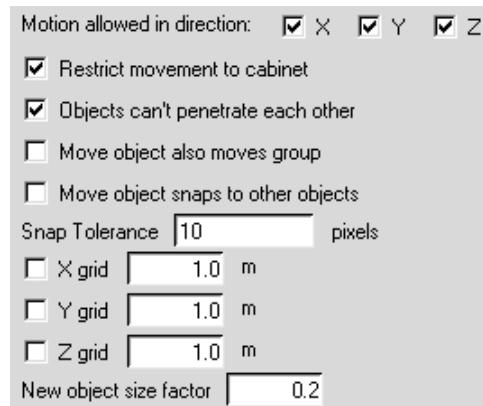



Figure 7.2.7: The Interaction Section of the Preferences Panel

X grid, Y grid, Z grid allow you to use a *grid-snap* technique to position the room at specified discrete distances along each axis. You can set the size of the grid independently along each axis. To use grid-snap along with the mouse to reposition the room, select X grid, Y grid, and/or Z grid and then enter values for X grid, Y grid, and/or Z grid in the relevant text entry fields. Airpak sets the values for the grid-snap distances to 1 by default, and uses the default length units for your model. If none of the grid-snap options are selected in the **Interactive editing** panel, the movement of the room using the mouse will be continuous.

To apply the chosen options to the current project only, click **This project**. To apply the chosen options to all projects, click **All projects**.

- Move the room using the **Move all objects in model** panel (Figure 7.2.6). To open this panel, right-click on **Room** under the **Model** node in the **Model manager** window and select **Move object** from the pull-down menu. Alternatively, you can select the room and click on the  button in the **Object modification** toolbar.

 Model →  Room → Move object

If multiple geometric transformations are selected, Airpak applies them in the order that they appear in the panel. For example, if both the **Rotate** and **Translate** options are selected, the room and objects are rotated first and then translated. Note that not all combinations of transformations are commutative; i.e., the result may be order-dependent, particularly if reflection is used.

Options available for moving objects using the **Move all objects in model** panel include the following:

Mirror allows you to obtain the mirror image of the room and all objects within the room. To mirror the room, turn on the **Mirror** option and specify the

Plane across which to reflect the room by selecting XY, YZ, or XZ. You can also specify the location about which the room is to be flipped by selecting **Centroid**, **Low end**, or **High end** next to **About**.

Rotate allows you to rotate the room and all objects within the room. You can rotate the room about any coordinate axis. Select X, Y, or Z next to **Axis**, and then select 90, 180, or 270 degrees of rotation.

Translate allows you to translate the room and all objects within the room. To translate the room, turn on **Translate** and define the distance of the translation from the current origin by specifying an offset in each of the coordinate directions: X offset, Y offset, and Z offset.

- Snap the room (and other objects) to a grid using the **Snap to grid** panel (Figure 7.2.8). To open this panel, select the room in the graphics window and then select **Snap to grid** in the **Model** menu.

Model→Snap to grid

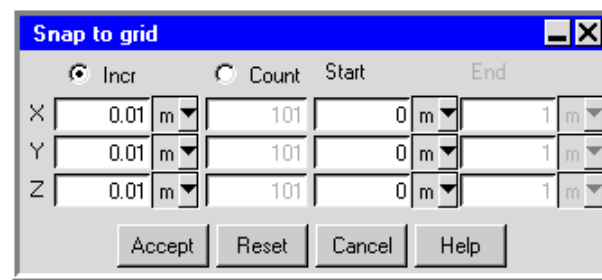


Figure 7.2.8: The Snap to grid Panel

Options available for snapping objects to a grid using the **Snap to grid** panel include the following:

Incr allows you to specify the increment unit and distance of the grid in each axis direction.

Count allows you to specify the number of grid points for in a specified range in each axis direction.

Start allows you to specify the coordinates of the origin of the grid. This option is available for both **Incr** and **Count**.

End allows you to specify the ending coordinates of the grid if you have selected the **Count** option.

After you click **Accept** the room (and all future objects) will be repositioned (or created) at the grid point nearest to the selection point rather than at the selection point itself.

i It is no longer necessary to snap all objects to a grid to eliminate small gaps in the mesh. That operation is automatically performed when you create a mesh, when the gaps are fixed in the mesh but the model is not changed. See Chapter 26 for more information about creating a mesh.

7.2.3 Changing the Walls of the Room

To give more complex physical properties to the room, **Airpak** allows you to change the definition of the room walls. By default, the walls of the room have no thickness and have zero velocity and heat flux boundary conditions.

For each side of the room (**Min x**, **Max x**, **Min y**, **Max y**, **Min z**, and **Max z**), you can specify how the wall is defined. To change the room walls, select a new option from the **Wall type** drop-down list in the **Properties** tab of the **Room** panel (Figure 7.2.4). The following options are available:

Default defines the specified room wall as an impermeable adiabatic boundary. This option is selected by default.

Wall defines the specified room wall as a wall object.

Opening defines the specified room wall as an opening object.

Vent defines the specified room wall as a vent object.

For each **Wall**, **Opening**, or **Vent** that you select, a new object will be added under the **Model** node in the **Model manager** window. To edit the properties of these objects, click the appropriate **Edit** button under **Properties** to open the appropriate **Object** panel. See Section 7.3.4 for more information about editing objects. (See Chapters 13, 11, and 10, respectively for details about wall, opening, and vent objects.)

7.2.4 Changing the Name of the Room

Airpak allows you to change the name of the room. The name of the room is displayed in the **Name** text entry box in the **Room** panel (Figure 7.2.2).



Model →



Room

You can change the name of the room by entering a new name in the **Name** text entry field. The default name is **room.1**.



There should never be more than one room in your **Airpak** model. Your **Airpak** model should always contain one (and only one) room.

Adding Notes About the Room

You can enter notes for room under **Notes for this object** in the **Notes** tab of the **Room** panel (Figure 7.2.5). There is no restriction on the number and type of text characters you can use. When you are finished entering or updating the text in this field, click **Update** to store this information along with the room object.

7.2.5 Modifying the Graphical Style of the Room

Airpak allows you to change the display of the room in the graphics window. You can change the color and line width of the room as described below.

Changing the Color

The **Color** of the room in the graphics window is shown as **default** in the **Room** panel (Figure 7.2.2). The color that will be applied to the room when **default** is selected is defined in the **Object types** section of the **Preferences** panel (see Section 6.3.6 for more details on changing the default color using this panel). To change the color of the room, select the **selected** option next to **Color** and click on the square button to the right of the **Color** text field. A color palette menu will open. You can select the new color in the color palette menu. To select a different color, click on the icon in the lower right corner of the palette menu. Select a new color using the method or panel that is appropriate to your system.

Changing the Line Width

The **Linewidth** of the room in the graphics window is shown as **default** in the **Room** panel (Figure 7.2.2). The width of the line used to display the room when **default** is selected is defined in the **Graphical styles** panel (see Section 6.3.6 for more details on changing the default line width using the **Graphical styles** panel). To change the width of the line for the room, select one of the options in the drop-down list next to **Linewidth** (**default**, 1, 2, 3, 4, or 5). Click **Update** at the bottom of the **Room** panel to change the width of the line for the room in the graphics window.


7.3 Configuring Objects Within the Room

The options for configuring objects within the room are described in detail in the following sections.

- Section 7.3.1: Overview of the Object Panels and Object Edit Windows
- Section 7.3.2: Creating a New Object
- Section 7.3.3: Selecting and Unselecting an Object

- Section 7.3.4: Editing an Object
- Section 7.3.5: Deleting an Object
- Section 7.3.6: Resizing an Object
- Section 7.3.7: Repositioning an Object
- Section 7.3.8: Aligning an Object With Another Object in the Model
- Section 7.3.9: Copying an Object

7.3.1 Overview of the Object Panels and Object Edit Windows

When you click on an object button in the Object creation toolbar, the Edit window in the lower right corner of the screen becomes the object Edit window for the type of object selected. For example, if you click on the  button in the Object creation toolbar, then the Edit window becomes the block Edit window. The object Edit window is similar for all types of objects, and is divided into sections for name and group information and relevant geometric information. Figure 7.3.1 shows an example of an object Edit window specific to blocks (the block Edit window).

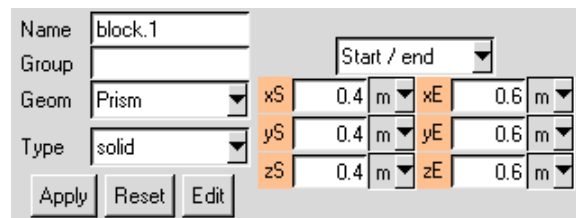



Figure 7.3.1: Example of an Object Edit Window

The object Edit window works in conjunction with the Object panel that opens at the bottom right of the screen when you click on the  button in the Object modification toolbar. Note that you can also open an Object panel by double-clicking on the name of the object under the Model node in the Model manager window or by clicking the Edit button in the object Edit window.

The Object panel allows you to specify physical characteristics and properties not available in the object Edit panel. In general, the Object panel is divided into four parts that are similar for all types of objects: description, geometry specification, and physical properties, and object notes. Figures 7.3.2–7.3.5 show examples of an Object panel specific to blocks (the Blocks panel).

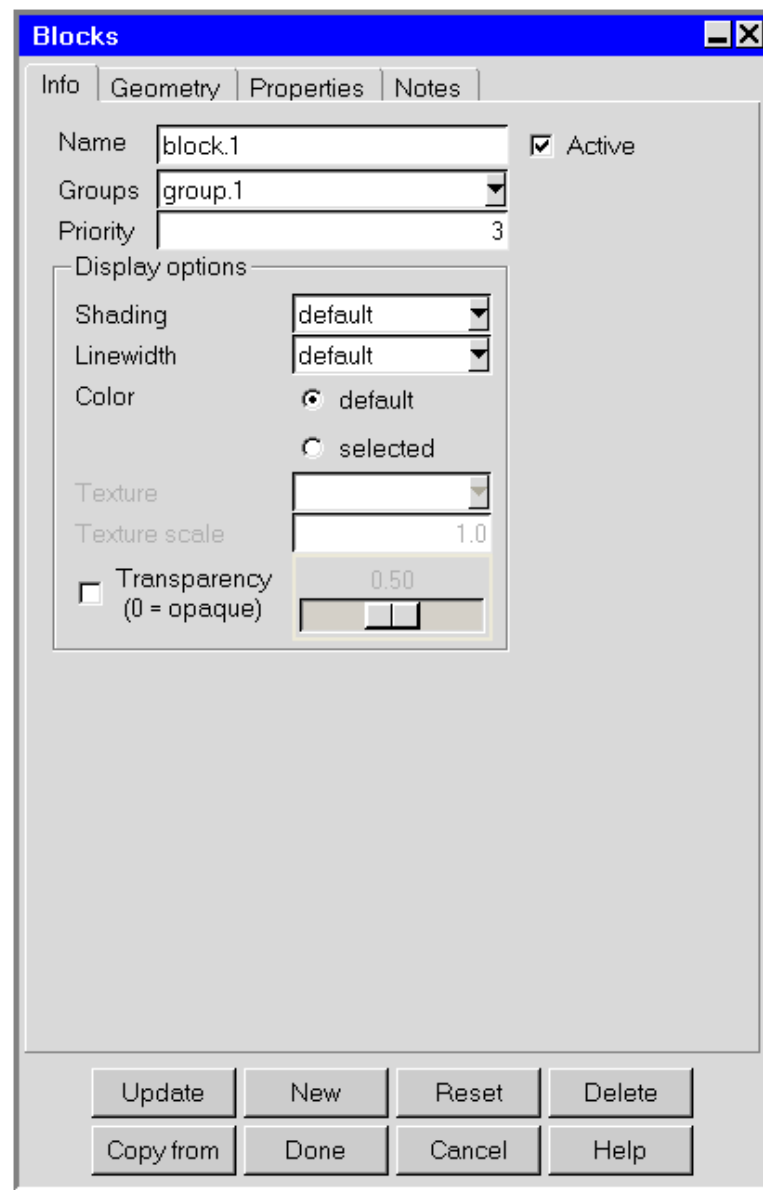


Figure 7.3.2: Example of an Object Panel (Info Tab)

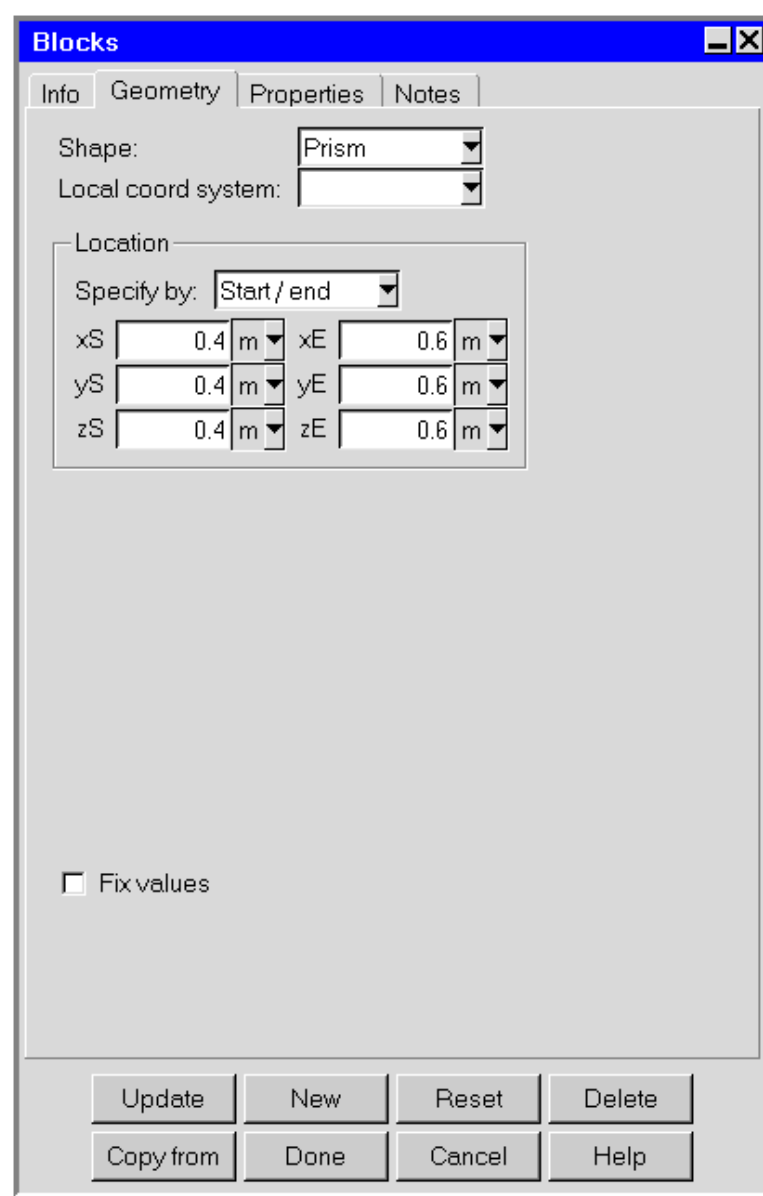


Figure 7.3.3: Example of an Object Panel (Geometry Tab)

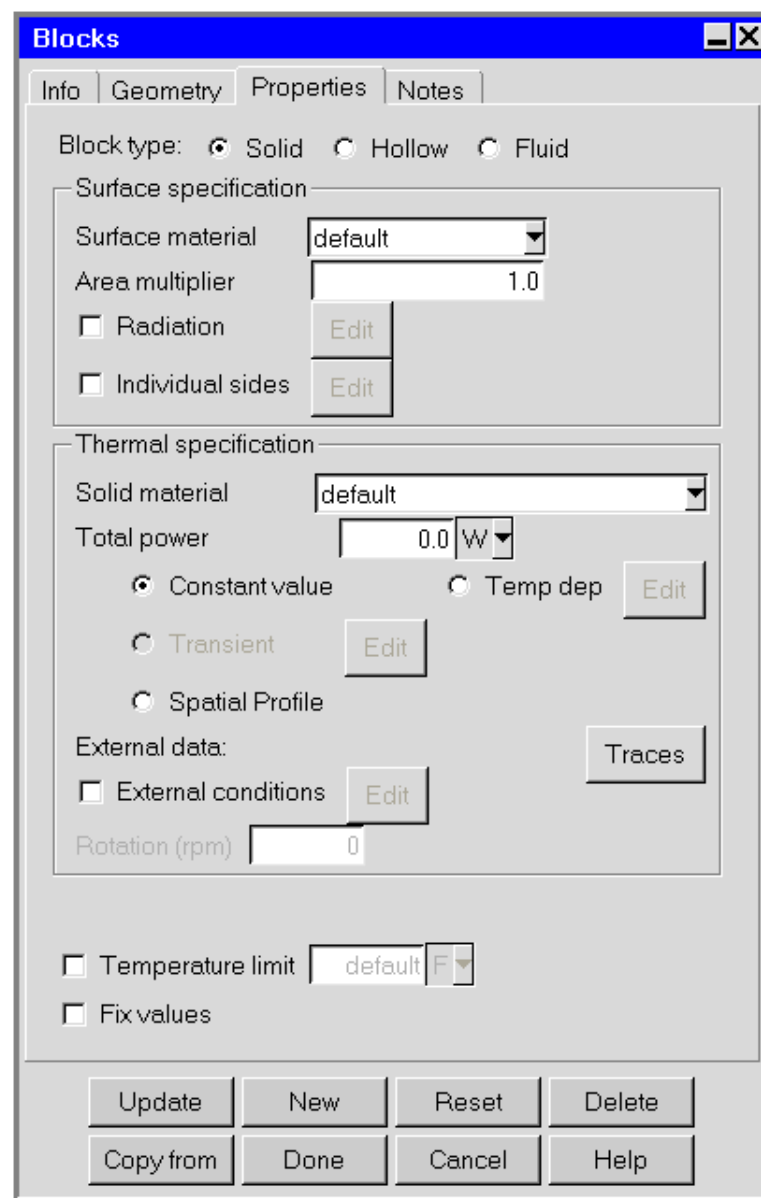


Figure 7.3.4: Example of an Object Panel (Properties Tab)

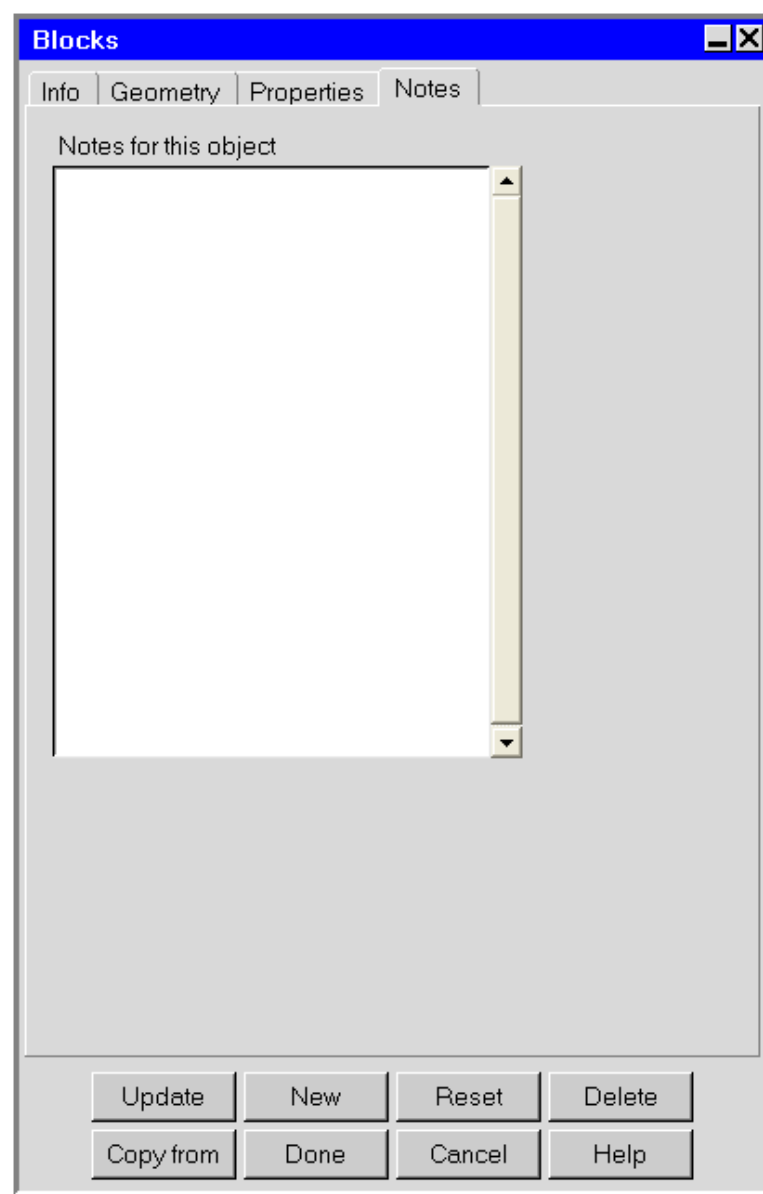


Figure 7.3.5: Example of an Object Panel (Notes Tab)

7.3.2 Creating a New Object

To create a new object, click on the appropriate button in the **Object creation toolbar** (Figure 7.1.1). You can also right-click on the **Model** node in the **Model manager** window, select **Create**, and then the object type from the subsequent pull-down menus. For example:



Model → Create → Block

A new object will be created with the default name **object.n**, where **n** is the next sequential number among numbered objects of the same type. The name of the new object will appear in the list of existing objects under the **Model** node in the **Model manager** window and in the **Name** text entry box in the **Object** panel and the object **Edit** window. You can rename an object by entering a new name in the **Name** text entry field.

Alternatively, you can create a new object by depressing the appropriate object button with the left mouse button, dragging the pointer to the desired location in the graphics window, and releasing the left mouse button. See Section 7.3.7 for details about repositioning objects once they have been created.

7.3.3 Selecting and Deselecting an Object


There are two ways to select an object:

- Select the name of the object in the object list under the **Model** node in the **Model manager** window using the left mouse button.
- Position the mouse cursor over the object in the graphics window, hold down the <Shift> key on the keyboard, and click the left mouse button.


The object will become highlighted in the object list and the characteristics of the object will be displayed in the object **Edit** window (e.g., Figure 7.3.1) and the **Object** panel (e.g., Figure 7.3.2).

To deselect an object that is currently selected, click on another item under the **Model** node in the **Model manager** window. When you select a new object, the previously selected object is automatically unselected.

7.3.4 Editing an Object

To edit an object, select the object and click on the  button in the **Object modification toolbar** (Figure 7.1.2). This opens the **Object** panel (e.g., Figure 7.3.2). The **Object** panel is specific to the type of object being configured and contains more functionality than the object **Edit** window. The **Object** panel allows you to specify physical characteristics and properties not available in the object **Edit** window.




An object must be selected before you click on the  button. See Section 7.3.3 for details on selecting an object.

Alternate ways of opening the **Object** panel after selecting an object are as follows:

- Click the **Edit** button in the object **Edit** window.
- In the **Edit** menu, select **Current object** and then **Edit** from the subsequent pull-down menus (**Edit**→**Current object**→**Edit**).
- Double-click on the object name in the object list under the **Model** node in the **Model manager** window.
- Right-click on the object name under the **Model** node and select **Edit object** from the pull-down menu.

7.3.5 Deleting an Object

To delete an object, select the object and click on the  button in the **Object modification** toolbar (Figure 7.1.2) or click **Delete** in the **Object** panel. The selected object will be permanently removed from the model and from the list of objects under the **Model** node in the **Model manager** window.

You can recover a deleted object immediately after the delete operation by selecting **Undo** in the **Edit** menu. See Section 2.1.2 for more details on using undo and redo operations.

To remove an object from the model only temporarily, you can deactivate it. See Section 7.4.1 for details.

Alternate ways of deleting a selected object are as follows:

- Select the object and press the <Delete> key on the keyboard.
- In the **Edit** menu, select **Current object** and then **Delete** from the subsequent pull-down menus (**Edit**→**Current object**→**Delete**).
- Right-click on the object name under the **Model** node and select **Delete** from the pull-down menu.

7.3.6 Resizing an Object

You can resize an object in **Airpak** in much the same way as you resize a room. See Section 7.2.1 for details on resizing a room. You should note the following differences between resizing an object and resizing a room:

- If you right-click on an object in the list under the **Model** node and select **Move object** in the pull-down menu, **Airpak** will open a **Move object** panel and not the **Move all objects in model** panel that is used for the room.
- The **Autoscale** option is not available for objects.

7.3.7 Repositioning an Object

You can reposition an object in **Airpak** in much the same way as you reposition a room. See Section 7.2.2 for details on repositioning a room. Note that you can also reposition an object (and the room) using a local coordinate system as described on page 7-28.

You should note the following differences between repositioning an object and repositioning a room:

- To modify the plane of the currently selected object (if relevant), click the **Plane** text entry field in the object **Edit** panel (e.g., Figure 7.3.1) using the left mouse button to open a list of available planes (**yz**, **xz**, and **xy**), and select a new plane from the list. The plane of the currently selected object can also be specified in the **Object** panel (e.g., Figure 7.3.3) using the **Plane** drop-down list, where you can choose **Y-Z**, **X-Z**, or **X-Y**.
- There are several options related to moving an object using the mouse that are available for objects but not for the room. These options are displayed in the **Interaction** section of the **Preferences** panel (Figure 7.2.7) and are listed below:

X, Y, Z allow you to select which combination of the three axes along which the object can be translated. For example, if you are positioning an opening on a wall, you would want to limit the motion of the opening to only two coordinate directions in order to move the opening across the surface of the wall.

Restrict movement to room allows you to restrict the motion of the object to within the room boundaries. If **In room** is not selected, **Airpak** will allow the object to project beyond the room boundaries. This option is on by default for all objects.

Objects can't penetrate each other instructs **Airpak** not to allow any penetration of the object by another object. This option is on by default for all objects.

Move object also moves group allows you to move entire groups using the <Shift> key and the middle mouse button.

Move object snaps to other objects allows you to snap objects to other objects using the <Shift> key and the middle mouse button.


Snap Tolerance specifies the distance, in pixels, within which a moved object will be snapped to another object. When this value has been set, and the model is oriented in one of the **X**, **Y**, or **Z** views, a dragged object will automatically

snap into alignment with a second object of similar shape when it comes within the specified number of pixels of a vertex, line, or plane of the second object.

New object size factor specifies the default size of a newly-created object in terms of the size of the room. For example, in a 1 m × 1 m × 1 m room, a value of 0.2 specifies that new objects will have side lengths of 0.2 m.



This value does not apply to package objects.

- If you click on the  button after selecting an object, **Airpak** will open a **Move object** panel and not the **Move all objects in model** panel that is used for the room. You can also open the **Move object** panel by right-clicking on an object in the list under the **Model** node and selecting **Move object** in the pull-down menu.

If multiple geometric transformations are selected, **Airpak** applies them in the order that they appear in the panel. For example, if both the **Rotate** and **Translate** options are selected, the new object is rotated first and then translated. Note that not all combinations of transformations are commutative; i.e., the result may be order-dependent, particularly if reflection is used.

If you specify a transformation that moves an object outside the room, **Airpak** opens the **Objects outside** panel (Figure 7.3.6), which contains the following options:

Allow out instructs **Airpak** to let the object remain outside the room boundary.

The object can be either completely outside the room or partly outside the room, as shown in Figure 7.3.7. This option can be used if the room you have created is too small and you want to resize the room at a later time.

Move instructs **Airpak** to move the object back inside the room. The object is moved back inside the room as shown in Figure 7.3.8, where the dashed line shows the position of the object before the move, and the solid line shows the position of the object after the move.

Resize instructs **Airpak** to resize the object so that it is inside the room. **Airpak** will change the object so that only the part of the object that is inside the room after the transformation remains in the model, as shown in Figure 7.3.9. The part that is outside will be discarded.

Resize room instructs **Airpak** to resize the room so that the object is inside the room, as shown in Figure 7.3.10.

Cancel instructs **Airpak** to cancel the move operation that caused the object to either exceed the room dimension or be placed outside the room.

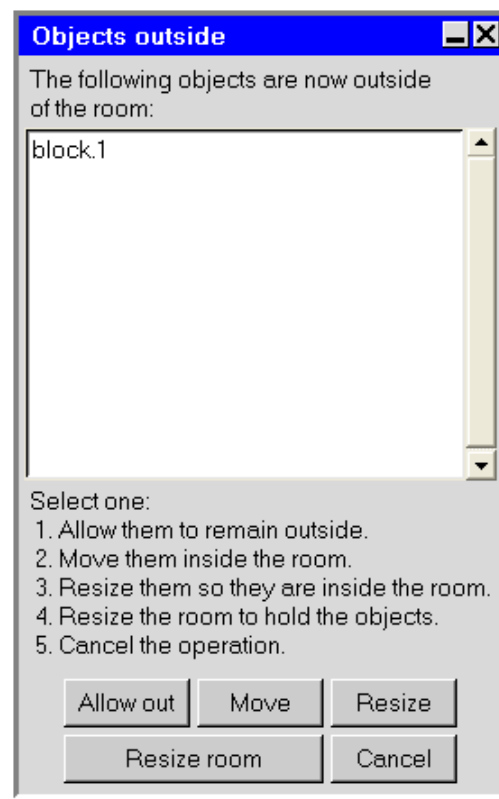


Figure 7.3.6: The Objects outside Panel

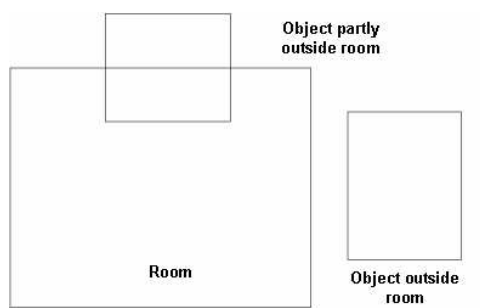


Figure 7.3.7: Object Outside Room Boundary

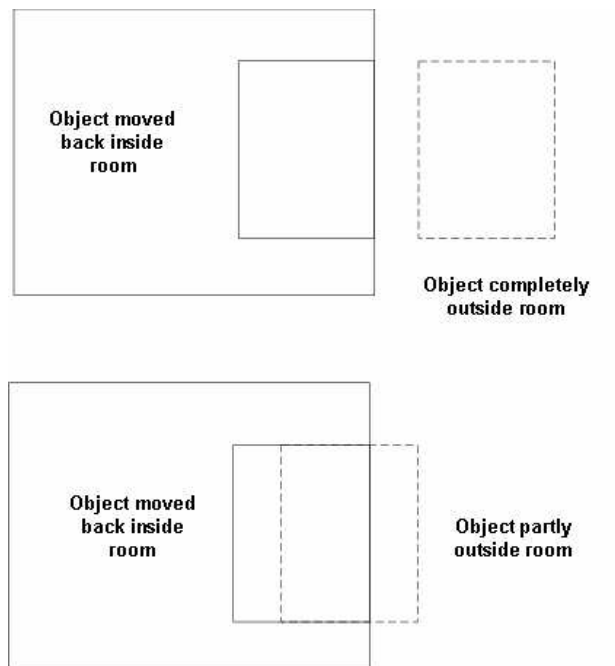


Figure 7.3.8: Moving the Object Back Inside the Room

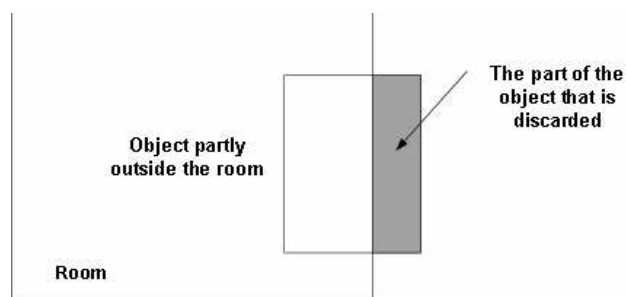


Figure 7.3.9: Resizing an Object that is Outside the Room

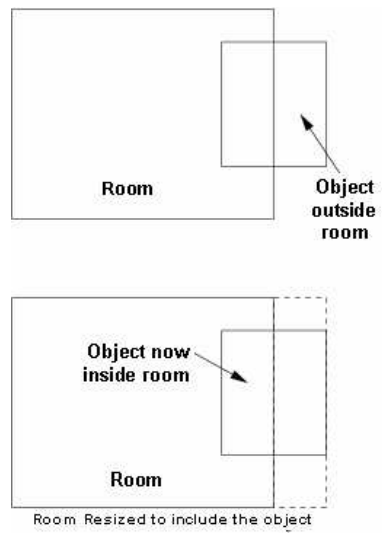


Figure 7.3.10: Resizing the Room to Include Outside Object

Local Coordinate Systems

The global coordinate system has an origin of (0, 0, 0) in **Airpak**. You can create local coordinate systems that can be used in your model. The origins of the local coordinate systems are specified with an offset from the origin of the global coordinate system.

The **Local coord systems** panel can be used to view and manage all local coordinate systems in your **Airpak** model. To open the **Local coord systems** panel (Figure 7.3.11), double-click on the **Local coords** item under the **Problem setup** node in the **Model manager** window.

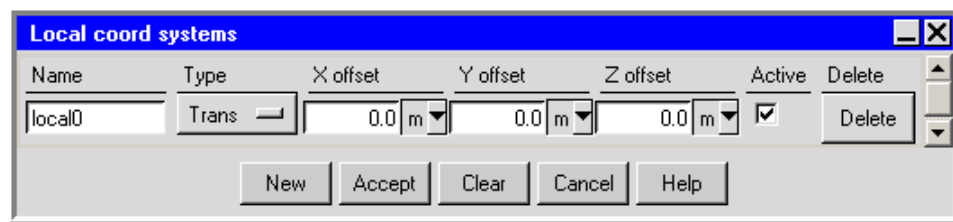
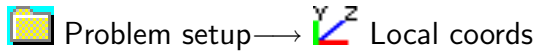


Figure 7.3.11: The Local coord systems Panel

The **Local coord systems** panel can be used to create new local coordinate systems, edit existing local coordinate systems, and delete or deactivate local coordinate systems. These operations are described below.

Each **Object** panel (e.g., Figure 7.3.3), and also the **Room** panel (Figure 7.2.3), contains a **Local coord system** drop-down list. To select a local coordinate system for an object (or for the room), open the **Local coord system** list and select a local coordinate system from the list. If the **Local coord system** field is empty, the global coordinate system will be used for the object (or room).

The **Local coord system** list can also be used to create a new local coordinate system, edit an existing local coordinate system, and view the definition of the selected local coordinate system. These operations are described below.

Creating a New Local Coordinate System

To create a new local coordinate system, click on **New** in the **Local coord systems** panel (Figure 7.3.11). A new local coordinate system will be created. Rename the local coordinate system, if desired. Keep the default **Type** (**Trans**), and specify the origin of the local coordinate system by entering values for the **X offset**, **Y offset**, and **Z offset** from the origin of the global coordinate system, which is (0, 0, 0). Repeat these steps to create any other local coordinate systems, and then click **Accept**.

You can also create a new local coordinate system in any Object panel (or the Room panel). To create a new local coordinate system, open the Local coord system drop-down list and select **Create new**. This will open the Local coords panel (Figure 7.3.12). Enter a name in the Name field for the new local coordinate system. Keep the default Type (Trans), and specify the origin of the local coordinate system by entering values for the X offset, Y offset, and Z offset from the origin of the global coordinate system, which is (0, 0, 0). Click **Accept** when you have finished creating the local coordinate system, and Airpak will close the Local coords panel and return to the Object panel (or the Room panel).

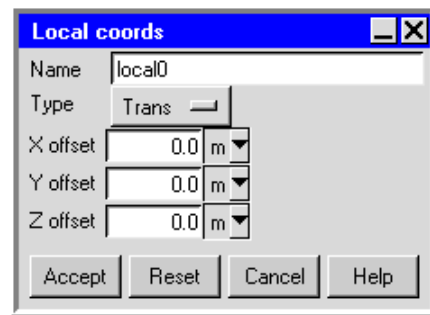


Figure 7.3.12: The Local coords Panel

Editing an Existing Local Coordinate System

You can use the Local coord systems panel (Figure 7.3.11) to rename an existing local coordinate system. You can also change the origin of a local coordinate system by entering new values for the X offset, Y offset, and Z offset from the origin of the global coordinate system, which is (0, 0, 0). Click **Accept** when you have finished editing the local coordinate systems.

You can also edit the definition of a local coordinate system in any Object panel (or in the Room panel). To edit the properties of the local coordinate system selected in the Local coord system list, open the list and select **Edit definition**. This will open the Local coords panel (Figure 7.3.12) and display the properties of the selected coordinate system. You can edit the definition of the coordinate system using the Local coords panel. Click **Accept** when you have finished editing the coordinate system, and Airpak will close the Local coords panel and return to the Object panel (or the Room panel).



Note that, if you have used a local coordinate system for an object (or the room) and you then edit the origin of the local coordinate system, the position of the object will not change in the graphics window, because the object's position with respect to the global coordinate system is still the same. Because the object is displayed with respect to the global coordinate system in the graphics window,

only its local coordinate values will be adjusted to account for its display location. For example, if the offset values for a local coordinate system were (0.1, 0, 0), then the coordinate location of any object using that local coordinate system would be adjusted by -0.1 units in the x direction so that the object is displayed in the same location in the graphics window. If the offset values are changed to be (0.2, 0, 0), then the coordinate location of the object(s) would be adjusted by a total of -0.2 units in the x direction, but the position of the object in the graphics window (with respect to the global coordinate system) will still remain the same.

Viewing the Definition of a Local Coordinate System

You can view the definition of a local coordinate system using any **Object** panel (or the **Room** panel). To view the properties of the local coordinate system selected in the **Local coord system** list, open the list and select **View definition**. The **Message** window will report the definition of the coordinate system.

Deleting Local Coordinate Systems

If there are local coordinate systems in the **Local coord systems** panel (Figure 7.3.11) that you no longer need, you can easily delete them. Click on the **Delete** button on the line of the coordinate system that you want to delete. The coordinate system will be permanently removed from your model.

To remove all local coordinate systems from your **Airpak** model, click **Clear** in the **Local coord systems** panel.

Note that, if you delete a local coordinate system that is being used by an object (or the room), the object (or room) will remain in the same position in the graphics window. **Airpak** will add the coordinates of the origin of the local coordinate system to the coordinates of the object (or room) and update the coordinates in the **Object** panel (or the **Room** panel) and the object **Edit** window (or the room **Edit** window).

Activating and Deactivating Local Coordinate Systems

By default, all local coordinate systems that you create will be available in your current **Airpak** model. You can remove a local coordinate system from your model temporarily by turning off the **Active** option for the coordinate system in the **Local coord systems** panel (Figure 7.3.11). You can repeat this for each local coordinate system that you want to temporarily remove from your model. When a local coordinate system is deactivated, it is simply removed from the **Local coord system** drop-down lists, *not* deleted from **Airpak**. Since it still exists, you can easily add it to the **Local coord system** drop-down lists again by turning the **Active** option back on.

Note that, if you deactivate a local coordinate system that is currently being used by an object (or the room), **Airpak** will still use the local coordinate system for the object

(or room). Deactivating a local coordinate system only removes it from the Local coord system lists so that it cannot be selected.

7.3.8 Aligning an Object With Another Object in the Model

You can align an object with another object in your Airpak model using the Edit window or the Object modification toolbar.

Aligning Objects With the Object Edit Window

Figure 7.3.13 shows an example of two objects that are close together. To align the right-hand side of `object.1` with the left-hand side of `object.2` using the object Edit window, follow the steps below.

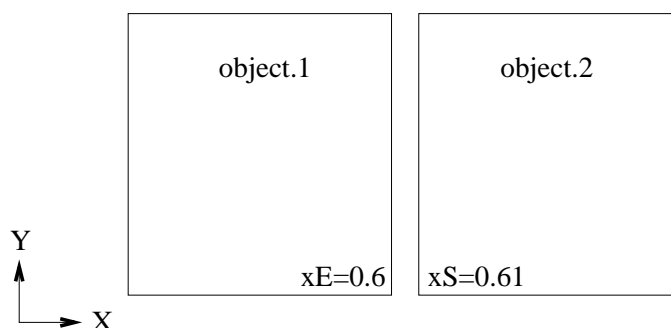


Figure 7.3.13: Two Objects That Are Close Together


1. Select the object you want to align (`object.1` in this example).
2. In the object **Edit** window, click on the coordinate that you want to change. The coordinates are displayed in orange in the object **Edit** window. For example, if you wanted to change the x_E coordinate of `object.1`, you would click x_E in the object **Edit** panel.
3. Use the left mouse button to click on the part of the object that you want to align your object with in the graphics window. In this example, you would click on the left-hand side of `object.2`.

Airpak will stretch `object.1` so that the right-hand side of `object.1` coincides with the left-hand side of `object.2`.

Aligning Objects With the Object modification Toolbar

You can align an object with another object in your Airpak model in several ways using the Object modification toolbar (Figure 7.1.2).

Aligning Object Faces



You can align two objects with faces that are in parallel planes using the  button in the **Object modification** toolbar. Alignment of one object with another using faces can be accomplished by two methods:

- resizing the selected object by stretching or contracting in the direction normal to the plane of its selected face, so that the selected face on the chosen object will be in the same plane as the selected face on the reference object
- translating the selected object in the direction normal to the plane of its selected face, so that the selected face on the chosen object will be in the same plane as the selected face on the reference object




The resizing option for aligning faces is available only for objects that have prism, cylindrical, or 3D polygon geometries. Objects with these geometries can then be resized only when they are being aligned to objects with similar geometry (e.g., prism objects cannot be resized to become aligned with cylindrical objects).


To align the faces of two objects, use the following procedure:

1. Decide whether you want to align the two desired objects by resizing or translating one of the objects.
 - To align objects by resizing one of the objects, left-click on the  button.
 - To align objects by translating one of the objects, right-click on the  button.
2. Use the left mouse button to select the face of the object that you want to change. If there are multiple faces joined together at the same edge or overlaid on the same plane, you can cycle through the faces by repeatedly clicking the left button until the desired face is selected. If you select the wrong face, click the right mouse button to deselect it.
3. When you have selected the desired face, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the face on the reference object with which you want the first object to be aligned.



Aligning Object Edges

You can align two objects with edges that run in the same direction using the  button in the **Object modification** toolbar. Aligning one object to another using edges can be accomplished by two methods for both 2D and 3D objects:

- 2D objects
 - resizing the selected object by stretching (or contracting) in the coordinate direction normal to the direction of its selected edge, so that the selected edge of the chosen object will be co-linear with the selected edge on the reference object
 - translating the selected object in the coordinate direction normal to the direction of its selected edge, so that the selected edge of the chosen object will be co-linear with the selected edge on the reference object
- 3D objects
 - resizing the selected object by stretching (or contracting) in up to two coordinate directions normal to the direction of its selected edge, so that the selected edge of the chosen object will be co-linear with the selected edge on the reference object
 - translating the selected object in up to two coordinate directions normal to the direction of its selected edge, so that the selected edge of the chosen object will be co-linear with the selected edge on the reference object

 Aligning two objects using edges is *not* available for objects with circular, cylindrical, ellipsoid, or elliptical cylinder geometry.

To align the edges of two objects, use the following procedure:

1. Decide whether you want to align the two desired objects by resizing or translating one of the objects.
 - To align objects by resizing one of the objects, left-click on the  button.
 - To align objects by translating one of the objects, right-click on the  button.
2. Use the left mouse button to select the edge of the object that you want to change. If there are multiple edges joined together at the same point or overlaid on the same line, you can cycle through the edges by repeatedly clicking the left button until the desired edge is selected. If you select the wrong edge, click the right mouse button to deselect it.

3. When you have selected the desired edge, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the edge on the reference object with which you want the first object to be aligned.

Figure 7.3.14 shows an example of aligning the edges of two objects using resizing and translation.

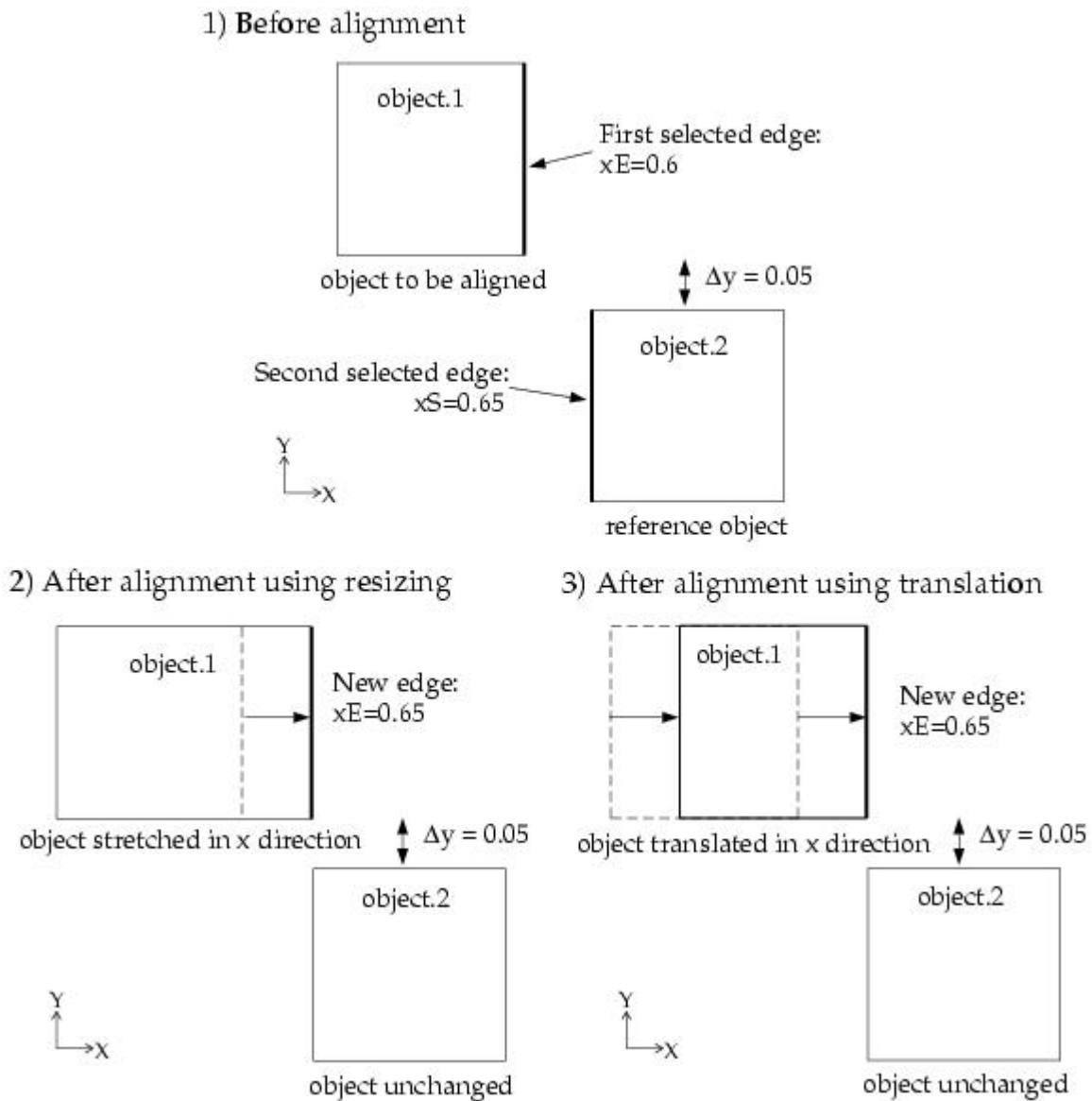



Figure 7.3.14: Alignment of Two Objects Using Edges

Aligning Object Vertices



You can align two objects by their individual vertices using the  button in the Object modification toolbar. Alignment of one object with another using vertices can be accomplished by two methods:

- resizing the selected object by stretching (or contracting) in up to three coordinate directions, so that the selected vertex of the chosen object will occupy the same point in the room space as the selected vertex on the reference object
- translating the selected object in up to three coordinate directions, so that the selected vertex of the chosen object will occupy the same point in the room space as the selected vertex on the reference object




Aligning two objects using vertices is *not* available for objects with circular or cylindrical geometry.

To align the vertices of two objects, use the following procedure:


1. Decide whether you want to align the two desired objects by resizing or translating one of the objects.
 - To align objects by resizing one of the objects, left-click on the  button.
 - To align objects by translating one of the objects, right-click on the  button.
2. Use the left mouse button to select the vertex of the object that you want to change. If there are multiple vertices occupying the same point, you can cycle through the vertices by repeatedly clicking the left button until the desired vertex is selected. If you select the wrong vertex, click the right mouse button to deselect it.
3. When you have selected the desired vertex, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the vertex on the reference object with which you want the first object to be aligned.

Aligning Object Centers


You can align two objects by their geometric centroids using the  button in the Object modification toolbar. Alignment of one object to another using centers occurs by translating the selected object in up to three coordinate directions so that the center


of the chosen object will occupy the same point in the room space as the center of the reference object.

To align the centers of two objects, use the following procedure:


1. Left-click on the  button.
2. Use the left mouse button to select the object that you want to change. If there are multiple objects in close proximity to each other, you can cycle through the objects by repeatedly clicking the left button until the desired object is selected. If you select the wrong object, click the right mouse button to deselect it.
3. When you have selected the desired object, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the reference object with which you want the first object to be aligned.

Aligning Object Face Centers


You can align two objects by the geometric centers of their faces using the  button in the **Object modification** toolbar. Alignment of one object to another using face centers occurs by translating the selected object in up to three coordinate directions so that the center of the face on the chosen object will occupy the same point in the room space as the center of the face on the reference object.

 Aligning two objects using face centers is *not* available for objects with ellipsoid geometry.


To align the face centers of two objects, use the following procedure:


1. Left-click on the  button.
2. Use the left mouse button to select the face on the object that you want to change. If there are multiple faces joined together at the same edge or overlaid on the same plane, you can cycle through the faces by repeatedly clicking the left button until the desired face is selected. If you select the wrong face, click the right mouse button to deselect it.
3. When you have selected the desired face, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the reference object with which you want the first object to be aligned.

Matching Object Faces


You can align two objects by matching their faces using the  button in the **Object modification** toolbar. Alignment of one object to another by matching faces occurs as necessary by some or all of the following methods:

- resizing the selected object by stretching (or contracting) in one or both of the coordinate directions of the plane of the selected face
- translating the selected object in up to three coordinate directions
- rotating the selected object so that the plane of the face on the selected object is the same as the plane of the face on the reference object


 Matching object faces is possible only when the selected faces are being matched to other object faces with similar geometry. For example, a face on a prism object cannot be matched to a face on a cylindrical object, but a circular face, such as a 2D fan, can be matched to a face on a cylindrical object. Circular faces in different planes (e.g., x - y and y - z), however, can not be matched.

 Aligning two objects by matching faces is *not* available for objects with ellipsoid geometry.


To align two objects by matching faces, use the following procedure:


1. Left-click on the  button.
2. Use the left mouse button to select the face on the object that you want to change. If there are multiple faces joined together at the same edge or overlaid on the same plane, you can cycle through the faces by repeatedly clicking the left button until the desired face is selected. If you select the wrong face, click the right mouse button to deselect it.
3. When you have selected the desired face, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the reference object with which you want the first object to be aligned.

Matching Object Edges


You can align two objects by matching their edges using the  button in the **Object modification** toolbar. Alignment of one object to another by matching edges occurs as necessary by one or both of the following methods:

- resizing the selected object by stretching (or contracting) in the coordinate direction of the selected edge
- translating the selected object in up to three coordinate directions

 Matching object edges is possible only when the selected edges are in the same general plane (i.e., x - y , y - z , x - z , or inclined).

 Aligning two objects by matching edges is *not* available for objects with circular, cylindrical, ellipsoid, or elliptical cylinder geometry.


To align two objects by matching edges, use the following procedure:

1. Left-click on the  button.
2. Use the left mouse button to select the edge on the object that you want to change. If there are multiple edges joined together at the same point or overlaid on the same line, you can cycle through the edges by repeatedly clicking the left button until the desired edge is selected. If you select the wrong edge, click the right mouse button to deselect it.
3. When you have selected the desired edge, click the middle mouse button to accept the choice.
4. Repeat steps 2 and 3 for the reference object with which you want the first object to be aligned.

7.3.9 Copying an Object

There are two ways to copy an object in Airpak: using the Copy object panel and using the Object selection panel.

Copying an Object Using the Copy object Panel

One way to copy an object is to use the Copy object panel. Figure 7.3.15 shows an example of a Copy object panel specific to copying a block (the Copy block panel). To open this panel, select the object in the graphics window and click on the  button in the Object modification toolbar (Figure 7.1.2).

Alternate ways of opening the **Copy object** panel are as follows:

- In the **Edit** menu, select **Current object** and then **Copy** from the subsequent pull-down menus (**Edit**→**Current object**→**Copy**).
- Right-click on the object name under the **Model** node and select **Copy** from the pull-down menu.



If you copy an object such that the new object is created outside the room, **Airpak** opens the **Objects outside** panel, as described in Section 7.3.7.

When you use a **Copy object** panel, a new object with a sequentially numbered name appears in the list of existing objects under the **Model** node in the **Model manager** window, and the new object appears in the graphics window. Only the options that are selected in the **Copy object** panel affect the copy creation.

If multiple geometric transformations are selected, **Airpak** applies them in the order that they appear in the panel. For example, if both the **Rotate** and **Translate** options are selected, the new object is rotated first and then translated. Note that not all combinations of transformations are commutative; i.e., the result may be order-dependent, particularly if reflection is used. If you want to perform a transformation that requires an order of operations different from that provided by the **Copy object** panel, you can create an initial object and then perform additional transformations in the order desired using the **Move object** panel. See Section 7.3.7 for more information on moving an object.

Options available for copying objects using the **Copy object** panel include the following:

Number of copies allows you to specify how many copies of the selected object you require.

Group name allows you to specify a name for the group to which the new objects will belong. No group will be created if this option is not selected.

Scale allows you to increase or decrease the size of the new object relative to the original object. To scale the object, turn on the **Scale** option and specify the scaling factor by entering a value in the **Scale** text entry box. The scaling factor must be a real number greater than zero. Values greater than 1 will increase the size, while values less than 1 will decrease the size. To scale the object by different amounts in different directions, enter the scaling factors separated by spaces. For example, if you enter 1.5 2 3 in the **Scale** text entry box, **Airpak** will scale the object by a factor of 1.5 in the x direction, 2 in the y direction, and 3 in the z direction. To scale the size of multiple copies successively (i.e., to scale the first copy relative to the original object, the second copy relative to the first copy, etc.), select **Scale copies cumulatively**.

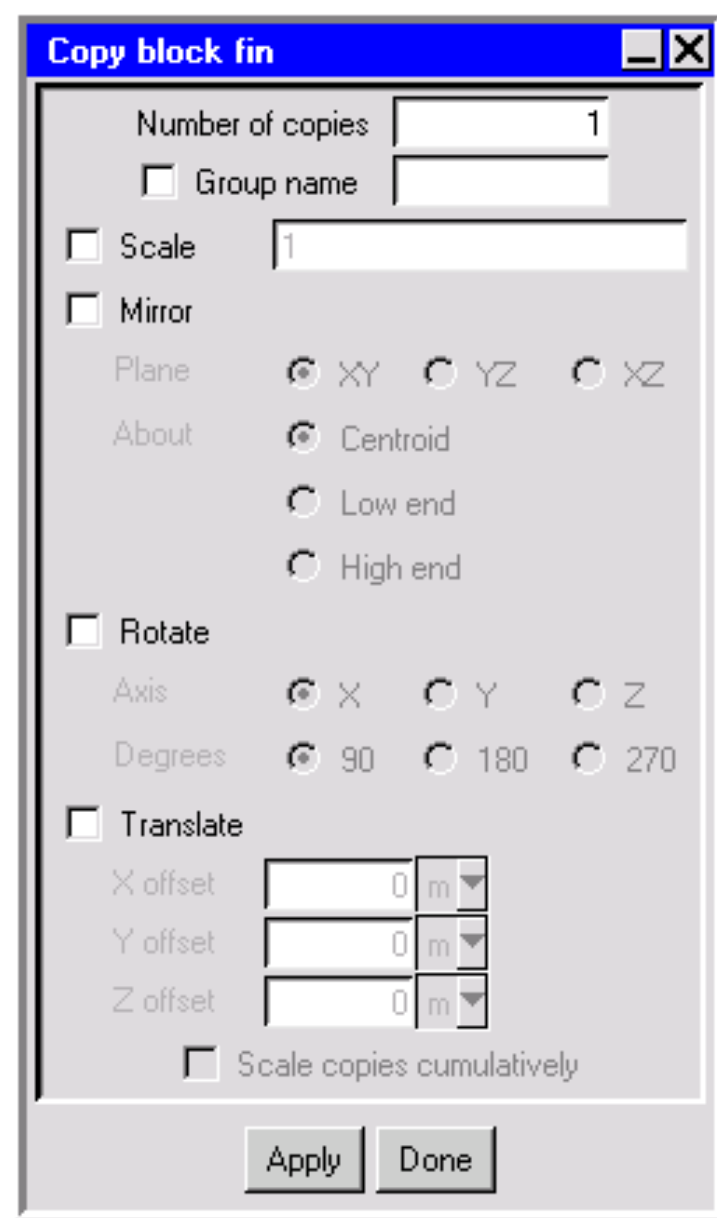


Figure 7.3.15: Example of a Copy object Panel

Mirror allows you to obtain the mirror image of the copied object. To mirror the object, turn on the **Mirror** option and specify the **Plane** across which to reflect the object by selecting **XY**, **YZ**, or **XZ**. You can also specify the location about which the object is to be flipped by selecting **Centroid**, **Low end**, or **High end** next to **About**.

Rotate allows you to rotate the new object about any coordinate axis from the object's original position. Select **X**, **Y**, or **Z** next to **Axis**, and then select **90**, **180**, or **270** degrees of rotation.

Translate allows you to translate the new object a specified distance from the original object's position. To translate the new object, turn on **Translate** and define the distance of the translation from the current object by specifying an offset in each of the coordinate directions: **X offset**, **Y offset**, and **Z offset**. If multiple copies are created, this translation is relative to the previous copy for each new object.

Copying an Object Using the Object selection Panel

You can also copy an object using the **Object selection** panel. Figure 7.3.16 shows an example of an **Object selection** panel. To open this panel, select **Current object** and then **Copy from** in the **Edit** menu (e.g., Figure 7.3.1) or click on **Copy from** in the **Object** panel (e.g., Figure 7.3.3).

The **Copy from** option is used when you want to transform an existing object of a specific type in a particular location into an object of another type in the same location. Consider, for example, that your model includes a block and a resistance and you want to transform the block into a 3D resistance. In the **Resistances** panel, click on the **Copy from** button. This opens the **Object selection** panel. In the **Copy from object** list, choose the block. When you click **Done**, **Airpak** will create a new object with the location and dimensions of the block and the physical properties of the resistance.

Other options available in the **Object selection** panel are as follows:

Deactivate other object deactivates the selected object in the **Copy from object** list.

This option is on by default. The object will be temporarily removed from the model, which renders the object inactive for purposes of the current analysis.

Delete other object deletes the selected object in the **Copy from object** list.

Neither does not delete or deactivate the selected object in the **Copy from object** list.

Copy shape info copies the shape of the object only.

Copy object info copies the shape and the parameters of the object.

Copy creation order ensures that the creation order of the new object is the same as that selected from the **Copy from object** list.

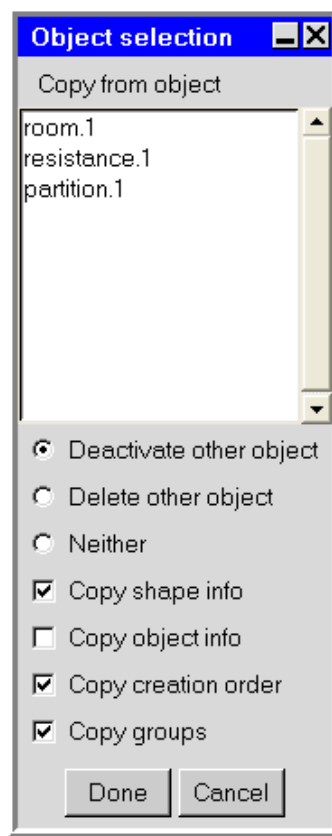


Figure 7.3.16: The Object selection Panel

Copy groups ensures that the new object is put into the same group as the selected object in the Copy from object list.

7.4 Object Attributes

When a new object is created in **Airpak**, default characteristics are applied to the object. **Airpak** gives the object the following characteristics, which are displayed in the object Edit window or the **Object** panel, or, in some cases, in both places.

- Description (name, notes, group, include or exclude the object)
- Graphical style (shading, line width, color, texture, transparency)
- Position and size
- Geometry
- Physical characteristics

The first four characteristics relate to all objects and are described below. The physical characteristics are specific to a particular object and are described in detail in the chapter related to that object.

7.4.1 Description

Airpak allows you to change the name of the object, specify the group to which it belongs, and include or exclude the object. These options are described below.

Changing the Name of an Object

Airpak allows you to change the name of an object. The name of an object is displayed in the **Name** text entry box in the object **Edit** window (e.g., Figure 7.3.1) and in the **Name** text entry box in the **Object** panel (e.g., Figure 7.3.2). To change the name of an object, enter a new name in the **Name** text entry box.

Adding Notes About the Object

You can enter notes for object under **Notes for this object** in the **Notes** tab of the **Room** panel (e.g., Figure 7.3.5). There is no restriction on the number and type of text characters you can use. When you are finished entering or updating the text in this field, click **Update** to store this information along with the object.

Assigning an Object to a Group

The group to which the object belongs is displayed in the **Group** text entry box in the object **Edit** window (e.g., Figure 7.3.1) and in the **Groups** text entry box in the **Object** panel (e.g., Figure 7.3.2). You can specify the group to which the object belongs (if applicable) by entering the name of the group in the **Group** text entry box.

Including or Excluding an Object

You can include an object in the model or exclude it by selecting or deselecting **Active** in the **Object** panel (e.g., Figure 7.3.2). The object is included by default. If the **Active** option is deselected, the object is temporarily removed from the model, which renders the object inactive for purposes of the current analysis.

7.4.2 Graphical Style

Airpak allows you to change the display of the object in the graphics window. The procedure for changing the color and line width of the object is the same as for changing the color and line width of the room, as described in Section 7.2.5.

Changing the Shading

The **Shading** of an object in the graphics window is shown as **default** in the **Object** panel (e.g., Figure 7.3.2). The type of shading that will be applied to the object when **default** is selected is defined in the **Object types** section of the **Preferences** panel (see Section 6.3.6 for more details on changing the default shading using the **Preferences** panel). To change the shading of the object, click on the square button to the right of the **Shading** text field in the **Object** panel. Select a shading type in the resulting drop-down list: **default**, **Wire**, **Solid**, **Solid/wire**, **Hidden line**, or **Invisible**. Click **Update** at the bottom of the **Object** panel to change the shading for the object in the graphics window.

Changing the Texture

The **Texture** of an object in the graphics window can be modified when **Solid** or **Solid/wire** has been selected in the **Shading** drop-down list. The default texture option is **No texture**. To modify the texture, select **Load from file** and select an available PPM image file in the resulting **File selection** dialog box (see Section 2.1.8).

Depending on its pixel size, the image, or a portion thereof, will be used to cover each side of the object. To change the size of the image used for the texture, specify a value for the **Texture scale**. Values greater than 1.0 will reduce the size of the image on each side of the object, resulting in a tiling effect at large enough values. Values less than 1.0 will increase the size of the image on each side of the object.

Changing the Transparency

The transparency of an object in the graphics window can be modified by turning on the **Transparency** option and moving the slider bar between values of 0.00 (opaque) and 0.99 (fully transparent). This option is useful when **Solid** has been selected in the **Shading** drop-down list.

7.4.3 Position and Size

An example of how **Airpak** displays the position and size of an object is described below for a rectangular object.

- The plane of the currently selected object is displayed in the object **Edit** window (e.g., Figure 7.3.1) next to **Plane** (if relevant). For 2D objects, **Plane** is the plane the object is in. (For 3D objects, **Plane** is not available.) To modify the plane, select a new plane in the drop-down list of available planes (**yz**, **xz**, and **xy**). The plane of the currently selected object can also be specified in the **Object** panel (e.g., Figure 7.3.3) by selecting **Y-Z**, **X-Z**, or **X-Y** in the **Plane** drop-down list.
- The starting and ending points of the selected object are displayed in the object **Edit** window (e.g., Figure 7.3.1) and in the **Object** panel (e.g., Figure 7.3.3) under

Location if **Start/end** is selected. For rectangular objects, the start and end points take the form (xS, yS, zS) and (xE, yE, zE). If **Start/length** is selected in the **Object** panel or the object **Edit** window, the starting point of the object (xS, yS, zS) and the lengths of the sides (xL, yL, and zL) are displayed in the **Object** panel and the object **Edit** window.

7.4.4 Geometry

Airpak displays the geometry of the currently selected object in the object **Edit** window (e.g., Figure 7.3.1) next to **Geom**. To modify the geometry, select a new geometry from the list of available geometries. The geometries available depend on the type of object selected. For example, the geometries available for a block are prism, cylinder, polygon, ellipsoid, and elliptical cylinder. The geometry of the currently selected object can also be specified in the **Object** panel (e.g., Figure 7.3.3) in the **Shape** drop-down list.

The geometries available in Airpak are listed below and described in the following sections.

- rectangular
- circular
- inclined
- polygon (2D and 3D)
- prism
- cylindrical
- ellipsoid
- elliptical cylinder
- CAD

Rectangular Objects

The location and dimension parameters for a rectangular object include the coordinate plane (Y-Z, X-Y, or X-Z) in which the object lies and its physical dimensions. A rectangular object is defined by the coordinates of its lower left and upper right corners (see Figure 7.4.1). These are referred to as the starting point (xS, yS, zS) and ending point (xE, yE, zE), respectively. The coordinate on the axis that is normal to the plane of the object is specified only for the starting point. For the ending point, this same value is used. For example, if the object is in the X-Z plane, you specify xS, yS, zS, xE, and zE, and Airpak automatically sets yE equal to yS.

Rectangular geometries are available for the following objects:

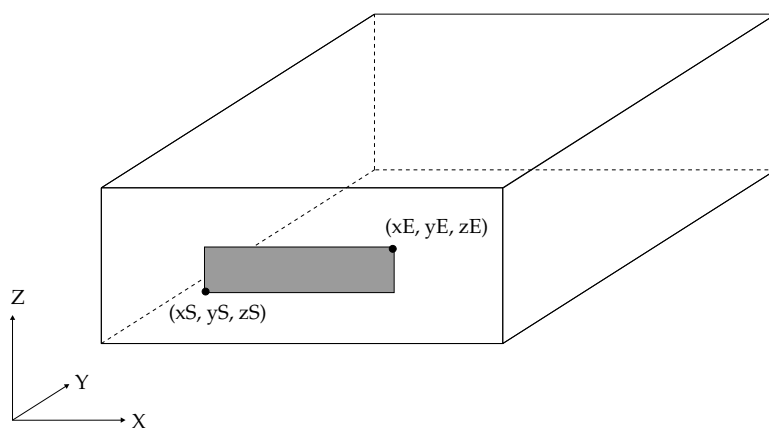


Figure 7.4.1: Rectangular Object Definition

- fans
- vents
- openings
- persons (faces)
- walls
- partitions
- sources
- resistances
- hoods (flanges, exhausts)

To specify a rectangular object, select **Rectangular** in the **Shape** drop-down list in the **Object** panel. The user inputs for a rectangular object are shown below.

Shape: Rectangular

Local coord system:

Plane: X-Y

Location

Specify by: Start / end

xS	0.0113773	m	xE	0.188623	m
yS	0.811377	m	yE	0.988623	m
zS	0.5	m	zE	0.5	m

Internal hub equiv. radius 0 m

Select the plane in which the object lies (Y-Z, X-Z, or X-Y) in the **Plane** drop-down list. Select **Start/end** in the **Specify by** drop down list and enter values for the start coordinates (x_S , y_S , z_S) and end coordinates (x_E , y_E , z_E) of the object, or select **Start/length** and enter values for the start coordinates (x_S , y_S , z_S) and lengths of the sides (x_L , y_L , and z_L) of the object.

Circular Objects

A circular object (see Figure 7.4.2) is defined by the coordinate location of its center (x_C , y_C , z_C), the plane (X-Y, Y-Z, or X-Z) in which the object lies, and its **Radius**.

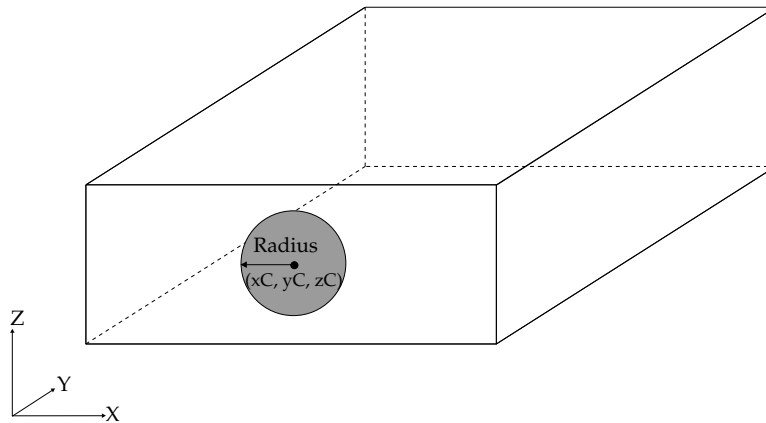


Figure 7.4.2: Circular Object Definition

Circular geometries are available for the following objects:

- fans
- vents
- openings
- walls
- partitions
- sources
- resistances

To specify a circular object, select **Circular** in the **Shape** drop-down list in the **Object** panel. The user inputs for a circular object are shown below.

Shape: Circular

Local coord system:

Plane: X-Y

Location

Center:

xC: 0.2 m

yC: 0.325 m

zC: 0.15 m

Radius: 0.0414593 m

Specify the location of the center of the object (x_C , y_C , z_C), the plane in which the object lies (Y-Z, X-Z, or X-Y), and the **Radius** of the object. For a circular fan, you can specify the size of the hub or inner radius (**IRadius**).

You can also define a circular object on the surface of another circular object. For example, you can create a circular fan on the circular face of a cylindrical block. Use the **Orient** menu or the **Orientation commands** toolbar to specify the desired orientation of the object. For the cylindrical block, you would choose the orientation such that the circular face of the block is in the plane of the graphics window. Then, for the fan, select **Circular** in the drop-down list next to **Geom** in the fan **Edit** window and click on **Select 3 points**. Using the left mouse button, select the first, second, and third points on the radius of the circular face of the cylindrical block in the graphics window. For a circular fan, you can also specify the size of the hub or inner radius by selecting an inner radius point in the graphics window using the left mouse button.

Inclined Objects

An inclined object has only two of its edges aligned with a coordinate axis (X, Y, or Z), and its physical dimensions are defined by the coordinates of a rectangular box that serves as its boundary (see Figure 7.4.3). The lower left corner and upper right corner of the box are referred to as the starting point (x_S , y_S , z_S) and ending point (x_E , y_E , z_E), respectively.

To complete the definition of an inclined object, you must specify the axis around which it is rotated and the orientation of the object (see Figure 7.4.4).

Inclined geometries are available for the following objects:

- fans
- vents
- openings
- walls

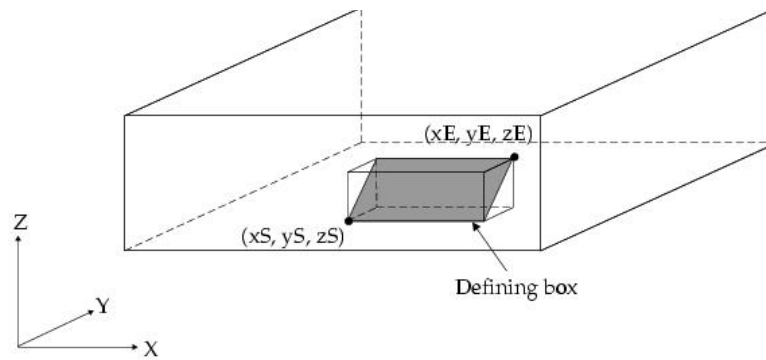


Figure 7.4.3: Inclined Object Definition

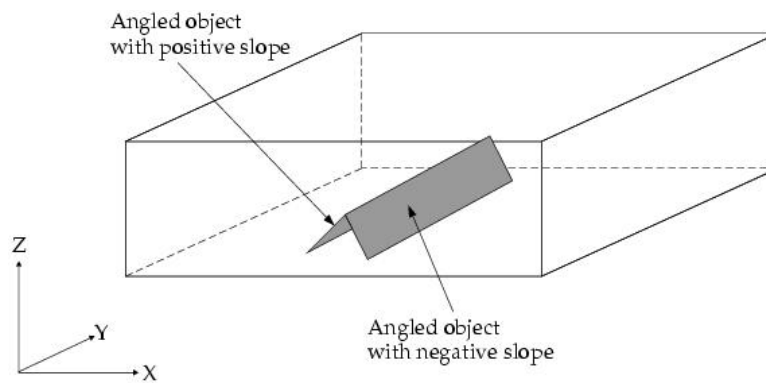


Figure 7.4.4: Inclined Object Slope Definition

- partitions
- sources
- resistances

To specify an inclined object, select **Inclined** in the **Shape** drop-down list in the **Object** panel. The user inputs for an inclined object are shown below.

The screenshot shows the 'Object' panel in a software interface. It contains several input fields and drop-down lists. At the top, 'Model as' is set to '2d', 'Shape' is set to 'Inclined', and 'Local coord system' is empty. Below these, 'Axis of rotation' is set to 'X'. A 'Location' section is expanded, showing 'Specify by' set to 'Start / end'. This section contains six input fields for coordinates: xS (0.18 m), xE (0.22 m), yS (0.305 m), yE (0.345 m), zS (0.13 m), and zE (0.17 m). At the bottom of the 'Location' section, 'Orientation' is set to 'Positive'.

Specify the **Axis of rotation** by selecting **X**, **Y**, or **Z** in the drop-down list. Under **Location**, select one of the following options from the **Specify by** drop-down list to specify the location of the inclined object.

Start/end Enter values for the start coordinates (x_S , y_S , z_S) and end coordinates (x_E , y_E , z_E), and specify the **Orientation** by selecting **Positive** or **Negative** in the drop-down list.

Start/length Enter values for the start coordinates (x_S , y_S , z_S) and lengths of the sides (x_L , y_L , and z_L) of the object, and specify the **Orientation** by selecting **Positive** or **Negative** in the drop-down list.

Start/angle Enter values for the start coordinates (x_S , y_S , z_S) and lengths of the sides (x_L , y_L , and z_L) of the object, and specify the **Angle** of inclination of the object.

Polygon Objects

Two-dimensional polygons and three-dimensional polygons are available in **Airpak**.

Two-Dimensional Polygons

A two-dimensional polygon object (see Figure 7.4.5) is described by the plane in which it lies (Y-Z, X-Z, or X-Y) and the coordinates of its vertices (e.g., vert 1, vert 2, vert 3). The coordinate on the axis that is normal to the plane of the object is specified only for the first vertex. For the remaining vertices, this same value is used. For example, if the object is in the X-Z plane, you specify $(x1, y1, z1)$ for vert 1, $(x2, z2)$ for vert 2, $(x3, z3)$ for vert 3, etc., and Airpak automatically sets $y2 = y3 = y1$, etc.

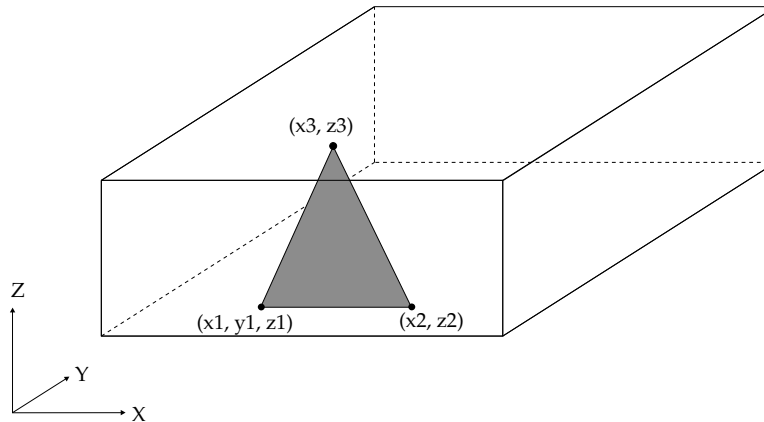


Figure 7.4.5: Definition of a 2D Polygon Object

Two-dimensional polygon geometries are available for the following objects:

- vents
- openings
- walls
- partitions
- sources
- resistances

To specify an object in the shape of a 2D polygon, select **Polygon** in the **Shape** drop-down list in the **Object** panel. The user inputs for a 2D polygon are shown below.

Shape: Polygon

Local coord system:

Plane: X-Y

Location

x1: 0.182828 m

y1: 0.283541 m

z1: 0.15 m

vert 1
vert 2
vert 3
vert 4
vert 5

Add Remove

Specify the plane in which the polygon lies by selecting Y-Z, X-Z, or X-Y in the Plane drop-down list and then specify the coordinates of the vertices (e.g., vert 1, vert 2, vert 3) of the polygon. You can add and remove vertices using the Add and Remove buttons.

To modify the position of any polygon vertex, either select its name (e.g., vert 1) in the vertex list and modify its x, y, and z coordinate values to the left of the vertex list, or select and move its point in the graphics window using the mouse.

Three-Dimensional Polygons

Three-dimensional polygon objects have top and bottom sides that are polygonal in shape and are parallel to each other. A uniform 3D polygon object (Figure 7.4.6) has a centerline aligned with one of the coordinate axes and a constant cross-section throughout the height of the block. It is described by the plane in which its base lies (Y-Z, X-Z, or X-Y), its Height, and the coordinates of the vertices (e.g., vert 1, vert 2, vert 3) on the base plane.

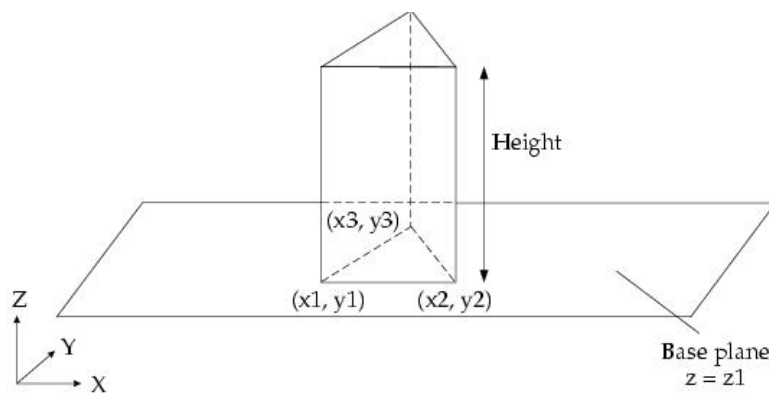


Figure 7.4.6: Definition of a Uniform 3D Polygon Object

A non-uniform 3D polygon object (Figure 7.4.7) can be skewed with respect to the coordinate axes and can possess top and bottom sides of different shapes. It is described by the plane in which its base lies (Y-Z, X-Z, or X-Y), its **Height**, and the coordinates of its vertices (e.g., low 1, low 2, low 3, high 1, high 2, high 3). The top and bottom sides of a non-uniform 3D polygon object can differ in shape but *must* be parallel to each other and possess an identical number of vertices.

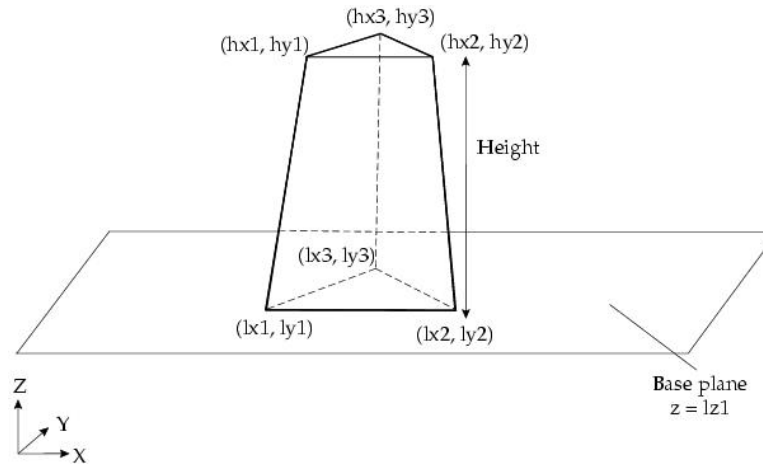


Figure 7.4.7: Definition of a Non-Uniform 3D Polygon Object

Three-dimensional polygon geometries are available for the following objects:

- blocks
- fans
- openings (recirculation only)
- hoods (canopy)

To specify an object in the shape of a 3D polygon, select **Polygon** in the **Shape** drop-down list in the **Object** panel. The user inputs for a 3D polygon are shown below.

Shape: Polygon

Local coord system:

Plane: X-Y

☐ Nonuniform

Location

Height: 0.06 m

x1: 0.179947 m

y1: 0.229947 m

z1: 0.13 m

vert 1
vert 2
vert 3

Add Remove

For a uniform 3D polygonal object, specify the plane in which the base lies by selecting Y-Z, X-Z, or X-Y in the **Plane** drop-down list. Under **Location**, specify the **Height** and the coordinates of the vertices (e.g., **vert 1**, **vert 2**, **vert 3**) on the base plane. You can add and remove vertices using the **Add** and **Remove** buttons.

If you select **None** in the **Plane** drop-down list, the polygonal block will be constructed with zero height and will take the shape of the base of the polygon.

For a non-uniform 3D polygon object, select **Nonuniform** and specify the plane in which the base lies by selecting Y-Z, X-Z, or X-Y in the **Plane** drop-down list. Under **Location** specify the **Height** and the coordinates of its vertices (e.g., **low 1**, **low 2**, **low 3**, **high 1**, **high 2**, **high 3**).

For a uniform 3D polygon object, the top and bottom sides are identical in shape and size, and the cross-section is uniform in one coordinate direction. (Therefore, a uniform 3D polygon object is completely defined by its height and the coordinates of its base vertices.) In a non-uniform 3D polygon object, the top and bottom sides lie in parallel planes and possess an identical number of vertices but are not identical in shape or size. Also, their centroids do not necessarily lie along a line parallel to any one of the coordinate axes.

The default 3D polygon object is uniform in type with a triangular base and a **Height** of 0.2 m. To modify the position of any polygon vertex, either select its name (e.g., **vert 1** or **low 2**) in the vertex list and modify its x, y, and z coordinate values to the left of the vertex list, or select and move its point in the graphics window using the mouse.



The mouse operations for a 3D polygon object in the graphics window do not restrict you from specifying vertices that render the top and bottom sides of the object non-parallel. However, **Airpak's** meshing procedures are not designed to mesh such shapes. To prevent the accidental creation of an unmeshable 3D polygon

object, restrict allowable point movement in the graphics window (using the **Motion allowed** toggle buttons in the **Interactive editing** panel as described in Section 7.3.7) before selecting and moving the polygon vertices.

Prism Objects

A prism object (Figure 7.4.8) has its sides aligned with the three coordinate planes. Its location is defined by the coordinates of the lower left front corner, referred to as the object starting point (xS , yS , zS), and the upper right corner, referred to as the object ending point (xE , yE , zE).

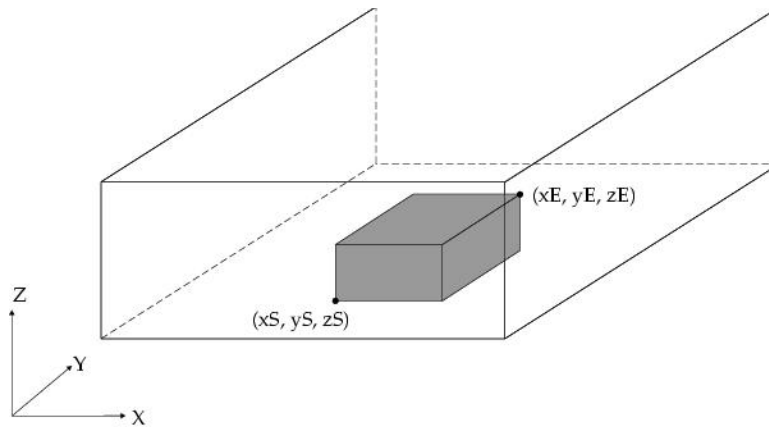


Figure 7.4.8: Definition of a Prism Object

Prism geometries are available for the following objects:

- blocks
- persons (head, body, arms, legs, thighs)
- sources
- resistances
- hoods (neck)

To specify an object in the shape of a prism, select **Prism** in the **Shape** drop-down list in the **Object** panel. The user inputs for a prism are shown below.

Shape:

Local coord system:

Location

Specify by:

xS	0.4	m	xE	0.6	m
yS	0.4	m	yE	0.6	m
zS	0.4	m	zE	0.6	m

Under **Specify by**, select **Start/end** and enter values for the start coordinates (xS , yS , zS) and end coordinates (xE , yE , zE) of the prism object, or select **Start/length** and enter values for the start coordinates (xS , yS , zS) and lengths of the sides (xL , yL , and zL) of the prism object.

Cylindrical Objects

Cylindrical objects can be specified as uniform or non-uniform. A uniform cylindrical object (Figure 7.4.9) has a constant radius throughout the height of the object, and can be described by its **Radius**, **Height**, the plane in which its base lies (Y-Z, X-Z, or X-Y), and the location of the center of its base (x_C , y_C , z_C).

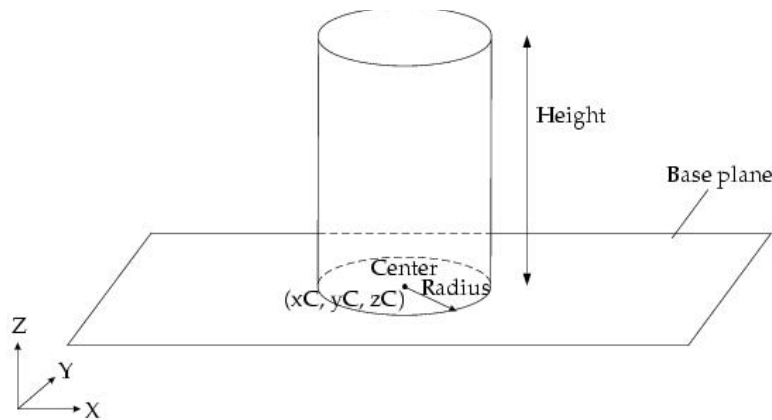


Figure 7.4.9: Definition of a Uniform Cylindrical Object

A non-uniform cylindrical object (Figure 7.4.10) has a radius that varies linearly with object height, and can be described by the parameters specified for a uniform cylinder and by the radius of the top of the cylinder (the side not located on the base plane).

Concentric cylinders are defined by the plane in which the circular base lies (X-Y, Y-Z, or X-Z), the outer radius of the cylinder (**Radius**), the inner radius (**Int Radius**), the coordinates of the center of the base (x_C , y_C , z_C), and the height of the cylinder. Additionally,

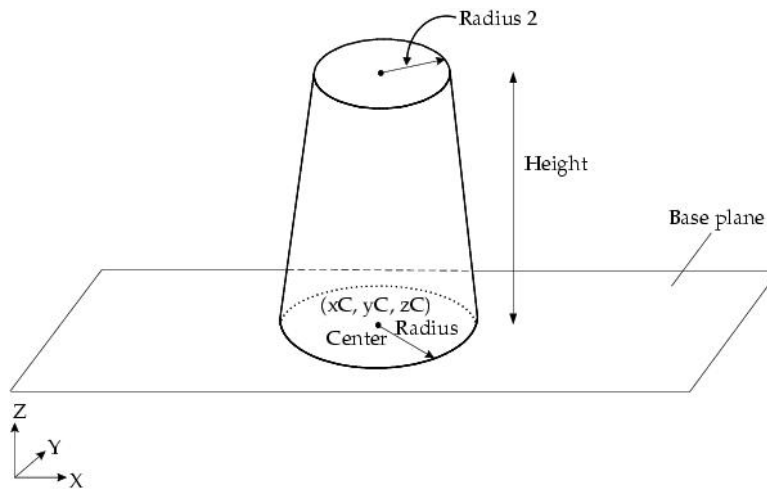


Figure 7.4.10: Definition of a Non-Uniform Cylindrical Object

the **Nonuniform radius** option can be enabled so that the cylinder can have varying radius along the length of its internal or external surfaces. The direction of increasing height is the normal direction to the specified plane. For example, if the circular base lies in the X-Z plane, the height will increase in the y direction.

Cylindrical geometries are available for the following objects:

- blocks
- fans
- openings (recirculation only)
- sources
- resistances

To specify an object in the shape of a cylinder, select **Cylinder** in the **Shape** drop-down list in the **Object** panel. The user inputs for a cylinder are shown below.

Shape: Cylinder

Local coord system:

Plane: Y-Z

☐ Nonuniform radius

Location

Center:

xC: 0.4 m

yC: 0.5 m

zC: 0.5 m

Height: 0.2 m

Radius: 0.112838 m

Int radius: 0 m

Radius 2: 0.112838 m

Int radius 2: 0 m

For a uniform cylinder, specify the plane in which the base lies by selecting Y-Z, X-Z, or X-Y in the Plane drop-down list. Under Location, specify the Radius, the Height (which can be negative), and the location of the center of the base (xC, yC, zC). For concentric cylinders, you should also specify a value for the inner radius (Int Radius).

For a non-uniform cylinder (an object in the shape of a truncated cone), select **Nonuniform radius** and specify the plane in which the base lies by selecting Y-Z, X-Z, or X-Y in the Plane drop-down list. Under Location, specify the Radius at the bottom of the cylinder, the radius (Radius 2) at the top of the cylinder (the circle *not* located on the base plane), the Height, and the location of the center of the base (xC, yC, zC). For concentric cylinders, you should also specify a value for the inner radius at the bottom of the cylinder (Int Radius) and the inner radius at the top of the cylinder (Int Radius 2).

Ellipsoid Objects

Ellipsoid objects are defined by a bounding box, just like a prism, and a set of flags that indicate whether each octant should be turned on or off. Any one or more of the eight octants can be activated or deactivated. The object is defined by the number of octants used as well as the actual dimensions specified by xS, xE, yS, yE, zS, and zE. The octants can be activated or deactivated using the xyz, Xyz, xYz, XYz, xyZ, XyZ, xYZ, and XYZ check boxes. The naming scheme follows the convention where, if the character is lower-case, then it activates the lower half, and if it is upper-case, then it activates the upper half.

i Note that ellipsoids can be meshed only with the tetrahedral mesher (see Section 26.3).

Ellipsoid geometries are available for the following objects:

- blocks
- sources

To specify an object in the shape of an ellipsoid, select **Ellipsoid** in the **Shape** drop-down list in the **Object** panel. The user inputs for an ellipsoid are shown below.

Shape: **Ellipsoid**

Local coord system:

Location

xS	0.4	m	xE	0.6	m
yS	0.387162	m	yE	0.612838	m
zS	0.387162	m	zE	0.612838	m

Corners

<input checked="" type="checkbox"/> xyz	<input checked="" type="checkbox"/> Xyz	<input checked="" type="checkbox"/> xYz	<input checked="" type="checkbox"/> XYz
<input checked="" type="checkbox"/> xyZ	<input checked="" type="checkbox"/> XyZ	<input checked="" type="checkbox"/> xYZ	<input checked="" type="checkbox"/> XYZ

Under **Location**, specify the dimensions of the object (xS, xE, yS, yE, zS, zE). Under **Corners**, activate or deactivate octants by selecting or deselecting xyz, Xyz, xYz, XYz, xyZ, XyZ, xYZ, or XYZ. The naming scheme follows the convention where, if the character is lower-case, then it activates the lower half, and if it is upper-case, then it activates the upper half.

Elliptical Cylinder Objects

Elliptical cylinders (**E. cylinder**) are defined with four quadrants. The two centers are specified by X, Y, Z for the bottom center point (**Bot cent**), and X, Y, Z for the top center (**Top cent**), as shown in Figures 7.4.11 and 7.4.12. Each center has two vectors that should be perpendicular, which define the two end caps of the cylinder. The four vectors are named **Bot vec 1** and **Bot vec 2** for the low center (**Bot cent**), and **Top vec 1** and **Top vec 2** for the high center (**Top cent**). One or more of the quadrants can be activated or deactivated using the check buttons -1 -2, -1 +2, +1 -2, and +1 +2. Here, -1 -2 is the quadrant between the vectors -**Bot vec 1** and -**Bot vec 2**, -1 +2 is the quadrant between the vectors -**Bot vec 1** and **Bot vec 2**, +1 -2 is the quadrant between the vectors **Bot vec 1** and -**Bot vec 2**, and +1 +2 is the quadrant between the vectors **Bot vec 1** and **Bot vec 2**.



Note that elliptical cylinders can be meshed only with the tetrahedral mesher (see Section 26.3).

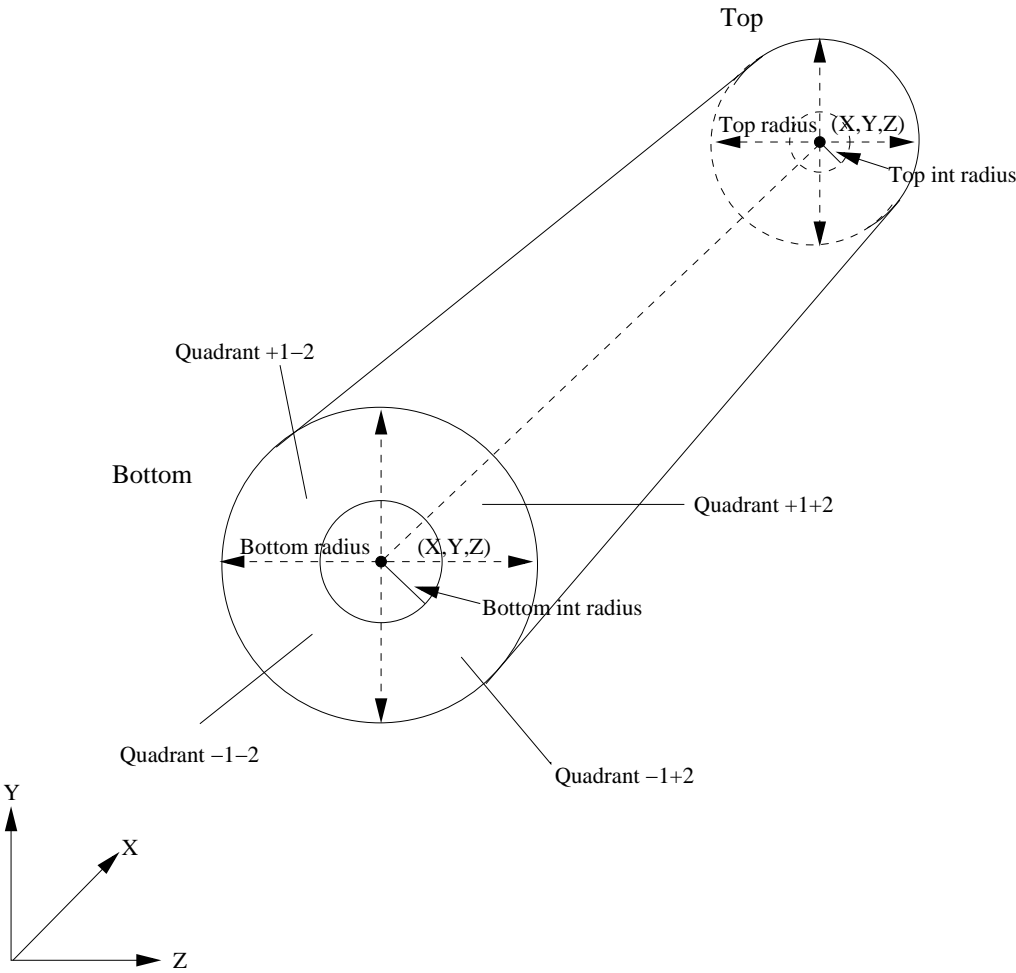


Figure 7.4.11: Nomenclature for Simplified Elliptical Cylinders

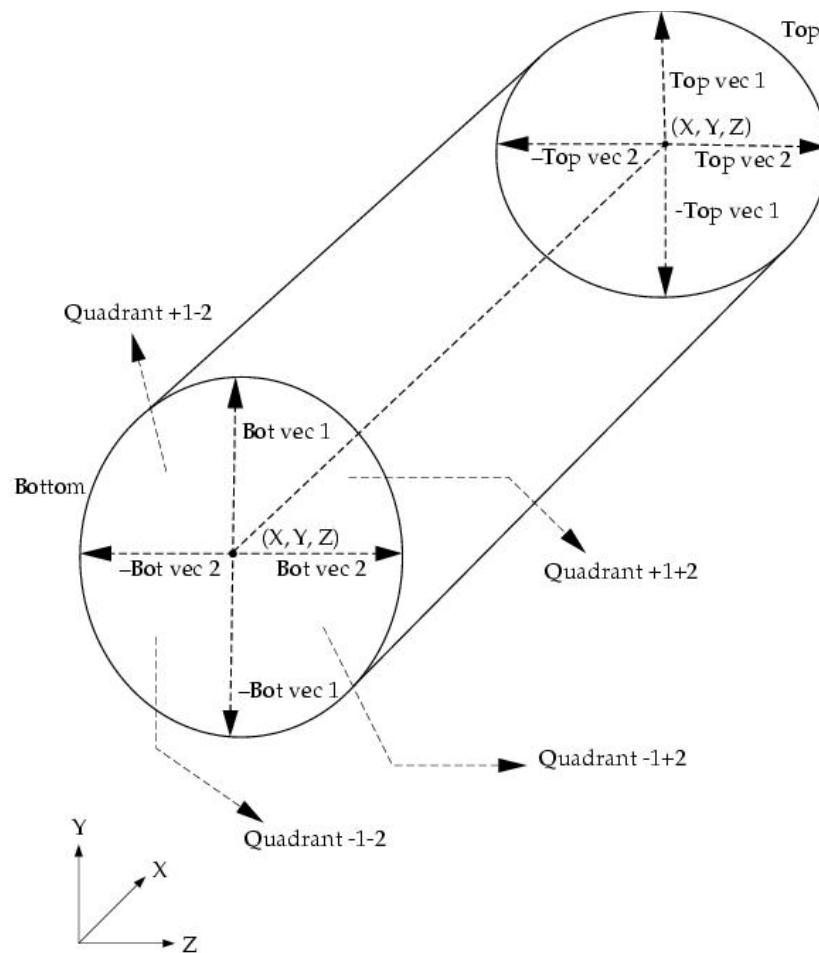


Figure 7.4.12: Nomenclature for Detailed Elliptical Cylinders

Elliptical cylinder geometries are available for the following objects:

- blocks
- sources

To specify an object in the shape of an elliptical cylinder, select **E. cylinder** in the **Shape** drop-down list in the **Object** panel. The user inputs for an elliptical cylinder are shown below.

The screenshot shows the configuration panel for an elliptical cylinder. It includes a 'Shape' dropdown set to 'E. cylinder' and a 'Local coord system' dropdown. Below these are input fields for 'Bot cent' and 'Top cent' with X, Y, and Z coordinates. A 'Corners' section has four checkboxes: '-1 -2', '-1 +2', '+1 -2', and '+1 +2', all of which are checked. There are two radio buttons: 'Simplified specification' (unselected) and 'Detailed specification' (selected). Under 'Simplified specification', there are input fields for 'Bot radius', 'Bot int radius', 'Top radius', and 'Top int radius'. Under 'Detailed specification', there is a note 'See the labels on the object for reference.' followed by input fields for 'Bot vec 1', 'Bot vec 2', 'Top vec 1', and 'Top vec 2', each with X, Y, and Z components.

Specify X, Y, Z for the bottom center point (**Bot cent**), and X, Y, Z for the top center point (**Top cent**).

The **Simplified specification** option allows you to define a cylinder with circular end caps of variable radius and also allows you to define tube-like geometry. For the **Simplified specification**, specify the **Bot radius** and **Bot int radius** for the bottom end cap, and the **Top radius** and **Top int radius** for the top end cap.

The **Detailed specification** allows you to define an elliptical cylinder with non-parallel end caps, because the vectors can encompass all three coordinate directions. For the **Detailed specification**, specify perpendicular vectors for the bottom center (**Bot vec 1** and **Bot vec 2**) and for the top center (**Top vec 1** and **Top vec 2**).

When you have specified the necessary radii or vectors, activate or deactivate quadrants by selecting or deselecting -1 -2, -1 +2, +1 -2, and +1 +2 next to **Corners**. For the **Detailed specification**, -1 -2 is the quadrant between the vectors -**Bot vec 1** and -**Bot vec 2**, -1 +2 is the quadrant between the vectors -**Bot vec 1** and **Bot vec 2**, +1 -2 is the quadrant between

the vectors **Bot vec 1** and **-Bot vec 2**, and **+1 +2** is the quadrant between the vectors **Bot vec 1** and **Bot vec 2**. For the **Simplified specification**, the four quadrants (**Corners**) are defined in the same way as for a detailed elliptical cylinder if you imagine that setting a value for the **Bot radius** is equivalent to setting **Bot vec 1** and **Bot vec 2** to be the same length and in the same plane.

CAD Objects

CAD objects are directly imported using the IGES/STEP import feature (see Section 4.2). CAD objects can possess the most general 2D (Figure 7.4.13) and 3D (Figure 7.4.14) shapes. The geometry of a CAD object cannot be specified or modified in Airpak. As such, any changes to the shape of the object must be made in the CAD tool before being exported as an IGES/Step file.

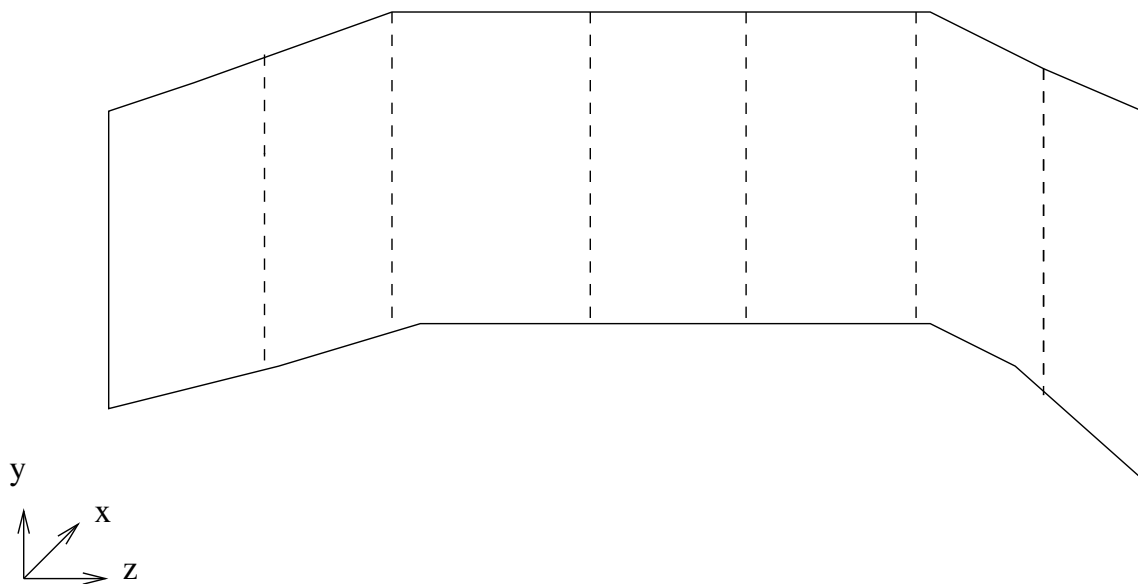


Figure 7.4.13: 2D CAD Object



CAD geometries are available only for blocks and partitions.

7.4.5 Physical Characteristics

Airpak allows you to specify the physical characteristics of an object in the **Object** panel (e.g., Figure 7.3.4). The physical characteristics are specific to the type of object being configured. For example, the **Object** panel for printed circuit boards allows you to specify whether there is a rack of boards, and, if so, how many boards are in the rack. Also, you can specify the number and height of components on each side of the board, as well as

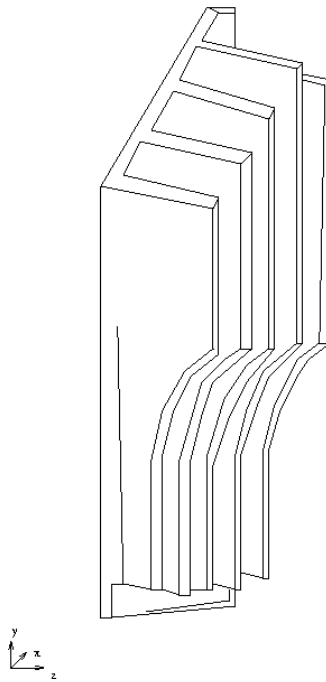


Figure 7.4.14: 3D CAD Object

physical properties for each side. Thermal characteristics of an object are also specified in the **Object** panel.

Specific information on the use of the individual **Object** panels appears in Section 7.5 within the discussion of each object type.

7.5 Adding Objects to the Model

Adding an object to your model is more complicated than defining a room, because, in addition to specifying dimensions, you must also assign physical properties, describe the components of some objects, and configure the object within the model.

Basic objects, including fans, vents, persons, hoods, and walls, are available in pre-constructed forms that you can edit to your specifications. More general kinds of objects, such as blocks, openings, sources, and resistances, are also provided in pre-constructed forms that you can customize into any object you might need in your model.

The first step in the process of adding an object to your model is to select the object type in the **Object creation** toolbar (Figure 7.1.1). Once you have selected an object, you can edit its dimensions, specify its physical characteristics, and choose whether to make it an active or inactive part of the model. You can also move it and position it precisely in the model or make multiple copies of it elsewhere in the model.

The basic steps for adding an object to your model are as follows:

1. Create the object.
2. Change the description of the object.
3. Change the graphical style of the object (optional).
4. Specify the geometry, position, and size of the object.
5. Specify the type of the object (if relevant).
6. Specify the physical and thermal characteristics of the object.

The objects available for your **Airpak** model are described in detail in the Chapters [8–19](#).

7.6 Grouping Objects

Airpak allows you to group objects in your model. Groups can be used for several purposes, including the following:

- defining a set of objects as a single design element so that you can use it repeatedly in a series of models
- modifying the properties of like objects
- designating a collection of objects as a set to test one part of the model independently from the whole model
- moving several objects simultaneously
- grouping objects together for postprocessing purposes
- turning defined sets of objects on and off at your discretion

Groups can be easily activated and deactivated, allowing you to control the parts of your model at a sub-assembly level by selecting the objects that are to be included in the current analysis. Groups can be copied or moved within the model using a variety of geometric transformations. **Airpak** allows you to copy the properties of one object to a collection of objects of the same type in a single operation.

You can use the **Edit** menu or the **Groups** node in the **Model manager** window (Figure [7.6.1](#)) to create and configure groups within your **Airpak** model.

Edit → **Current group**

The procedures for creating and configuring groups are described in the following sections.

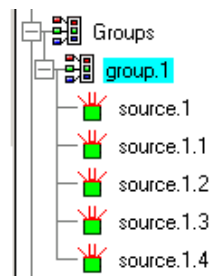


Figure 7.6.1: The Groups Node in the Model manager Window

7.6.1 Creating a Group

To create a new group, double-click on the **Groups** node in the **Model manager** window (Figure 7.6.1) or select **Create** in the **Edit**→**Current group** menu. The new group node will be added under the **Groups** node. It will not contain any objects until you add them to it, as described in Section 7.6.4.

Alternatively, you can use the <Control> and <Shift> keys in combination with the mouse to select multiple objects in the **Model manager** window (see Section 2.2.2). To create a group from the selected items, right-click on one of the items and select **Create group** in the pull-down menu. In the resulting **Query** dialog box (Figure 7.6.2), enter a name and click **Done**.



Figure 7.6.2: The Query Dialog Box for Creating a Group

Airpak will create a new group with the default name **group.n**, where **n** is the next sequential number among numbered groups. The **Groups** node lists the groups that already exist in the current model. Selecting a group in this list and expanding the node will display the list of objects it contains.

7.6.2 Renaming a Group

To rename an existing group, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Rename** in the pull-down menu or select **Rename** in the **Edit**→**Current group** menu. In the resulting **Query** dialog box (Figure 7.6.3), enter a new name and click **Done**.

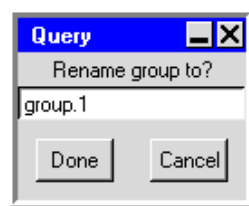


Figure 7.6.3: The Query Dialog Box for Renaming a Group

7.6.3 Changing the Graphical Style of a Group

Airpak allows you to change the display of a group in the graphics window. You can change the shading, color, and line width of the group in the **Group parameters** panel (Figure 7.6.4).

To open the **Group parameters** panel, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Properties** in the pull-down menu.

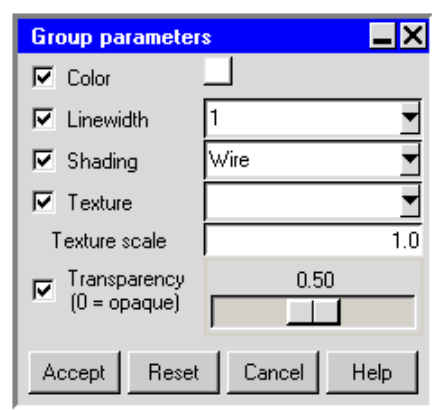


Figure 7.6.4: The Group parameters Panel

Changing the Color

To change the color of a group, turn on the **Color** option in the **Group parameters** panel and click on the square button to the right of the **Color** text option. A color palette menu will open. See Section 7.2.5 for details about selecting colors. Click **Accept** in the **Group parameters** panel to change the color of the group in the graphics window.

Changing the Line Width

To change the width of the lines for a group, turn on the **Linewidth** option and select a line width (1, 2, 3, 4, or 5) from the drop-down list. Click **Accept** in the **Group parameters** panel to change the line width of the group in the graphics window.

Changing the Shading

To change the shading of a group, turn on the **Shading** option in the **Group parameters** panel and select a shading type (**View**, **Wire**, **Solid**, **Solid/wire**, **Hidden line**, or **Invisible**) from the drop-down list. Click **Accept** in the **Group parameters** panel to change the shading of the group in the graphics window.

Note that the type of shading that will be applied to the group when **View** is selected is taken from the **Shading** sub-menu in the **View** menu, described in Section 2.1.2.

Changing the Texture

The surface texture of a group can be modified when **Solid** or **Solid/wire** has been selected in the **Shading** drop-down list. The default texture option is **No texture**. To modify the texture, turn on the **Texture** option in the **Group parameters** panel. Select **Load from file**, and then select an available PPM image file in the resulting **File selection** dialog box (see Section 2.1.8).

Depending on its pixel size, the image, or a portion thereof, will be used to cover each side of the object. To change the size of the image used for the texture, specify a value for the **Texture scale**. Values greater than 1.0 will reduce the size of the image on each side of the object, resulting in a tiling effect at large enough values. Values less than 1.0 will increase the size of the image on each side of the object.

Changing the Transparency

The transparency of a group can be modified by turning on the **Transparency** option and moving the slider bar between values of 0.00 (opaque) and 0.99 (fully transparent). This option is most useful when **Solid** has been selected in the **Shading** drop-down list.

7.6.4 Adding Objects to a Group

To add an object to a group, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Add** in the pull-down menu or select **Add to group** in the **Edit**→**Current group** menu.

There are three ways to add objects to a group using these menus:

Screen select adds individually selected objects to a group. To add an object to a group, select **Screen select** and then use the <Shift> key and the left mouse button to select the object you want to add to the group in the graphics window. To deactivate the **Screen select** mode, press the <Shift> key and click the middle mouse button or the right mouse button in the graphics window.

Screen region adds objects to a group based on their location in the model. This option allows you to add all objects in an entire region of the model to a group. To add objects, select **Screen region** and then select the objects in the model to be added to the group by defining a rectangular box on the screen. Position the mouse pointer at a corner of the area where the objects to be included are located, hold down the left mouse button and drag open a selection box to enclose the objects to be included, and then release the mouse button. The objects within the bounded area will be added to the currently-selected group.

Name/pattern adds any object whose name matches a specified text pattern to a group. To add an object, select **Name/pattern**. This opens a **Query** dialog box (Figure 7.6.5).

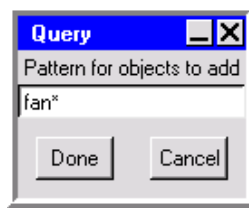


Figure 7.6.5: The **Query** Dialog Box for Adding Objects to a Group

You can type an object name that contains an asterisk or a question mark in place of characters or a character, respectively. For example, typing **fan*** will add all objects whose names start with **fan** to the group; typing **vent?** will add all objects whose names consist of the word **vent** plus one character to the group. Any object in the model whose name matches this text pattern will be added to the group when you click **Done** in the **Query** dialog box.

You can also add an object to a group by typing the name of the group in the **Groups** text entry box in the **Object** panel (see Section 7.4.1).

7.6.5 Removing Objects From a Group

To remove an object from a group, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Remove** in the pull-down menu or select **Remove from group** in the **Edit**→**Current group** menu.

There are three ways to remove objects from a group using these menus. For all methods, the object(s) will be removed only from the group, *not* from the model.

Screen select removes individually selected objects from an existing group. To remove an object from a group, select **Screen select** and use the <Shift> key and the left mouse button to select the object you want to remove from the group in the graphics window. To deactivate the **Screen select** mode, press the <Shift> key and click the middle mouse button or the right mouse button in the graphics window.

Screen region removes objects from a group based on their location in the model. This option allows you to remove all objects in an entire region of the model from a group. To remove an object, select **Screen region** and then select the objects in the model to be removed from the group by defining a rectangular box on the screen. Position the mouse pointer at a corner of the area where the objects to be removed

are located, hold down the left mouse button and drag open a selection box to enclose the objects to be removed, and then release the mouse button. The objects within the bounded area will be removed from the currently-selected group.

Name/pattern removes any object whose name matches a specified text pattern from a group. To remove an object, select **Name/pattern**. This will open a **Query** dialog box (Figure 7.6.6).

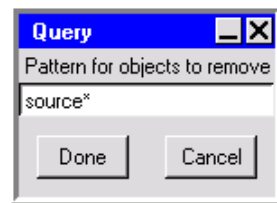


Figure 7.6.6: The **Query** Dialog Box for Removing Objects from a Group

You can type an object name that contains an asterisk or a question mark in place of characters or a character, respectively. For example, typing **fan*** will remove all objects whose names start with **fan** from the group; typing **vent?** will remove all objects whose names consist of the word **vent** plus one character from the group. Any object in the model whose name matches this text pattern will be removed from the group when you click **Done** in the **Query** dialog box.

Note that you can also remove an object from a group by right-clicking on the object name under the specific group node in the **Model manager** window and selecting **Remove from group** at the bottom of the pull-down menu.

7.6.6 Copying Groups

There are two options related to copying groups in the **Edit**→**Current group** menu.

Copy opens the **Copy group** panel. You can make one or more copies of the group while applying various geometric transformations to the group. The operations in the **Copy group** panel are the same as those in the **Copy object** panel (Figure 7.3.15) and are described in Section 7.3.9. There is one additional option in the **Copy group** panel: you can specify a name for the new group by turning on the **Group name** option and entering a name in the text entry field.

Copy params applies the parameters of the currently-selected object to all the objects of the same type in the currently-selected group. This option is useful if you want to change all the objects in a group at the same time. Note that these parameters are the physical characteristics of the object, *not* the geometric definition.

Select a group under the **Groups** node in the **Model manager** window. To select the object whose properties you wish to apply to the remaining objects of similar type in the group, click on the name of the object under the specific group node using the left mouse button. Select the **Copy params** option in the **Edit**→**Current group** menu to apply the physical characteristics of the currently-selected object to all the objects of the same type in the currently-selected group.

Note that you can also access these items using the **Model manager** window by right-clicking on the specific group node and selecting **Copy group** or **Copy params** in the pull-down menu.

7.6.7 Moving a Group

To move a group, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Copy group** in the pull-down menu or select **Copy** in the **Edit**→**Current group** menu. This opens the **Move group** panel. You can apply various transformations to the group. The operations in the **Move group** panel are the same as those in the **Move all objects in model** panel (Figure 7.2.6). The **Scale** option is described in Section 7.2.1, and the **Mirror**, **Rotate**, and **Translate** options are described in Sections 7.2.2 and 7.3.7.

You can also move a group using the mouse by selecting the group in the **Model manager** window, holding down the <Shift> key and dragging the group with the middle mouse button in the graphics window.

7.6.8 Editing the Properties of Like Objects in a Group

You can edit the properties of like objects simultaneously within a group. Note that the objects in the group must be of the same type (e.g., all blocks), and you cannot change the geometry of the objects. To edit the properties of like objects in a group, right-click on the group under the **Groups** node in the **Model manager** window and select **Edit objects** in the pull-down menu or select **Edit** in the **Edit**→**Current group** menu. This will open the **Object** panel (e.g., the **Blocks** panel if the group contains blocks) and display the properties of the selected objects. You can edit any properties of the objects that are not grayed out in the **Object** panel. Click **Done** when you have finished editing the objects. **Airpak** will close the **Object** panel and apply the changes to all the objects in the group. You can check the changes that **Airpak** has made to the objects in the group by viewing the properties of the objects in the **Parameter summary** panel, described in Section 7.9.1.

7.6.9 Deleting a Group

There are two options for deleting groups in the **Edit**→**Current group** menu:

Delete removes the selected group from under the **Groups** node in the **Model manager** window.

i Deleting a group does not cause the objects associated with it to be deleted.

Delete all permanently removes all the objects in a group. The objects in the group will be removed from the graphics window, and from the model.

i When you delete the objects in a group using **Delete all**, the objects are deleted from the model. They can be retrieved using the **Undo** button in the **Edit** menu or by moving them out of the **Trash** node in the **Model manager** window.

Note that you can also access these items in the **Model manager** window by right-clicking on the specific group node and selecting **Delete** or **Delete all** in the pull-down menu.

7.6.10 Activating or Deactivating a Group

You can activate or deactivate a group within your current **Airpak** model in the **Edit**→**Current group** menu:

Activate all makes the objects in a group an active part of your model.

Deactivate all allows you to temporarily remove the objects in a group from the model. This renders the group inactive for the purpose of the current analysis. The group remains in the model, but is no longer active or visible.

Note that you can also access these items using the **Model manager** window by right-clicking on the specific group node and selecting **Activate all** or **Deactivate all** in the pull-down menu.

7.6.11 Using a Group to Create an Assembly

To create an assembly using a group, right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Create assembly** in the pull-down menu or select **Create assembly** in the **Edit**→**Current group** menu. See Section 7.8 for information on assemblies.

7.6.12 Saving a Group as a Project

To save a group of objects as a separate project (model), right-click on the group node in the **Model manager** window (Figure 7.6.1) and select **Save as project** in the pull-down menu or select **Save as project** in the **Edit**→**Current group** menu. This will open the **Save project** panel. (See Section 3.4 for information on the **Save project** panel.) You can save the project containing the group to any directory, but it is recommended that you create a local groups directory where you can save all the groups you create.

7.7 Material Properties

An important step in the setup of your model is the definition of the physical properties of the materials. Material properties are defined in the **Materials** panel, which allows you to input values for the properties that are relevant to the problem you have defined in **Airpak**. These properties can include the following:

- Density and/or molecular weights
- Viscosity
- Specific heat capacity
- Thermal conductivity
- Diffusivity
- Volumetric expansion coefficient
- Surface roughness
- Emissivity
- Solar and diffuse absorptance and transmittance

Materials can be defined from scratch, or they can be used directly from the **Airpak** material property database.

Some properties may be temperature-dependent, and **Airpak** allows you to define a property as:

- a constant value
- a linear function of temperature
- a piecewise-linear function of temperature, described as a series of points defining the property variation. These points can be described in two ways:
 - using the graph editor
 - specifying points as coordinate pairs

You can also define the thermal conductivity of a fluid to be velocity-dependent, if required.

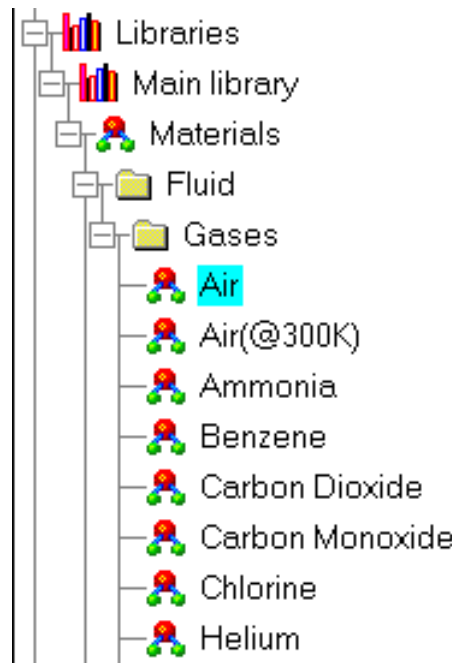





Figure 7.7.1: The Materials Node in the Model manager Window

7.7.1 Using the Materials Library and the Materials Panel

You will use the Materials library node in the Model manager window and the Materials panel to define materials within your Airpak model.

 Libraries →  Main library →  Materials


The Materials library node in the Model manager window works in conjunction with the Materials panel. The Materials panel allows you to edit the physical characteristics of the material selected under the Materials library node. The Materials panel will have different inputs depending on the material selected.

7.7.2 Editing an Existing Material

A common operation that you will perform with the Materials panel is the modification of an existing material. Items in the Materials library, themselves, cannot be edited. Instead, you can create a copy of an existing material for use in your model and then edit the copy.

The steps for this procedure are as follows:

1. Expand the node for the type of the material (e.g., Fluid, Solid, Surface) in the Model manager window. Expand the node for the sub-type of the material (e.g., Other

in Figure 7.7.1) and select the material item under the sub-type node. Right-click on the material item and select **Edit** in the pull-down menu to open the **Materials** panel. You can also open the **Materials** panel by selecting the material item in the **Model manager** window and clicking on the  button in the **Object modification** toolbar. Figure 7.7.2 shows the **Materials** panel for a solid material.

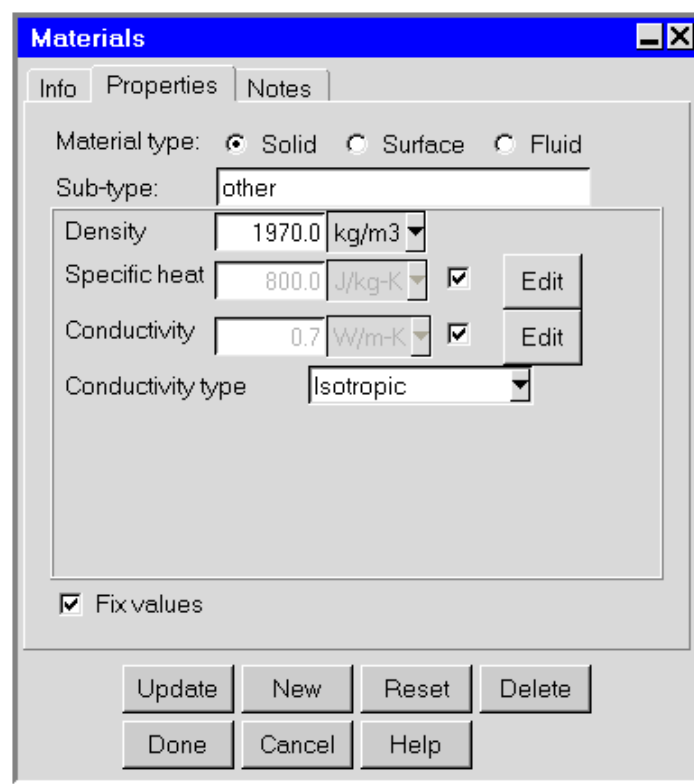


Figure 7.7.2: The Materials Panel for a Solid Material (Properties Tab)

The **Material type** and the **Sub-type** for the material will be displayed in the **Properties** tab of the **Materials** panel. The lower part of the panel will change depending on your selection of material type.

2. Change the name of the material, if required. The name of the currently selected material is displayed in the **Name** text entry box in the **Info** tab of the **Materials** panel. You can change the name of the material by entering a new name in the **Name** text entry field.

Alternatively, you can rename a material using the **Model manager** window. Right-click on the material item and select **Rename** in the pull-down menu. In the subsequent **Query** dialog box, enter a new name in the text entry field and click **Done**.

3. In the **Properties** tab, specify whether you want to fix the values for the units in the panel. See Section 5.4.2 for details.

4. Define the properties of the material. The properties for different types of materials are described below.
5. Click **Update** to save the changes to the material for the current project and keep the **Materials** panel open, or click **Done** to save the changes to the material for the current project and close the **Materials** panel.

You can also edit the definition of a material in any panel that contains a materials list. For example, the **Walls** panel contains a list called the **External material** list. To edit the properties of the material selected in the **External material** list, open the list and select **Edit definition**. This will open the **Materials** panel and display the properties of the selected material. You can edit the definition of the material as described above. Click **Done** when you have finished editing the material, and **Airpak** will close the **Materials** panel and return to the **Walls** panel.



Note that any changes you make to the materials in the current project will be saved as part of the current project only. These changes will be available whenever you work on this project, but not for any other project. See Section 7.7.6 for details on saving material properties to a library.

Editing a Solid Material

The inputs for editing a solid material are shown in Figure 7.7.2. The following properties are defined for solid materials:

Density is the density of the solid.

Specific heat is the specific heat capacity of the solid.

Conductivity is the thermal conductivity of the solid.

Conductivity type contains a list of options for specifying the thermal conductivity. To specify an isotropic conductivity, select **Isotropic** from the **Conductivity type** drop-down list and enter a value for the **Conductivity**. You can specify a non-isotropic conductivity in **Airpak**, where the thermal conductivity varies with respect to one or more coordinate axes. There are three options available in **Airpak**:

- To define an orthotropic thermal conductivity, select **Orthotropic** from the **Conductivity type** drop-down list and specify the thermal **Conductivity**. To define the degree to which the orthotropic conductivity varies in each coordinate direction, specify a scaling factor for the conductivity in each of the X, Y, and Z coordinate directions.

- To specify an anisotropic thermal conductivity, specify the thermal **Conductivity** and then select **Anisotropic** from the **Conductivity type** drop-down list. To define the degree to which the anisotropic conductivity varies in each direction, specify a scaling factor for the conductivity in each of the XX, XY, XZ, YX, YY, YZ, ZX, ZY, and ZZ directions, respectively, next to **Tensor**. Alternatively, you can click on the **Edit** button to the right of the **Tensor** text entry field. This opens the **Anisotropic tensor** panel (Figure 7.7.3).

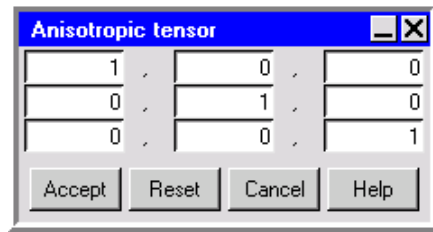


Figure 7.7.3: The Anisotropic tensor Panel

Enter the scaling factors for the conductivity in the following way in the **Anisotropic tensor** panel:

XX	XY	XZ
YX	YY	YZ
ZX	ZY	ZZ

- To specify a biaxial thermal conductivity, select **Biaxial** from the **Conductivity type** drop-down list and specify the thermal **Conductivity**. To define the degree to which the biaxial conductivity varies in each direction, specify a scaling factor for the conductivity in the **Normal** and **In-plane** directions.

The thermal conductivity is used when temperature is included in the problem (see Section 6.4.2). When modeling a transient problem, density and specific heat will also be used. See Chapter 20 for details on transient simulations.

Editing a Fluid Material

The inputs for editing a fluid material are shown in Figure 7.7.4.

The following properties are defined for fluid materials:

Vol. expansion is the volumetric expansion coefficient of the fluid.

Viscosity is the fluid dynamic viscosity.

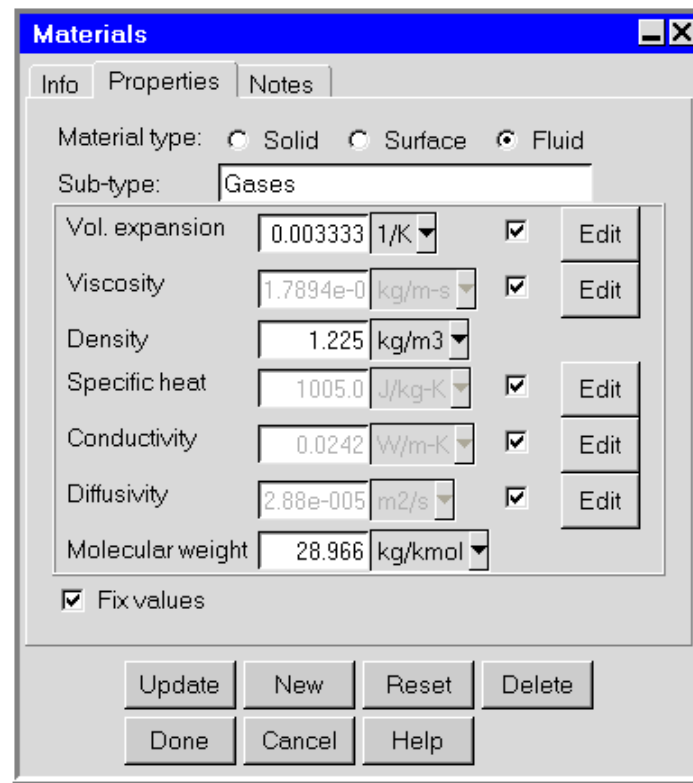


Figure 7.7.4: The Materials Panel for a Fluid Material

Density is the fluid density.

Specific heat is the specific heat capacity of the fluid.

Conductivity is the thermal conductivity of the fluid (which can be velocity- or temperature-dependent).

Diffusivity is the diffusivity of the fluid.

Molecular weight is the molecular weight of the fluid.

The density and viscosity of the fluid are used in all flow problems, but **Airpak** automatically sets an effective viscosity when the flow is turbulent, as described in Section 6.4.3. The thermal conductivity and specific heat are used when temperature is included in the problem (see Section 6.4.2). Again, **Airpak** automatically sets an effective conductivity when the flow is turbulent. The volumetric expansion coefficient of the fluid is used when natural convection due to gravity effects is present (see Section 6.4.4). If gravity is not activated, the volumetric expansion coefficient is not used. By default, **Airpak** uses the Boussinesq approximation (see Section 31.3.1) to model the buoyancy force and in this

case **Airpak** uses a constant density in its calculations. If you select the ideal gas law to model the buoyancy force, **Airpak** uses a density that varies as $\frac{p}{RT}$, as described in Section 31.3.2.

i If you use the ideal gas law and you have created a new fluid material, make sure that you specify the correct molecular weight for the new material.

Editing a Surface Material

The inputs for editing a surface material are shown in Figure 7.7.5.

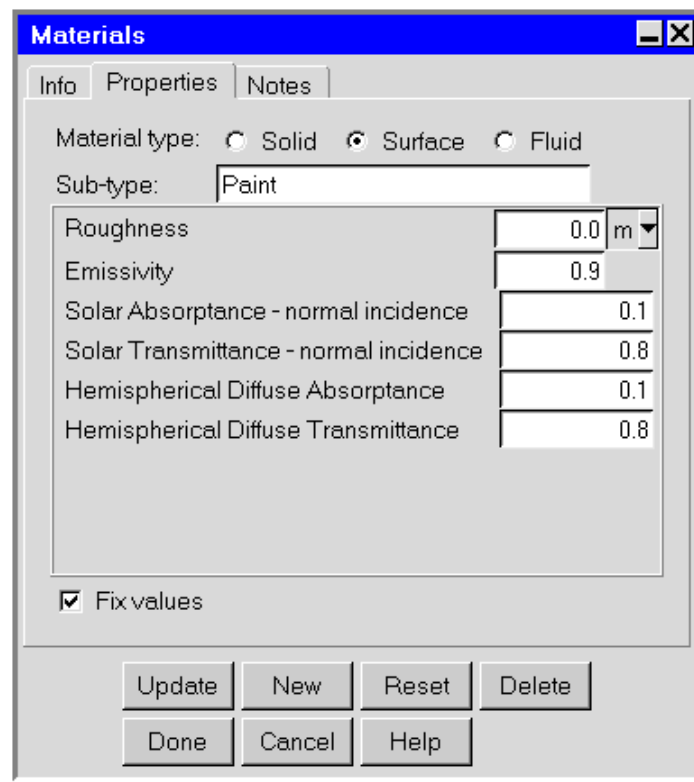


Figure 7.7.5: The Materials Panel for a Surface Material

The following properties are defined for surface materials:

Roughness is the roughness of the entire surface, where zero represents a perfectly smooth surface.

Emissivity is the emissivity of the surface.

Solar absorptance - normal incidence is the fraction of direct solar irradiation energy that is absorbed by the semi-transparent surface for a direct solar irradiation beam normal to the surface.

Solar transmittance - normal incidence is the fraction of direct solar irradiation energy that passes through the semi-transparent surface for a direct solar irradiation beam normal to the surface.

Diffuse hemispherical absorptance is the fraction of diffuse solar irradiation energy that is absorbed by the semi-transparent surface.

Diffuse hemispherical transmittance is the fraction of diffuse solar irradiation energy that passes through the semi-transparent surface.

The roughness of the surface is used when the flow is turbulent, as described in Section 6.4.3. The emissivity is used when radiation is modeled (see Chapter 22). Solar and diffuse absorptance and transmittance properties are used when solar loading is modeled (see Section 22.5). Properties of semi-transparent surface radiation account for its solar performance relative to direct solar irradiation and diffuse solar irradiation. One semi-transparent surface material may be used to represent the lumped effect of fenestration containing several glazing layers. For any semi-transparent surface, the sum of absorptance, transmittance, and reflectance must equal 1.0 for each type of solar irradiation. Therefore, it is sufficient to specify only two of the three solar performance properties for each type of radiation.


7.7.3 Viewing the Properties of a Material

You can view the properties of a material using any panel that contains a materials list. For example, the Walls panel contains a list called the **External material** list. To view the properties of the material selected in the **External material** list, open the list and select **View definition**.

The **Message** window will report the properties of the material, e.g., for the default material in the **External material** list in the Walls panel:

```
Material name: "Al-Polished Partition-surface", type surface
  Roughness = 0.0 m
  Emissivity = 0.041
```

7.7.4 Copying a Material

To copy a material, right-click on the material item in the **Model manager** window and select **Copy material** in the pull-down menu. You can also copy a material by selecting the material item in the **Model manager** window and clicking on the  button. A new material will be created under the **Model** node (Figure 7.7.6) with the

name of the material that you have copied and the suffix `.1`. For example, if you make a copy of the solid material `Al-Pure`, the new material will have the default name `Al-Pure.1`. The properties of the new material are the same as the properties of the material that was copied. The properties of the new material can be edited as described in Section 7.7.2.

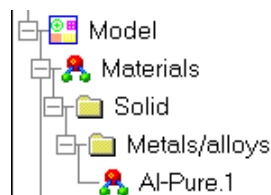



Figure 7.7.6: The **Materials** Node Under the **Model** Node

i Note that any changes you make to the materials in the current project will be saved as part of the current project only. These changes will be available whenever you work on this project, but not for any other project. See Section 7.7.6 for details on saving material properties to a library.

7.7.5 Creating a New Material

To create a new material, click on the  button in the **Object creation** toolbar. A new material will be created with the default name `material.n`, where `n` is the next sequential number among numbered materials. The name of the new material will appear under the **Model** node in the **Model manager** window (Figure 7.7.6) and in the **Name** text entry box in the **Materials** panel. See Section 7.7.2 for details on renaming a material.

You can also create a new material in any panel that contains a materials list. For example, the **Walls** panel contains a list called the **External material** list. To create a new material, open the **External material** list and select **Create material**. This will open the **Materials** panel and create a material called `wall.1_ext_material` in this example. You can rename and edit the material using the **Materials** panel. Click **Done** when you have finished editing the material, and **Airpak** will close the **Materials** panel and return to the **Walls** panel.

i Note that any changes you make to the materials in the current project will be saved as part of the current project only. These changes will be available whenever you work on this project, but not for any other project. See Section 7.7.6 for details on saving material properties to a library.

7.7.6 Saving Materials and Properties

By default, **Airpak** searches for the materials file that is provided with the standard **Airpak** distribution. This file includes a large collection of material definitions that are available to all **Airpak** users. It is located in the `AIRPAK_ROOT/airpak_lib` directory and is called the **materials** file. Different materials files can be accessed using different libraries. If you wish to modify the properties of materials that are included with **Airpak**, there are two ways to do this:

- create a new library and copy materials to the library
- copy materials from the main library into the model

You can access as many different materials files as you like. Note that all the materials files must have the same name (**materials**), and so must be located in different directories.

When you use the **Model manager** window or the **Materials** panel to create, modify, or delete an existing material, these modifications are saved as part of the current project only. These changes will be available whenever you work on this project, but not for any other project.

To create a new library of materials, you have two options:

- Create your own local materials file using the following procedure:
 1. Select **Create material library** in the **Model** menu.
Model → Create material library
 2. Select all the materials that you want to save from the list in the **Selection** panel (Figure 7.7.7). Click **All** if you want to select all the materials in the list. Click **None** to deselect all of the materials in the list.
 3. Click **Done** when you have finished selecting the materials to be saved. This will open the **File selection** dialog box, in which you can specify the filename (**materials**) and directory to which the materials file is to be saved. See Section 2.1.8 for details about the file selection dialog box.

To use the new materials file, add the path for the **materials** file to the list of library paths in the **Library path** panel so that the new and edited materials can be made available for all your **Airpak** projects. See Section 6.3.5 for details.

This allows you to create new materials or edit previously defined materials using the **Materials** panel. If a material is defined in the global materials file and also in a user-defined file, **Airpak** will take the definition of the material from the first file that it loads and will ignore any subsequent redefinitions of the material. The order in which the files are loaded is the same as the order in which they are specified in the **Library path** panel (see Section 6.3.5).

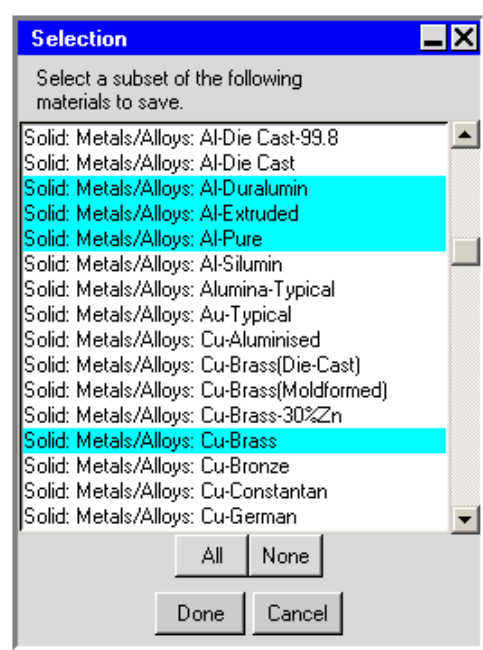



Figure 7.7.7: The Selection Panel for Saving Materials

- Create a new library using the Library path panel as described in Section 6.3.5.

Once you have created a new library, the name of the library will appear under the Libraries node in the Model manager window. To add new or previously-defined materials to a new library, select the material item and drag it into the new library node. This will create a new materials file. If you drag any new materials into the same library, the materials file will be appended with the new materials.

7.7.7 Deleting a Material

If there are materials in your local materials library that you no longer need, you can easily delete them. Select the material under the Materials (or Model) node and click on the  button in the Object modification toolbar or click Delete in the Materials panel (e.g., Figure 7.7.2). The selected material will be removed from your model.

7.7.8 Defining Properties Using Velocity-Dependent Functions

You can define the thermal conductivity of a fluid in Airpak to be a function of velocity. The thermal conductivity of a fluid can be defined using a power law form:

$$k = Bv^n \quad (7.7-1)$$

where k is the thermal conductivity of the fluid, v is the velocity, and B and n are constants.

To define the thermal conductivity of the fluid as velocity-dependent, enable the check box next to **Conductivity** and click the adjacent **Edit** button for a fluid material in the **Materials** panel. This will open the **Temperature or velocity dependent fluid conductivity** panel. Select **Velocity** at the top of the panel (Figure 7.7.8).

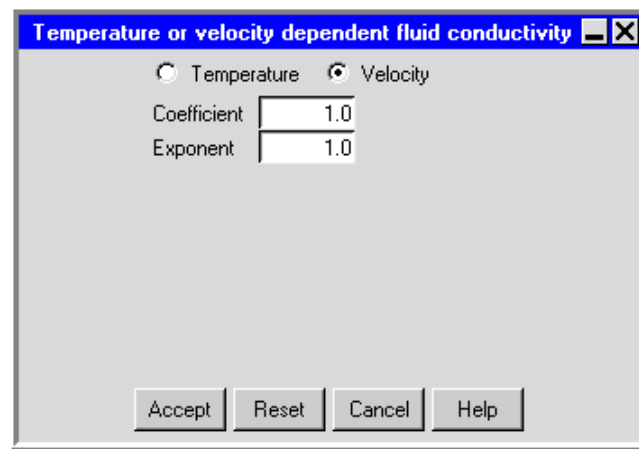


Figure 7.7.8: The Temperature or velocity dependent fluid conductivity Panel
Showing the Velocity-Dependent Inputs

Specify the **Coefficient** (B in Equation 7.7-1) and the **Exponent** (n in Equation 7.7-1) and click **Accept** to define the velocity-dependent thermal conductivity of the fluid.

i You must specify these inputs in SI units.

7.7.9 Defining Properties Using Temperature-Dependent Functions

Many material properties in **Airpak** can be defined as functions of temperature. For most properties, you can define a constant, linear, or piecewise-continuous function of temperature. To define a property as temperature-dependent, enable the check box next to the property and click the adjacent **Edit** button in the **Materials** panel. This will open the **Temperature dependent parameter** panel. Figure 7.7.9 shows an example of a temperature-dependent parameter panel (the **Temperature dependent solid specific heat** panel).

Note that if you are defining a temperature-dependent thermal conductivity for a fluid, you will need to select the **Temperature** option at the top of the **Temperature or velocity dependent fluid conductivity** panel.

There are three options for specifying the temperature dependence of a parameter:

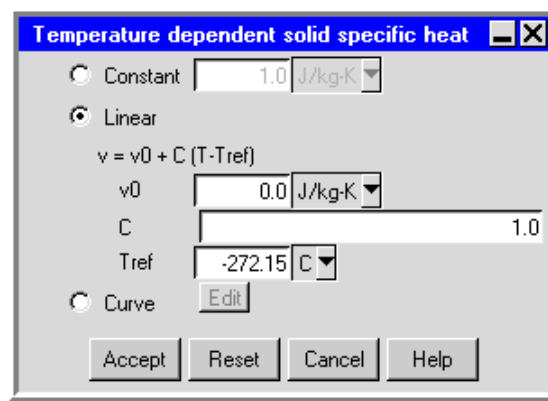


Figure 7.7.9: Example of a Temperature dependent Parameter Panel

Constant allows you to define the property as a constant.

Linear allows you to define the property in terms of a linear equation:

$$v = v_0 + C(T - T_{\text{ref}}) \quad (7.7-2)$$

where v_0 (**v0**) is a reference value, C (**C**) is a constant, and T_{ref} (**Tref**) is the reference temperature.

Curve allows you to define the property as a curve consisting of piecewise-continuous line segments. **Airpak** allows you to describe the curve either by positioning a series of points on a graph using the **Temperature/value curve** window or by specifying a list of temperature/value coordinate pairs using the **Curve specification** panel. These options are described below.

Using the Temperature/value curve Window

You can specify a piecewise linear variation of a property with temperature using the **Temperature/value curve** graphics display and control window (Figure 7.7.10). To open the **Temperature/value curve** window, select **Curve** in the **Temperature dependent** parameter panel (Figure 7.7.9) and click on **Edit**. Select **Graph editor** from the resulting list.

The following functions are available for creating, editing, and viewing a curve:

- To create a new point on the curve, click on the curve with the middle mouse button.
- To move a point on the curve, hold down the middle mouse button while positioned over the point, and move the mouse to the new location of the point.

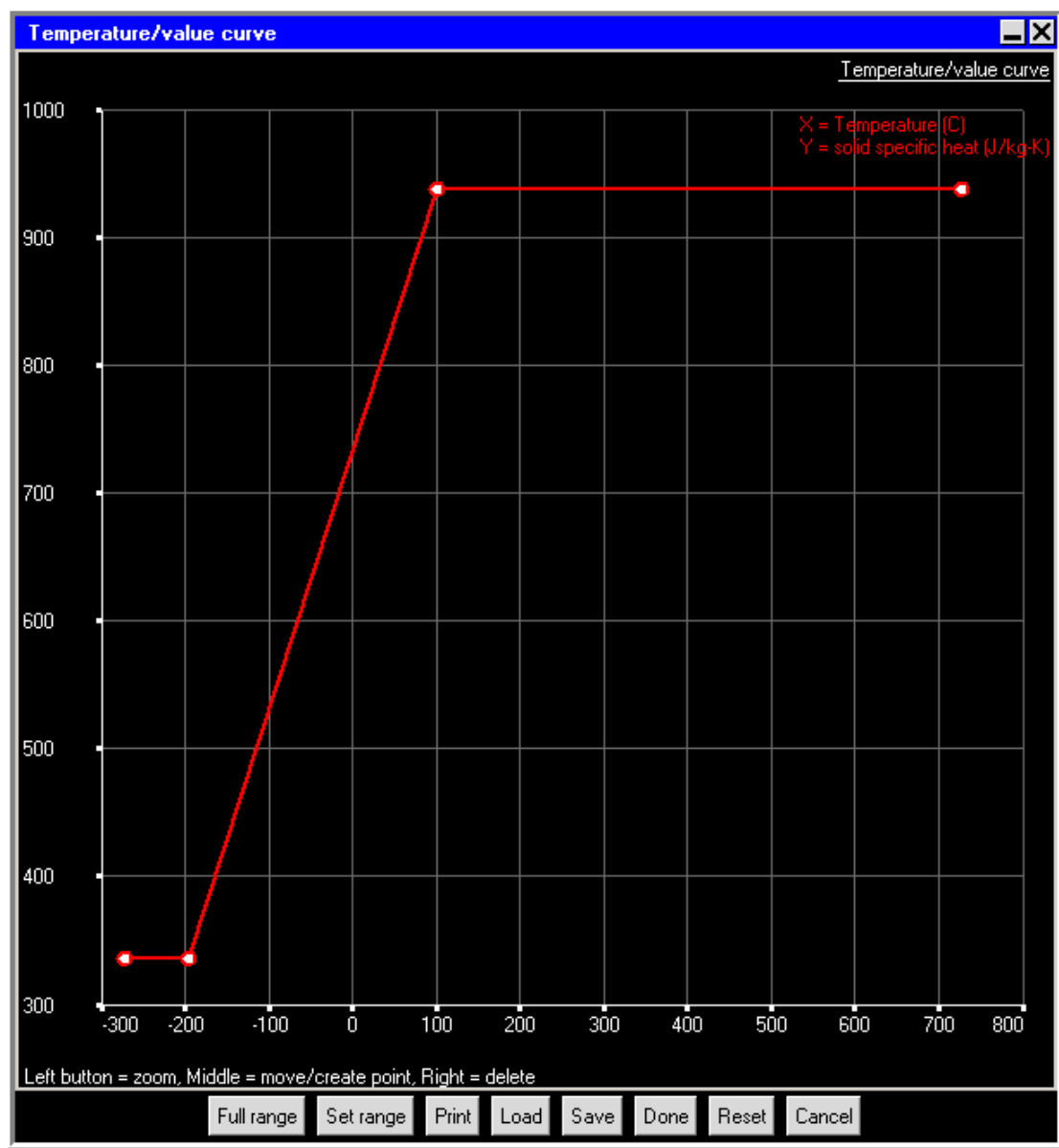


Figure 7.7.10: The Temperature/value curve Graphics Display and Control Window

- To delete a point on the curve, click the right mouse button on the point.
- To zoom into an area of the curve, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the **Temperature/value curve** window, with appropriate changes to the axes. After you have zoomed into an area of the model, click on **Full range** to restore the graph to its original axes and scale.
- To set the minimum and maximum values for the scales on the axes, click on **Set range**. This will open the **Set range** panel (Figure 7.7.11).

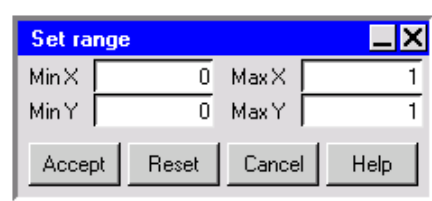


Figure 7.7.11: The **Set range** Panel

Enter values for **Min X**, **Min Y**, **Max X**, and **Max Y** and click **Accept**.

- To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.
- To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

You can use the **Print** button to print the curve. See Section 3.5 for details on saving hardcopy files.

Click **Done** when you have finished creating the curve; this will store the curve and close the **Temperature/value curve** graphics display and control window. Once the curve is defined, you can view the pairs of coordinates defining the curve in the **Curve specification** panel (see Figure 7.7.12 for the pairs of coordinates for the curve shown in Figure 7.7.10).

Using the Curve specification Panel

You can define a piecewise linear variation of a property with temperature using the **Curve specification** panel (Figure 7.7.12). To open the **Curve specification** panel, select **Curve** in the **Temperature dependent parameter** panel (Figure 7.7.9) and click on **Edit**. Select **Text editor** from the resulting list.

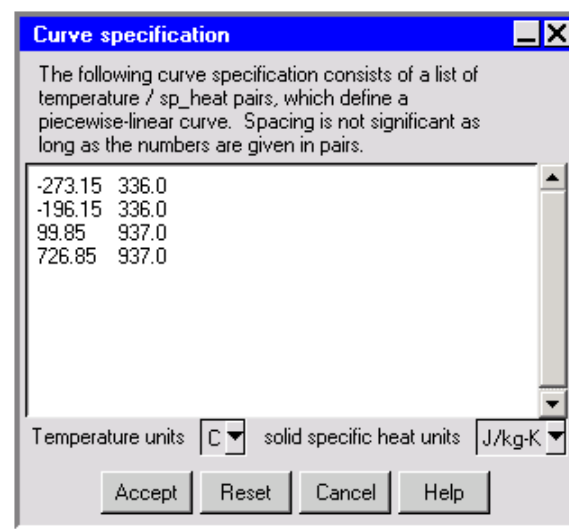


Figure 7.7.12: The Curve specification Panel

To define a curve, specify a list of coordinate pairs in the **Curve specification** panel. It is important to give the numbers in pairs, but the spacing between numbers is not important. Click **Accept** when you have finished entering the pairs of coordinates; this will store the values and close the **Curve specification** panel.

Once the pairs of coordinates have been entered, you can view the curve in the **Temperature/value curve** graphics display and control window. See Figure 7.7.10 for the curve for the values shown in Figure 7.7.12.

7.8 Custom Assemblies

In HVAC models, it is not uncommon to encounter cases where a given combination of modeling objects appears in more than one model. It may be necessary, for example, to model the use of a standard diffuser in conjunction with several different types of rooms and/or other indoor equipment. **Airpak** allows you to create the standard assembly once, and then use it in other models.

Assemblies are collections of **Airpak** objects (e.g., vents, fans, blocks) that have been defined together as a group (see Section 7.6 for information on grouping objects) and stored as a single unit.

7.8.1 Creating and Adding an Assembly

You can create an assembly to be used in your **Airpak** model using one of the following methods:

- Create a group containing all the objects that you want to be in the assembly and then create the assembly from the group. See Section 7.6 for details on creating groups.
- Create an assembly first and then add individual objects or groups to the assembly.

Creating an Assembly From a Group of Objects

To create an assembly from an existing group of objects, select **Create assembly** in the **Edit**→**Current group** menu. The objects in the group will be added as items under the new assembly node in the **Model manager** window (Figure 7.8.1).

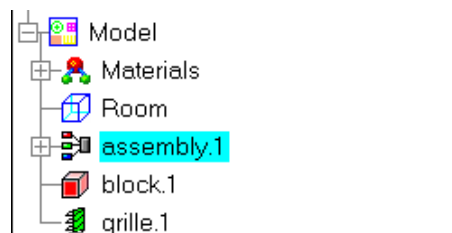



Figure 7.8.1: An Assembly Node in the **Model manager** Window

Alternatively, you can create a new assembly by right-clicking on the group item under the **Groups** node in the **Model manager** window and selecting **Create assembly** in the pull-down menu.

Creating an Assembly Item Without Using a Group of Objects

To create an empty assembly item under the **Model** node in the **Model manager** window (Figure 7.8.1), click on the  button in the **Object creation** toolbar. If the **Assemblies** panel is already open, you can also click on **New** in the **Assemblies** panel to create a new assembly.

Alternatively, you can use the <Control> and <Shift> keys in combination with the mouse to select multiple objects in the **Model manager** window (see Section 2.2.2). To create an assembly from the selected items, right-click on one of the items and select **Create assembly** in the pull-down menu.


Adding Objects to an Assembly

To add objects to an existing assembly, use one of the following methods:

- Select one or more object items in the **Model manager** window and drag them into the desired assembly node.
- Select one or more objects in the graphics window by holding down the <Shift> key and dragging open a selection box around the objects with the left mouse button. In the **Model manager** window, drag the highlighted item(s) into the desired assembly node.
- Right-click on the assembly item in the **Model manager** window and select **Create** in the first pull-down menu. In the subsequent pull-down menu, select the type of object you wish to add to the assembly. See Chapters 8–19 for information about the different objects in **Airpak**.

7.8.2 Editing Properties of an Assembly

To edit properties of an assembly item, use the following procedure:

1. Click on the  button in the **Object modification** toolbar or right-click on the assembly item in the **Model manager** window and select **Edit object** in the pull-down menu. This will open the **Assemblies** panel (Figures 7.8.2 and 7.8.3).
2. Change the description of the assembly, if required, in the **Info** tab of the **Assemblies** panel. Changing the description of an assembly is the same as changing the description of an object. See Section 7.4.1 for details.
3. Change the graphical style of the assembly, if required. Changing the graphical style of an assembly is the same as changing the graphical style of an object. See Section 7.4.2 for details.

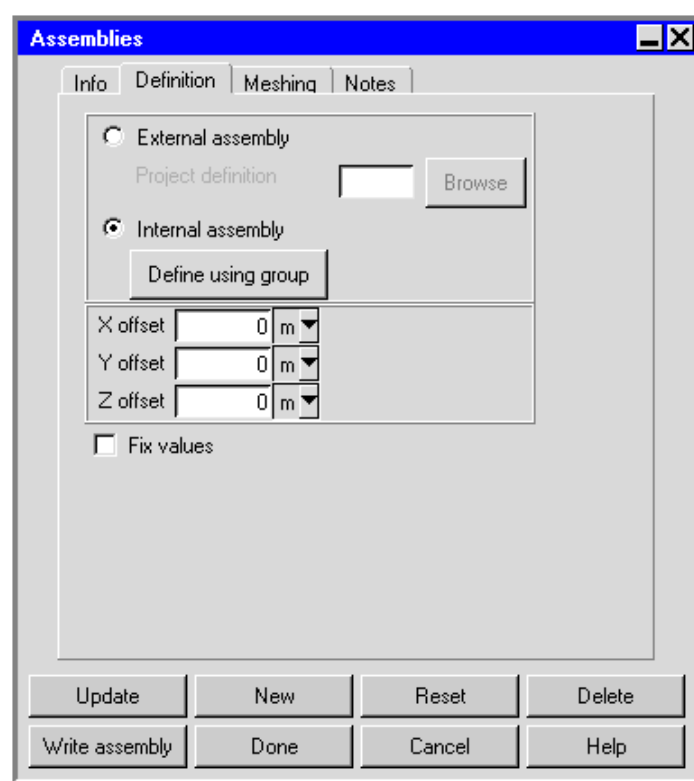


Figure 7.8.2: The Assemblies Panel (Definition Tab)

4. In the **Definition** tab, select the assembly to be added to the model (Figure 7.8.2). There are two options:

External Assembly allows you to use an assembly that was created in a previous **Airpak** session. Specify the name of the project where the assembly was created in the **Project definition** text entry field. You can enter your own filename, which can be a full pathname to the file (beginning with a / character on a UNIX system or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started. Alternatively, you can choose a filename by clicking on the square button located next to the **Project definition** text field and then selecting the file in the resulting **File selection** dialog box. See Section 2.1.8 for more information on the **File selection** dialog box.



When you use the external assembly option to define your assembly, the assembly is not copied into your model. **Airpak** creates a link to the project that contains the assembly. To copy the assembly into the model and store it in the model, you should create the link to the external assembly using the **External Assembly** option as described above, then select the **Internal Assembly** option and click **Update**. If the external assembly is not copied

into the current model, all modifications made to the external assembly will be lost.

Internal Assembly allows you to specify a group that is defined in the current model to be used as an assembly. This option is selected by default. To specify a group, click the **Define using group** button and choose a group from the list in the subsequent **Selection** panel.

5. (optional) To translate the assembly by a specified distance from its original position, define the distance of the translation from the original position by specifying an offset in each of the coordinate directions: **X offset**, **Y offset**, and **Z offset**.

Note that if you want to perform a transformation that is different from that provided by the **Assemblies** panel, you can create an initial assembly and then perform additional transformations using the **Move assembly** panel. The **Move assembly** panel is identical to the **Move all objects in model** panel described in Sections 7.2.1 and 7.2.2. You should also note the extra details for moving an object in Section 7.3.7.



If you scale, rotate, mirror, or translate the assembly, these operations will be performed with respect to the assembly's local coordinate system and not the global coordinate system.

6. In the **Meshing** tab, specify whether you want to have the assembly meshed separately from the rest of your **Airpak** model (Figure 7.8.3).
 - (a) To allow **Airpak** to generate a non-conformal mesh, turn on the **Mesh separately** option. See Chapter 26 for details about generating a mesh.
 - (b) If you selected **Mesh separately**, specify the **Slack** distance around the bounding box of the assembly (**Min X**, **Max X**, **Min Y**, **Max Y**, **Min Z**, **Max Z**). The bounding box will be moved outward by the amount of slack that is specified. Additionally, each of the slack distances can be assigned by aligning the edges of the assembly bounding box to existing edges of objects or other assemblies. To do so, follow the procedure described in Section 7.3.8.

For example, to set the slack distance in the **Min X** direction, you would first click **Min X** displayed in orange in the **Meshing** tab of the **Assemblies** panel, then click the edge of the object in the graphics window that you want to align with the **Min X** edge of the assembly bounding box.

- (c) If you selected **Mesh separately**, you can specify the **Minimum gap** that separates objects in the assembly in the **X**, **Y**, and **Z** directions. This specification is used by **Airpak** whenever the distance between two objects inside the assembly is less than this value, but greater than the model's zero tolerance. If you don't specify the **Minimum gap** (i.e., the specified values are equal to 0), the separation values will be inherited from the global mesh settings.

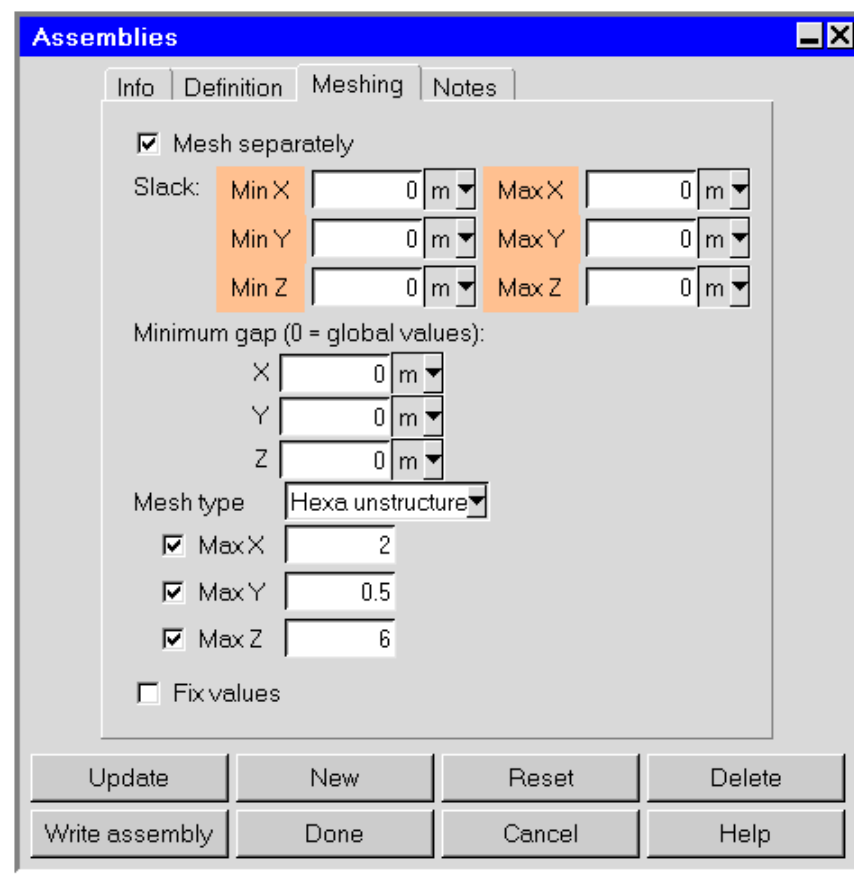


Figure 7.8.3: The Assemblies Panel (Meshing Tab)

- (d) If you selected **Mesh separately**, you can specify the **Mesh type** for the objects in the assembly. Assemblies meshed separately can have a mesh type different from the type used to mesh the outside room.

For example, if the global mesh type has been specified as **Hexa unstructured** in the **Mesh control** panel, a **Tetra** mesh type can be used to mesh the objects inside the assembly. By default, **Airpak** uses the same mesh type specified in the **Mesh control** panel to mesh the objects inside the assembly.

- (e) If you selected **Mesh separately**, you can specify a limit on the element size of the assembly mesh in the x , y , and z coordinate directions. To do so, select **Max X**, **Max Y**, and **Max Z** size specifications and set each one to the desired maximum element length in each direction. If no values are specified, **Airpak** uses the global values specified in the **Mesh control** panel.

7. Specify whether you want to fix the values for the units in the panel. See Section 5.4.2 for details.

8. Click **Update** to position the assembly according to the specifications in the **Assemblies** panel. Click **Done** to position the assembly and close the **Assemblies** panel.

7.8.3 Assembly Viewing Options

There are several ways to view an assembly in your **Airpak** model. Expanding the assembly node in the **Model manager** window will display the objects in the assembly as they would normally look. Closing the assembly node will display only the bounding box of the assembly.


In a more complicated model, you can choose to view only one particular assembly while hiding the rest of your **Airpak** model. To change the display in this manner, right-click on the desired assembly node under the **Model** node in the **Model manager** window and turn on the **View separately** option in the pull-down menu. The assembly node will be temporarily moved up one level in the tree and the **Model** node will be closed, which results in only the assembly being displayed. To return to the default display settings, right-click on the assembly node and turn off the **View separately** option.

7.8.4 Selecting an Assembly

There are two ways to select an assembly:


- Select the name of assembly under the **Model** node in the **Model manager** window using the left mouse button.
- Position the mouse cursor over the assembly in the graphics window, hold down the <Shift> key on the keyboard, and click the left mouse button.

7.8.5 Editing Objects in an Assembly

Once you have created an assembly or read an assembly into your **Airpak** model, you can edit the individual objects in the assembly. To edit an object, select the object item under the assembly node and click on the  button in the **Object modification** toolbar or select **Edit** in the **Edit—>Current object** menu. This will open the **Object** panel, where you can edit any of the properties of any individual object.

Alternatively, you can edit an assembly by right-clicking on the assembly item in the **Model manager** window and selecting **Edit object** in the pull-down menu.


7.8.6 Copying an Assembly

To copy an assembly in your model, select the assembly and click on the  button in the **Object modification** toolbar or select **Copy** in the **Edit—>Current object** menu. This will open the **Copy assembly** panel. The procedure for copying an assembly is the same as copying an object using the **Copy object** panel, as described in Section 7.3.9.

Alternatively, you can copy an assembly by right-clicking on the assembly item in the **Model manager** window and selecting **Copy object** in the pull-down menu.

i If you scale, rotate, mirror, or translate the assembly, these operations will be performed with respect to the assembly's local coordinate system and not the global coordinate system.

7.8.7 Moving an Assembly

To move an assembly, select the assembly and click on the  button in the Object modification toolbar or select **Move** in the **Edit—>Current object** menu. This opens the Move assembly panel. You can apply various transformations to the assembly. The operations in the Move assembly panel are the same as those in the Move all objects in model panel (Figure 7.2.6). The Scale option is described in Section 7.2.1, and the Rotate, Mirror, and Translate options are described in Sections 7.2.2 and 7.3.7.

Alternatively, you can move an assembly by right-clicking on the assembly item in the Model manager window and selecting **Move object** in the pull-down menu.

i If you scale, rotate, mirror, or translate the assembly, these operations will be performed with respect to the assembly's local coordinate system and not the global coordinate system.

7.8.8 Saving an Assembly

To save an assembly to be used in another model, click **Write assembly** in the Assemblies panel (Figure 7.8.2). This will open the Save project panel. See Section 3.4 for information on the Save project panel. You can save the project containing the assembly to any directory, but it is recommended that you create a local assemblies directory where you can save all the assemblies you create.

Alternatively, you can save an assembly as a separate project by right-clicking on the assembly item in the Model manager window and selecting **Save as project** in the pull-down menu. See Section 3.4 for details about saving projects.


7.8.9 Loading an Assembly

To load an assembly that was created in another model, right-click on the assembly item in the Model manager window and select **Load assembly** in the pull-down menu. This will open the Load project panel, which is similar to the Open project panel. See Section 6.2 for information on the Open project panel.

7.8.10 Merging an Assembly With Another Project

To merge an assembly with another project, right-click on the assembly item in the Model manager window and select **Merge project** in the pull-down menu. This will open the Merge project panel. See Section 3.3 for information about using this panel.

7.8.11 Deleting an Assembly

To delete an assembly, select the assembly and click on the  button in the Object modification toolbar, select **Delete** in the **Edit**→**Current object** menu, or click **Delete** in the **Assemblies** panel (Figure 7.8.2). The selected assembly will be removed from the model and from the **Model manager** window.

Alternatively, you can delete an assembly by right-clicking on the assembly item in the **Model manager** window and selecting **Delete** in the pull-down menu.

To recover a deleted assembly, you can undo the delete operation by selecting the **Undo** option in the **Edit** menu. See Section 2.1.2 for more details on using undo and redo operations.

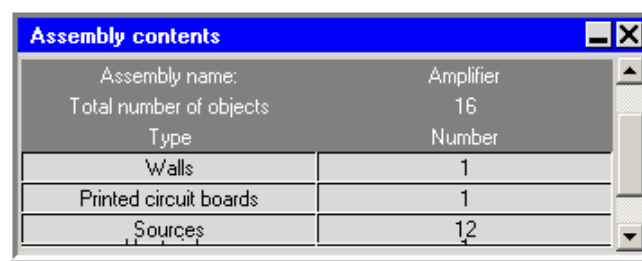
7.8.12 Expanding an Assembly Into Its Components

Once you have created an assembly or read an assembly into your **Airpak** model, you can expand the assembly into its individual components. To expand an assembly, simply expand the assembly node in the **Model manager** window. The assembly will be expanded into its individual components that can be edited separately.

Alternatively, you can expand an assembly into its components by right-clicking on the assembly item in the **Model manager** window and selecting **Open subtree** in the pull-down menu. To view the assembly as a single object, you can close the node or select **Close subtree** in the pull-down menu.

7.8.13 Summary Information for an Assembly

To view a brief summary of the contents of an assembly, right-click the assembly item in the **Model manager** window and select **Summary information** in the pull-down menu. This will open the **Assembly contents** panel, as shown in Figure 7.8.4. This panel lists the total number of objects in the assembly along with the number of objects of each individual object-type.



Assembly contents	
Assembly name:	Amplifier
Total number of objects	16
Type	Number
Walls	1
Printed circuit boards	1
Sources	12

Figure 7.8.4: The Assembly contents Panel

7.8.14 Total Volume of an Assembly

To display the total volume occupied by all the objects in an assembly, right-click the assembly item in the **Model manager** window and select **Total volume** in the pull-down menu. The total volume will be displayed in the **Message** window.

7.8.15 Total Area of an Assembly

To display the total area occupied by all the objects in an assembly, right-click the assembly item in the **Model manager** window and select **Total area** in the pull-down menu. The total area will be displayed in the **Message** window.

7.9 Checking the Design of Your Model

Airpak provides two ways to check your model for design problems: object and material summaries, and design checks. Object and material summaries, and design checks are described in detail below.

7.9.1 Object and Material Summaries

Object and material summaries provide an on-screen catalog of all objects in the model, (including names, descriptions, dimensions, locations, and all physical and thermal properties associated with each object) and any materials you have created or edited. The summary allows you to review the contents of your model as a means of examining the design. To create an object and material summary, select **Summary (HTML)** in the **View** menu. The HTML version of the summary will be displayed in your web browser. Airpak will automatically launch your web browser (Netscape, IE, or Mozilla), as shown in (Figure 7.9.1).

View—>Summary (HTML)

The summary displays a list of all the objects in the model and all the parameters that have been set for each object. The information is grouped by object types (e.g., blocks, fans, vents). For each object type, the number of objects of that type is indicated, along with the **Name** and **Shape** of every object. If specified, other information will be listed, such as **Material**, **Power**, **Radiation**, **Loss specification**, etc. If certain properties (e.g., radiation) are specified, you can view the detailed version of the summary by clicking the appropriate object names or property specifications. For example, if you click **block.3** in the summary shown in Figure 7.9.1, the detailed version of the summary will be displayed as shown in Figure 7.9.2.

7.9.2 Design Checks

Design checks test the model for problems in the design. To perform an automated design check, select **Check model** in the **Model** menu.

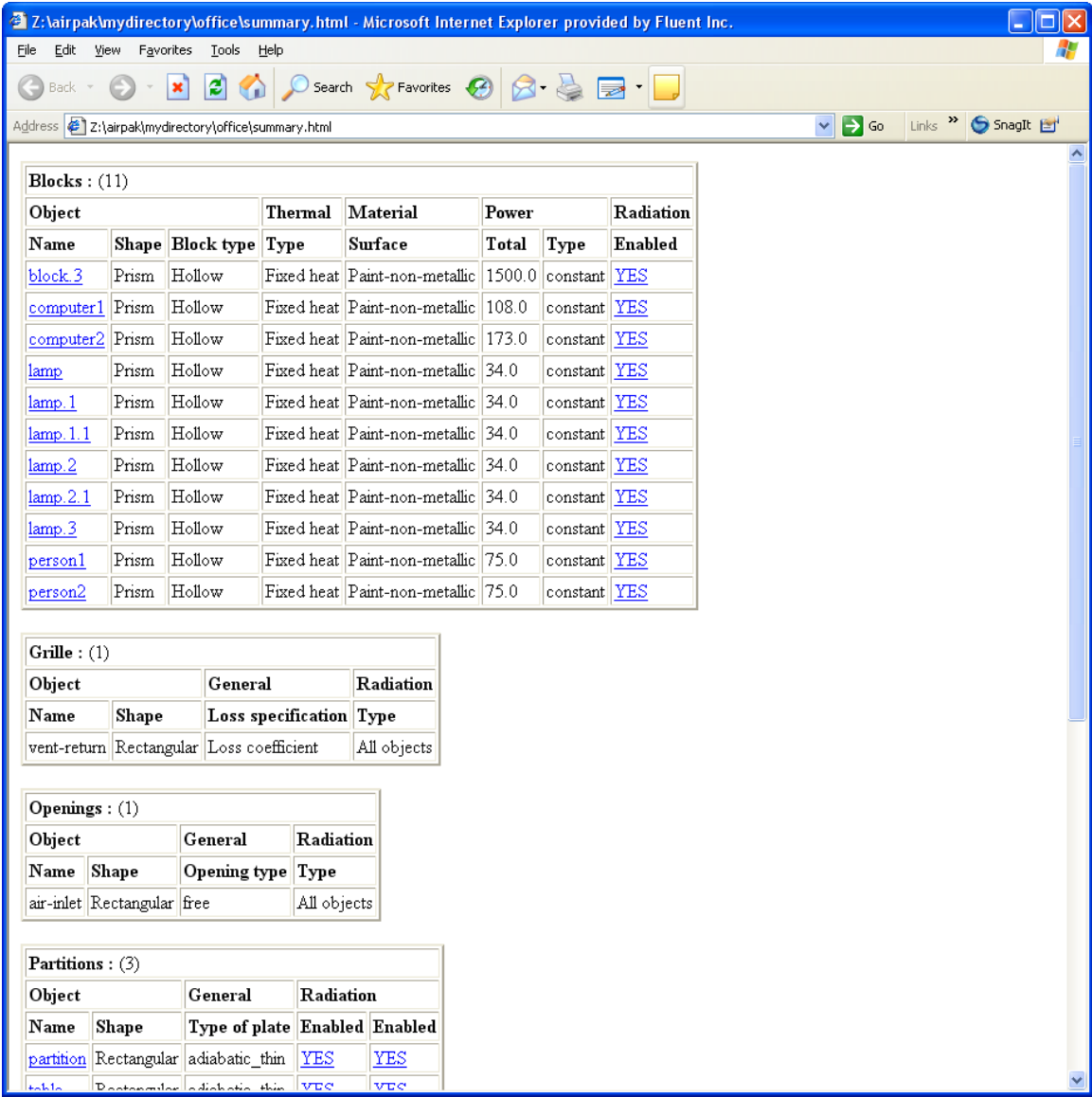


Figure 7.9.1: Model Summary

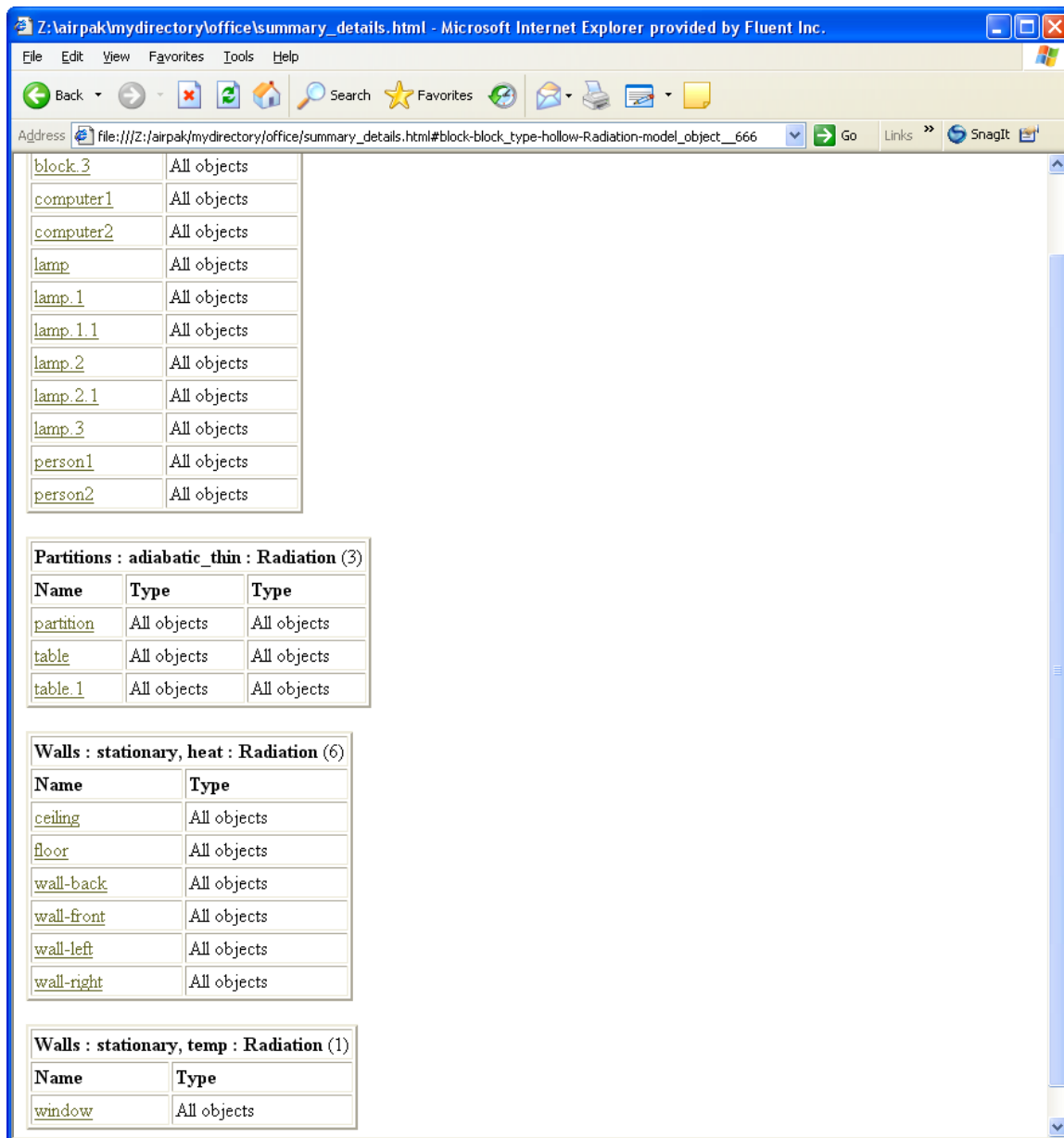


Figure 7.9.2: Detailed Summary

Model—→Check model

In the design check, **Airpak** searches for incidences of overlapping objects, violations of physics (e.g., openings not associated with walls), and unacceptable data (e.g., values out of range for individual specifications).

The results of the design check are reported sequentially in the **Airpak Message** window. Informative messages appear as blue text, while flaws or design errors detected by the check appear as red text. **Airpak** highlights the overlapping sections of all intersecting objects in red in the graphics window. To restore the model colors to their defaults, select **Check model** again.

If **Airpak** detects a large number of flaws in the model, the on-screen report may exceed the size of the **Message** window. To review the entire list of flaws, you can either expand the **Message** window, or print the contents of the **Message** window to a file using the **Log** option in the **Message** window (see Section 2.1.6).

Note that the design check is performed automatically when you generate a mesh for the model.

A block is a three-dimensional modeling object. Block geometries include prism (rectangular box), cylinder, 3D polygon, ellipsoid, elliptical cylinder and 3D CAD. Block types include solid, hollow, and fluid.

Physical, species, and thermal characteristics that need to be specified vary according to the block type. All types of blocks (or individual sides of the blocks) can exchange radiation with other objects in the model. Blocks exist within the room, so any part of their non-contact surfaces may be exposed to the fluid in the room. By default, the no-slip condition for fluid velocity applies at all block surfaces. For turbulent flows, you can specify a roughness parameter.

To configure a solid, hollow, or fluid block in your model, you must specify its geometry (including location and dimensions) and type, as well as its physical and thermal characteristics.

In this chapter, information about the characteristics of a block is presented in the following sections:

- Section 8.1: Geometry, Location, and Dimensions
- Section 8.2: Block Type
- Section 8.3: Surface Roughness
- Section 8.4: Physical and Thermal Specifications
- Section 8.5: Block-Combination Thermal Characteristics
- Section 8.6: Adding a Block to Your Airpak Model

8.1 Geometry, Location, and Dimensions

Block location and dimension parameters vary according to block geometry. Block geometries include prism, cylinder, 3D polygon, ellipsoid, elliptical cylinder, and 3D CAD. These geometries are described in Section 7.4.4. Note that network blocks can only have prism geometries.

8.2 Block Type

There are three types of **Airpak** blocks: solid, hollow, and fluid. Although they share certain specifications, each is unique in purpose and characteristics:

- Solid blocks represent actual solid objects and can possess physical and thermal characteristics such as density, specific heat, thermal conductivity, and total heat flux. **Airpak** considers the interior of a solid block to be part of the computational domain and includes the block internal temperature distribution as part of the model solution.
- Hollow blocks represent three-dimensional regions of the model for which only side characteristics are important. **Airpak** does not mesh or solve for temperature or flow within regions bounded by the sides of a hollow block. Hollow-block surfaces can be specified as adiabatic (impervious to heat flow) or as possessing a fixed, uniform temperature or heat flux.
- Fluid blocks are regions of the model where fluid properties can be specified independently of those specified for the **Default fluid** in the **Basic parameters** panel (see Section 6.4.7). Individual side parameters for fluid blocks are specified in the same manner as those for solid and hollow blocks. If individual side parameters are specified on a side of a fluid block, a zero thickness wall is defined for that side of the block.

8.3 Surface Roughness

In fluid dynamics calculations, it is common practice to assume that boundary surfaces are perfectly smooth. In laminar flow, this assumption is valid, because the length scales of typical rough surfaces are much smaller than the length scales of the flow. In turbulent flow, however, the length scales of the flow eddies are much smaller than laminar length scales; therefore, it is sometimes necessary to account for surface roughness. Surface roughness acts to increase resistance to flow, leading to higher rates of heat transfer.

Airpak assumes, by default, that all surfaces of a block are hydrodynamically smooth, and applies standard no-slip boundary conditions. For turbulent-flow simulations in which roughness is significant, however, you can specify a roughness factor for the entire block or (for solid, hollow, and fluid blocks) each individual side of the block. The roughness factor is defined as part of the properties of the surface material specified for the block. The purpose of the roughness factor is to approximate the average height of the surface texture on the block.

8.4 Physical and Thermal Specifications

Block physical and thermal specifications vary according to block type:

- Solid block specifications include parameters related to material properties.
- Hollow block specifications, on the other hand, include only parameters related to the block surface itself, such as temperature, heat flux, species, and whether or not the surface is adiabatic.
- Fluid block specifications relate to the properties of the fluid within the block.

In addition to specifying parameters for the entire block, **Airpak** allows you to specify parameters for each individual side of the block. Side-specific parameters include those related to surface thermal characteristics (e.g., temperature and heat flux) as well as radiation.

If you specify the side of a block as adiabatic, **Airpak** treats it as impervious to heat flow in the direction normal to its surface. When a block side is specified as fixed temperature, **Airpak** assigns a constant temperature to the side. If you specify a block side as fixed heat, **Airpak** assumes that it emits or absorbs heat uniformly at a constant rate. You can specify a fixed heat flux in terms of either total heat flux or heat flux per unit area. If the side of the block is subject to solar loading, you can calculate the solar radiation flux using the solar flux macro (described in Section 22.5) and then use the value of the solar radiation flux as the specified heat flux for the side of the block.

8.5 Block-Combination Thermal Characteristics

Blocks can be combined with other blocks and objects to achieve a wide variety of complex shapes. In combining blocks to create custom objects, special care must be taken to ensure that the assignment of thermal characteristics achieves the intended representation of the object being modeled. The following examples illustrate the general rules that govern heat transfer in objects created by combining two or more objects.

8.5.1 Blocks with Coincident Surfaces

When a model involves two blocks with coincident surfaces (see Figure 8.5.1), **Airpak** employs two basic rules to govern the thermal characteristics in the region of contact:

- If both blocks are specified as conducting solid blocks, **Airpak** computes the amount of heat transferred between the blocks. To apply a thermal resistance between the blocks at the contact surface (representing, for example, a coating layer between the blocks), locate and specify a partition at the region of coincidence (see Chapter 14).
- When one (or both) of the blocks is specified as hollow, no heat transfer occurs across the coincident surface. Figure 8.5.2 shows an example configuration where Block A is a non-conducting, solid block and Block B is a hollow block with a specified constant heat flux. Block A insulates the coincident region with respect

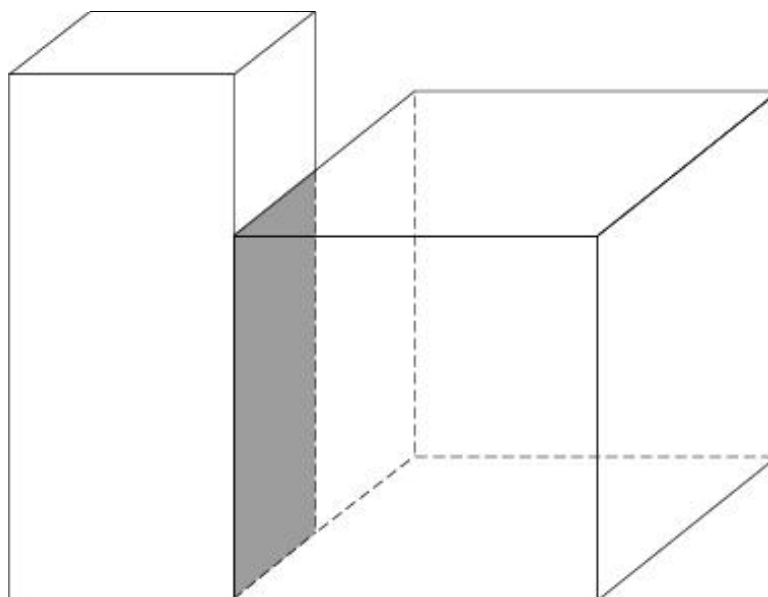


Figure 8.5.1: Blocks with Coincident Surfaces

to heat flow, so heat is transferred away from Block B only on those surfaces exposed to the fluid in the room.

8.5.2 Blocks with Intersecting Volumes

Blocks with intersecting volumes follow rules similar to those outlined above for blocks with coincident surfaces. Intersecting blocks, such as those shown in Figure 8.5.3, however, differ from blocks with coincident surfaces in that the characteristics of the intersecting region are those of the last block created.

Consider, for example, a block configuration such as that shown in Figure 8.5.4, where Block B is created after Block A. The intersecting region (shaded) possesses the characteristics and parameters of Block B, regardless of which block is solid, hollow, or fluid in type, because Block B was the last block created.

If Block A is specified as a conducting solid block with a specified heat flux and Block B is a hollow block, **Airpak** distributes the entire specified heat flux for Block A only in its non-intersecting volume. If, on the other hand, Block A is hollow and Block B is a conducting solid block with a specified heat flux, **Airpak** distributes the power for Block B throughout its entire volume, including the coincident region. If both blocks are conducting solid blocks, **Airpak** distributes specified heat flux and calculates temperature distributions throughout the entire volumes of both blocks, using the thermal properties of Block B in the coincident region.

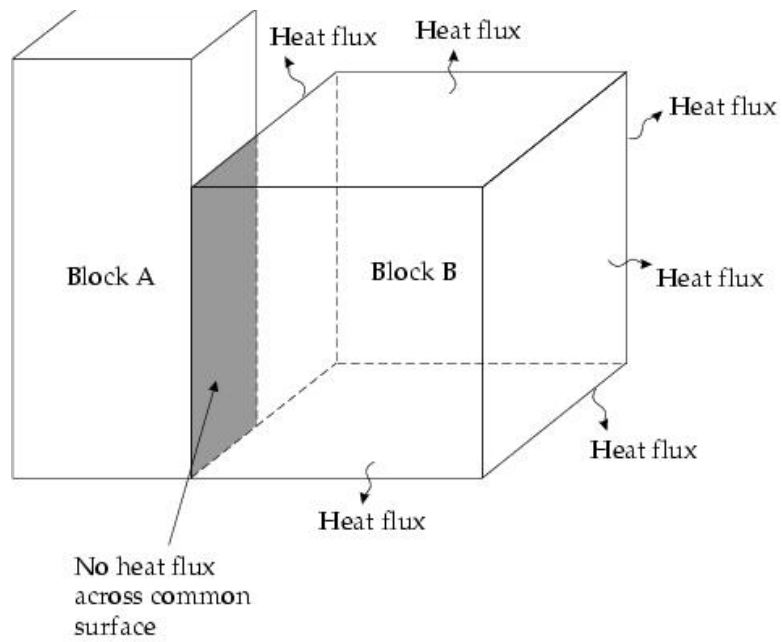


Figure 8.5.2: Heat Flux for Blocks with Coincident Surfaces

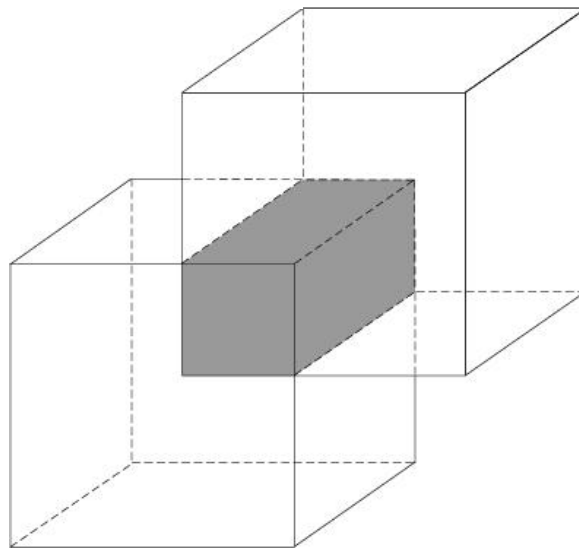


Figure 8.5.3: Blocks with Intersecting Volumes

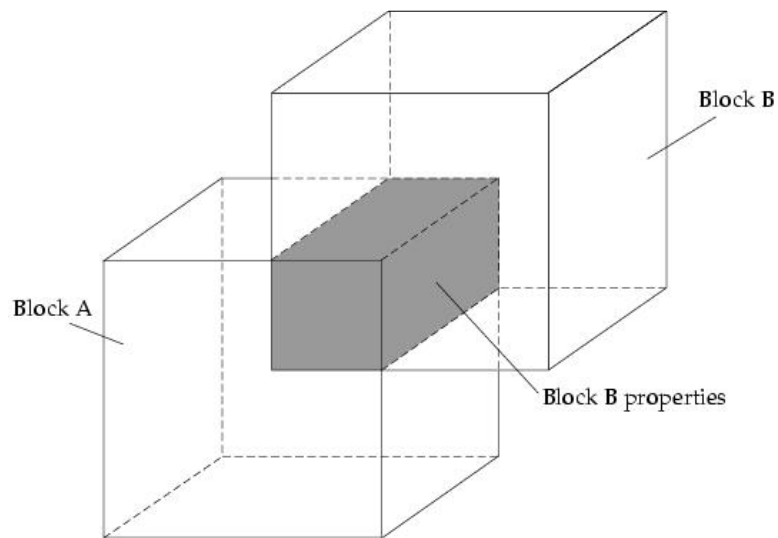


Figure 8.5.4: Intersecting-Volume Properties

A Block and a Partition with Coincident Surfaces

When a block is positioned on a partition (see Figure 8.5.5), two rules govern the manner in which the thermal characteristics of the partition affect those of the block:

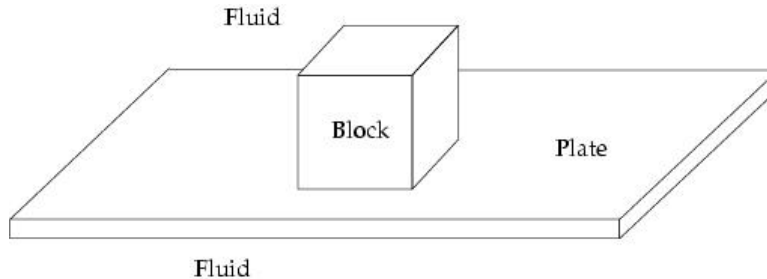


Figure 8.5.5: Block Positioned on a Partition

- If the block is a conducting solid block and the partition has a non-zero thickness, the partition can be considered as a conducting solid block, and the rules governing blocks with coincident surfaces apply. If the partition is specified with zero thickness, there is no heat transfer from the block to the partition.
- If the block is specified as fixed heat or fixed temperature, it can transfer heat to the partition but cannot receive heat from the partition.

8.5.3 A Block and an Intersecting Partition

Blocks and partitions can be combined in a number of ways to produce complex objects. For example, a partition can be embedded in a block to introduce anisotropic conductivity. When a block and a partition intersect (see Figure 8.5.6) two rules govern the manner in which the thermal characteristics of the partition affect those of the block:

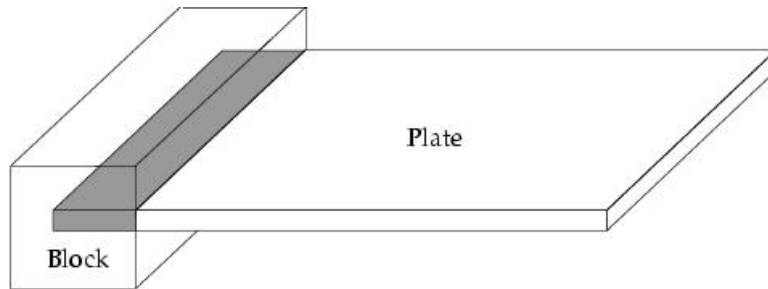


Figure 8.5.6: Partition Intersecting a Block

- The thermal characteristics of a partition intersecting a conducting solid block override those of the block, regardless of which object is created first. A partition with zero conductivity, for example, acts as an insulator within the block.
- If the block is specified as adiabatic, fixed heat, or fixed temperature, the presence of the partition has no effect (i.e., the portion of the partition within the block is ignored).

8.5.4 Blocks Positioned on an External Wall

When a block is located on an external wall (see Figure 8.5.7), the thermal interaction between the wall and block is governed by the following rules:

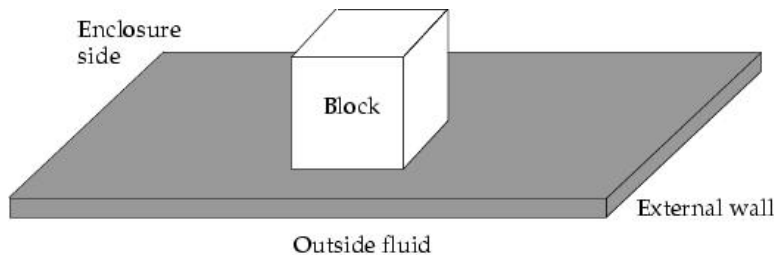


Figure 8.5.7: Block on an External Wall

- A conducting solid block on an external wall exchanges heat with the wall. If the block is specified as dissipating power, the side coincident with the wall transfers its heat to the wall.
- If a block in contact with a wall is specified as adiabatic, there is no heat transfer between the block and the wall. If the block is specified as fixed heat or fixed temperature, it can transfer heat to the wall but cannot receive heat transferred from the wall.

8.5.5 Cylinder, Polygon, Ellipsoid, or Elliptical Cylinder Blocks Positioned on a Prism Block

When a prism block is in surface contact with a cylinder, polygon, ellipsoid, or elliptical cylinder block (see Figure 8.5.8), the rules outlined above governing blocks with coincident surfaces apply:

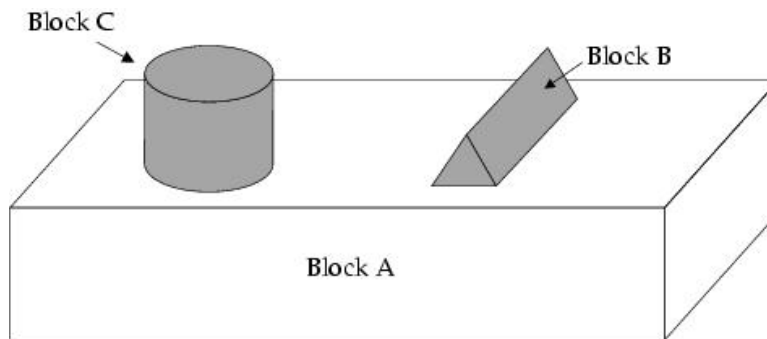




Figure 8.5.8: Multiple Blocks in Contact

- If both blocks with coincident surfaces are designated as conducting solid blocks, **Airpak** calculates the amount of heat transferred between the blocks.
- If one (or both) blocks is non-conducting, no heat transfer occurs between the blocks, and **Airpak** computes heat transfer only from those surfaces exposed to the enclosure fluid as specified for each block. In Figure 8.5.8, for example, if Block A is a nonconducting solid block and Blocks B and C are conducting blocks, heat is not allowed to flow from Block B to Block C (or vice versa) through Block A. Furthermore, the entire heat flux (if any) specified for Blocks B and C is transferred to the fluid through the non-coincident surfaces.

8.6 Adding a Block to Your Airpak Model

To include a block in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Blocks panel, shown in Figures 8.6.1 and 8.6.2.

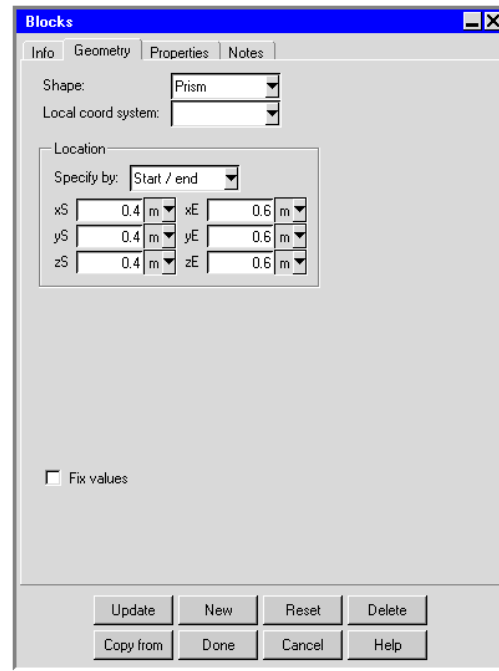


Figure 8.6.1: The Blocks Panel (Geometry Tab)

The procedure for adding a block to your Airpak model is as follows:

1. Create a block. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the block, if required. See Section 7.4.1 for details.
3. Change the graphical style of the block, if required. See Section 7.4.2 for details.
4. In the **Geometry** tab, specify the geometry, position, and size of the block. There are five different kinds of geometry available for blocks in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.
5. In the **Properties** tab, specify the type of the block by selecting **Solid**, **Hollow**, or **Fluid** next to **Block type**. The lower part of the panel will change depending on your selection of **Block type**.

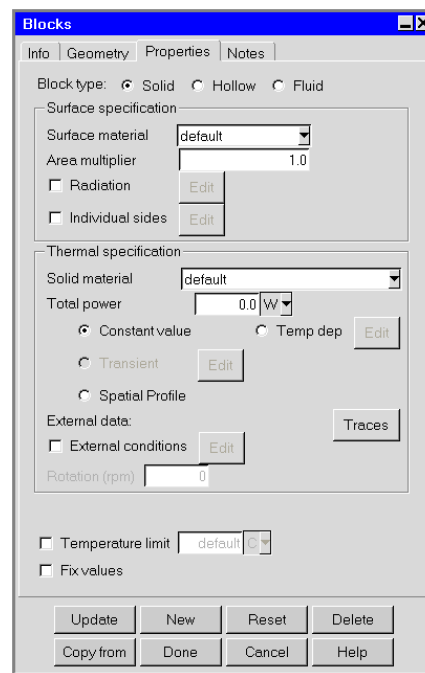
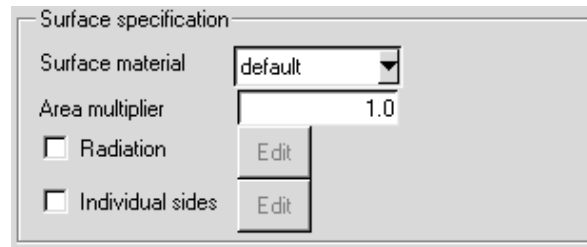


Figure 8.6.2: The Blocks Panel (Properties Tab)

6. Define the **Surface specification** for a block of the selected type. The options for surface specification are described below.
7. Define the **Thermal specification** (or **Fluid specification**) for a block of the selected type. The options for thermal (or fluid) specification are described in Section 8.6.2.
8. (hollow blocks only) Specify the concentration of species at the surface of the block, if required. You can input the concentrations of the species at the surface of the block using the **Species concentration** panel. To open this panel, select **Species** in the **Blocks** panel and click **Edit**. See Chapter 21 for details on modeling species transport.

8.6.1 User Inputs for the Block Surface Specification

Define the **Surface specification** for a solid, hollow, or network block. The **Surface specification** can optionally be included for a fluid block also. The surface specification allows you to specify thermal and physical surface properties for the block. The user inputs for the surface specification are shown below.



1. Specify the **Surface material** to be used for the block. This material defines the roughness and emissivity of the surface of the block. By default, the **Surface material** is specified as **default** for each type of block. This means that the material specified on the surface of the block is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **Surface material** for the block, select a material from the **Surface material** drop-down list. The surface roughness parameters and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.
2. Choose the options to be included for the specification of the surface. The following options are available:
 - (solid and hollow blocks only) Specify the value of the **Area multiplier**. This is the factor by which the surface area of all sides of the block will be increased (e.g., for modeling serrated surfaces), and is set to 1 by default.
 - Select **Radiation** to specify radiation as an active mode of heat transfer to and from the block. This option is available if you have selected **On** next to **Radiation** in the **Basic parameters** panel. You can modify the default radiation characteristics of the block (e.g., the view factor) by using the **Radiation specification** panel. To open this panel, select **Radiation** in the **Properties** tab of the **Blocks** panel and then click **Edit**. See Chapter 22 for details on radiation modeling.
 - Select **Individual sides** to specify thermal and physical surface properties for individual sides of the block using the **Individual side specification** panel (Figure 8.6.3). To open this panel, select **Individual sides** under **Surface specification** in the **Blocks** panel and then click **Edit**.

The steps for specifying thermal and physical properties for individual sides of the block are as follows:

- (a) Select the side of the block where you want to define individual properties by selecting one of the options under **Which side**. The **Which side** options vary according to block geometry as shown in Table 8.6.1.

Note that the numbers of the sides of a polygon block are defined relative to the lowest numbered adjacent vertex. Side 1, for example, is the side

Individual side specification

Which side:

☒ Min X: ☐ Min Y: ☐ Min Z:
☐ Max X: ☐ Max Y: ☐ Max Z:

Surface material: default

☒ Thermal properties:

☒ Fixed heat: ☒ Total power: 0.0 W
☐ Power / area: 0.0 W/m2

☐ Fixed temperature: ambient C

☐ External conditions: Heat transfer coefficient: 1.0 W/K-m2
Ref. temperature: ambient C

Area multiplier: 1.0

☒ Additional resistance: Conductance
Conductance: 1.0 W/C

☐ Conducting thin side

☒ Radiation properties:

Radiate to:

☒ Reference temp: ambient C
View factor: 1.0

☐ All objects

☐ Selected objects: Select

Accept Reset Cancel Help

Figure 8.6.3: The Individual side specification Panel

Table 8.6.1: Which side Options for Block Geometries

Geometry	Which side options
Prism	Min X, Max X, Min Y, Max Y, Min Z, Max Z
Cylinder	Bottom, Top, Sides
Polygon	Bottom, Top, Side 1, Side 2, Side 3, etc.
Ellipsoid	Sides, Inner
E. cylinder	Bottom, Top, Sides

of the block located between vertices 1 and 2. Side 2 is the side of the block located between vertices 2 and 3.

- (b) Specify the **Surface material** to be used for the currently selected side of the block. This material defines the roughness and emissivity of the surface of the block. By default, the **Material** is specified as **default**. This means that the material specified on the currently selected surface is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **Material** for an individual side, select a material from the **Material** drop-down list. See Section 7.7 for details on material properties.
- (c) Specify the **Thermal properties** for the currently selected side of the block. The following options are available:

Fixed heat specifies a constant, uniform heat flux into or out of the block surface. You can define the heat flux as either per unit surface area (**Power / area**) or as a fixed value (**Total power**) for the currently selected side of the block.

Fixed temperature specifies a constant uniform temperature for the currently selected side of the block. The value of the **ambient** temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

External conditions (solid blocks only) allows you to specify a **Heat transfer coefficient** and an **Ambient temp** for the volume of space on the immediately exterior of the currently selected side of the solid block.

Internal conditions (hollow blocks only) allows you to specify a **Heat transfer coefficient** and an **Ambient temp** for the volume of space on the immediate interior of the currently selected side of the hollow block.

Area multiplier (solid or hollow blocks only) allows you to specify a factor by which the surface area of the currently selected side of the block will be increased (e.g., for modeling serrated surfaces).

Additional resistance (solid or hollow blocks only) allows you to specify an additional resistance to heat transfer for the currently selected side of the block. You can specify the resistance by selecting one of the following options from the drop-down list and entering appropriate values:

Conductance allows you to specify a value for the **Conductance** ($= hA$). The inverse of this value will be used to calculate the additional resistance of the current side of the partition.

Thermal resistance allows you to specify a value for the thermal resistance ($\text{Thermal resistance} = \frac{1}{hA}$). This value will be used to calculate the additional resistance of the current side of the partition.

Thermal impedance allows you to specify a value for the thermal impedance ($\text{Thermal impedance} = \frac{1}{h}$). This value will be used to compute the additional resistance of the current side of the partition.

Thickness allows you to specify a value for the **Thickness** of a specified **Solid material**. The values of the thickness and the thermal conductivity of the solid material will be used to calculate the additional resistance of the current side of the partition.

Additionally, you can specify the current side to be a **Conducting thin side**. If this option is enabled, the additional resistance to heat transfer can be applied anisotropically according to the thermal conductivities specific to each direction.

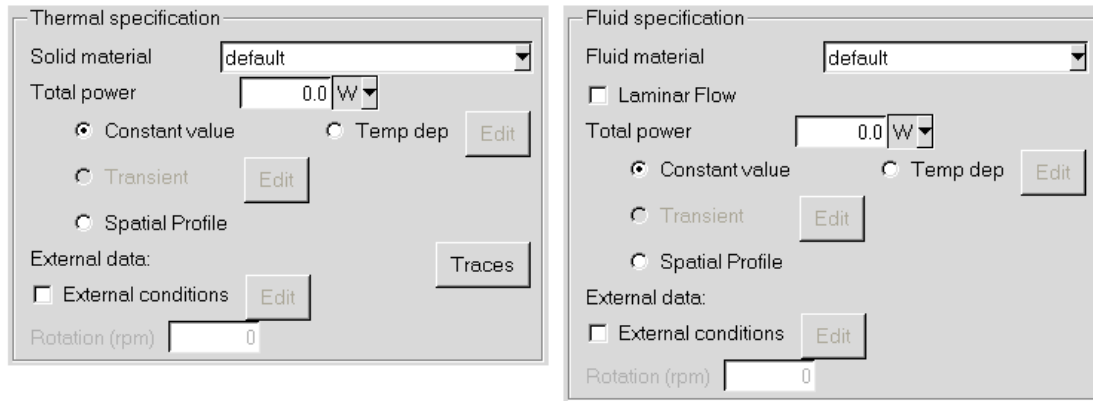
- (d) Specify the **Radiation properties** for the currently selected side of the block. This option is available if you have selected **On** next to **Radiation** in the **Basic parameters** panel and you have not selected the **Discrete ordinates radiation model** option. The properties that can be specified for individual sides are the same as those that can be specified for the whole block. See Chapter 22 for details on radiation modeling.
- (e) Click **Accept** in the **Individual side specification** panel.
- (f) Repeat these steps for each side of the block.

8.6.2 User Inputs for the Block Thermal Specification

Define the **Thermal specification** for the block. The thermal specification allows you to specify thermal properties for the block.

Solid and Fluid Blocks

The user inputs for the thermal specification for a solid or fluid block are shown below.



1. Specify the **Solid material** for a solid block or the **Fluid material** for a fluid block. By default, both are specified as **default**. This means that the solid and fluid materials specified are defined in the **Basic parameters** panel (see Section 6.4.7). To change the **Solid material** or **Fluid material**, select a new material from the relevant material drop-down list. See Section 7.7 for details on material properties.
2. (fluid blocks only) Specify whether the interior of the block is to be modeled as a laminar zone by toggling the **Laminar Flow** option. Note that this option is only available when one of the turbulence models has been enabled in the **Basic parameters** panel.
3. Specify the total power dissipated by the solid or fluid block. There are five options for specifying the total power:

Constant value allows you to specify a constant value for the **Total power**.

Temp dep allows you to specify the power as a linear function of temperature. Select **Temp dep** under **Total power** in the **Blocks** panel, and enter a value of the **Total power**. Click **Edit** next to **Temp dep**; this opens the **Temperature dependent power** panel (Figure 8.6.4).

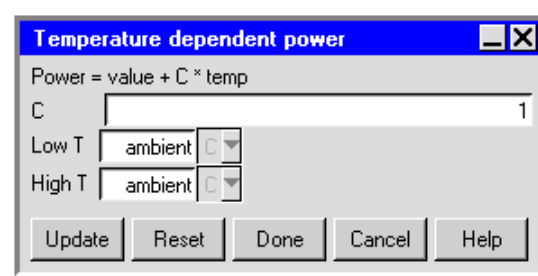


Figure 8.6.4: The Temperature dependent power Panel

Specify a value for the constant **C**. The **value** in the equation at the top of the **Temperature dependent power** panel is the **Total power** specified in the

Blocks panel. Define the temperature range for which the function is valid by entering values (in Kelvin) for **Low T** and **High T**. The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). If the temperature exceeds the specified value of **High T**, then the power is given by substituting the value of **High T** into the equation at the top of the **Temperature dependent power** panel. If the temperature falls below the specified value of **Low T**, then the power is given by substituting the value of **Low T** into the equation at the top of the **Temperature dependent power** panel. Click **Update** to update the thermal specification of the block.

Transient allows you to specify the total power as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. Select **Transient** under **Thermal specification** and enter a value for the **Total power**. To edit the transient parameters for the block, click **Edit** next to **Transient**. See Chapter 20 for more details on transient simulations.

4. (optional) Specify external heat transfer parameters for the block by turning on the **External conditions** option under **External data**. Click the **Edit** button to open the **Block thermal conditions** panel (Figure 8.6.5).

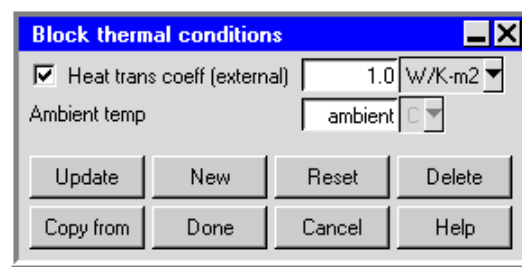


Figure 8.6.5: The **Block thermal conditions** Panel for a Solid or Fluid Block

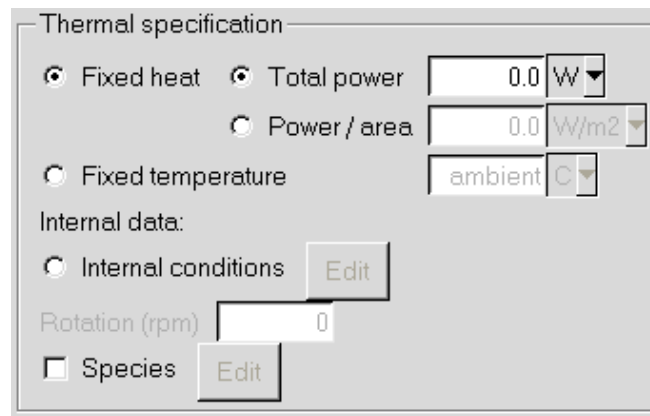
Heat trans coeff allows you to specify the **Heat trans coeff (external)** for the space surrounding the block.

Ambient temp allows you to specify the corresponding **Ambient temp**.

5. (solid cylindrical blocks only) Specify the speed of **Rotation (rpm)**.

Hollow Blocks

Define the thermal specification for the hollow block. There are two options: fixed heat and fixed temperature. The user inputs for the thermal specification of the hollow block are shown below.



Fixed heat specifies a constant uniform heat flux into or out of all block surfaces. You can define the heat flux either as per unit surface area (**Power / area**) or as a fixed value (**Total power**) for the entire block.

Fixed temperature specifies a constant uniform temperature for all the block surfaces. The value of the **ambient** temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

Internal conditions (optional) allows you to specify internal heat transfer parameters for the block. Click the adjacent **Edit** button to open the **Block thermal conditions** panel (Figure 8.6.6).

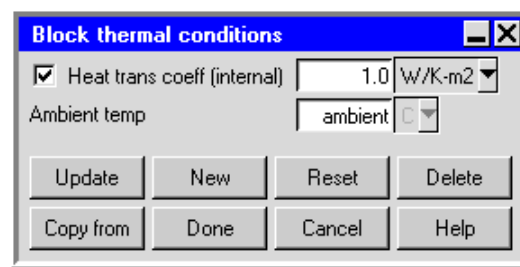


Figure 8.6.6: The Block thermal conditions Panel for a Hollow Block

In the **Block thermal conditions** panel, specify the **Heat trans coeff (internal)** for the space immediately inside the block and the corresponding **Ambient temp**.

Rotation (rpm) (for cylindrical blocks only) allows you to specify the rotational speed of a cylindrical block.

Fans are two- or three-dimensional modeling objects. A fan is used to move fluid into, out of, or within the room. Fan geometries include circular, rectangular, inclined, and 2D polygon. Fan types include fixed flow and characteristic curve. Fixed flow fans are always located on a room wall and must be specified either as intake (drawing fluid into the room) or exhaust (expelling fluid from the room). Characteristic curve fans can be located anywhere within the room or on the room boundary.

Fans are always associated with a magnitude and direction of flow. Circular fans can possess a central hub of non-zero radius and rectangular fans can have a rectangular hub. The hub is impervious to flow but can transfer heat to and from the fluid. The magnitude of the flow can be specified either as a fixed value or as a function of pressure drop across the fan.

In this chapter, information about the characteristics of a fan is presented in the following sections:

- Section 9.1: Defining a Fan in Airpak
- Section 9.2: Geometry, Location, and Dimensions
- Section 9.3: Flow Direction
- Section 9.4: Fans in Series
- Section 9.5: Fans in Parallel
- Section 9.6: Fans on Blocks
- Section 9.7: Specifying Swirl
- Section 9.8: Fixed Flow
- Section 9.9: Fan Characteristic Curve
- Section 9.10: Adding a Fan to Your Airpak Model

9.1 Defining a Fan in Airpak

Figure 9.1.1 shows two fans on the room boundary; one defined as an intake fan and the other defined as an exhaust fan. The intake fan draws fluid into the room. By default, **Airpak** assumes that the intake fluid is at ambient temperature. The exhaust fan expels fluid from the room in a direction determined by the local flow conditions. By default, the fluid exits the room at the temperature computed for the fluid within the room at the intake side of the fan.

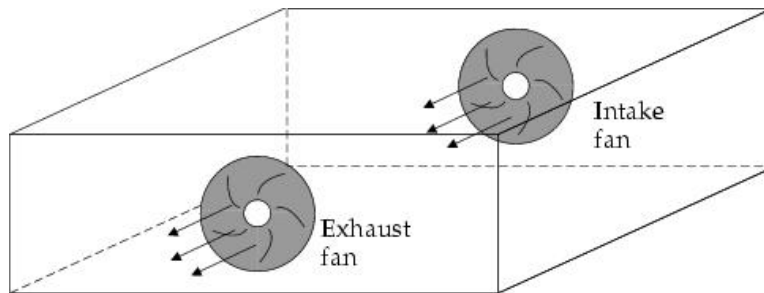


Figure 9.1.1: Intake and Exhaust Fans

Characteristic curve fans can be defined as exhaust, intake, or internal. Internal fans are located entirely within the room (see Figure 9.1.2) and are surrounded by fluid on all sides. The direction of flow through an internal 2D fan can be specified as positive or negative, relative to the coordinate axis normal to the plane of the fan.

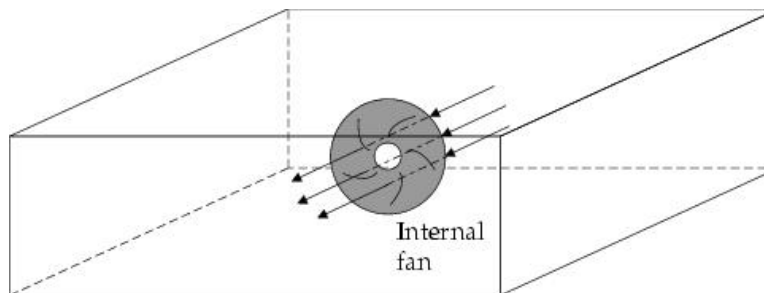


Figure 9.1.2: Internal Fan Placement

To configure a fan in the model, you must specify its geometry (including location and dimensions), its type, the flow rate associated with the fan, and the swirl. For a transient simulation, you must also specify parameters related to the strength of the fan for a characteristic curve fan. You can also specify the species concentrations and turbulence parameters at the fan.

9.2 Geometry, Location, and Dimensions

Fan location and dimension parameters vary according to fan geometry. Fan geometries include circular, rectangular, inclined, cylindrical, and polygonal. These geometries are described in Section 7.4.4.

9.2.1 Simple Fans

A simple fan is a two-dimensional fan that can have circular, rectangular, inclined, or polygonal geometry.

A polygon fan is a two-dimensional fan. To create a 2D polygon fan, you should first create a polygonal fan and then specify a height of zero.

Circular fans can include hubs. You must specify the size of the hub or inner radius (Int radius), and its overall size or outer radius (Radius), as shown in Figure 9.2.1.

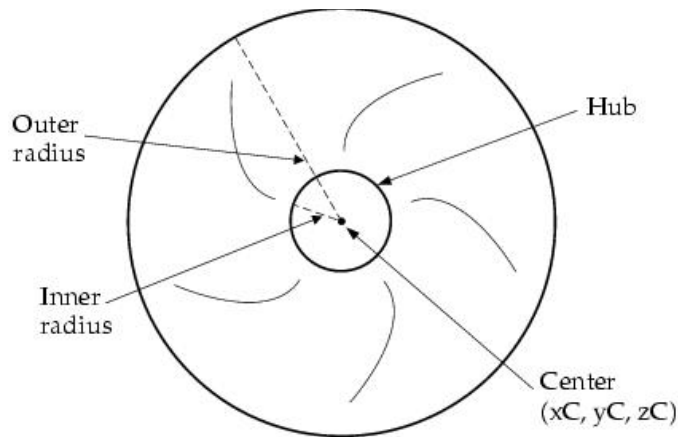


Figure 9.2.1: Circular Fan Definition

Rectangular fans can also include a hub. To create a hub for a rectangular fan, you must specify the equivalent radius for the hub if it were to be created as a circular hub for a circular fan; i.e., you specify r in Figure 9.2.2.

Airpak will create a rectangular hub with the area πr^2 . Airpak uses the area of the rectangular hub and the ratio of the lengths of the sides of the rectangular fan (d_1 and l_1 in Figure 9.2.2) to calculate the lengths of the sides of the rectangular hub (d_2 and l_2 in Figure 9.2.2):

$$d_2 l_2 = \pi r^2 \quad \text{and} \quad \frac{d_2}{l_2} = \frac{d_1}{l_1}$$

If the fan is square, then $d_2 = l_2 = x$, and

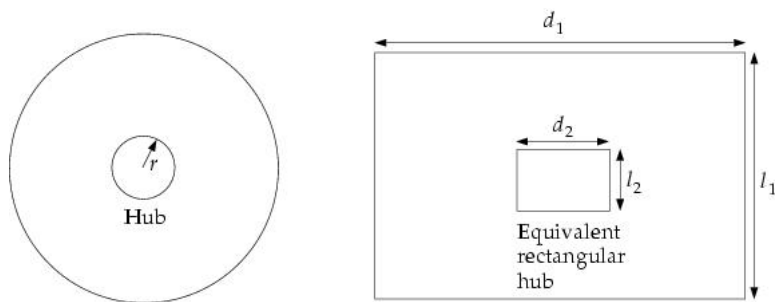


Figure 9.2.2: Rectangular Fan Hub Definition

$$x^2 = \pi r^2 \quad \text{i.e.,} \quad x = \sqrt{\pi r^2}$$

9.3 Flow Direction

Exhaust fans expel fluid from the room. The flow exits the room in a direction perpendicular to the fan.

By default, the direction of flow through an intake fan is normal to the plane of the fan. *Airpak* allows you to specify the flow direction for a fan and, thereby, model the effect of of an inclined fan (see Figure 9.3.1).

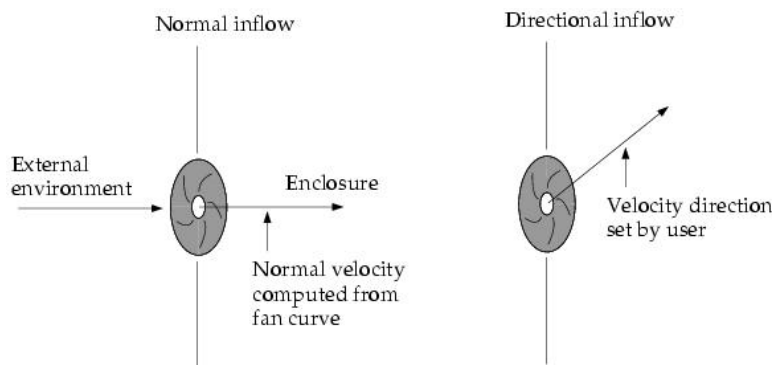


Figure 9.3.1: Intake Fan Flow Direction

Internal fans can be located anywhere inside the room. Fluid flows through the fan in a direction perpendicular to it. You must specify the direction of flow (positive or negative) for an internal fan. For a rectangular, polygon, or circular fan, if flow is in the direction of increasing axis coordinates, the inward direction is positive. If flow is in the direction of decreasing coordinates, the inward direction is negative. For an inclined fan, the positive

and negative directions are defined with respect to the axis of rotation for the inclined fan:

- If the axis of rotation of the inclined fan is the x axis, the positive inward direction is the positive y direction, and the negative inward direction is the negative y direction.
- If the axis of rotation of the inclined fan is the y axis, the positive inward direction is the positive z direction, and the negative inward direction is the negative z direction.
- If the axis of rotation of the inclined fan is the z axis, the positive inward direction is the positive x direction, and the negative inward direction is the negative x direction.

9.4 Fans in Series

Fans can be arranged in series (see Figure 9.4.1). There are two major advantages of placing several small fans in series rather than using a single large fan to achieve a given flow rate:

- Fans placed in series can be made active and inactive individually, so you can create different air flow patterns.
- Fans placed in series can direct air flows into localized streams.

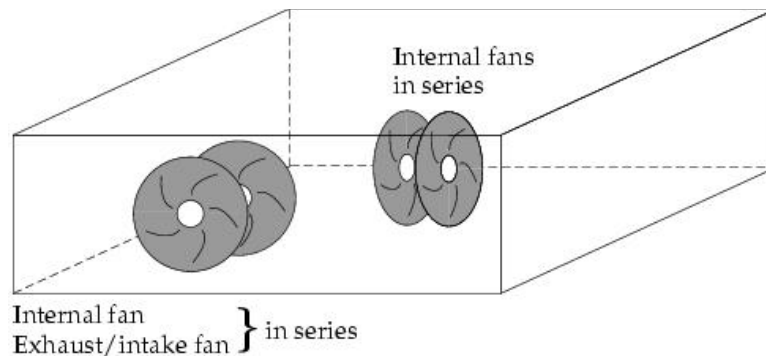


Figure 9.4.1: Fans in Series

9.5 Fans in Parallel

Fans can also be positioned in parallel (see Figure 9.5.1). Two or more fans placed in parallel can increase flow with respect to a single fan of equivalent power.

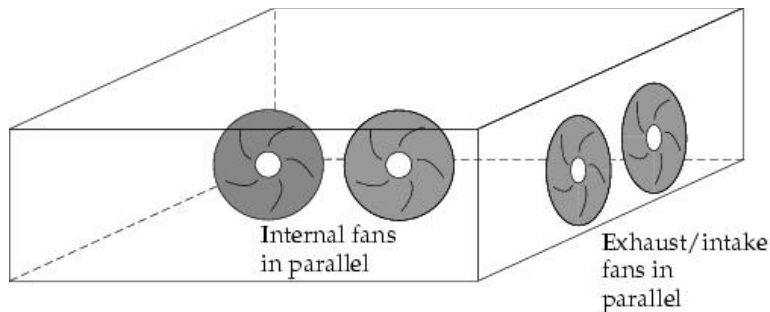


Figure 9.5.1: Fans in Parallel

9.6 Fans on Blocks

Fans can be combined with blocks to account for the dimensions of an actual fan. The treatment of fans used in conjunction with blocks varies according to the fan type. When a hollow block is placed against a room wall to mask a region of the room, an exhaust/intake fan can be positioned on any one of the exposed surfaces of the block (see Figure 9.6.1). Fans used in this way must be exhaust/intake in type despite the fact that they are located within the room and not on a room wall, because, in effect, the interior surfaces of this block are room walls (i.e., they represent part of the room).

i Exhaust/intake fan and block combinations cannot involve a conducting solid block.

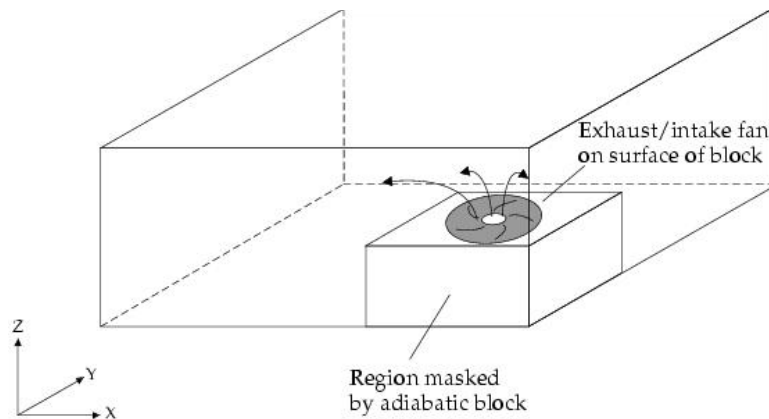


Figure 9.6.1: Fan on a Block

Internal fans can be placed on any conducting solid block within the room to give the fan a thickness. **Airpak** will automatically “bore a hole” through the block on both sides of the fan to allow fluid flow to and from the fan. Note that the face through which the fan bores a hole must not be located on the room boundary. Note that only conducting thick partitions and conducting solid blocks can be used for this purpose.

9.7 Specifying Swirl

To specify the swirl for a fan, you must specify the swirl magnitude or the rotational speed of the fan.

9.7.1 Swirl Magnitude

By default, **Airpak** assumes that fluid exits the fan in the direction normal to the plane of the fan. Alternatively, you can specify a swirl magnitude. This skews the flow direction in the θ direction (i.e., the direction of blade revolution). Swirl magnitude is defined by

$$u_{\theta}(r) = u_z(r) \left(\frac{r}{R} \right) S \quad (9.7-1)$$

where $u_{\theta}(r)$ is the velocity in the direction of revolution, $u_z(r)$ is the velocity in the direction normal to the fan, r is the radial coordinate, R is the outer radius of the fan, and S is the swirl magnitude ($S = 0$ by default).

9.7.2 Fan RPM

Instead of using a specified swirl magnitude, **Airpak** can also allow the swirl factor to change as a function of the operating point on the fan curve. This is achieved by specifying the RPM of the fan. The swirl magnitude (the ratio of the tangential velocity to the axial velocity) can then change as the fan operating point changes on the fan curve. For example, if the flow rate through the fan decreases, the swirl magnitude will increase, and if the flow rate through the fan increases, the swirl magnitude will decrease. Fan RPM is defined by

$$u_{\theta}(r) = \left[(\text{RPM}) \times \frac{2\pi}{60} \times r \right] \frac{1}{20} \quad (9.7-2)$$

where r is the radial coordinate, and it is assumed that only 5% of the maximum tangential velocity of the fan is transferred to the fluid.

9.8 Fixed Flow

In real-world applications, the performance of a fan is described by its characteristic curve, as described below. In **Airpak**, you can also specify a constant total mass flow or volume flow rate.

9.9 Fan Characteristic Curve

The relationship between volumetric flow rate and the pressure drop across the fan (static pressure) is described by the fan characteristic curve, which is usually supplied by the fan manufacturer. Figure 9.9.1 shows a characteristic curve for a common tube-axial fan. The total volumetric flow rate, Q , is plotted against fan static pressure, p_{fs} .

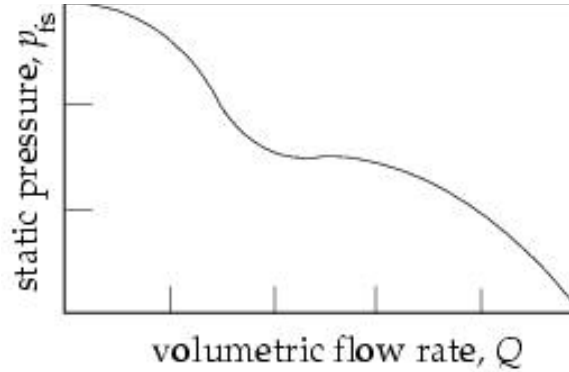


Figure 9.9.1: Tube-axial Fan Curve

For a linear fan characteristic curve, only the volume flow rate at zero static pressure, Q_0 , and the fan static pressure at zero flow rate, p_0 , need to be specified. The equation for a linear fan characteristic curve is

$$Q = Q_0(p_0 - p_{fs})/p_0 \quad (9.9-1)$$

In most cases, the linear characteristic curve does not adequately approximate the true fan characteristic curve over its entire operational range, so it is best to specify the actual fan curve, if possible.

Fan static pressure is computed by:

$$p_{fs} = p_{\text{discharge}} - p_{\text{intake}} \quad (9.9-2)$$



where p_{intake} is the pressure averaged over the face of the intake side of the fan, and $p_{\text{discharge}}$ is the pressure averaged over the face of the discharge side of the fan.

For internal fans, both p_{intake} and $p_{\text{discharge}}$ are computed by **Airpak**. For an intake fan, $p_{\text{discharge}}$ is computed by **Airpak** and p_{intake} is the ambient pressure. The value of the ambient pressure is specified under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). For an exhaust fan, $p_{\text{discharge}}$ is the ambient pressure, and p_{intake} is computed by **Airpak**. The default ambient pressure is zero (gauge pressure) and this should be satisfactory in almost all situations.

The accuracy of the fan flow rate used by **Airpak** is directly related to the accuracy with which the fan static pressure is computed. This, in turn, depends on how accurately pressure losses in the entire system are modeled. Therefore, care should be taken to model all features of the system that contribute to the overall nature of the pressure distribution in the system.

You can create a report of the fan operating point (pressure rise and volume flow rate) for a characteristic curve fan, as described in [Section 29.8](#).

9.10 Adding a Fan to Your Airpak Model

To include a fan in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Fans panel, shown in Figures 9.10.1 and 9.10.2.

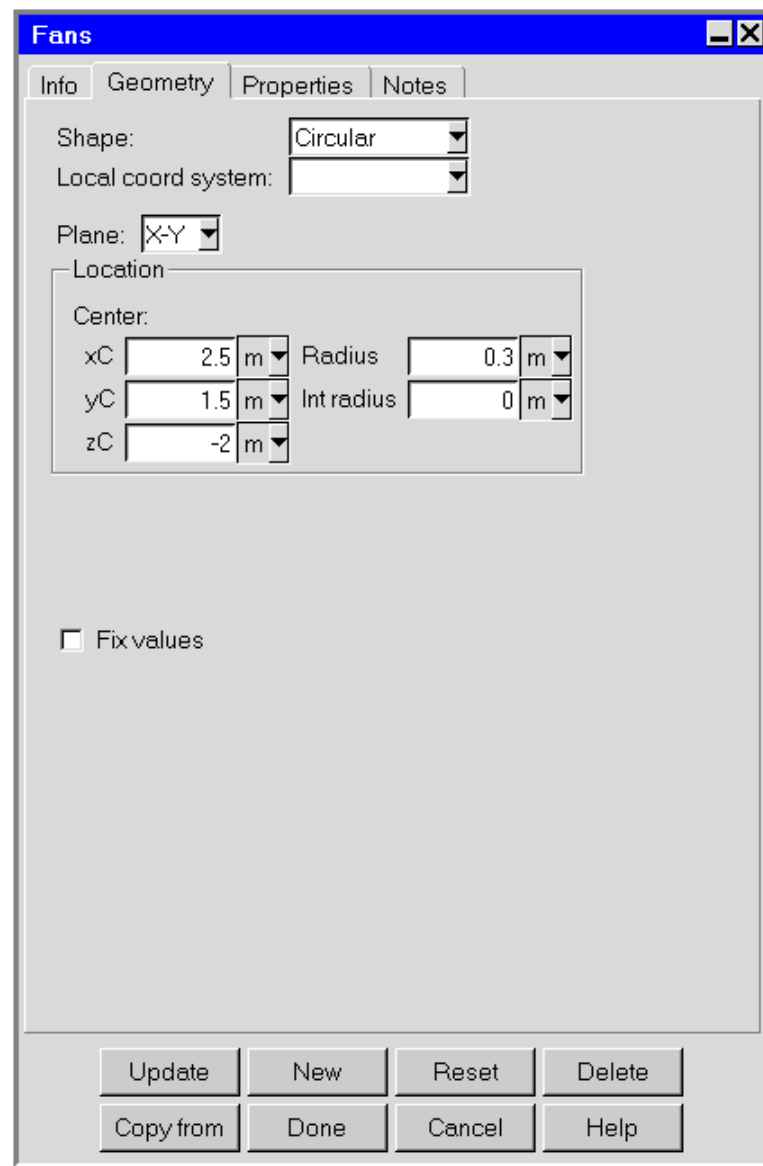


Figure 9.10.1: The Fans Panel (Geometry Tab)

The procedure for adding a fan to your Airpak model is as follows:

1. Create a fan. See Section 7.3.2 for details on creating a new object and Section 7.3.9

for details on copying an existing object.

2. Change the description of the fan, if required. See Section 7.4.1 for details.
3. Change the graphical style of the fan, if required. See Section 7.4.2 for details.
4. In the **Info** tab, enter the **Manufacturer** and **Model number**, if known.
5. Specify the geometry, position, and size of the fan in the **Geometry** tab. There are five different kinds of geometry available for fans in the **Shape** drop-down list for fans. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.

A polygon fan can be two-dimensional or three-dimensional. To create a 3D polygon fan, select **Polygon** in the **Fans** panel and specify a non-zero **Height**. To create a 2D polygon fan, select **Polygon** and specify a **Height** of zero.



Note that if you specify a circular fan, you can specify the size of the hub or inner radius (**IRadius**). If you specify a rectangular fan, you can specify the equivalent radius for the rectangular hub (**Internal hub equiv. radius**).

6. In the **Properties** tab, select the **Fan type**.. This will specify whether the fan is on the room boundary or inside the room. There are different options for **Fixed flow** and **Characteristic curve** fans.

Exhaust specifies that the fan is an exhaust fan. For 2D and 3D fans, the fluid is defined to exit the room through the fan in a direction determined by the local flow conditions. You do not need to specify a flow direction for an exhaust fan.

Intake specifies that the fan is an intake fan. The following inputs are required for an intake fan.

- (a) Specify the **Fluid temp**. This is the temperature of the fluid being drawn into the model, and is specified as **ambient** by default. The value of the **ambient** temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).
- (b) (2D fans only) Specify the flow direction. . There are two options:
 - If the fluid flows into the room normal to the fan, select **Normal**.
 - To specify the flow angle of the fluid entering the room through the fan, select **Given**. There are two options:

Vector allows you to specify the direction vector (X, Y, Z) for the flow. Only the direction of the vector is used by **Airpak**; the magnitude is ignored.

Angles allows you to specify the direction of the flow entering the room in terms of angles relative to the inward-pointing normal direction. You can specify two angles, **A** (α) and **T** (θ), to account for incoming flow not aligned with the normal direction. If the normal inward vector points in the positive direction, then α and θ must have values in the range $-90^\circ < \text{angle} < 90^\circ$, with $\alpha=\theta=0^\circ$ representing flow in the positive normal direction. If the normal inward vector points in the negative direction, then α and θ must have values in the range $90^\circ < \text{angle} < 270^\circ$, with $\alpha=\theta=180^\circ$ representing flow in the negative normal direction. Since the fan may be oriented in any one of three planes (x - y , y - z , or x - z), the roles of α and θ will vary depending on the orientation of the plane of the fan. An example is shown in Figure 9.10.3, where the fan is in the x - y plane and the normal inward vector is pointing in the positive direction.

Internal (2D characteristic curve fans only) specifies that the fan is an internal fan. Specify the **Facing** direction of the intake relative to the normal direction. The **Positive** and **Negative** options specify the inward direction normal to the fan as pointing toward high or low coordinates, respectively.

7. Specify the **Flow rate** for the fan. There are different options for the fixed flow and characteristic curve fans.

- For a fixed flow fan, you can specify a **Volume** flow rate or a **Mass** flow rate.
- for a characteristic curve fan, the following options are available:

Linear allows you to specify values defining a linear fan curve. Specify the **Volume flow rate** at zero fan static pressure, and a static pressure at zero flow (**Static pressure (head)**). The value of the **ambient** pressure is defined under **Ambient** values in the **Problem setup** panel (see Section 6.4.6).

Curve allows you to define the characteristic curve as a curve consisting of piecewise-continuous line segments. **Airpak** allows you to describe the curve either by positioning a series of points on a graph using the **Fan curve** graphics display and control window (described below), or by specifying a list of fan static pressure/volume flow rate coordinate pairs using the **Curve specification** panel (described below). These options are available under **Edit**.

To load a previously defined curve, click **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.

To save a curve, click **Save**. This will open the **Save curve** dialog box, and you can specify the filename and directory where the curve data is to be saved.



Note that the box to the right of **Edit** will be empty if you have not defined a curve for the fan. This box will contain the first volume flow value if you have defined a curve.

8. (characteristic curve fans only) Specify the **Transient strength**. This option is available only if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the fan, click **Edit** next to **Transient strength**. See Chapter 20 for details on transient simulations.
9. Specify the **Swirl** for the fan. You can specify a **Magnitude** or **RPM** for the fan.
10. Specify the inlet or outlet species concentrations for the fan, if required. Select **Species** and click the **Edit** button to open the **Species concentrations** panel (Figure 21.2.4). For details on setting species parameters, see Chapter 21.
11. Specify the turbulence parameters for the fan, if required. This option is available only if you selected the **Two equation** or **RNG k - ϵ** model in the **Basic parameters** panel (Figure 6.4.1). See Section 6.4.3 for details on determining turbulence parameters for objects in your **Airpak** model.

Select **Turbulence** in the **Fans** panel and click **Edit** to open the **Turbulence equation parameters** panel (Figure 9.10.4).

There are two options for specifying turbulence parameters:

Turbulent intensity/turbulent length scale allows you to specify the turbulence intensity and length scale for the fan.

Specify values for the **Turbulent intensity** and the **Turbulent length scale**. The default value of the turbulence intensity is 10%. By default, **Airpak** will use a default value for the length scale. The default value calculated by **Airpak** is 10% of the length of the shortest side of the fan.

Turbulent energy/turbulent dissipation allows you to specify the **Turbulent energy** and **Turbulent dissipation rate** for the fan.

12. Specify the **Solar behavior** for the fan. This option is available if you have enabled solar loading in the **Basic parameters** panel. See Section 22.5 for details.

9.10.1 Using the Fan curve Window to Specify the Curve for a Characteristic Curve Fan Type

You can specify a curve for a **Characteristic curve** fan type using the **Fan curve** graphics display and control window (Figure 9.10.5). To open the **Fan curve** window, select **Curve** under **Characteristic curve** in the **Fans** panel and click on **Edit**. Select **Graph editor** from the resulting list.

The following functions are available for creating, editing, and viewing a curve:

- To create a new point on the curve, click on the curve with the middle mouse button.
- To move a point on the curve, hold down the middle mouse button while positioned over the point, and move the mouse to the new location of the point.
- To delete a point on the curve, click the right mouse button on the point.
- To zoom into an area of the curve, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the **Fan curve** window, with appropriate changes to the axes. After you have zoomed into an area of the model, click on **Full range** to restore the graph to its original axes and scale.
- To set the minimum and maximum values for the scales on the axes, click on **Set range**. This will open the **Set range** panel (Figure 9.10.6). Enter values for **Min X**, **Min Y**, **Max X**, and **Max Y** and click **Accept**.
- To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 9.10.2 for details about creating a curve file outside of Airpak. See Section 2.1.8 for details on selecting a file.
- To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

You can use the **Print** button to print the curve. See Section 3.5 for details on saving hardcopy files.

Click **Done** when you have finished creating the curve; this will store the curve and close the **Fan curve** window. Once the curve is defined, you can view the pairs of coordinates defining the curve in the **Curve specification** panel. See Figure 9.10.7 for the pairs of coordinates for the curve shown in Figure 9.10.5.



Note that the curve you specify *must* intersect both the x and y axes.

9.10.2 Using the Curve specification Panel to Specify the Curve for a Characteristic Curve Fan Type

You can define a curve for a Characteristic curve fan type using the **Curve specification** panel (Figure 9.10.7). To open the **Curve specification** panel, select **Curve** under **Characteristic curve** in the **Fans** panel and click on **Edit**. Select **Text editor** from the resulting list.

To define a curve, specify a list of coordinate pairs in the **Curve specification** panel. It is important to give the numbers in pairs, but the spacing between numbers is not important. Click **Accept** when you have finished entering the pairs of coordinates; this will store the values and close the **Curve specification** panel.

To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the units used in the curve data you are loading, you should select the appropriate units in the **Curve specification** panel before you load the curve. If you want to view the imported data after you have loaded them, using different units than the default units in the **Curve specification** panel, select **Fix values** for **Volume flow units** and/or **Pressure units** and select the appropriate units from the unit definition list.

If you want to load a curve file that you have created outside of **Airpak**, you will need to make sure that the first three lines of the file before the data contain the following information:

1. the number of data sets in the file (usually 1)
2. the unit specifications for the file, which can be obtained from the **Curve specification** panel (e.g., **units m3/s N/m2**)
3. the number of data points in the file (e.g., 10)

Using the above example, the first three lines of the curve file would be

```
1
units m3/s N/m2
10
```

The actual data points should be entered in the same way as you would enter them in the **Curve specification** panel.



If you want to load a curve from an Excel file, make sure that you also save the file as formatted text (space delimited) before reading it into **Airpak**.

To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

Once the pairs of coordinates have been entered, you can view the curve in the **Fan curve** graphics display and control window. See Figure 9.10.5 for the curve for the values shown in Figure 9.10.7.

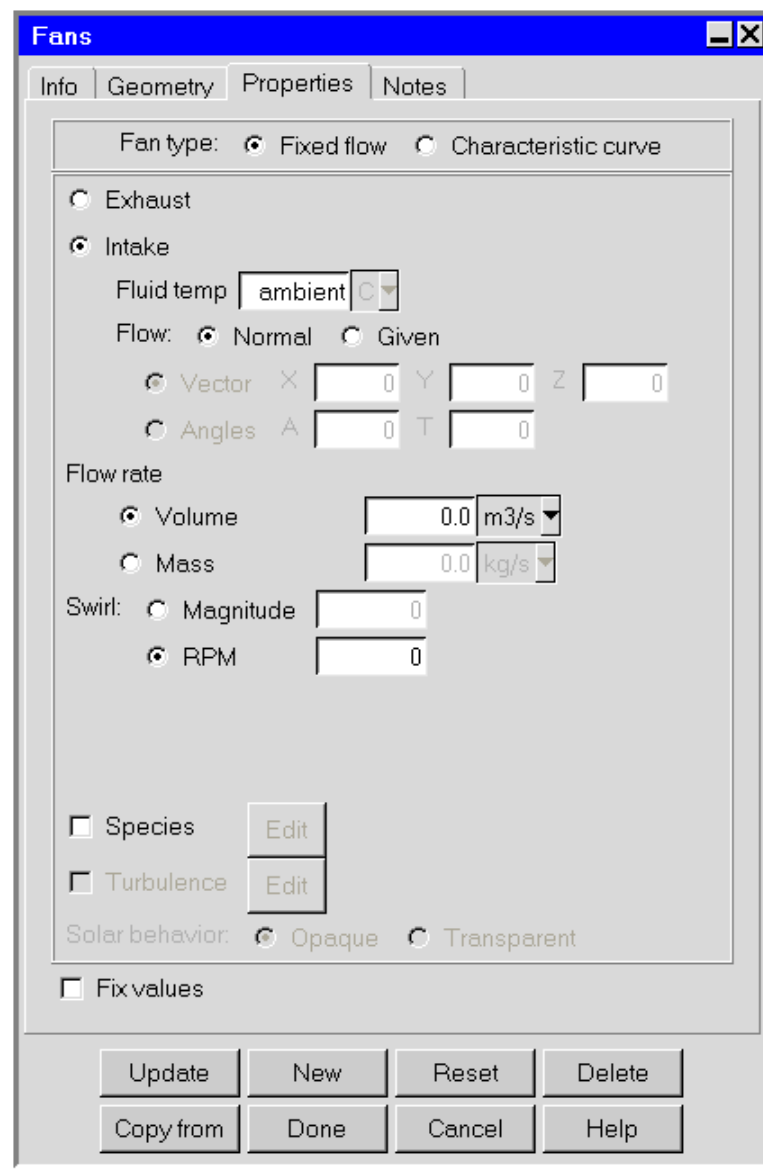


Figure 9.10.2: The Fans Panel (Properties Tab)

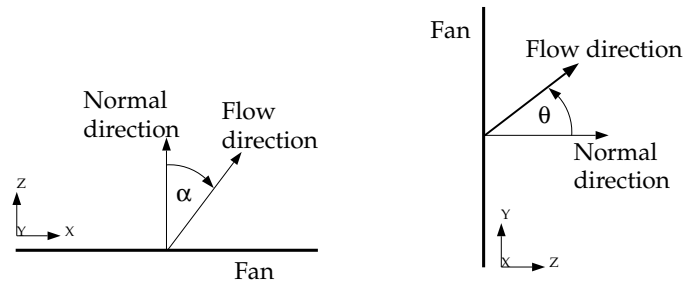


Figure 9.10.3: Definition of Angles for a Fan

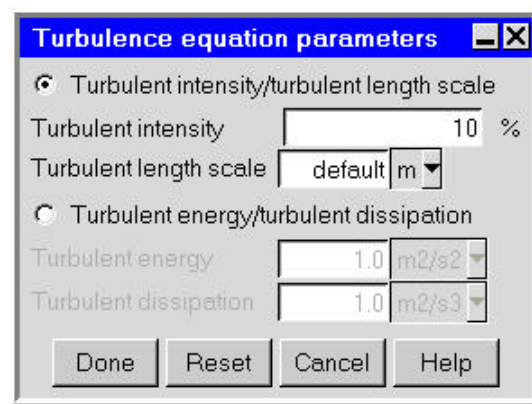


Figure 9.10.4: The Turbulence equation parameters Panel

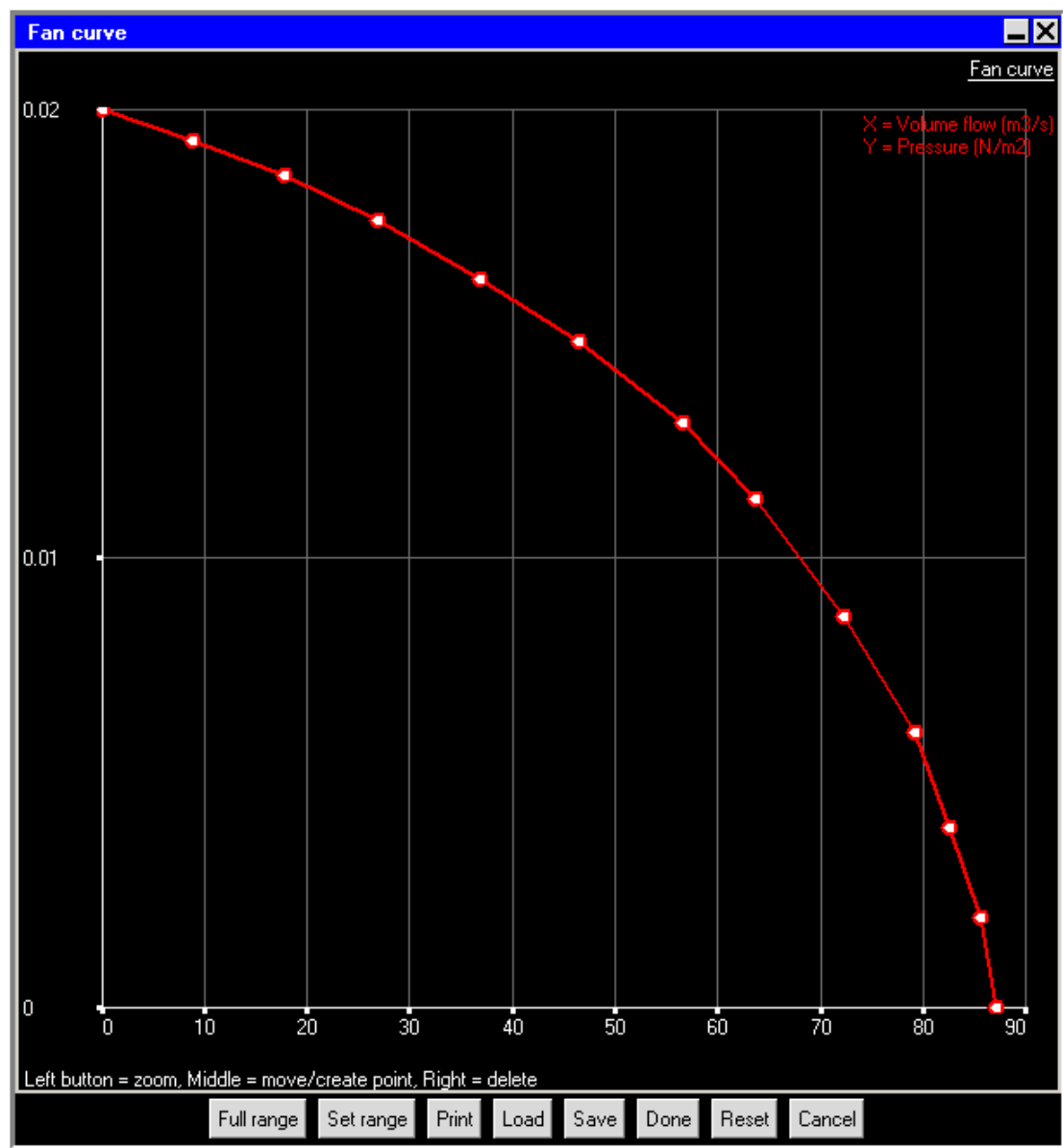


Figure 9.10.5: The Fan curve Graphics Display and Control Window

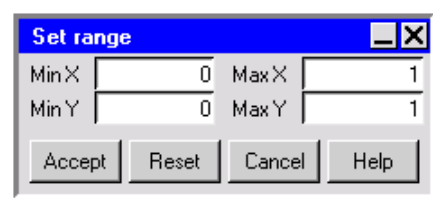


Figure 9.10.6: The Set range Panel

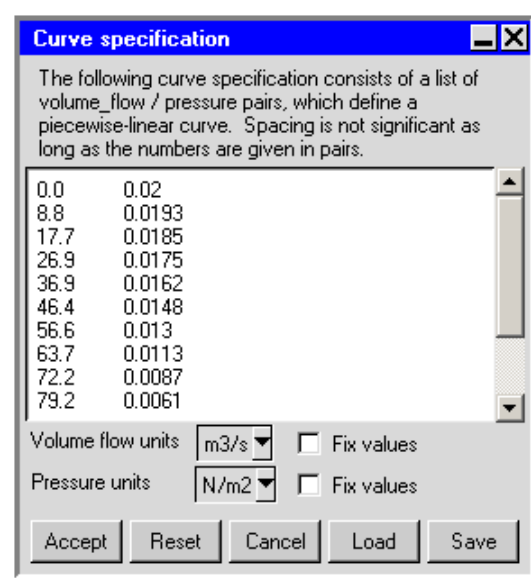


Figure 9.10.7: The Curve specification Panel

10.1 Overview

Vents represent holes through which fluid can enter or leave the room. They are always located on enclosure boundaries (i.e., either room walls or the surfaces of blocks used to modify the shape of the room) See Figure 10.1.1. Vent geometries include rectangular, circular, 2D polygon, and inclined.

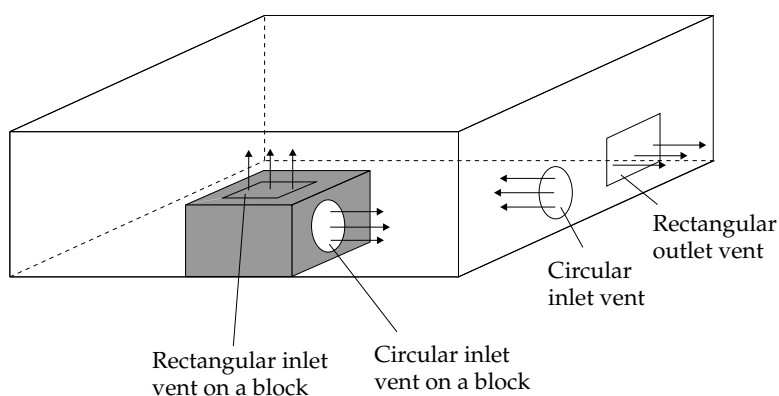


Figure 10.1.1: Vent Examples

In most real-world cases, vents have coverings (e.g., screen meshes, angled slats, wire vents). These coverings result in a pressure drop across the plane of the vent. **Airpak** treats the pressure drop in the same way that it treats a pressure drop through a resistance modeling object.

Fluid can enter or exit the room through a vent. The actual flow direction through the vent is computed by **Airpak**. In some cases, fluid can enter and exit through different areas of a single vent. In cases where the amount of flow in both directions is nearly equal, the vent may be modeled as two separate but adjoining vents.

Airpak computes both the temperature and the direction of the fluid exiting the room through a vent. By default, the temperature of the fluid entering the room through a vent is assumed to be the ambient temperature specified under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). If flow is leaving the domain through the vent, the temperature is ignored. By default, the direction of flow of the fluid through a vent is computed by **Airpak**. If you know the direction of flow at the vent, you can specify it in the **Vents** panel.

To configure a vent in the model, you must specify its geometry (including location and dimensions). For best results, the vent size and geometry should closely match the size and geometry of the actual vent. In some cases, you must also specify the temperature, pressure, species concentration, turbulence parameters, and/or method used to calculate the pressure drop through the vent covering.

10.2 Planar Resistances

Planar resistances represent partial obstructions to flow within the room. Planar resistance geometries include rectangular, circular, inclined, and 2D polygon and are designed to model planar flow obstructions such as screens and permeable baffles.

For a planar resistance in **Airpak**, the effect of any resistance is modeled as a pressure drop through its area or volume. **Airpak** provides a list of resistance types that you can select to model the pressure drop through a planar resistance. Alternatively, the pressure drop across the resistance can be calculated using either the approach-velocity method or the device-velocity method, both of which require a user-specified velocity loss coefficient. The approach-velocity and device-velocity methods differ from each other only by virtue of a factor called the free area ratio. The calculated pressure drop can be proportional either to the fluid velocity itself, or to the square of the velocity. It is common practice to employ the linear relationship for laminar flow and the quadratic relationship for turbulent flow. In the general case, a combination of the linear and quadratic relationships may more accurately model the pressure drop/speed curve.

i Note that you cannot use the hexahedral mesher if a planar resistance is placed on an inclined conducting thick partition. The tetrahedral mesher, on the other hand, can be used with all types of resistances.

To configure a flow resistance in the model, you must specify its geometry (including location and dimensions), the pressure drop model, and the relationship between resistance and velocity.

10.3 Geometry, Location, and Dimensions

Vent location and dimension parameters vary according to the vent geometry. Vent geometries include rectangular, circular, inclined, and 2D polygon. These geometries are described in Section 7.4.4.

10.4 Pressure Drop Calculations for Vents

Airpak computes the speed, direction, and temperature of the fluid exiting the room through a vent (see Figure 10.4.1). The calculations are based on the assumption that the external pressure is static. If you do not specify a value for the external pressure, Airpak uses the ambient pressure specified under **Ambient** values in the **Basic parameters** panel (see Section 6.4.6).

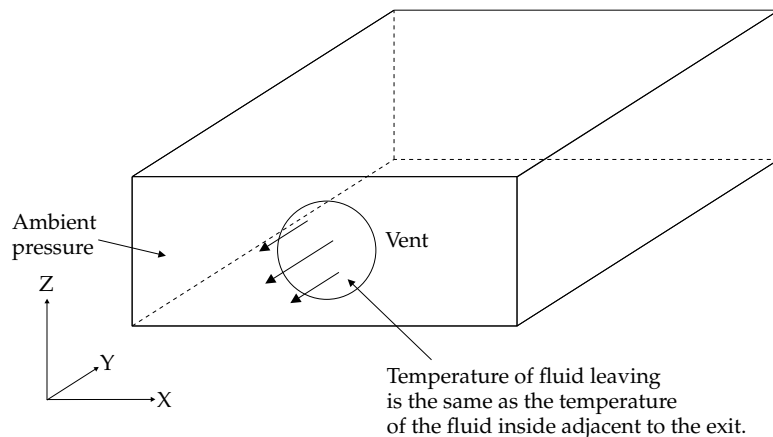


Figure 10.4.1: Outlet Vent Conditions

Fluid entering the room through a vent is drawn in from the external environment. By default, the external fluid is at the ambient temperature specified under **Ambient** values in the **Basic parameters** panel (see Section 6.4.6), and the flow enters the room in a direction computed by Airpak. However, you can impose a flow direction, as shown in Figure 10.4.2. You can also specify a temperature for the fluid entering the vent.

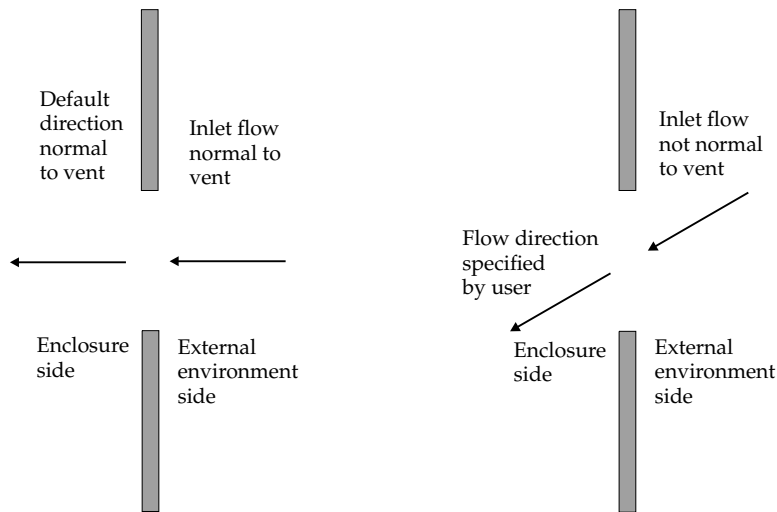


Figure 10.4.2: Vent Flow Direction

To account for pressure losses due to the presence of mesh screens or angled slats on the vent, you must specify a loss coefficient or select a vent type. See [14] for a compilation of loss coefficients applicable to most situations encountered in electronic enclosures.

Airpak can calculate loss coefficients for different types of vents based on the free area ratio of the vent. The following vent types are available in **Airpak**:

- a perforated thin vent, with a loss coefficient of

$$l_c = \frac{1}{A^2} \left[0.707(1 - A)^{0.375} + 1 - A \right]^2 \quad (10.4-1)$$

where A is the free area ratio.

- a circular metal wire screen, with a loss coefficient of

$$l_c = 1.3(1 - A) + \left(\frac{1}{A} - 1 \right)^2 \quad (10.4-2)$$

where A is the free area ratio.

- a two-plane screen with cylindrical bars, with a loss coefficient of

$$l_c = \frac{1.28(1 - A)}{A^2} \quad (10.4-3)$$

where A is the free area ratio.

i Note that setting the ambient temperature and flow direction has no effect on the calculations if fluid flows out of the room through the vent.

Alternatively, **Airpak** can calculate the pressure drop resulting from a resistance either by the approach-velocity method or by the device-velocity method.

The approach-velocity method relates the pressure drop to the fluid velocity:

$$\Delta p = \rho \frac{l_{c1}}{2} v_{\text{app}}^n \quad (10.4-4)$$

where l_c is the user-specified loss coefficient, ρ is the fluid density, and v_{app} is the approach velocity. The approach velocity is the calculated velocity at the plane of the vent. The velocity dependence can be linear ($n = 1$), quadratic ($n = 2$), or a combination of linear and quadratic.

The device-velocity method relates the pressure drop induced by the vent to the fluid velocity:

$$\Delta p = \rho \frac{l_{c2}}{2} v_{\text{dev}}^n \quad (10.4-5)$$

where v_{dev} is the device velocity. The velocity dependence can be linear ($n = 1$), quadratic ($n = 2$), or a combination of linear and quadratic.

The difference between the approach-velocity and device-velocity methods is in the velocity used to compute the pressure drop. The device velocity is related to the approach velocity by

$$v_{\text{dev}} = \frac{v_{\text{app}}}{A} \quad (10.4-6)$$

where A is the free area ratio. The free area ratio is the ratio of the area through which the fluid can flow unobstructed to the total planar area of the obstruction.

i Note that the loss coefficient used in the equation for the device velocity is not the same as the loss coefficient used in the equation for the approach velocity.

The loss coefficients in Equations 10.4-4 and 10.4-5 are related to the flow regime of the problem:

- For a viscous flow regime (e.g., laminar flow, slow flow, very dense packing), you should select a linear velocity relationship:

$$\Delta p = \frac{\rho}{2} l_{c1} v \quad (10.4-7)$$

- For an inertial flow regime (e.g., turbulent flow), you should select a quadratic velocity relationship:

$$\Delta p = \frac{\rho}{2} l_{c2} v^2 \quad (10.4-8)$$



- For a combination of these two types of flow, you should select a linear+quadratic velocity relationship:

$$\Delta p = \Delta p_{\text{linear}} + \Delta p_{\text{quadratic}} \quad (10.4-9)$$

You can obtain the loss coefficients in several ways:

- experimental measurements
- computational measurements
- from a reference (The loss coefficients for many vent configurations are available in [14].)

10.5 Adding a Vent to Your Airpak Model

To include a vent in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Vents panel, shown in Figures 10.5.1 – 10.5.3.

The procedure for adding a vent to your Airpak model is as follows:

1. Create a vent. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the vent, if required. See Section 7.4.1 for details.
3. Change the graphical style of the vent, if required. See Section 7.4.2 for details.

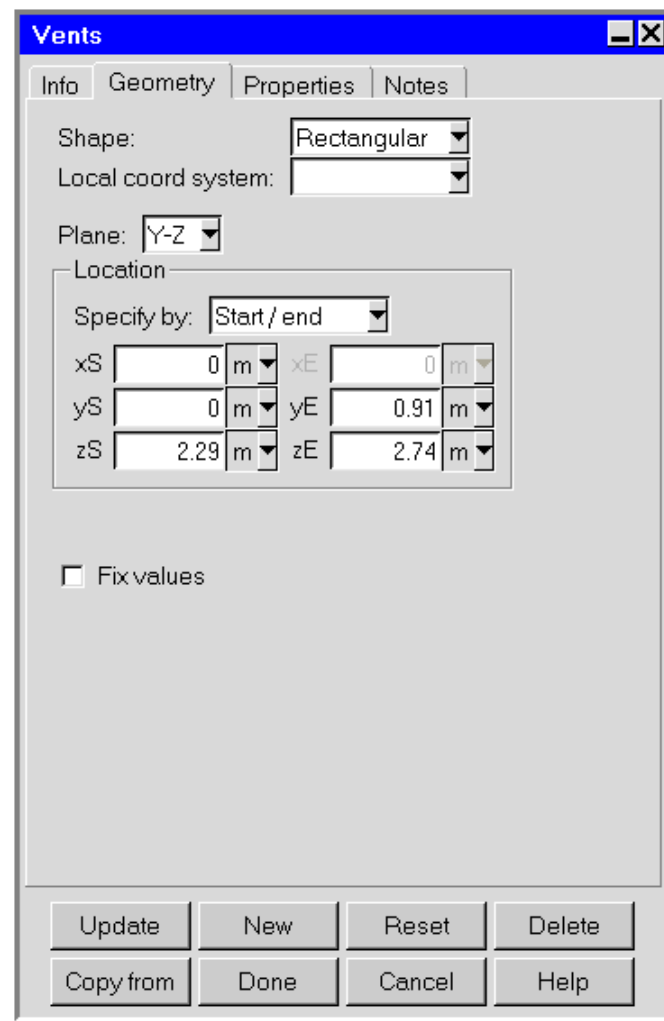


Figure 10.5.1: The Vents Panel (Geometry Tab)

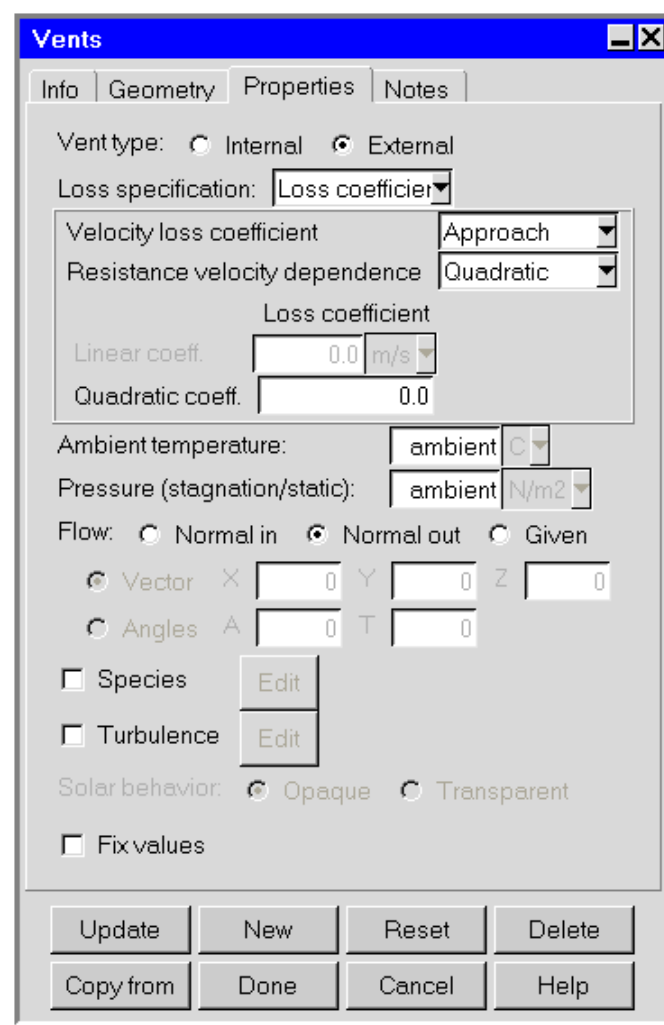


Figure 10.5.2: The Vents Panel (Properties Tab) for an External Vent

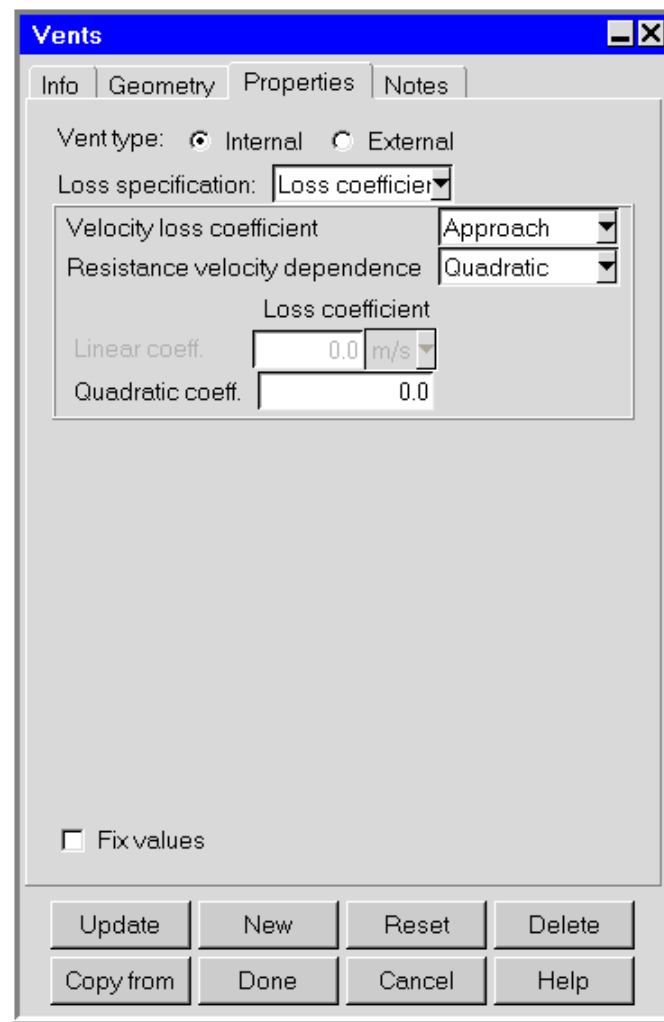


Figure 10.5.3: The Vents Panel (Properties Tab) for an Internal 2D Resistance Vent

4. In the **Geometry** tab, specify the geometry, position, and size of the vent (Figure 10.5.1). There are four different kinds of geometry available for vents in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.
5. In the **Properties** tab, specify the characteristics for the vent.
 - (a) Specify the **Vent type** as **Internal** (Figure 10.5.3) or **External** (Figure 10.5.2). Choose **Internal** for a 2D resistance vent.
 - (b) (**Internal and External Vents**) Select the **Loss specification** in the drop-down list. The following options are available:
 - To specify the loss coefficient, select **Loss coefficient** and then select the method to be used to calculate the velocity loss coefficient. The following options are available in the **Velocity loss coefficient** drop-down list.
 - To have **Airpak** calculate the loss coefficient for a particular vent type based on the free area ratio of the vent, select **Automatic**. Then select the type of vent (**Perforated thin vent**, **Circular metal wire screen**, or **Two-plane screen, cyl. bars**) in the **Resistance type** drop-down list and specify the **Free area ratio**.
 - To use the device-velocity method, select **Device** and select the method to be used to calculate the **Resistance velocity dependence**. There are three options in the drop-down list: **Linear**, **Quadratic**, and **Linear+quadratic**. Finally, specify the appropriate **Loss coefficient** and **Free area ratio** for the **Linear coeff.** and/or **Quadratic coeff.**
 - To use the approach-velocity method, select **Approach** and select the method to be used to calculate the **Resistance velocity dependence**. Finally, specify the appropriate **Loss coefficient** for the **Linear coeff.** and/or **Quadratic coeff.**
 - To define a piecewise-linear profile for the pressure drop as a function of the speed of the fluid through the vent, select **Loss curve**. **Airpak** allows you to describe the curve either by positioning a series of points on a graph using the **Resistance curve** graphics display and control window (described below), or by specifying a list of vent speed/pressure coordinate pairs using the **Curve specification** panel (described below). These options are available under **Edit**.

To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.

To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

i Note that the box to the right of **Edit** will be empty if you have not defined a curve for the vent. This box will contain the first speed value if you have defined a curve.

The following parameters can be set for **External** vents, only.

- (c) Enter a value for the **Ambient temperature** of the external fluid. The default value of **Ambient temperature** is the ambient temperature specified under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). This temperature is used if fluid flows into the room through the vent, and is ignored if fluid flows out of the room through the vent.
- (d) To specify the external pressure, enter a value for the **Pressure (stagnation/static)**. For flow into a vent, the external pressure is the stagnation pressure. For flow out of a vent, the external pressure is the static pressure. By default, the external pressure is the ambient pressure specified under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).
- (e) Specify the **Flow direction**. There are three options:
 - If the fluid flows into the room normal to the vent, select **Normal in**.
 - If the fluid flows out of the room normal to the vent, select **Normal out**.
 - To specify the flow angle of the fluid entering the room through the vent, select **Given**. There are two options:

Vector allows you to specify the direction vector (**X**, **Y**, **Z**) for the flow. Only the direction of the vector is used by **Airpak**; the magnitude is ignored.

Angles allows you to specify the direction of the flow entering the room in terms of angles relative to the inward-pointing normal direction. You can specify two angles, **A** (α) and **T** (θ), to account for incoming flow not aligned with the normal direction. If the normal inward vector points in the positive direction, then α and θ must have values in the range $-90^\circ < \text{angle} < 90^\circ$, with $\alpha=\theta=0^\circ$ representing flow in the positive normal direction. If the normal inward vector points in the negative direction, then α and θ must have values in the range $90^\circ < \text{angle} < 270^\circ$, with $\alpha=\theta=180^\circ$ representing flow in the negative normal direction. Since the vent may be oriented in any one of three planes (x - y , y - z , or x - z), the roles of α and θ will vary depending on the orientation of the plane of the vent. An example is shown in Figure 10.5.4, where the vent is in the x - y plane and the normal inward vector is pointing in the positive direction.

- (f) Specify the inlet or outlet species concentrations for the vent, if required. You can input the species concentrations for the vent using the **Species concentrations** panel. To open this panel, select **Species** in the **Vents** panel and then click **Edit**. See Chapter 21 for details on modeling species transport.

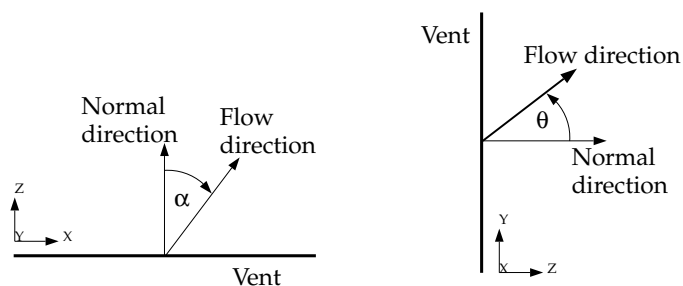


Figure 10.5.4: Definition of Angles for a Vent

- (g) Specify the turbulence parameters for the vent, if required. This option is available only if you selected the **Two equation** or **RNG $k-\epsilon$** model in the **Basic parameters** panel (Figure 6.4.1). See Section 6.4.3 for details on determining turbulence parameters for objects in your **Airpak** model.

Select **Turbulence** in the **Vents** panel and click **Edit** to open the **Turbulence equation parameters** panel (Figure 10.5.5).

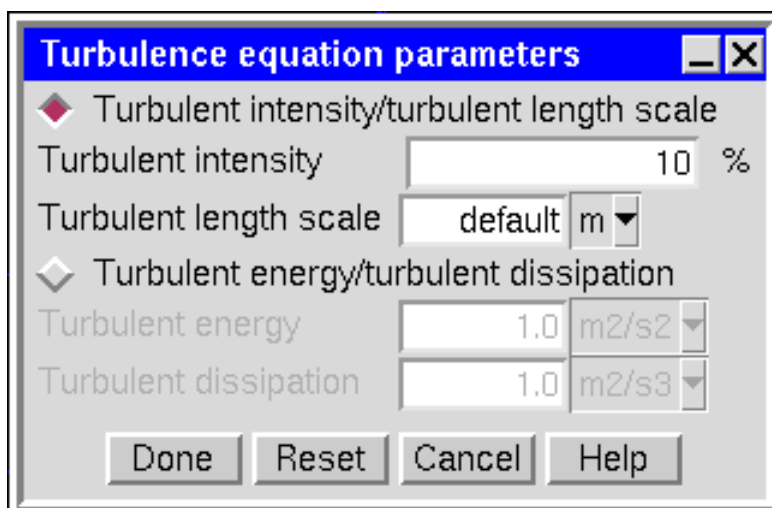


Figure 10.5.5: The Turbulence equation parameters Panel

There are two options for specifying turbulence parameters:

Turbulent intensity/turbulent length scale allows you to specify the turbulence intensity and length scale for the vent.

Specify values for the **Turbulent intensity** and the **Turbulent length scale**. The default value of the turbulence intensity is 10%. By default, **Airpak** will use a **default** value for the length scale. The **default** value calculated by **Airpak** is 10% of the length of the shortest side of the vent.

Turbulent energy/turbulent dissipation allows you to specify the Turbulent energy and Turbulent dissipation rate for the vent.

- (h) If you have enabled solar loading in the **Basic parameters** panel, specify the Solar behavior for the vent. See Section 22.5 for details.

10.5.1 Using the Resistance curve Window to Specify the Curve for a Vent

You can specify a resistance curve for a vent using the Resistance curve graphics display and control window (Figure 10.5.6). To open the Resistance curve window, select **Loss curve** from the **Loss specification** drop-down list in the **Vents** panel and click on **Edit**. Select **Graph editor** from the resulting list.

The following functions are available for creating, editing, and viewing a curve:

- To create a new point on the curve, click on the curve with the middle mouse button.
- To move a point on the curve, hold down the middle mouse button while positioned over the point, and move the mouse to the new location of the point.
- To delete a point on the curve, click the right mouse button on the point.
- To zoom into an area of the curve, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the **Resistance curve** window, with appropriate changes to the axes. After you have zoomed into an area of the model, click on **Full range** to restore the graph to its original axes and scale.
- To set the minimum and maximum values for the scales on the axes, click on **Set range**. This will open the **Set range** panel (Figure 10.5.7). Enter values for **Min X**, **Min Y**, **Max X**, and **Max Y** and click **Accept**.
- To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.
- To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

You can use the **Print** button to print the curve. See Section 3.5 for details on saving hardcopy files. Click **Done** when you have finished creating the curve; this will store the curve and close the **Resistance curve** window. Once the curve is defined, you can view the pairs of coordinates defining the curve in the **Curve specification** panel. See Figure 10.5.8 for the pairs of coordinates for the curve shown in Figure 10.5.6.

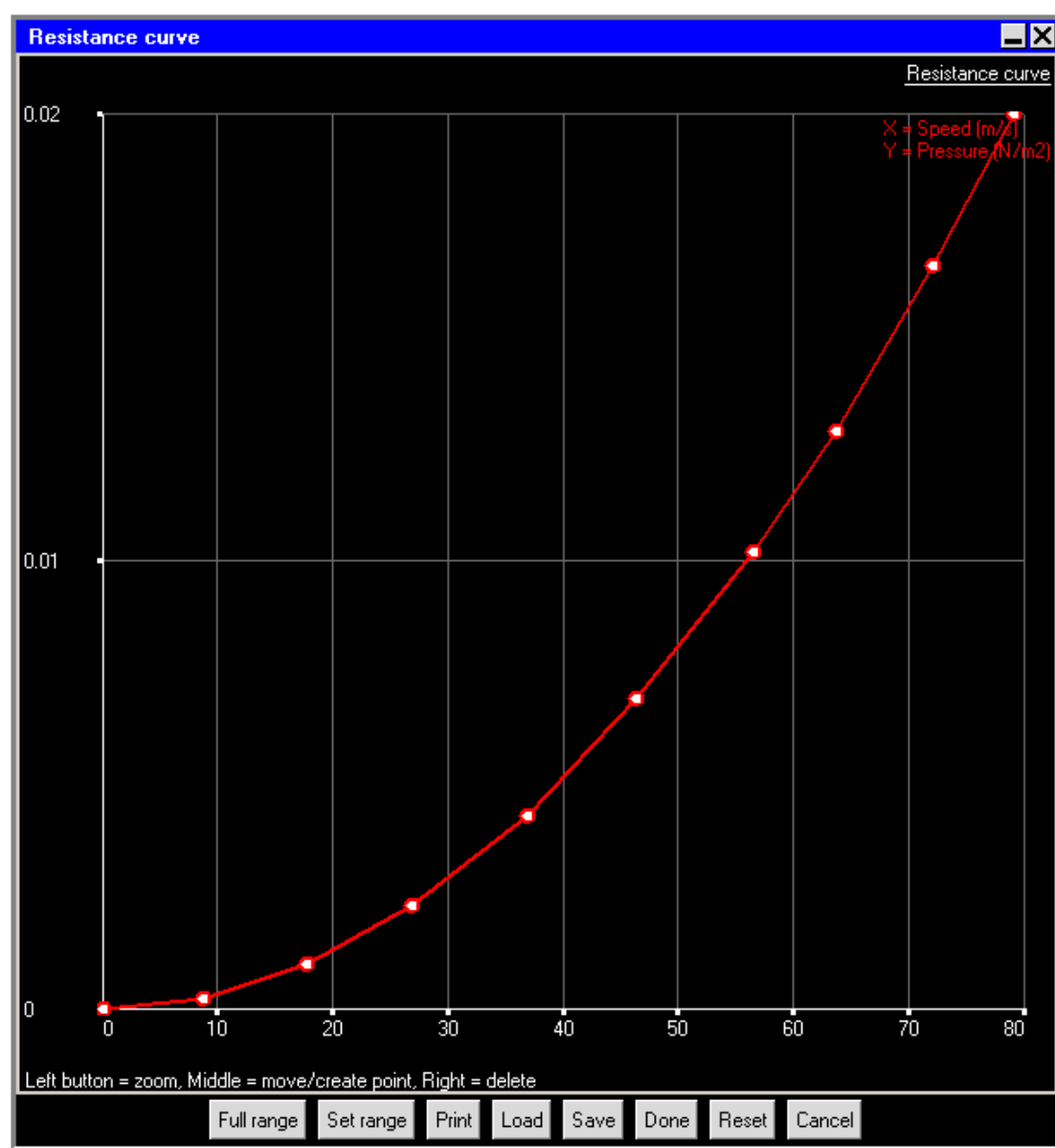


Figure 10.5.6: The Resistance curve Graphics Display and Control Window

The figure shows a window titled "Set range" with a blue title bar and a white background. It contains four input fields for Min X, Max X, Min Y, and Max Y. The values for Min X and Min Y are 0, and for Max X and Max Y are 1. Below the input fields are four buttons: Accept, Reset, Cancel, and Help.

Field	Value
Min X	0
Max X	1
Min Y	0
Max Y	1

Figure 10.5.7: The Set range Panel

i Note that the curve you specify *must* intersect both the x and y axes.

10.5.2 Using the Curve specification Panel to Specify the Curve for a Vent

You can define a resistance curve for a vent using the Curve specification panel (Figure 10.5.8). To open the Curve specification panel, select Loss curve from the Loss specification drop-down list in the Vents panel and click on Edit. Select Text editor from the resulting list.

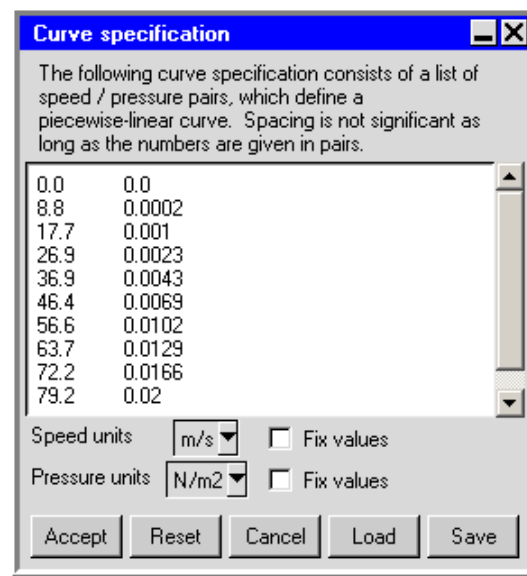


Figure 10.5.8: The Curve specification Panel

To define a curve, specify a list of coordinate pairs in the Curve specification panel. It is important to give the numbers in pairs, but the spacing between numbers is not important. Click **Accept** when you have finished entering the pairs of coordinates; this will store the values and close the Curve specification panel.

To load a previously defined curve, click on **Load**. This will open the Load curve file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the units used in the curve data you are loading, you should select the appropriate units in the Curve specification panel before you load the curve. If you want to view the imported data after you have loaded them, using different units than the default units in the Curve specification panel, select **Fix values** for Speed units and/or Pressure units and select the appropriate units from the unit definition list.

If you want to load a curve file that you have created outside of Airpak, you will need to make sure that the first three lines of the file before the data contain the following information:

1. the number of data sets in the file (usually 1)
2. the unit specifications for the file, which can be obtained from the **Curve specification** panel (e.g., `units m/s N/m2`)
3. the number of data points in the file (e.g., 10)

Using the above example, the first three lines of the curve file would be

```
1
units m/s N/m2
10
```

The actual data points should be entered in the same way as you would enter them in the **Curve specification** panel.



If you want to load a curve from an Excel file, make sure that you also save the file as formatted text (space delimited) before reading it into **Airpak**.

To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

Once the pairs of coordinates have been entered, you can view the curve in the **Resistance curve** graphics display and control window. See Figure 10.5.6 for the curve for the values shown in Figure 10.5.8.

Openings are two-dimensional modeling objects representing areas of the model through which fluid can flow. Opening geometries include rectangular, circular, 2D polygon, and inclined. Opening types include free and recirculation. Free openings are specified individually, but recirculation openings must be specified in pairs. Recirculation opening pairs consist of two sections:

- an extract section, representing the location at which fluid is removed from the enclosure
- a supply section, representing the location at which fluid is returned to the enclosure

Openings should be located on an enclosure boundary (i.e., either a room wall or the surface of a hollow block that has been used to mask a portion of the enclosure). Free openings can also be located on the surfaces of blocks or partitions within the room.

Free openings represent holes either on the boundary of the room or on blocks or partitions. Recirculation openings model devices (e.g., heaters, refrigeration circuits, or filters) that extract fluid from the room at one location, and supply it to a different location in the model with altered thermal or species properties.

Note that the mass flow rates for the extract and supply sections of a recirculation opening must be the same. You can specify different mass fluxes for the two sections if they differ in size, but the mass flow rates (mass flux \times area) for the sections must be the same.

To configure an opening in the model, you must specify its geometry (including location and dimensions) and type. For free openings, you can also specify temperature, static pressure, species concentrations, velocity, and turbulence parameters at the opening. For transient problems, you can specify the variation of temperature, pressure, species concentration, velocity, and turbulence parameters with time. For recirculation openings, you must specify the mass flow rate and thermal treatment of the fluid in the recirculation loop. The thermal treatment can be specified as a constant temperature increase (or decrease), a fixed heat input (or extraction), or using a conductance and the external temperature. You can also specify the direction of flow out of the supply section as well as an increase or decrease in species in the recirculation loop.

In this chapter, information about the characteristics of an opening is presented in the following sections:

- Section 11.1: Geometry, Location, and Dimensions
- Section 11.2: Free Openings
- Section 11.3: Recirculation Openings
- Section 11.4: Adding an Opening to Your **Airpak** Model

11.1 Geometry, Location, and Dimensions

The location and dimension parameters for an opening vary according to the geometry of the opening. Opening geometries include rectangular, circular, 2D polygon, and inclined. These geometries are described in Section 7.4.4.



The geometries of the extract and supply sections for a recirculation opening can differ from one another.

11.2 Free Openings

A free opening represents an area on the surface of a solid object (e.g., block, partition), or an area on a planar object such as a wall, through which a fluid is free to flow in any direction. In most cases, an opening represents a hole on the boundary of the room where the model fluid is exposed to the external environment.

By default, **Airpak** calculates the rate of flow through a free opening as part of the solution.

For free openings located on room walls, **Airpak** computes the rate of flow through the opening based on an external static pressure. The default external pressure and temperature are the ambient temperature specified under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). For cases where the external fluid velocity is not perpendicular to the plane of the opening, **Airpak** also allows you to specify the velocity directional components as well as specify species concentrations and turbulence parameters at the opening. In addition, you can specify boundary profiles for pressure, temperature, velocity, species concentrations, and turbulence parameters at a free opening.

11.3 Recirculation Openings

Recirculation openings model recirculation devices such as heating or cooling units. In such a device, the fluid is withdrawn from the room through the extract section of the opening and returned to the room through the supply section, as shown in Figure 11.3.1. The extract and supply sections of the recirculation opening can differ from one another in both size and geometry.

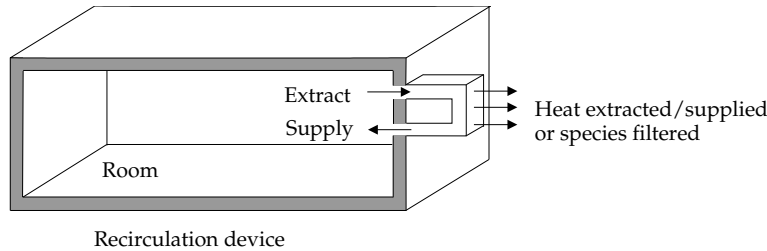


Figure 11.3.1: External Recirculation Cooling Device

You can model an internal recirculation device by placing the extract and supply sections of a recirculation opening on two different sides of an adiabatic block, as shown in Figure 11.3.2.

i A conducting solid block cannot be used to represent an internal recirculation device.

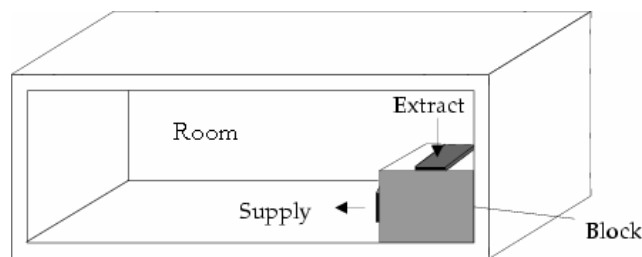


Figure 11.3.2: Internal Recirculation Device

11.3.1 Recirculation Mass Flow Rate

Airpak provides three methods of specifying the flow rate through a recirculation device:

- total mass flow through the opening
- mass flow rate per unit area of the opening
- total volumetric flow through the opening

Because the extract and supply sections can differ in size, the per-unit-area specification can result in different mass fluxes but the same mass flow rate for each section of the opening. Such specifications can be used to model devices that extract or supply a fluid anywhere in the recirculation loop.

11.3.2 Flow Direction for Recirculation Openings

By default, the flow direction of the fluid passing through the extract or supply section of a recirculation opening is normal to the plane of the section. However, you can specify that the flow leaves the supply section at an angle, as shown in Figure 11.3.3. Note that Airpak uses the direction parameters only to determine the direction of fluid flow; they do not affect the magnitude of the velocity.

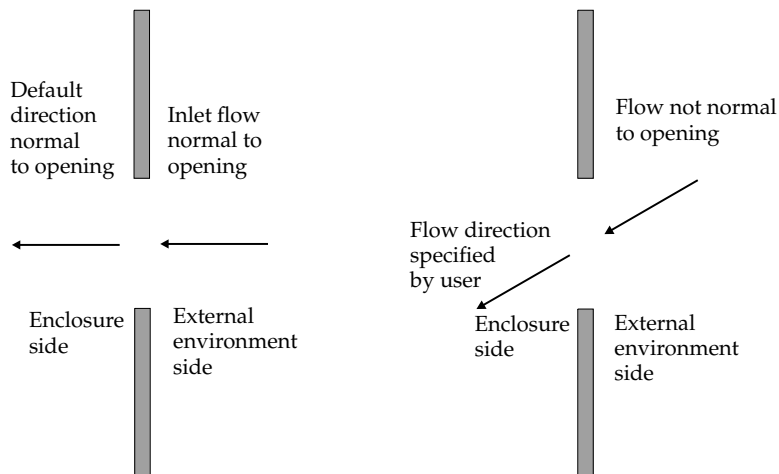


Figure 11.3.3: Flow Direction Through Opening Section

11.3.3 Recirculation Opening Thermal Specifications

When fluid exits and re-enters the enclosure through a recirculation device, its temperature can increase or decrease. Airpak computes the temperature of the re-entering fluid

(T_{supply}) based on the temperature of the exiting fluid (T_{extract}) and the thermal change applied to the fluid as it passes through the device.

Airpak provides three methods for computing T_{supply} . The first method requires the specification of a constant temperature change (ΔT) applied to the fluid within the device. In this case, T_{supply} is computed by

$$T_{\text{supply}} = T_{\text{extract}} + \Delta T \quad (11.3-1)$$

where T_{extract} is the temperature of the enclosure fluid averaged over the face of the extract section.

The second method requires the specification of the amount of heat (ΔH) input to or extracted from the fluid by the recirculation device. In this case, T_{supply} is computed by

$$T_{\text{supply}} = T_{\text{extract}} + \frac{\Delta H}{c_p \dot{m}} \quad (11.3-2)$$

where c_p is the fluid specific heat of the default fluid material selected in the **Basic parameters** panel (see Section 6.4.7) and \dot{m} is the mass flow rate through the device.

The third method requires the specification of the conductance ($h_1 A$) and the external temperature (T_{external}). In this case, T_{supply} is computed by



$$T_{\text{supply}} = T_{\text{extract}} - \frac{h_1 A (T_{\text{extract}} - T_{\text{external}})}{c_p \dot{m}} \quad (11.3-3)$$

where c_p is the specific heat of the default fluid material selected in the **Basic parameters** panel (see Section 6.4.7), and \dot{m} is the mass flow rate through the device.

11.3.4 Recirculation Opening Species Filters

When fluid exits and re-enters the room through a recirculation device, you can specify an increase or decrease of species in the recirculation loop. See Chapter 21 for details on modeling species transport.

11.4 Adding an Opening to Your Airpak Model

To include an opening in your **Airpak** model, click on the  button in the **Object** creation toolbar and then click on the  button to open the **Openings** panel, shown in Figures 11.4.1 and 11.4.2.

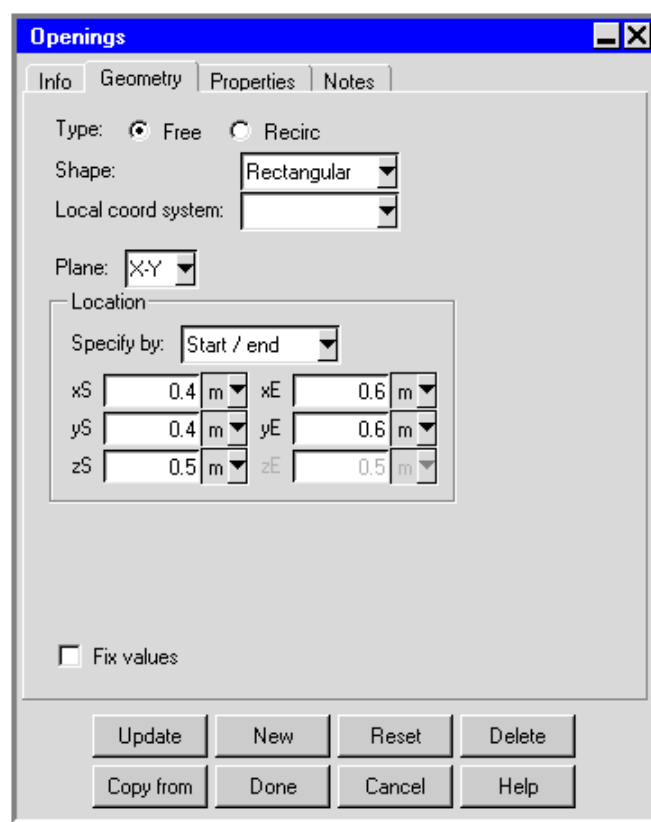


Figure 11.4.1: The **Openings** Panel for a Free Opening (**Geometry** Tab)

The procedure for adding an opening to your **Airpak** model is as follows:

1. Create an opening. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the opening, if required. See Section 7.4.1 for details.
3. Change the graphical style of the opening, if required. See Section 7.4.2 for details.
4. In the **Geometry** tab, specify the type of the opening by selecting **Free** or **Recirc** next to **Type**. The lower part of the panel will change depending on your selection of the **Type**.

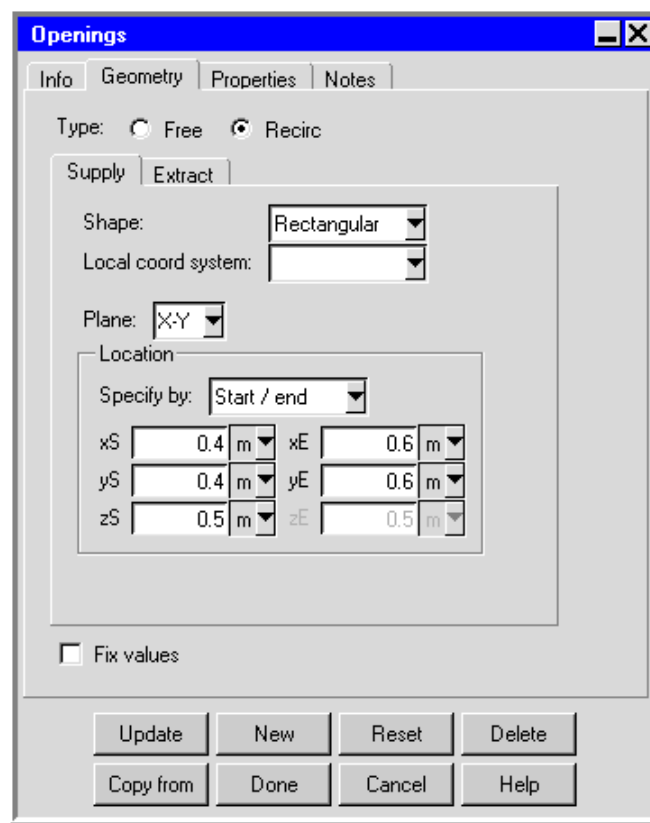


Figure 11.4.2: The Openings Panel for a Recirculation Opening (Geometry Tab)

- Specify the geometry, position, and size of the opening. There are four different kinds of geometry for **Free** type opening and five different kinds of geometry for **Recirc** type opening available in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.

i You can specify different geometries and dimensions for the **Supply** and **Extract** sections of a recirculation opening.

- In the **Properties** tab, specify the characteristics related to the selected opening **Type**. These options are described below.

11.4.1 User Inputs for a Free Opening

To specify a free opening, select **Free** next to **Type** in the **Openings** panel. The user inputs for a free opening are shown in Figure 11.4.3.

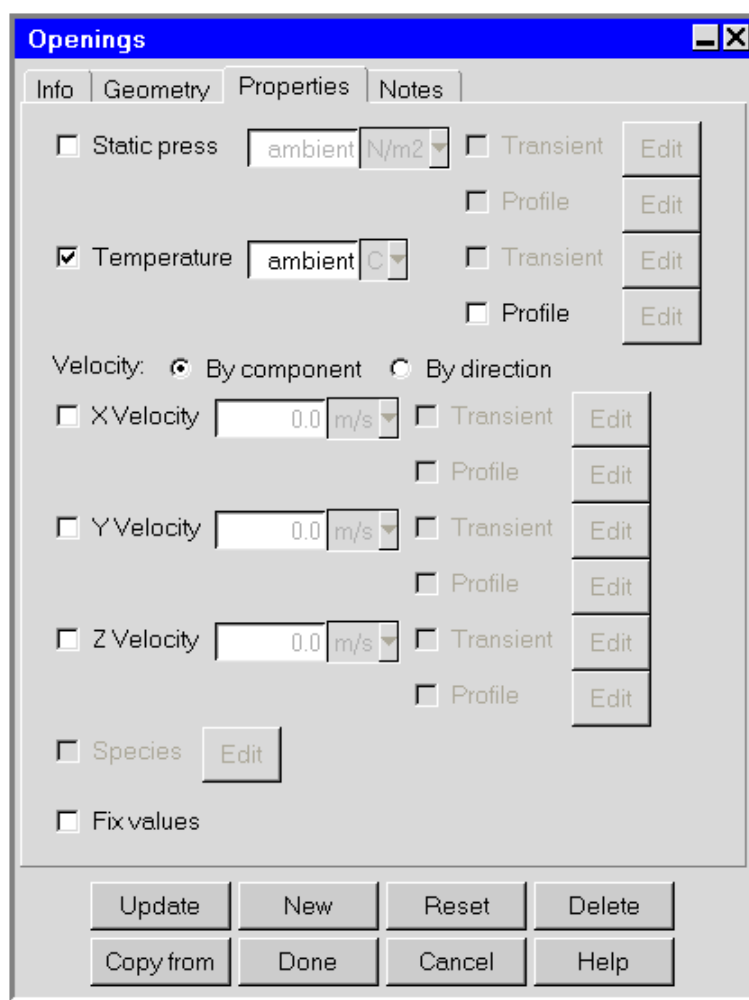


Figure 11.4.3: The Openings Panel for a Free Opening (Properties Tab)

The steps for defining a free opening are as follows:

1. Specify the static pressure (**Static press**) external to the room. To specify a uniform static pressure, enter a value in the **Static press** text entry box. To define a spatial profile for the static pressure external to the room, select **Profile** next to **Static press** and click **Edit** to open the **Curve specification** panel (described below).
2. Specify the **Temperature** of the fluid external to the room. By default, the external temperature is the temperature specified under **Ambient values** in the **Basic param-**

eters panel (see Section 6.4.6). To specify a uniform temperature, enter a value in the **Temperature** text entry box. To define a spatial profile for the temperature external to the room, select **Profile** next to **Temperature** and click **Edit** to open the **Curve specification** panel (described below).

3. Specify the velocity vector for flow across the opening. To specify a uniform x velocity, y velocity, or z velocity, enter a value in the **X Velocity**, **Y Velocity**, or **Z Velocity** text entry box. To define a spatial profile for the x velocity, y velocity, or z velocity, select **Profile** next to **X Velocity**, **Y Velocity**, or **Z Velocity** and click **Edit** to open the **Curve specification** panel (described below).
4. If you are setting up a transient simulation, you can specify the **Static press**, **Temperature**, **X Velocity**, **Y Velocity**, or **Z Velocity** as a function of time. These options are available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the static pressure, temperature, x velocity, y velocity, or z velocity, select **Transient** and click **Edit** in the **Openings** panel. See Chapter 20 for more details on transient simulations.



Note that **Airpak** cannot use both a spatial profile and a transient profile for the static pressure, temperature, x velocity, y velocity, or z velocity. If you specify both profile types for any of these parameters, **Airpak** will use the transient profile and ignore the spatial profile.

5. Specify the inlet or outlet species concentrations for the opening if **X Velocity**, **Y Velocity**, and/or **Z Velocity** are defined, if required. You can input the species concentrations for the opening using the **Species concentrations** panel. To open this panel, select **Species** in the **Openings** panel and then click **Edit**. See Chapter 21 for details on modeling species transport.
6. Specify the turbulence parameters for the opening, if required. This option is available only if you selected the **Two equation** or **RNG k - ϵ** model in the **Problem setup** panel (Figure 6.4.1). See Section 6.4.3 for details on determining turbulence parameters for objects in your **Airpak** model.

Select **Turbulence** in the **Openings** panel and click **Edit** to open the **Turbulence equation parameters** panel (Figure 11.4.4).

There are two options for specifying turbulence parameters:

Turbulent intensity/turbulent length scale allows you to specify the turbulence intensity and length scale for the opening.

Specify values for the **Turbulent intensity** and the **Turbulent length scale**. The default value of the turbulence intensity is 10%. By default, **Airpak** will use a **default** value for the length scale. The **default** value calculated by **Airpak** is 10% of the length of the shortest side of the opening.

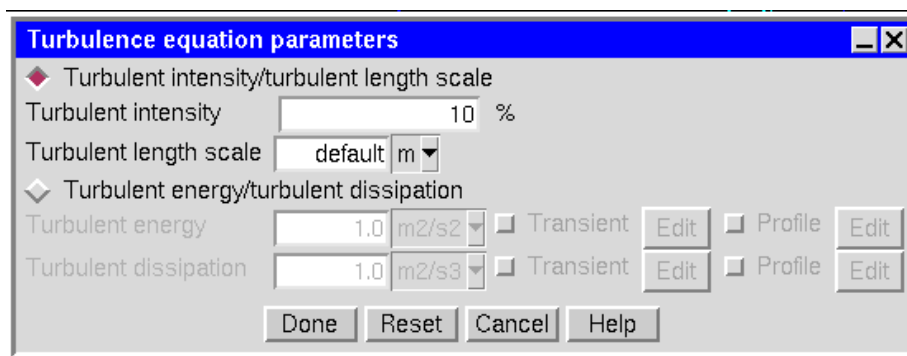


Figure 11.4.4: The Turbulence equation parameters Panel

Turbulent energy/turbulent dissipation allows you to specify the **Turbulent energy** and **Turbulent dissipation** rate for the opening. To specify a uniform turbulent kinetic energy or turbulent dissipation rate, enter a value in the **Turbulent energy** or **Turbulent dissipation** text entry box. To define a spatial profile for the turbulent kinetic energy or turbulent dissipation rate, select **Profile** next to **Turbulent energy** or **Turbulent dissipation** and click **Edit** to open the **Curve specification** panel (described below).

If you are setting up a transient simulation, you can specify the **Turbulent energy** or **Turbulent dissipation** as a function of time. These options are available if you have selected **Transient** under **Time variation** in the **Problem setup** panel. To edit the transient parameters for the turbulent kinetic energy or turbulent dissipation rate, select **Transient** and click **Edit** in the **Turbulence equation parameters** panel. See Chapter 20 for more details on transient simulations.

i Note that **Airpak** cannot use both a spatial profile and a transient profile for the turbulent kinetic energy or turbulent dissipation rate. If you specify both profile types for either of these parameters, **Airpak** will use the transient profile and ignore the spatial profile.

i Note that **Airpak** cannot use both a spatial profile and a transient profile for the turbulent kinetic energy or turbulent dissipation rate. If you specify both profile types for either of these parameters, **Airpak** will use the transient profile and ignore the spatial profile.

7. If you have enabled solar loading in the **Basic parameters** panel, specify the **Solar** behavior for the opening. See Section 22.5 for details.

Using the Curve specification Panel to Specify a Spatial Boundary Profile

You can define a spatial boundary profile using the Curve specification panel (Figure 11.4.5). To open the Curve specification panel, select Profile in the Openings panel and click Edit.

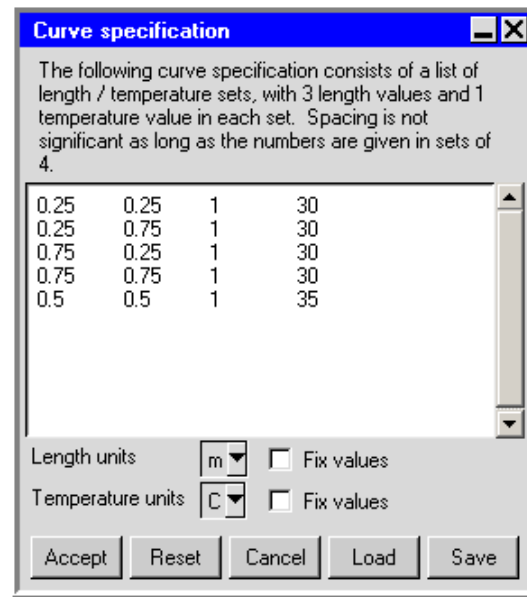


Figure 11.4.5: The Curve specification Panel

To define a profile, specify a list of (x, y, z) coordinates and the corresponding values in the Curve specification panel. For example, the first line in Figure 11.4.5 specifies a temperature of 30°C at (0.25, 0.25, 1). The data in Figure 11.4.5 specify a variation of temperature on the plane $0.25 \leq x \leq 0.75$, $0.25 \leq y \leq 0.75$, $z = 1$. The values in the right-hand column are the external static pressure, external temperature, x velocity, y velocity, or z velocity, depending on your selection in the Openings panel. Click **Accept** when you have finished defining the profile; this will store the values and close the Curve specification panel. Airpak will interpolate the data you provide in the Curve specification panel to create a profile for the whole boundary.

i If the starting point of the opening object is located at (x_0, y_0, z_0) , then the first point of the profile should be (x_0, y_0, z_0, a_0) , where a_0 is the corresponding value for that point. However, if the first point in the profile is has a different value, for example (x_1, y_1, z_1, a_0) , Airpak will automatically translate the first point to (x_0, y_0, z_0, a_0) , and the rest of the profile points will be shifted by $(x_1 - x_0, y_1 - y_0, z_1 - z_0)$. This translation of point locations will not affect the values of the variables (a_n), and is also useful if the opening is ever translated within the model. In this way, you will not have to recreate the profile file or re-enter values in the Curve specification panel.

This translation feature also applies to other objects that allow the specification of point profiles (i.e., walls, blocks, and resistances).

To load a previously defined profile, click on **Load**. (See Section 28.2.8 for details on saving contour data and using them as a profile.) This will open the **Load curve** file selection dialog box. Select the file containing the profile data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the units used in the profile data you are loading, you should select the appropriate units in the **Curve specification** panel before you load the profile. If you want to view the data after you have loaded it, using different units than the default units in the **Curve specification** panel, select the relevant **Fix values** options and then select the appropriate units from the unit definition lists.

If you want to load a curve file that you have created outside of **Airpak**, you will need to make sure that the first three lines of the file before the data contain the following information:

1. the number of data sets in the file (usually 1)
2. the unit specifications for the file, which can be obtained from the **Curve specification** panel (e.g., **units m C**)
3. the number of data points in the file (e.g., 10)

Using the above example, the first three lines of the curve file would be

```
1
units m C
10
```

The actual data points should be entered in the same way as you would enter them in the **Curve specification** panel.



If you want to load a curve from an Excel file, make sure that you also save the file as formatted text (space delimited) before reading it into **Airpak**.

To save a profile, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the profile data are to be saved.

11.4.2 User Inputs for a Recirculation Opening

To specify a recirculation opening, select **Recirc** next to **Type** in the **Geometry** tab of the **Openings** panel. The user inputs for the physical properties of a recirculation opening are shown in Figure 11.4.6.

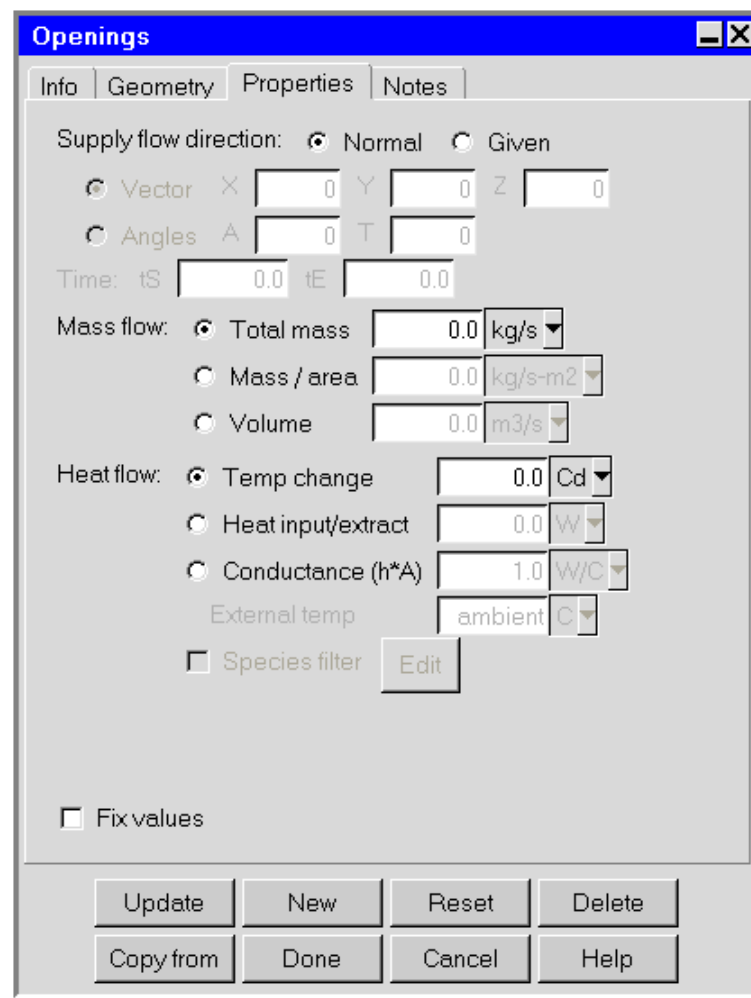


Figure 11.4.6: The Openings Panel for a Recirculation Opening (Properties Tab)

The steps for defining a recirculation opening are as follows:

1. Specify the **Supply flow direction**. There are two options:

- If the fluid flows into the room normal to the opening, select **Normal**.
- To specify the flow angle of the fluid entering the room through the supply section of the opening, select **Given**. There are two options:

Vector allows you to specify the direction vector (X, Y, Z) for the flow. Only the direction of the vector is used by **Airpak**; the magnitude is ignored.

Angles allows you to specify the direction of the flow entering the room in terms of angles relative to the inward-pointing normal direction. You can specify two angles, **A** (α) and **T** (θ), to account for incoming flow not aligned with the normal direction. If the normal inward vector points in the positive direction, then α and θ must have values in the range $-90^\circ < \text{angle} < 90^\circ$, with $\alpha = \theta = 0^\circ$ representing flow in the positive normal direction. If the normal inward vector points in the negative direction, then α and θ must have values in the range $90^\circ < \text{angle} < 270^\circ$, with $\alpha = \theta = 180^\circ$ representing flow in the negative normal direction. Since the opening may be oriented in any one of three planes (x - y , y - z , or x - z), the roles of α and θ will vary depending on the orientation of the plane of the opening. An example is shown in Figure 11.4.7, where the opening is in the x - y plane and the normal inward vector is pointing in the positive direction.

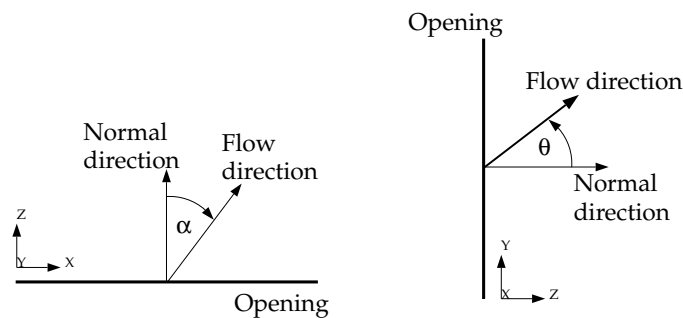


Figure 11.4.7: Definition of Angles for the Supply Section of an Opening

2. (transient problems only) Specify the start time (**tS**) and the end time (**tE**) of the period when the opening is active.
3. Specify the mass flow rate through the opening. You can specify either a **Total mass** flow rate, **Mass / area** flow rate, or **Volume** flow rate. If you specify a **Mass / area** flow rate, and you have specified different cross-sectional areas for the **Supply** and **Extract** sections of the opening, then these sections will have the same total mass

flow rate but different mass fluxes. The extract side mass flux is used to compute the mass flow rate.

4. Specify the thermal change applied to the fluid in the recirculation loop. There are three options:

Temp change specifies a constant increase or decrease of temperature applied to the fluid in the recirculation loop (see Equation 11.3-1).

Heat input/extract specifies a constant heat flow into or out of the fluid as it passes through the recirculation loop (see Equation 11.3-2).

Conductance ($h \cdot A$) specifies a conductance to model the heat input/output of the fluid from your system. Enter the value of the conductance ($h_1 A$ in Equation 11.3-3) in the **Conductance ($h \cdot A$)** text entry field.

Specify the external temperature (T_{external} in Equation 11.3-3) next to **External temp.** The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

5. Specify the change of species in the recirculation loop. You can specify an increase or decrease in the species in the recirculation loop using the **Species filter efficiency** panel. To open this panel, select **Species filter** in the **Openings** panel and then click **Edit**. See Chapter 21 for details on modeling species transport.

A person object is a three-dimensional modeling object. You can specify the shape and size of the person in **Airpak** either by specifying the relative proportions of the person, or by specifying the sizes of the individual parts of the body. You can choose whether the person is sitting or standing (see Figure 12.0.1). A person can exchange radiation with other objects in the model.

To configure a person in the model, you must specify its position and dimensions, and whether the person is standing or seated. You must also specify the thermal conditions of the person.

In this chapter, information about the characteristics of the person object in **Airpak** is presented in the following sections:

- Section 12.1: Location and Dimensions
- Section 12.2: Thermal Options
- Section 12.3: Adding a Person to Your **Airpak** Model

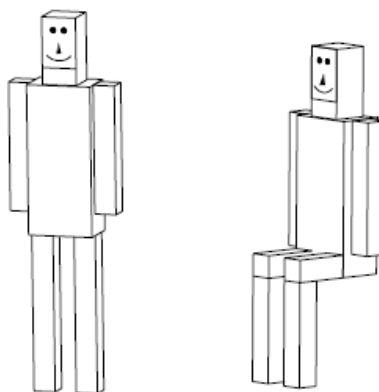


Figure 12.0.1: Standing Person and Seated Person

12.1 Location and Dimensions

Airpak allows you to specify the position of the person in your model. The position consists of four specifications:

- the posture of the person (i.e., whether the person is standing or seated)
- the location of the person in the room
- the direction in which the person is facing
- the “up” direction for the person

You can specify the following proportions for a person:

- the total height of the person, as defined in Figure 12.1.1. Note that the total height of a seated person includes the length of the lap.
- the width of the person’s body, as defined in Figure 12.1.1
- the depth of the person’s body, as defined in Figure 12.1.1

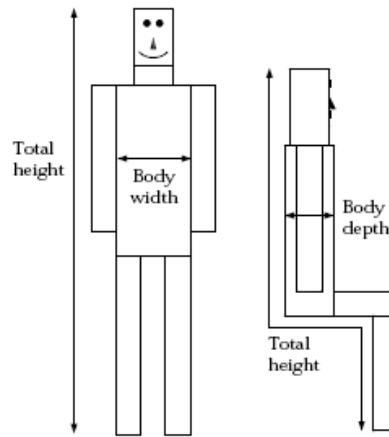


Figure 12.1.1: Proportions of a Person

- the head fraction, which is given by

$$\text{head fraction} = \frac{\text{height of head}}{\text{total height}} \quad (12.1-1)$$

- the body fraction, which is given by

$$\text{body fraction} = \frac{\text{height of body}}{\text{total height}} \quad (12.1-2)$$

- the lap fraction (if seated), which is given by

$$\text{lap fraction} = \frac{\text{length of thigh}}{\text{total height}} \quad (12.1-3)$$

You can also specify the individual sizes of the following parts of a person's body:

- head
- face
- body
- arms
- legs (vertical part only, if seated)
- thighs (if seated)

12.2 Thermal Options



You can specify the thermal boundary conditions of a person using one of three options:

Heat/area sets the flux of heat per unit area of the person. This is the default thermal boundary condition for the person, and the default value is 1 met (1 metabolic rate unit = 58.2 W/m²).

Total heat sets the total power output of the person.

Temperature sets the fixed value of the person's temperature.

12.3 Adding a Person to Your Airpak Model

To include a person in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Persons panel, shown in Figure 12.3.1.

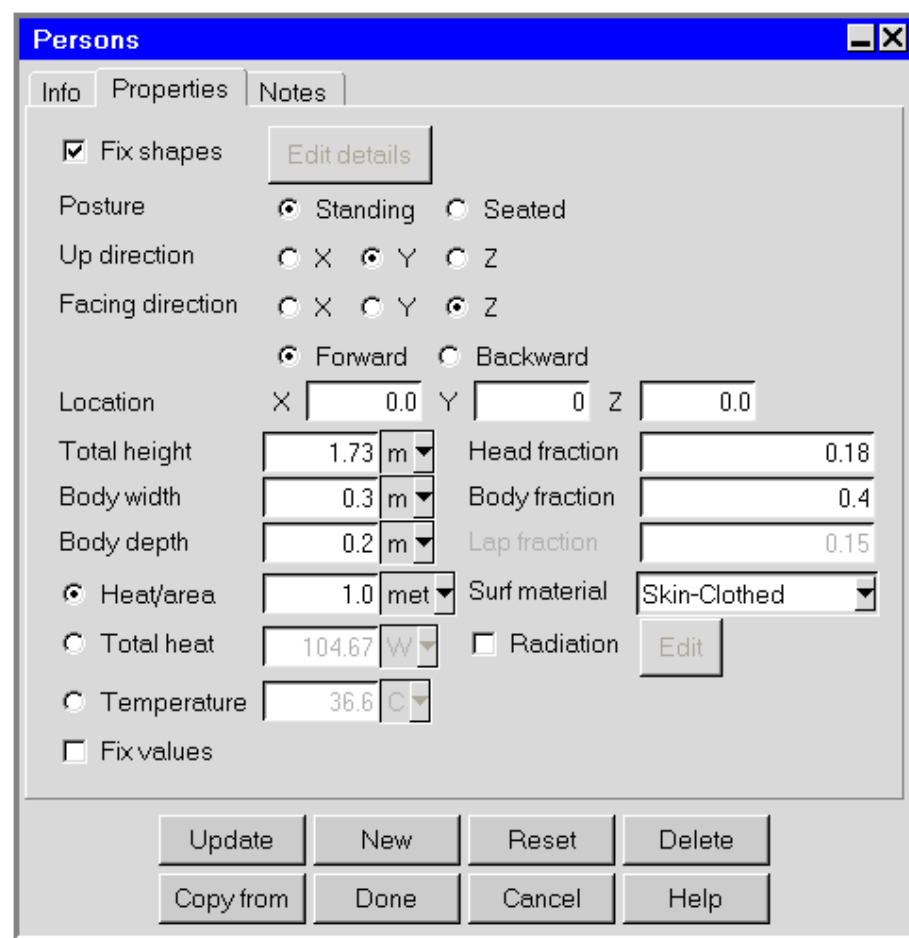


Figure 12.3.1: The Person Panel

The procedure for adding a person to your Airpak model is as follows:

1. Create a person. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the person, if required. See Section 7.4.1 for details.
3. Change the graphical style of the person, if required. See Section 7.4.2 for details. Note that you can turn off the display of the person's face by deselecting the **Decoration** option for the person in the **Graphical styles** panel (see Section 6.3.6 for details about using the **Graphical styles** panel).
4. Specify the position of the person in the room. (See also Section 7.3.7 for details on repositioning an object.)
 - (a) Select whether the person is **Standing** or **Seated**.
 - (b) Select the **Up direction** for the person. The up direction is the direction of the vector from the person's feet to the person's head.
 - (c) Select the direction in which the person is facing next to **Facing direction**. You can also specify whether the person is facing **Forward** or **Backward**. If you specify that the person is facing **Forward**, they will face toward the positive direction of the axis selected next to **Facing direction**.
 - (d) Specify the **Location** of the person in the model.
5. Specify the size of the person. (See also Section 7.3.6 for details on resizing an object.) You can specify the size of the person in two ways: by specifying the relative proportions of the person, or by specifying the sizes of the individual parts of the body.
 - To specify the relative proportions of the person:
 - (a) Make sure that **Fix shapes** is selected.
 - (b) Specify the **Total height** of the person. Note that for a seated person, this height includes the length of the lap.
 - (c) Specify the **Body width** and **Body depth** of the person.
 - (d) Specify the **Head fraction**, which is the height of the head as a fraction of the total height.
 - (e) Specify the **Body fraction**, which is the height of the body as a fraction of the total height.
 - (f) Specify the **Lap fraction** (if the person is seated), which is the length of the thighs as a fraction of the total height.
 - To specify the sizes of the individual parts of the body:

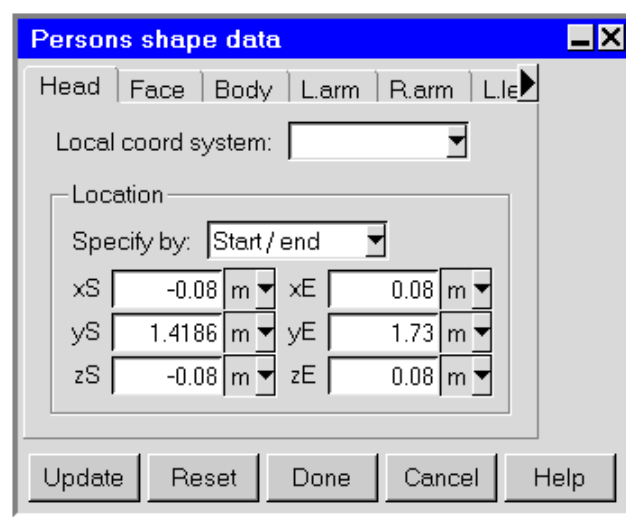


Figure 12.3.2: The Person shape data Panel

- (a) Deselect **Fix shapes** and click **Edit details** to open the **Person shape data** panel (Figure 12.3.2).
- (b) Select one of the body parts under **Section**.
- (c) Keep the default **Origin** of (0, 0, 0), or specify the **Origin** for the body part (if required).
- (d) Select **Start/end** and enter values for the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE) of the body part, or select **Start/length** and enter values for the start coordinates (xS, yS, zS) and lengths of the sides (xL, yL, and zL) of the body part.

Note that the left thigh (L.thigh) and right thigh (R.thigh) are specified as **None** if the person is standing. If you want to specify a thigh for a standing person, select **Default** and specify the starting and ending points (or the starting point and the lengths of the sides) for the thigh.

Note that the face is rectangular and not a prism like the other body parts. Specify the plane in which the face lies (Y-Z, X-Z, or X-Y). Select **Start/end** and enter values for the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE) of the face, or select **Start/length** and enter values for the start coordinates (xS, yS, zS) and lengths of the sides (xL, yL, and zL) of the face.

- (e) Click **Done** to save the changes to the person and close the **Person shape data** panel.
6. Specify the thermal conditions of the person in the **Person** panel. There are three options:

Heat/area allows you to enter the flux of heat per unit area of the person.

Total heat allows you to enter a value for the total heat output of the person.

Temperature allows you to specify the fixed value of the person's temperature.

7. Specify the surface material (**Surf material**) to be used for the person. This material defines the roughness and emissivity of the person. By default, the surface material is specified as **Skin-Clothed**. To change the surface material, select a material from the **Surf material** drop-down list. The surface roughness parameters and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.
8. Select **Radiation** if the person is subject to radiative heat transfer. This option is available if you have selected **On** next to **Radiation** in the **Problem setup** panel and you have not selected the **Discrete ordinates radiation model** option. You can modify the default radiation characteristics of the person (e.g., view factor) by using the **Radiation specification** panel. To open this panel, select **Radiation** in the **Person** panel and then click **Edit**. See Chapter 22 for details on radiation modeling.

Walls are objects that constitute all or part of the room boundary. Walls can be specified with respect to their thickness, velocity, and heat flux. Wall geometries include rectangular, 2D polygon, circular, and inclined.

By default, the room sides are zero-thickness walls with zero velocity and zero heat flux boundary conditions. To modify the characteristics of the room boundaries, you must create and specify thermal conditions on external walls. To construct a wall internal to the enclosure, you must specify a thickness for the wall. The inner surface of an external wall is in direct contact with the enclosure fluid, and its outer surface is exposed to the external environment. A no-slip velocity boundary condition is applied at the inner surface of the wall. For turbulent flows, you can also specify the surface roughness of the wall, the effect of which is to increase resistance to the flow.

Throughout this section, reference will be made to the inside and outside of a wall. The inside of the wall is the side in contact with the fluid in the room; the outside of the wall is the side exposed to the conditions external to the room. For a wall with zero thickness, the inner and outer sides of the wall coincide; however, the inner and outer surface materials can be different.

To configure a wall in the model, you must specify its geometry (including location and dimensions), velocity, thickness, thermal characteristics, and the material the wall is made from.

In this chapter, information about the characteristics of a wall is presented in the following sections:

- [Section 13.1: Geometry, Location, and Dimensions](#)
- [Section 13.2: Surface Roughness](#)
- [Section 13.3: Wall Velocity](#)
- [Section 13.4: Thermal Boundary Conditions](#)
- [Section 13.5: External Thermal Conditions](#)
- [Section 13.6: Constructing Multifaceted Walls](#)
- [Section 13.7: Adding a Wall to Your Airpak Model](#)

13.1 Geometry, Location, and Dimensions

Wall location and dimension parameters vary according to the wall geometry. Wall geometries include rectangular, 2D polygon, circular, and inclined. These geometries are described in Section 7.4.4.

13.1.1 Wall Thickness

For walls with non-zero thickness, **Airpak** automatically extends the wall inward or outward from the specified plane of the wall. The direction of this extension is determined by the sign of the specified thickness relative to the coordinate axis normal to the plane of the wall. If the thickness value is positive, then the expansion is in the positive direction of the coordinate axis normal to the wall, as shown in Figure 13.1.1. If the thickness value is negative, the expansion is in the negative direction.

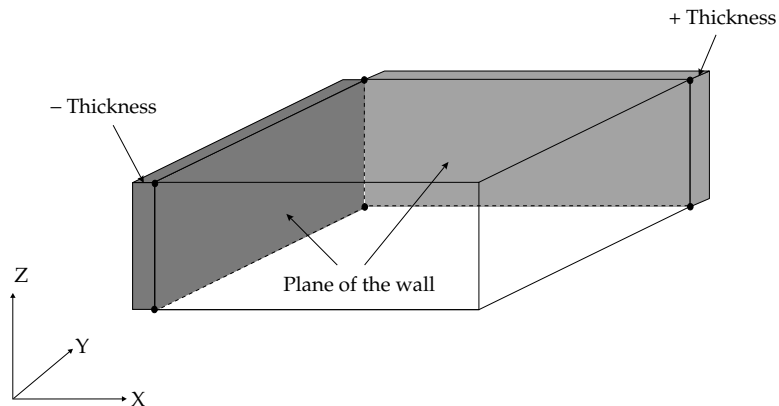


Figure 13.1.1: Wall Thickness Direction

Walls with non-zero thickness can conduct heat either through or along the plane of the wall, and can do so anisotropically, i.e., according to thermal conductivities specific to each direction (defined as part of the properties of the solid material specified for the wall). They must possess a physical thickness so that **Airpak** can mesh the interior of the wall. Effective-thickness walls have the same properties as non-zero-thickness walls, except that they have no physical thickness; they can possess only an effective thickness.

13.2 Surface Roughness

In fluid dynamics calculations, it is common practice to assume that boundary surfaces are perfectly smooth. In laminar flow, this assumption is valid, because the length scales of typical rough surfaces are much smaller than the length scales of the flow. In turbulent flow, however, the length scales of the flow eddies are much smaller than laminar length scales; therefore, it is sometimes necessary to account for surface roughness. Surface roughness acts to increase resistance to flow, leading to higher rates of heat transfer.

Airpak assumes, by default, that all surfaces of a wall in contact with a fluid are hydrodynamically smooth, and applies standard no-slip boundary conditions. For turbulent-flow simulations in which roughness is significant, however, you can specify a roughness factor for the entire wall. This roughness factor is defined as part of the properties of the surface material specified for the wall. The purpose of the roughness factor is to approximate the average height of the surface texture on the wall.

13.3 Wall Velocity

In most cases, walls represent stationary objects, but occasionally circumstances arise in which the model requires walls that move. For example, if the moving belt shown in Figure 13.3.1 is located at the room boundary, it can be represented as a wall moving at a fixed velocity. Moving walls always have zero thickness.

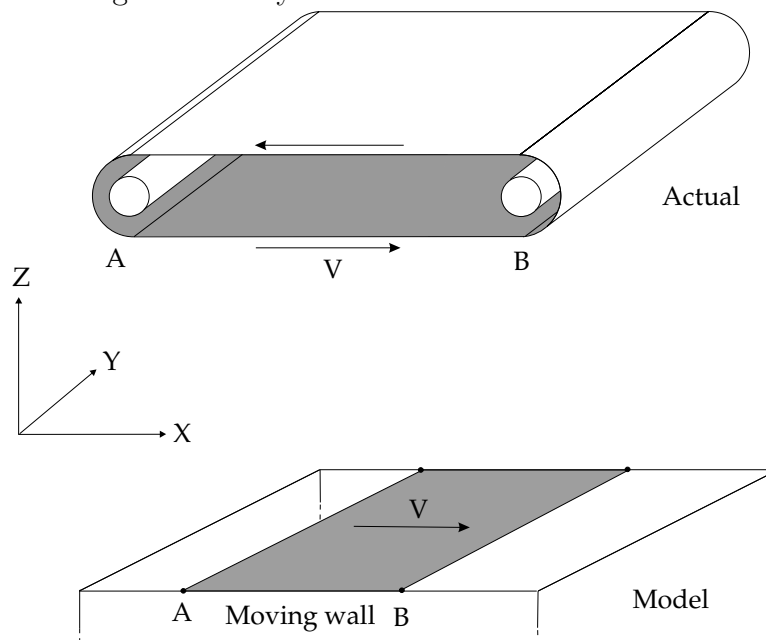


Figure 13.3.1: Moving Walls

When a wall is specified as moving, it is allowed to move only in the plane of the wall, i.e., there is no translation of the wall outside its plane. Also, fluid in contact with the wall is pulled along with the wall because of the no-slip condition. In such cases, the velocity of the wall relative to the stationary enclosure must be set in the plane of the wall. For the example shown in Figure 13.3.1, the wall must be specified with a velocity of V in the x direction and zero velocity in the y direction. **Airpak** automatically imposes a velocity of zero in the direction normal to the plane of the wall (the z direction in this example).

13.4 Thermal Boundary Conditions

External walls can have two distinct thermal boundary conditions: a specified heat flux or a fixed temperature. In both cases, the wall is assumed to have zero thickness by default. When neither parameter is known, the external wall can model conditions applied to the outer side of the wall that enable **Airpak** to compute the heat flux and the temperature on the inside of the wall, as shown in Figure 13.4.1.

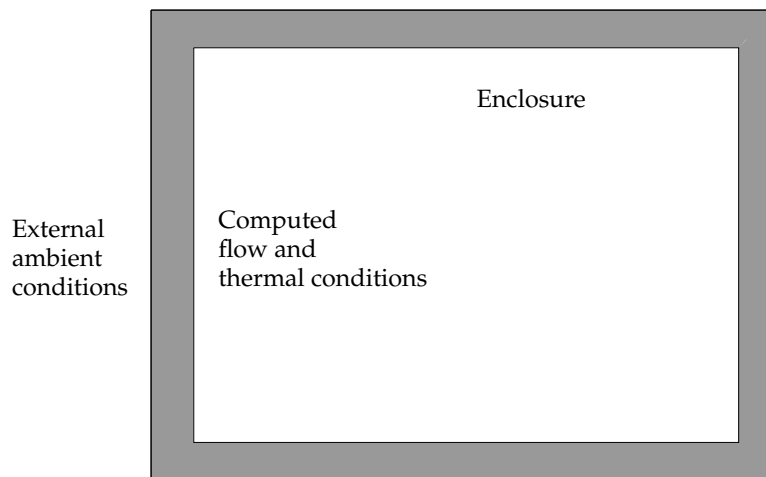


Figure 13.4.1: Thermal Boundary Conditions

13.4.1 Specified Heat Flux

The simplest thermal boundary condition for a wall is that of a specified heat flux. In this case, the amount of heat that can pass through the wall is specified as a constant value or as a spatial boundary profile, expressed as power per unit area (heat flux). For an adiabatic wall, the heat flux is zero. For walls with non-zero thickness, the heat flux is applied on the outer surface of the wall, as shown in Figure 13.4.2. An example of the use of this capability might be modeling solar loading. For transient problems, you can specify the variation of heat flux with time.

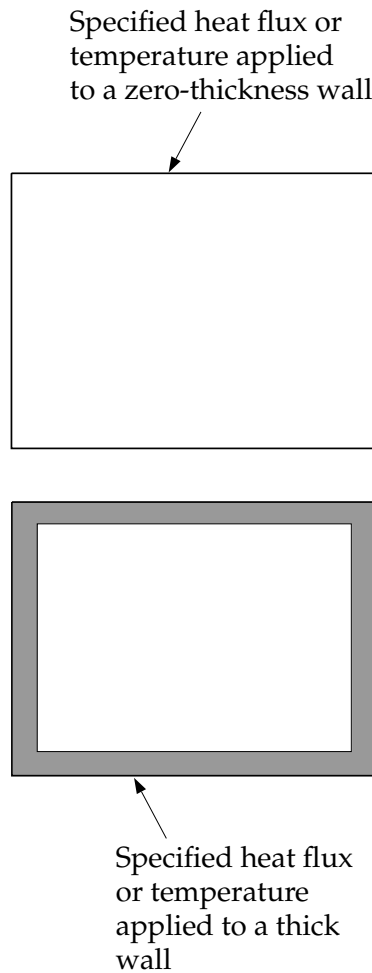


Figure 13.4.2: Specified Heat Flux or Specified Temperature

13.4.2 Specified Temperature

In most cases, the temperature at the inside surface of a wall is unknown. However, if the temperature is known (e.g., if the wall abuts some material or object whose absolute temperature is known or easily determined), the temperature can be directly applied as the thermal boundary condition at the inside surface of the wall, either as a constant value or as a spatial boundary profile. For walls with non-zero thickness, the temperature is applied on the outer surface of the wall, as shown in Figure 13.4.2. For transient problems, you can specify the variation of temperature with time.

13.5 External Thermal Conditions

In some situations, neither the heat flux nor the temperature on the inner side of the external wall is known. In this case, the external wall and the heat transfer from the surface of the wall to the external environment can be modeled as a wall with a specified thickness. Alternatively, a zero-thickness wall can be specified with a convective and/or radiative heat transfer condition applied directly to the wall.

To compute heat transfer through a wall with non-zero thickness to the external environment, the energy equation is used. For these calculations, you must specify the conductivity of the wall (k , defined as part of the properties of the solid material specified for the wall) and the wall thickness (see Figure 13.5.1). For transient problems, you must also specify the density and specific heat of the wall (defined as part of the properties of the solid material specified for the wall).

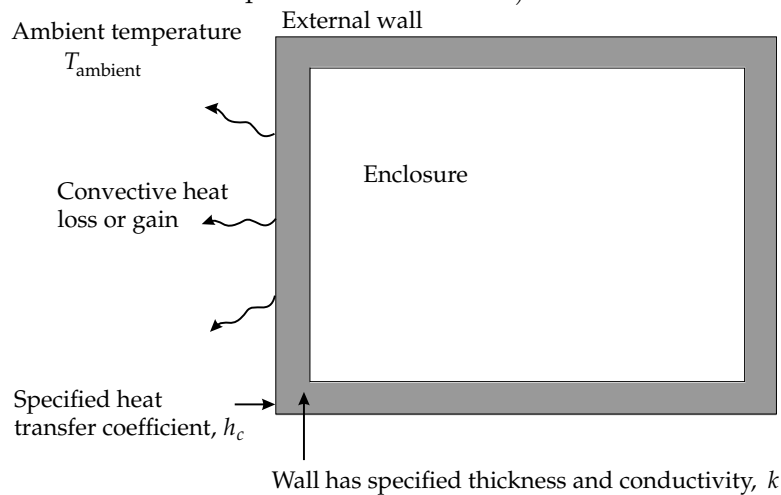


Figure 13.5.1: Wall with Non-Zero Thickness

Heat can also be lost or gained through the outside surface of an external wall by convective heat transfer and radiative heat transfer. The inner surface of an external wall can also convect heat or radiate to objects within the room. These two types of heat transfer conditions can be applied to walls with either zero or non-zero thickness.

13.5.1 Convective Heat Transfer

The convective heat transfer boundary condition can be written as

$$q_{\text{conv}} = h_c(T_{\text{wall}} - T_{\text{ambient}}) \quad (13.5-1)$$

where q_{conv} is the heat gain or loss, h_c is the heat transfer coefficient, T_{wall} is the computed wall temperature, and T_{ambient} is the external specified ambient temperature, i.e., the

temperature of the external fluid. From Equation 13.5-1, if the wall temperature is greater than the ambient temperature, an amount of heat proportional to the temperature difference is lost to the environment from the room. Similarly, if the wall temperature is lower than the external temperature, heat is transferred into the room. In contrast, the specified heat flux option prescribes a specified heat gain or loss at the wall independent of the wall or ambient temperatures.

The heat transfer coefficient can be specified as a constant value, as a spatial profile, or as a function of temperature. For transient problems, you can specify the variation of the heat transfer coefficient with time.

In circumstances where heat transfer through the wall is not required (e.g., if the wall is thin and made of a highly conducting material), the thickness of the wall can be set to zero. In this case, the convective heat transfer boundary condition is applied directly to the inside surface of the external wall.

13.5.2 Radiative Heat Transfer

The radiative heat transfer boundary condition provides for heat transfer between the room wall and a remote surface. It can be written as

$$q_{\text{rad}} = e\sigma F(T_{\text{wall}}^4 - T_{\text{remote}}^4) \quad (13.5-2)$$

where q_{rad} is the heat gain or loss due to radiation (i.e., the net radiant flux from the wall surface), T_{remote} is the temperature of the remote surface, σ is the Stefan-Boltzmann constant, F is a view factor specifying the fraction of radiant energy that is intercepted by the wall, and e is the surface emissivity of the wall (defined as part of the properties of the surface material specified for the wall). An example of the use of this capability might be a room within a room whose walls are at a constant temperature, as shown in Figure 13.5.2. In this case, the room walls exchange radiant energy with the remote surfaces.

In circumstances where heat transfer through the wall is not required (e.g., if the wall is thin and made of a highly conducting material), the thickness of the wall can be set to zero. In this case the radiative heat transfer boundary condition is applied directly to the inside surface of the external wall.

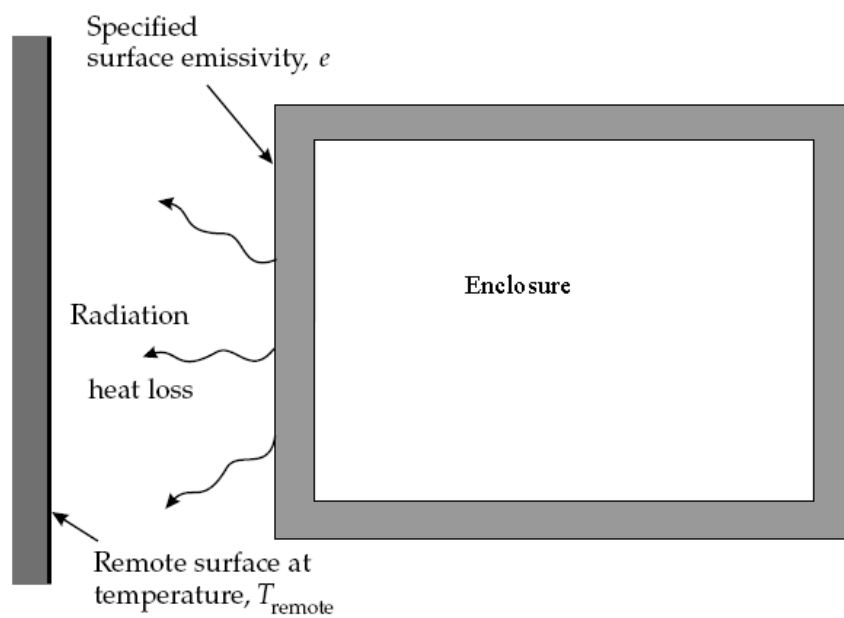


Figure 13.5.2: Radiative Heat Transfer

13.6 Constructing Multifaceted Walls

When an adiabatic block is used to change the shape or size of a room, or to mask a portion of the room to be excluded from the computational domain, the sides of the block represent enclosure boundaries. You can position an external wall on one or more of the block surfaces, as shown in Figure 13.6.1. The main reason for using a wall in this way is that an external wall might provide better control of heat transfer behavior than the surface of a block.

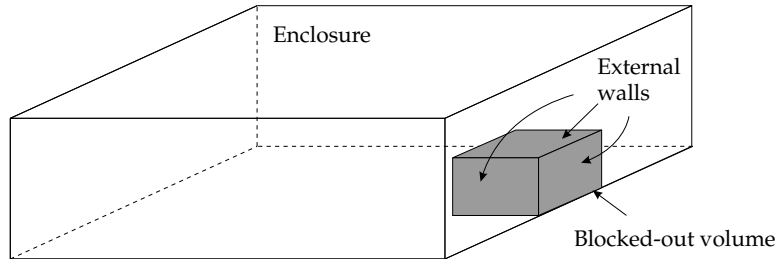


Figure 13.6.1: Blocked-Out Volumes

It is not necessary to separate an external wall into sections in order to accommodate vents, openings, or fans. **Airpak** automatically removes the wall definition from the space occupied by the vents, openings, or fans; in effect, it cuts the appropriately sized holes in the wall to accommodate these objects, as shown in Figure 13.6.2.

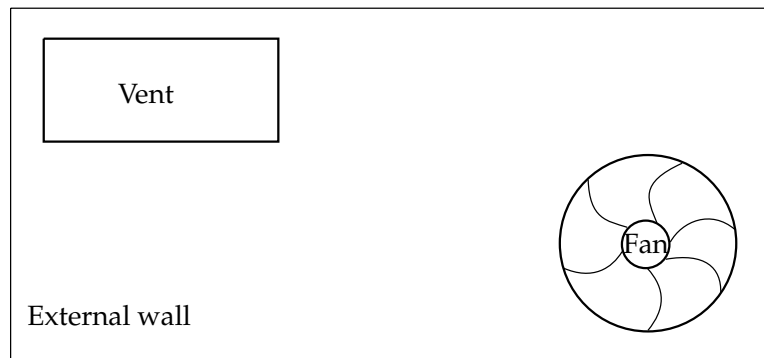


Figure 13.6.2: Wall with Overlaid Objects

This same approach can be used to construct walls that are made up of different materials. Consider the example shown in Figure 13.6.3, where a wall is composed of two different materials: Wall 1 is made of Material 1 and Wall 2 is made of Material 2. Wall 1 should be created first, then Wall 2. The second wall definition will override the first when Wall 2 is overlaid on Wall 1, effectively cutting a comparably-sized area out of Wall 1 and substituting Wall 2 into the hole.

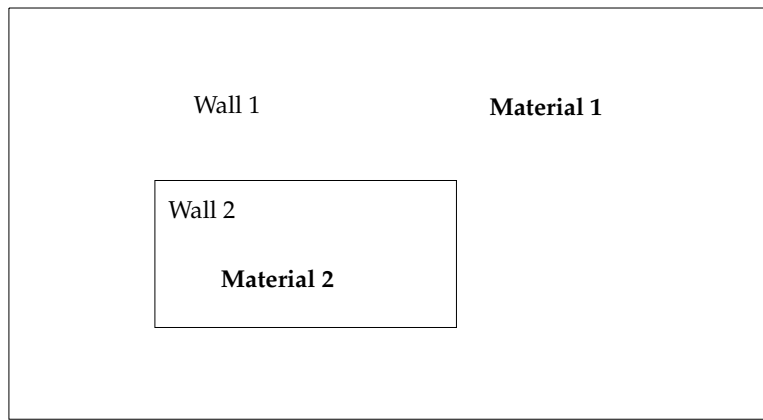




Figure 13.6.3: Multifaceted Walls

13.7 Adding a Wall to Your Airpak Model

To include a wall in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Walls panel, shown in Figures 13.7.1 and 13.7.2.

The procedure for adding a wall to your Airpak model is as follows:

1. Create a wall. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the wall, if required. See Section 7.4.1 for details.
3. Change the graphical style of the wall, if required. See Section 7.4.2 for details.
4. In the Geometry tab, specify the geometry, position, and size of the wall. There are three different kinds of geometry available for walls in the Shape drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.
5. In the Properties tab, specify the type of the wall by selecting Stationary, Symmetry, or Moving next to Wall type. The lower part of the panel will change depending on your selection of the Wall type.
6. Specify the characteristics related to the selected Wall type. These options are described below.

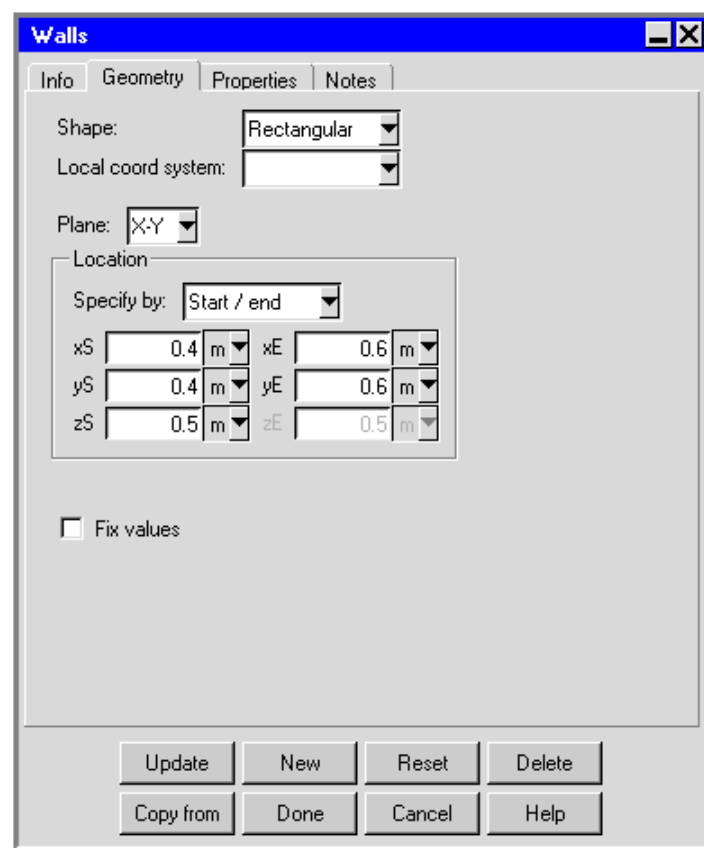


Figure 13.7.1: The Walls Panel (Geometry Tab)

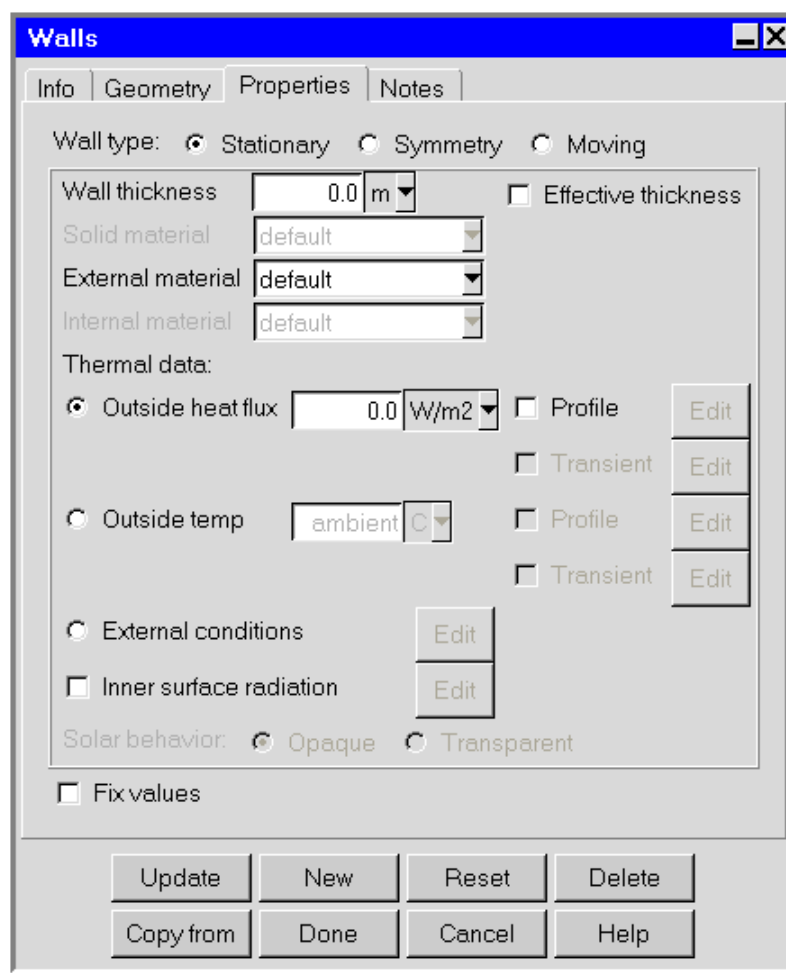


Figure 13.7.2: The Walls Panel (Properties Tab)

13.7.1 User Inputs for a Symmetry Wall

The velocity component normal to a symmetry wall is set to zero. This option allows you to model half of a room in the case where the room is geometrically symmetric about a center plane. When the **Symmetry** option is selected, you cannot specify any thermal data and Airpak automatically specifies a zero heat flux condition at the symmetry plane.

There are no additional inputs for a symmetry wall.

13.7.2 User Inputs for a Stationary or Moving Wall

To specify a stationary wall or a moving wall, select **Stationary** or **Moving** next to **Wall type** in the **Walls** panel. The user inputs for a stationary wall are shown below. The user inputs for a moving wall are almost identical to those for a stationary wall, except that the motion of the wall must also be specified.

Wall type: ☒ Stationary ☐ Symmetry ☐ Moving

Wall thickness: 0.0 m ☐ Effective thickness

Solid material: default

External material: default

Internal material: default

Thermal data:

☒ Outside heat flux: 0.0 W/m2 ☐ Profile

☐ Outside temp: ambient C ☐ Transient

☐ External conditions

☐ Inner surface radiation

Solar behavior: ☒ Opaque ☐ Transparent

☐ Fix values

The steps for defining a stationary wall or a moving wall are as follows:

1. Select **Stationary** or **Moving** to specify whether the wall is stationary or moving. For a stationary wall, the flow velocity is zero at the wall.
2. (stationary wall only) Specify the **Wall thickness**. When the wall thickness is non-zero, the heat flow through the wall is computed based on the conductivity of the

wall, the computed inner wall temperature on the inner surface of the wall, and the conditions applied at the outer surface of the wall.

If the thickness is non-zero, the wall is expanded inward or outward from the plane of the wall. If the thickness is positive, the wall is expanded to the specified thickness in the positive direction of the coordinate axis that is normal to the plane of the wall. If the thickness is specified with a negative sign, the expansion is in the negative direction of the coordinate axis normal to the plane of the wall.

3. (stationary rectangular walls only) Select the **Effective thickness** option if you want the specified **Wall thickness** to be only an effective thickness.
4. (stationary wall only) If you specify a non-zero thickness for the wall, you must also specify the **Solid material** for the wall. By default, this is specified as **default** for the wall. This means that the material specified as the **Solid material** for the wall is defined under **Default solid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Solid material** for the wall, select a material from the **Solid material** drop-down list. See Section 7.7 for details on material properties.
5. Specify the **External material** for the wall. By default, this is specified as **default** for the wall. This means that the material specified as the **External material** for the wall is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **External material** for the wall, select a material from the **External material** drop-down list. The surface roughness and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.
6. If you specify a non-zero thickness for a stationary wall, or if you select a moving wall, you can specify the **Internal material** for the wall. By default, this is specified as **default** for the wall. This means that the material specified as the **Internal material** for the wall is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **Internal material** for the wall, select a material from the **Internal material** drop-down list. The surface roughness and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.



For a zero-thickness wall, changing the **External material** does not automatically change the **Internal material**. In these cases, the **Internal material** will be set to the default surface material unless you have changed the **Internal material** prior to specifying the zero-thickness wall.

7. (moving wall only) Set the **Velocity** vector (**X**, **Y**, **Z**) for the movement of the wall. Only the components of velocity in the plane of the wall can be set to non-zero values. The component of velocity normal to the wall is automatically set to zero.

8. Specify the **Thermal data** for the wall. There are three options:

- Specify the **Outside heat flux** for the wall. To specify a uniform fixed rate of heat transfer through the wall, enter a value in the **Outside heat flux** text entry box. To define a spatial profile for the rate of heat transfer through the wall, select **Profile** next to **Outside heat flux** and click **Edit** to open the **Curve specification** panel (described below).

If you are setting up a transient simulation, you can specify the **Outside heat flux** as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the heat flux, select **Transient** and click **Edit** in the **Walls** panel. See Chapter 20 for more details on transient simulations.



Note that **Airpak** cannot use both a spatial profile and a transient profile for the outside heat flux. If you specify both profile types, **Airpak** will use the transient profile and ignore the spatial profile.

- Specify the **Outside temp** of the wall. To specify a uniform fixed temperature on the outer surface of the wall, enter a value in the **Outside temp** text entry box. To define a spatial profile for the temperature on the outer surface of the wall, select **Profile** next to **Outside temp** and click **Edit** to open the **Curve specification** panel (described below).

If you are setting up a transient simulation, you can specify the **Outside temp** as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the outside temperature, select **Transient** and click **Edit** in the **Walls** panel. See Chapter 20 for more details on transient simulations.



Note that **Airpak** cannot use both a spatial profile and a transient profile for the outside temperature. If you specify both profile types, **Airpak** will use the transient profile and ignore the spatial profile.

- Specify the **External conditions** for the wall. The **External conditions** option allows you to account for heat loss or gain at the outer surface of the wall through convective or radiative heat transfer. To specify the external conditions for the wall, select **External conditions** and click **Edit**. **Airpak** will open the **Wall external thermal conditions** panel (Figure 13.7.3).
- (a) To activate convective heat transfer at the outer surface of the wall, select **Enabled** in the **Wall external thermal conditions** panel. To specify a uniform heat transfer coefficient, enter the value of the heat transfer coefficient (h_c in Equation 13.5-1) in the **Base value** text entry field and select the **Constant value** option (default). To define a spatial profile for the heat transfer coefficient, select the **Spatial profile** option and click **Edit** to open the **Curve specification** panel (described below). To define a heat transfer coefficient that varies as a

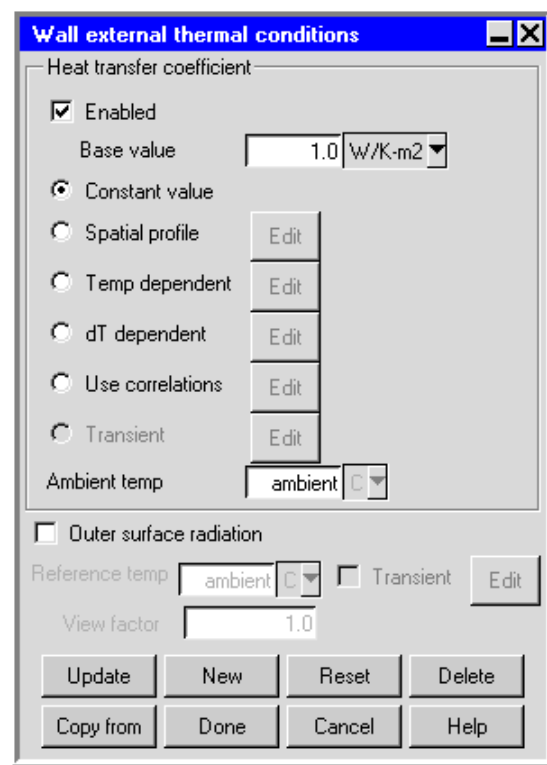


Figure 13.7.3: The Wall external thermal conditions Panel

function of temperature, select the **Temp dependent** option and click **Edit** to open the **Curve specification** panel (described below). You can also define the heat transfer coefficient as a function of temperature difference (relative to the ambient temperature). To define a heat transfer coefficient that varies as a function of temperature difference, select **dT dependent** and click **Edit** to open the **Curve specification** panel (described below).

The heat transfer coefficient can also be set automatically using existing empirical Nusselt correlations for forced convection and natural convection flows. These correlations depend on the geometry of the wall, fluid properties, Reynolds number (for forced convection flows), and Rayleigh number (for natural convection flows). To specify the heat transfer coefficient using correlations select the **Use correlations** option and click **Edit** to open the **Flow dependent heat transfer** panel (Figure 13.7.4).

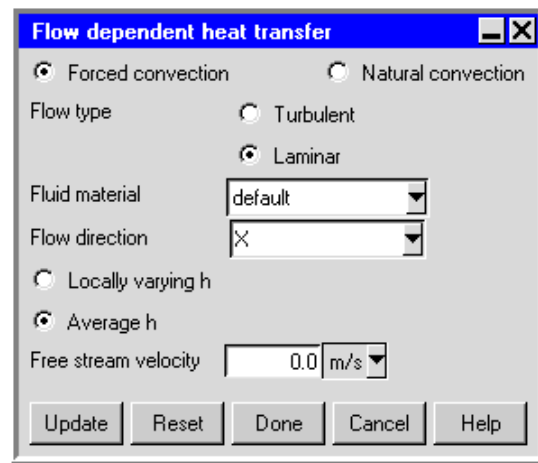


Figure 13.7.4: The Flow dependent heat transfer Panel

In this panel, you can select either **Forced convection** (default) or **Natural convection**.

- If you have selected the **Forced convection** option
 - i. Select **Turbulent** or **Laminar** flow regime next to the **Flow type**.
 - ii. To change the default **Fluid material** for the external flow, select a material from the **Fluid material** drop-down list. See Section 7.7 for details on material properties.
 - iii. To specify the flow direction, select the appropriate direction from the **Flow direction** drop-down list. For a wall object, the flow can be in either direction in the plane of the wall.
 - iv. Specify the type of the heat transfer coefficient used. For forced convection situations, there are empirical correlations to define a local value of the heat transfer coefficient. To specify a locally varying heat transfer coefficient, select the **Locally varying h** option. To specify an average heat transfer coefficient for the entire wall surface, select the **Average h** option.
 - v. To specify the free stream velocity, enter the value of the external flow velocity in the **Free stream velocity** text entry field.
- If you have selected the **Natural convection** option
 - i. To change the default **Fluid material** for the external natural convection flow, select a material from the **Fluid material** drop-down list. See Section 7.7 for details on material properties.
 - ii. Specify the ambient external temperature for the fluid surrounding the wall in the **Ambient temp** entry field.

- iii. Specify the orientation of the surface with respect to gravity. The heat transfer coefficients for natural convection flow depend on the orientation of the wall surface relative to the direction of gravity. To specify the orientation of the surface, select one of the following three options under **Surface: Vertical, Top, and Bottom**.

For example, if gravity is in the negative Y direction, the XY and YZ planes are the **Vertical** surfaces, the high XZ plane is the **Top** surface, and the low XZ plane is the **Bottom** surface. To specify the direction of the gravity, select the appropriate axis from the **Gravity direction** drop-down list.

If you are setting up a transient simulation, you can specify the **Heat trans coeff** as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the heat transfer coefficient, select **Transient** and click **Edit** in the **Wall external thermal conditions** panel. See Chapter 20 for more details on transient simulations.



Note that **Airpak** cannot use both a spatial profile and a transient profile for the heat transfer coefficient. If you specify both profile types, **Airpak** will use the transient profile and ignore the spatial profile.

Specify the ambient external temperature (i.e., T_{ambient} in Equation 13.5-1) next to **Ambient temp** for the convective heat transfer boundary condition. The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

- (b) To activate radiative exchange of heat between the exterior surface of the wall and a remote surface, select **Outer surface radiation** in the **Wall external thermal conditions** panel. Then specify the following parameters: Reference temp (T_{remote} in Equation 13.5-2) and View factor. The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

If you are setting up a transient simulation, you can specify the reference temperature as a function of time. To define the variation of reference temperature as a function of time, select **Transient** next to **Reference temp**, and click **Edit**. This will open the **Transient temperature** panel shown in Figure 20.1.6. For details on specifying the transient temperature see Section 20.1.

View factor of the remote surface (i.e., the fraction of radiant energy that is intercepted by the wall). The default **View factor** is 1.0.

- (c) Click **Done** in the **Wall external thermal conditions** panel to accept your changes and close the panel.

9. (optional) Specify the radiative heat transfer parameters for the inner surface of the wall. This option is available if you have selected **On** next to **Radiation** in the **Basic parameters** panel. You can modify the default radiation characteristics of the inner surface of the wall (e.g., the view factor) by using the **Radiation specification** panel. To open this panel, select **Inner surface radiation** in the **Walls** panel and then click **Edit**. See Chapter 22 for details on radiation modeling.

Using the Curve specification Panel to Specify a Spatial Boundary Profile

You can define a spatial boundary profile using the Curve specification panel (Figure 13.7.5). To open the Curve specification panel, select **Profile** in the **Walls** panel or **Spatial profile** in the **Wall external thermal conditions** panel, and click **Edit**.

You can also define the variation of the heat transfer coefficient with temperature or temperature difference using the Curve specification panel. To open the Curve specification panel, select **Temp dependent** or **dT dependent** in the **Wall external thermal conditions** panel, and click **Edit**.

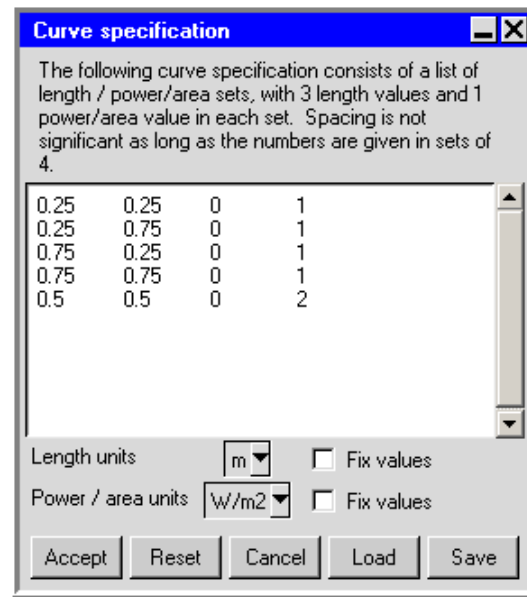


Figure 13.7.5: The Curve specification Panel

To define a spatial boundary profile, specify a list of (x, y, z) coordinates and the corresponding values in the **Curve specification** panel. For example, the first line in Figure 13.7.5 specifies an outside heat flux of 1 W/m^2 at $(0.25, 0.25, 0)$. The data in Figure 13.7.5 specify a variation of heat flux on the plane $0.25 \leq x \leq 0.75$, $0.25 \leq y \leq 0.75$, $z = 0$. The values in the right-hand column are the rate of heat transfer through the wall, the temperature on the outer surface of the wall, or the heat transfer coefficient at the outer surface of the wall, depending on your selection under **Thermal data** in the **Walls** panel.

To define a temperature/heat transfer coefficient curve, specify a list of temperature/heat transfer coefficient pairs in the **Curve specification** panel. It is important to give the numbers in pairs, but the spacing between numbers is not important.

Click **Accept** when you have finished defining the profile or curve; this will store the values and close the **Curve specification** panel. **Airpak** will interpolate the data you provide in the **Curve specification** panel to create a profile for the whole boundary.

i If the starting point of the wall object is located at (x_0, y_0, z_0) , then the first point of the profile should be $(x_0 \ y_0 \ z_0 \ a_0)$, where a_0 is the corresponding value for that point. However, if the first point in the profile has a different value, for example $(x_1 \ y_1 \ z_1 \ a_0)$, **Airpak** will automatically translate the first point to $(x_0 \ y_0 \ z_0 \ a_0)$, and the rest of the profile points will be shifted by $(x_1 - x_0, y_1 - y_0, z_1 - z_0)$. This translation of point locations will not affect the values of the variables (a_n), and is also useful if the opening is ever translated within the model. In this way, you will not have to recreate the profile file or re-enter values in the **Curve specification** panel. This translation feature also applies to other objects that allow the specification of point profiles (i.e., openings, blocks, and resistances).

To load a previously defined profile or curve, click on **Load**. (See Section 28.2.8 for details on saving contour data and using them as a profile.) This will open the **Load curve** file selection dialog box. Select the file containing the profile or curve data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the units used in the profile or curve data you are loading, you should select the appropriate units in the **Curve specification** panel before you load the profile or curve data. If you want to view the imported data after you have loaded it, using different units than the default units in the **Curve specification** panel, select the relevant **Fix values** options and then select the appropriate units from the unit definition lists.

If you want to load a curve file that you have created outside of **Airpak**, you will need to make sure that the first three lines of the file before the data contain the following information:

1. the number of data sets in the file (usually 1)
2. the unit specifications for the file, which can be obtained from the **Curve specification** panel (e.g., `units m W/m2`)
3. the number of data points in the file (e.g., 5)

Using the above example, the first three lines of the curve file would be

```
1
units m W/m2
5
```

The actual data points should be entered in the same way as you would enter them in the **Curve specification** panel.



If you want to load a curve from an Excel file, make sure that you also save the file as formatted text (space delimited) before reading it into **Airpak**.

To save a profile or curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the profile or curve data are to be saved.

Partitions are objects that are impervious to fluid flow. They can possess a thickness and are defined by both their geometry and their type. Partition geometries include rectangular, 2D polygon, circular, inclined and 2D CAD.

Partition types are defined by their associated thermal models, including adiabatic thin, conducting thick, conducting thin, hollow thick, or contact resistance. Adiabatic thin partitions do not conduct heat either across or in the plane of the partition. Conducting thick partitions can conduct heat in either direction and they possess a thickness. Conducting thin partitions can conduct heat in either direction and have no physical thickness. Hollow thick partitions can conduct heat in the plane of the partition but not across the partition. Contact resistance partitions model resistances to heat transfer due to barriers such as surface coatings or glues. Fluid partitions are also available and can be used to cut a hole into a solid partition.

In this chapter, information about the characteristics of a partition is presented in the following sections:

- Section [14.1](#): Defining a Partition in **Airpak**
- Section [14.2](#): Geometry, Location, and Dimensions
- Section [14.3](#): Thermal Model Type
- Section [14.4](#): Surface Roughness
- Section [14.5](#): Using Partitions in Combination with Other Objects
- Section [14.6](#): Adding a Partition to Your **Airpak** Model

14.1 Defining a Partition in Airpak

In Airpak, partition sides are referred to as high and low, relative to the coordinates of the plane perpendicular to the partition (see Figure 14.1.1). The no-slip boundary condition applies at any partition surface in contact with the fluid and, for turbulent flows, you can specify a surface roughness for each side of the partition. The sides of a partition can exchange radiation with other objects in the model.

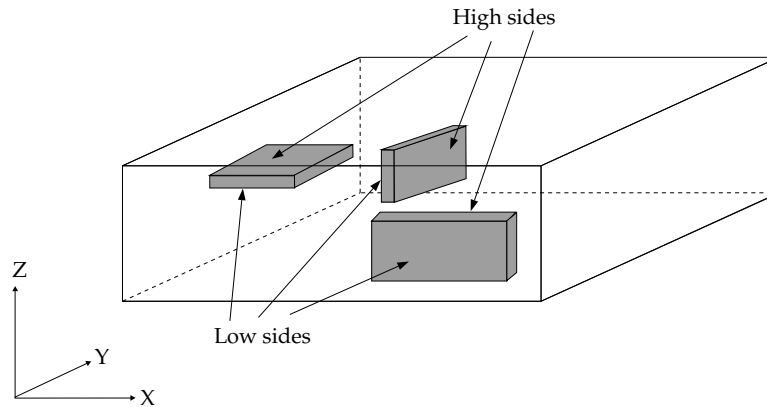


Figure 14.1.1: High/Low Side Partition Definition

To configure a partition in the model, you must specify its geometry (including location and dimensions) and type, as well as the thermal characteristics, and the material from which each side is made.

14.2 Geometry, Location, and Dimensions

The location and dimension parameters for a partition vary according to the geometry of the partition. Partition geometries include rectangular, 2D polygon, circular, inclined and 2D CAD. These geometries are described in Section 7.4.4.

14.2.1 Partition Thickness

When a rectangular, 2D polygon, or circular partition is specified with a non-zero thickness, the thickness extends in either the positive or the negative coordinate direction (normal to the plane of the partition), depending on whether the thickness is specified as a positive or negative value. For an inclined partition, Airpak distributes the specified thickness in equal portions on both sides of the partition.

14.3 Thermal Model Type

Partitions are defined according to their associated thermal models, which can be specified as adiabatic thin, conducting thick, conducting thin, hollow thick, or contact resistance.

Adiabatic thin partitions have zero thickness and do not conduct heat in any direction, either normal to the partition or along the plane of the partition.

Conducting thick partitions can conduct heat either through or along the plane of the partition and can do so anisotropically, i.e., according to thermal conductivities specific to each direction (defined as part of the properties of the solid material specified for the partition). They must possess a physical thickness; the thickness must be physical so that **Airpak** can mesh the interior of the partition. For transient simulations, you can specify both density and specific heat for the partition (defined as part of the properties of the solid material specified for the partition), thereby imparting a thermal mass. If a conducting thick partition is specified without thermal mass, it can conduct heat but it cannot accumulate heat.

Conducting thin partitions have the same properties as conducting thick partitions, except that they have no physical thickness. They can possess only an effective thickness.

Contact resistance partitions represent barriers to heat transfer either between objects or between an object and the adjacent fluid. You can specify the resistance of the barrier in terms of a thermal conductivity or a contact resistance. The thermal-conductivity-based resistance is defined by the thermal conductivity (defined as part of the properties of the solid material specified for the partition) and the partition thickness. The partition must be specified with a constant material conductivity; i.e., the conductivity must not be a function of temperature. Contact resistance partitions can possess an effective thickness, in which case **Airpak** will not mesh the interior of the partition.

Hollow thick partitions represent three-dimensional regions of the model for which only side characteristics are important. **Airpak** does not mesh or solve for temperature or flow within regions bounded by the sides of a hollow partition.

14.4 Surface Roughness



In fluid dynamics calculations, it is common practice to assume that boundary surfaces are perfectly smooth. In laminar flow, this assumption is valid, because the length scales of typical rough surfaces are much smaller than the length scales of the flow. In turbulent flow, however, the length scales of the flow eddies are much smaller than laminar length scales; therefore, it is sometimes necessary to account for surface roughness. Surface roughness acts to increase resistance to flow, leading to higher rates of heat transfer.

Airpak assumes, by default, that all surfaces of a partition are hydrodynamically smooth, and applies standard no-slip boundary conditions. For turbulent-flow simulations in which roughness is significant, however, you can specify a roughness factor for the entire partition or for each individual side of the partition. The roughness factor is defined as part of the properties of the surface material for the partition. The purpose of the roughness factor is to approximate the average height of the surface texture on the partition.

14.5 Using Partitions in Combination with Other Objects

Partitions can be used alone or in conjunction with other modeling objects to create complex objects in order to perform sophisticated thermal simulations. For example, partitions and blocks can be used to build compartment, offices, or even railroad cars of varying complexity.

14.6 Adding a Partition to Your Airpak Model

To include a partition in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Partitions panel, shown in Figures 14.6.1 and 14.6.2.

The procedure for adding a partition to your Airpak model is as follows:

1. Create a partition. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the partition, if required. See Section 7.4.1 for details.
3. Change the graphical style of the partition, if required. See Section 7.4.2 for details.
4. In the Geometry tab, specify the geometry, position, and size of the partition. There are four different kinds of geometry available for partitions in the Shape drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.

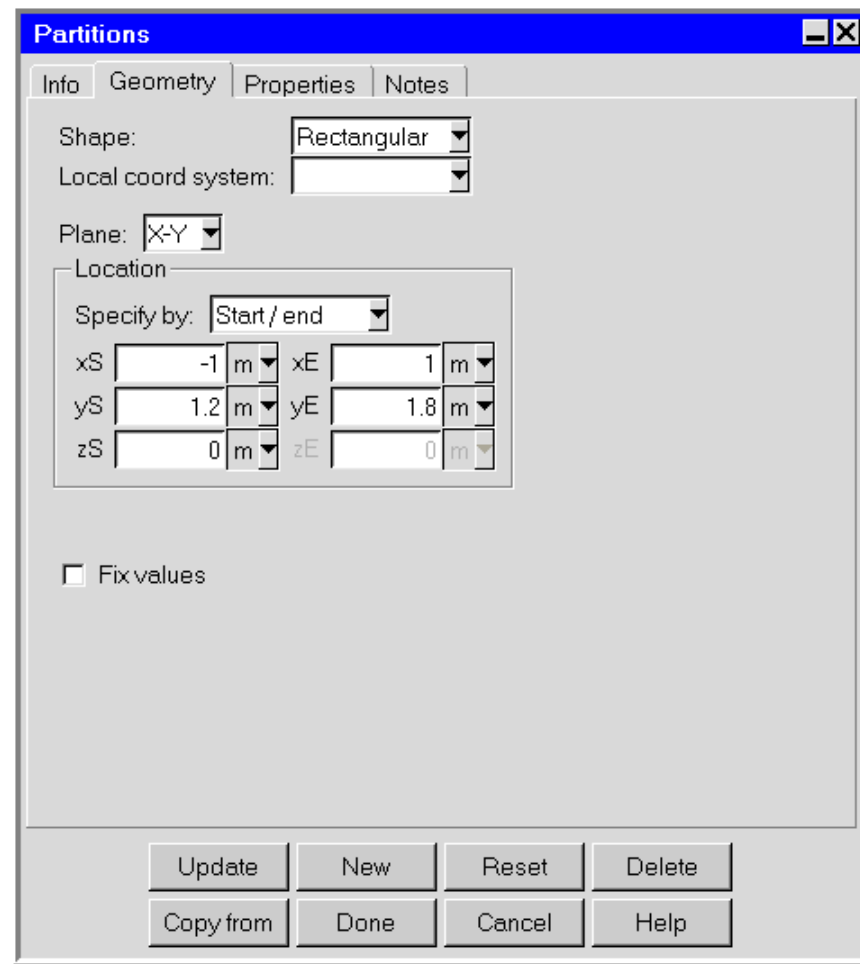


Figure 14.6.1: The Partitions Panel (Geometry Tab)

5. In the **Properties** tab, specify the thermal model for the partition by selecting **Conducting thick**, **Hollow thick**, **Contact resistance**, **Conducting thin**, **Adiabatic thin**, or **Fluid under Thermal model**. The lower part of the panel will change depending on your selection of the **Thermal model**.
6. Specify the characteristics related to the selected **Thermal model**. These options are described below.
7. Specify the properties for the **Low side** and the **High side** of the partition. These options are described in Section 14.6.2.
8. (circular partitions only) Specify the speed of **Rotation (rpm)**.
9. If you have enabled solar loading in the **Basic parameters** panel, specify the **Solar behavior** for the partition. See Section 22.5 for details.

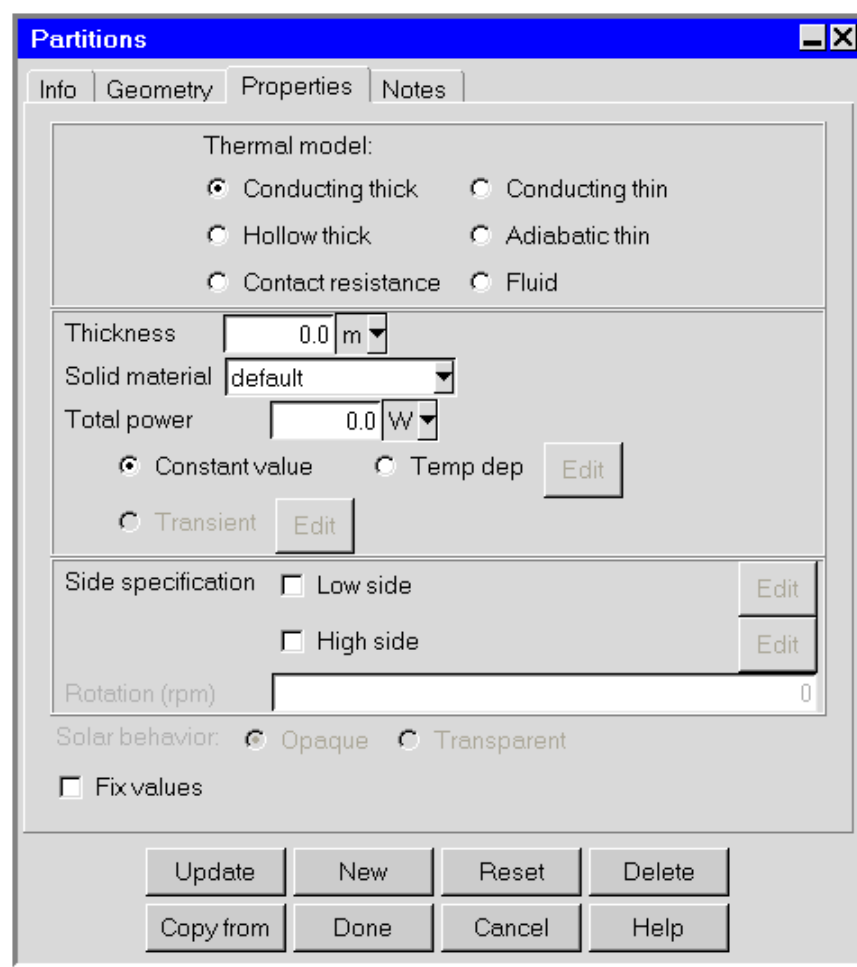


Figure 14.6.2: The Partitions Panel (Properties Tab)

14.6.1 User Inputs for the Thermal Model

Conducting Thick Partitions

To specify a conducting thick partition, select **Conducting thick** under **Thermal model** in the **Partitions** panel. The user inputs for the **Conducting thick** thermal model are shown below.

The screenshot shows a software interface for defining a thermal model. It is titled "Thermal model:" and contains several radio button options: "Conducting thick" (selected), "Conducting thin", "Hollow thick", "Adiabatic thin", "Contact resistance", and "Fluid". Below these options are three input fields: "Thickness" with a value of "0.0" and unit "m", "Solid material" with a dropdown menu showing "default", and "Total power" with a value of "0.0" and unit "W". There are also two sets of radio buttons for specifying power: "Constant value" (selected) and "Temp dep", and a "Transient" option. Each of these three options has an "Edit" button next to it.

The steps for defining a partition with a **Conducting thick** thermal model are as follows:

1. Specify the **Thickness** of the partition.
2. Specify the **Solid material** for the partition. By default, this is specified as **default** for the partition. This means that the material specified as the **Solid material** for the partition is defined under **Default solid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Solid material** for the partition, select a material from the **Solid material** drop-down list. See Section 7.7 for details on material properties.
3. Specify the total power dissipated by the partition. There are three options for specifying the total power:

Constant value allows you to specify a constant value of the **Total power**.

Temp dep allows you to specify heat as a function of temperature. This option is not available if the **Transient** option is selected.

Click **Edit** next to **Temp dep** in the **Partitions** panel to open the **Temperature dependent power** panel (Figure 14.6.3).

Specify a value for the constant **C**. The value in the equation shown in the panel is either the **Total power** or the power **Per unit area/volume** specified in the **Partitions** panel. Define the temperature range for which the function is valid by entering values (in Kelvin) for **Low T** and **High T**. The value of the **ambient**

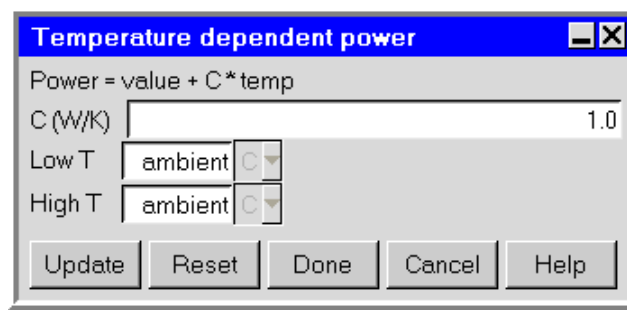


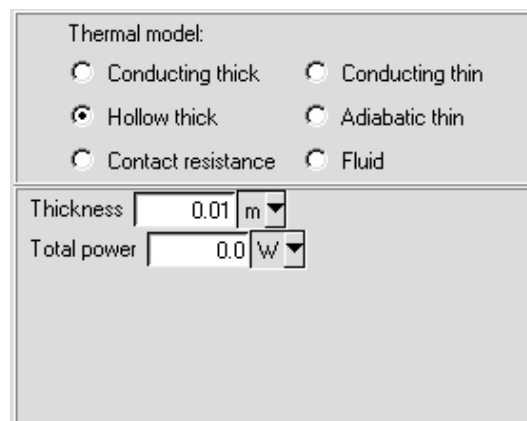
Figure 14.6.3: The Temperature dependent power Panel

temperature is defined under **Ambient** values in the **Basic parameters** panel (see Section 6.4.6). If the temperature exceeds the specified value of **High T**, then the power is given by substituting the value of **High T** into the equation at the top of the **Temperature dependent power** panel. If the temperature falls below the specified value of **Low T**, then the power is given by substituting the value of **Low T** into the equation at the top of the **Temperature dependent power** panel.

Transient allows you to specify the total power as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. Select **Transient** under **Total power** and enter a value for the **Total power**. To edit the transient parameters for the partition, click **Edit** next to **Transient**. See Chapter 20 for more details on transient simulations.

Hollow Thick Partitions

To specify a hollow thick partition, select **Hollow thick** under **Thermal model** in the **Partitions** panel. The user inputs for the **Hollow thick** thermal model are shown below.



The steps for defining a partition with a **Hollow thick** thermal model are as follows:

1. Specify the **Thickness** of the partition.
2. Specify the **Total power** dissipated by the partition.

Contact Resistance Partitions

To specify a contact resistance partition, select **Contact resistance** under **Thermal model** in the **Partitions** panel. The user inputs for the **Contact resistance** thermal model are shown below.

The screenshot shows a dialog box for the 'Contact resistance' thermal model. It includes radio buttons for 'Conducting thick', 'Conducting thin', 'Hollow thick', 'Adiabatic thin', 'Contact resistance' (which is selected), and 'Fluid'. Below these, there is a checked checkbox for 'Additional resistance' and a dropdown menu set to 'Thickness'. Further down, there is a field for 'Effective thickness' set to '1000.0 mm' and a dropdown for 'Solid material' set to 'default'. At the bottom, there is a 'Total power' field set to '0.0 W'.

The steps for defining a partition with a **Contact resistance** thermal model are as follows:

1. If you want to specify an additional resistance, select **Additional resistance**. You can specify the resistance by selecting one of the following options from the drop-down list and entering appropriate values:

Conductance allows you to specify a value for the **Conductance**. The inverse of this value will be used to calculate the contact resistance of the partition.

Thermal resistance allows you to specify a value for the **Thermal resistance** to heat transfer.

Thermal impedance allows you to specify a value for the **Thermal impedance**. Airpak computes the thermal resistance of the partition as Z/A where Z is the thermal impedance of the partition and A is the area of the partition.

Thickness allows you to specify a value for the **Effective thickness** of a specified **Solid material**. The values of the thickness and the thermal conductivity of the solid material will be used to calculate the contact resistance of the partition. The resistance is computed as d/k , where d is the effective thickness of the partition and k is the thermal conductivity of the solid material (defined as part of the properties of the solid material specified for the partition).

By default, the **Solid material** is specified as **default** for the partition. This means that the material specified as the **Solid material** for the partition is defined under **Default solid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Solid material** for the partition, select a material from the **Solid material** drop-down list. See Section 7.7 for details on material properties.

2. Enter a value for the **Total power** dissipated by the partition.

Conducting Thin Partitions

To specify a conducting thin partition, select **Conducting thin** under **Thermal model** in the **Partitions** panel. The user inputs for the **Conducting thin** thermal model are shown below.

The screenshot shows a software interface for configuring a partition's thermal model. It is titled 'Thermal model:' and contains several radio button options: 'Conducting thick', 'Conducting thin' (which is selected), 'Hollow thick', 'Adiabatic thin', 'Contact resistance', and 'Fluid'. Below these options are three input fields: 'Effective thickness' with a value of '0.0' and a unit dropdown set to 'mm'; 'Solid material' with a dropdown menu showing 'default'; and 'Total power' with a value of '0.0' and a unit dropdown set to 'W'. At the bottom, there are two radio buttons for 'Constant value' (selected) and 'Temp dep', followed by an 'Edit' button.

The steps for defining a partition with a **Conducting thin** thermal model are as follows:

1. Specify the **Effective thickness** for the partition.
2. Specify the **Solid material** for the partition. By default, this is specified as **default** for the partition. This means that the material specified as the **Solid material** for the partition is defined under **Default solid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Solid material** for the partition, select a material from the **Solid material** drop-down list. See Section 7.7 for details on material properties.
3. Specify the total power dissipated by the partition. There are two options for specifying the total power:

Constant value allows you to specify a constant value of the **Total power**.

Temp dep allows you to specify the power as a linear function of temperature. Select **Temp dep** under **Total power** in the **Partitions** panel, and enter a value of the **Total power**. Click **Edit** next to **Temp dep** to open the **Temperature dependent power** panel (Figure 14.6.3). Specify a value for the constant **C**.

The **value** in the equation at the top of the **Temperature dependent power** panel is the **Total power** specified in the **Partitions** panel. Define the temperature range for which the function is valid by entering values (in Kelvin) for **Low T** and **High T**. The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). If the temperature exceeds the specified value of **High T**, then the power is given by substituting the value of **High T** into the equation at the top of the **Temperature dependent power** panel. If the temperature falls below the specified value of **Low T**, then the power is given by substituting the value of **Low T** into the equation at the top of the **Temperature dependent power** panel. Click **Update** to update the thermal specification of the partition.

Adiabatic Thin Partitions

An adiabatic partition does not conduct heat in any direction, either normal to the partition or along the plane of the partition.

To specify an adiabatic thin partition, select **Adiabatic thin** under **Thermal model** in the **Partitions** panel. There are no additional inputs for an adiabatic thin partition.

Fluid Partitions

To specify a fluid partition, select **Fluid** under **Thermal model** in the **Partitions** panel. The only input for this model is the **Thickness** of the partition.

Note that the **Side specification** options in the **Partitions** panel are not available with the **Fluid** thermal model. A fluid partition can only be used to cut a hole into a solid partition, as shown in Figure 14.6.4.

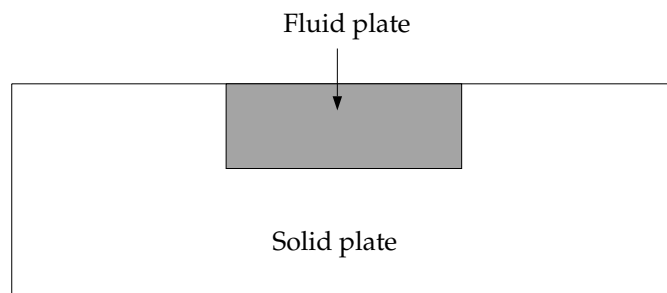


Figure 14.6.4: Using a Fluid Partition

14.6.2 User Inputs for the Low- and High-Side Properties of the Partition

Airpak allows you to specify different physical characteristics for each side of the partition. If you select **Low side** next to **Side specification** in the **Partitions** panel (Figure 14.6.2) and

then click **Edit**, Airpak will open the **Low side surface properties** panel (Figure 14.6.5). If you select **High side** and then click **Edit**, Airpak will open the **High side surface properties** panel, which is identical to the **Low side surface properties** panel.

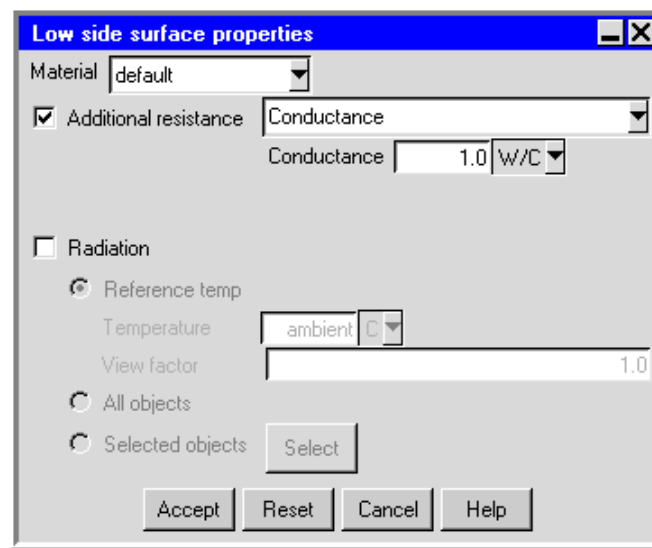


Figure 14.6.5: The Low side surface properties Panel

To define the physical characteristics for the low side or the high side of the partition, follow the steps below.

1. Specify the surface **Material** to be used for the current side of the partition. This material defines the roughness and emissivity for this side of the partition. By default, this is specified as **default**, which means that the material specified for the side of the partition is defined in the **Basic parameters** panel (see Section 6.4.7). To change the material for the current side of the partition, select a material from the **Material** drop-down list. See Section 7.7 for details on material properties.
2. (conducting thick or hollow thick partitions only) If you want to specify an additional resistance to heat transfer for the current side of the partition, select **Additional resistance**. You can specify the resistance by selecting one of the following options from the drop-down list and entering appropriate values:

Conductance allows you to specify a value for the **Conductance** ($= hA$). The inverse of this value will be used to calculate the additional resistance of the current side of the partition.

Thermal resistance allows you to specify a value for the thermal resistance (**Thermal resistance** $= \frac{1}{hA}$). This value will be used to calculate the additional resistance of the current side of the partition.

Thermal impedance allows you to specify a value for the **Thermal impedance**.

Airpak computes the thermal resistance of the partition as Z/A where Z is the thermal impedance of the partition and A is the area of the partition.

Thickness allows you to specify a value for the **Thickness** of a specified **Solid material**. The values of the thickness and the thermal conductivity of the solid material will be used to calculate the additional resistance of the current side of the partition.

3. If the side of the partition is subject to radiative heat transfer, select **Radiation**. You can modify the default radiation characteristics of the partition (e.g., the view factor). See Chapter [22](#) for details on radiation modeling.

Sources represent regions in the model within which heat flux originates. Source geometries include prism, cylinder, ellipsoid, elliptical cylinder, rectangular, circular, 2D polygon, and inclined. Sources can be used to specify the primary field variable of temperature. For transient problems, you can also define a period during which the source is active. Two-dimensional sources can exchange radiation with other objects in the model.

To configure a source in the model, you must specify its geometry (including location and dimensions) and temperature options. For a transient simulation, you must also specify parameters related to the source coefficients.

In this chapter, information about the characteristics of a source is presented in the following sections:

- Section [15.1](#): Geometry, Location, and Dimensions
- Section [15.2](#): Thermal Options
- Section [15.3](#): Source Usage
- Section [15.4](#): Adding a Source to Your Airpak Model

15.1 Geometry, Location, and Dimensions

Source location and dimension parameters vary according to source geometries. Source geometries include rectangular, circular, 2D polygon, inclined, prism, cylinder, ellipsoid, and elliptical cylinder. These geometries are described in Section [7.4.4](#).

15.2 Thermal Options

Energy sources are specified using one of the following options: total heat, per unit area/volume, fixed value, temperature dependent, or transient. These options are described below and in the section describing user inputs for heat source parameters. Note that the variable s represents the input value in each text field.

Total heat sets the total power output over the plane or through the volume to the value s . **Airpak** then computes the source per-unit-area (or volume) value by dividing by the area of the plane or the volume of the 3D region.

Per unit area/volume sets the flux of heat per unit area of the source (positive) or sink (negative) or per unit volume of the volume to the value s .

Fixed value sets the temperature of the fluid on the plane to the value s .

Temperature dependent sets the flux of heat per unit area of the source (positive) or sink (negative) or per unit volume of the volume to the value

$$s + CT \tag{15.2-1}$$



where T is the temperature computed by **Airpak**, and C and s are user-specified values. Note that the **Temperature dependent** option for temperature sources is designed to model temperature-maintaining sources. For such sources, the coefficient C must be negative. A positive value for the coefficient C will result in runaway temperatures.

15.3 Source Usage

Some general points regarding source usage are as follows:

- A volumetric source placed within a flow region can be regarded as a “transparent” object; i.e., the fluid flows through it. Its only effect is to add an appropriate source term to one of the governing equations being solved.
- If a 2D source is suspended in a fluid, it takes on the properties of the fluid, but it does not allow any fluid flow to pass through it; i.e., it behaves like an impermeable wall element.
- In general, 2D flux sources should be placed on the surface of another object, such as a wall, block, or partition.

15.4 Adding a Source to Your Airpak Model

To include a source in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Sources panel, shown in Figures 15.4.1 and 15.4.2.

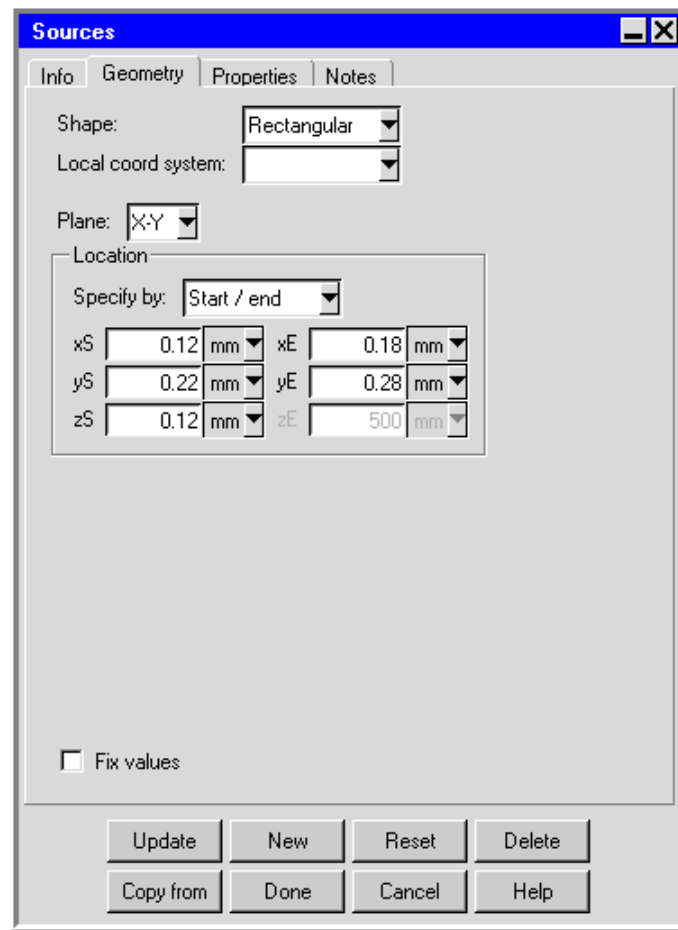


Figure 15.4.1: The Sources Panel (Geometry Tab)

The procedure for adding a source to your Airpak model is as follows:

1. Create a source. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the source, if required. See Section 7.4.1 for details.
3. Change the graphical style of the source, if required. See Section 7.4.2 for details.

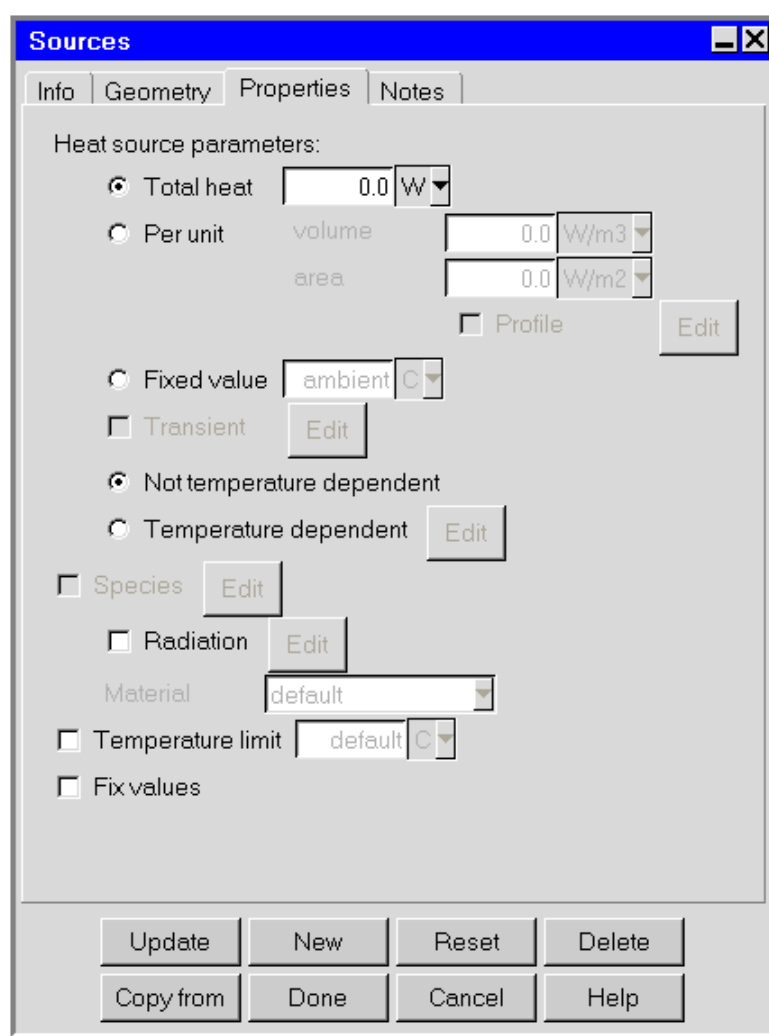


Figure 15.4.2: The Sources Panel (Properties Tab)

4. In the **Geometry** tab, specify the geometry, position, and size of the source. There are eight different kinds of geometry available for sources in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.
5. In the **Properties** tab, specify the **Heat source parameters** for the source. These options are described below.
6. Specify a source of species within the computational domain, if required. You can input the species concentrations for the source by using the **Species concentrations** panel. To open this panel, select **Species** in the **Sources** panel and click **Edit**. See Chapter 21 for details on modeling species transport.

7. (2D sources only) Select **Radiation** to specify radiation as an active mode of heat transfer to and from the source, if required. You can modify the default radiation characteristics of the source (e.g., the view factor) by using the **Radiation specification** panel. To open this panel, select **Radiation** and then click **Edit**. See Chapter 22 for details on radiation modeling.

15.4.1 User Inputs for Heat Source Parameters

The following options are available for specifying heat source parameters in the **Sources** panel.

Total heat allows you to enter a value for the total heat output over the plane of a 2D source or through the volume of a 3D source.

Per unit allows you to specify:

- the flux of heat per unit volume of the source for a 3D source by entering a value for the flux of heat per unit volume next to **volume**.
- the flux of heat per unit area of the source for a 2D source by entering a value for the flux of heat per unit area next to **area**.

Fixed value allows you to specify the temperature of the source. The value of the ambient temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6). This option is available only for 2D sources.

Transient allows you to specify the power as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. To edit the transient parameters for the source, click **Edit** next to **Transient**. See Chapter 20 for more details on transient simulations.

Not temperature dependent specifies that the heat input is independent of temperature.

Temperature dependent allows you to specify heat as a function of temperature. This option is not available if the **Transient** option is selected. There are two options to specify the temperature dependence of power: linear and piecewise linear. Select **Temperature dependent** under **Heat source parameters** in the **Sources** panel. Click **Edit** next to **Temperature dependent** to open the **Temperature dependent power** panel (Figure 15.4.3).

Choose either the linear option or the piecewise linear option. If you choose the linear option specify a value for the constant **C**. The **value** in the equation shown in the panel is either the **Total power** or the power **Per unit area/volume** specified in the **Sources** panel. Define the temperature range for which the function is valid by entering values (in Kelvin) for **Low T** and **High T**. The value of the **ambient**

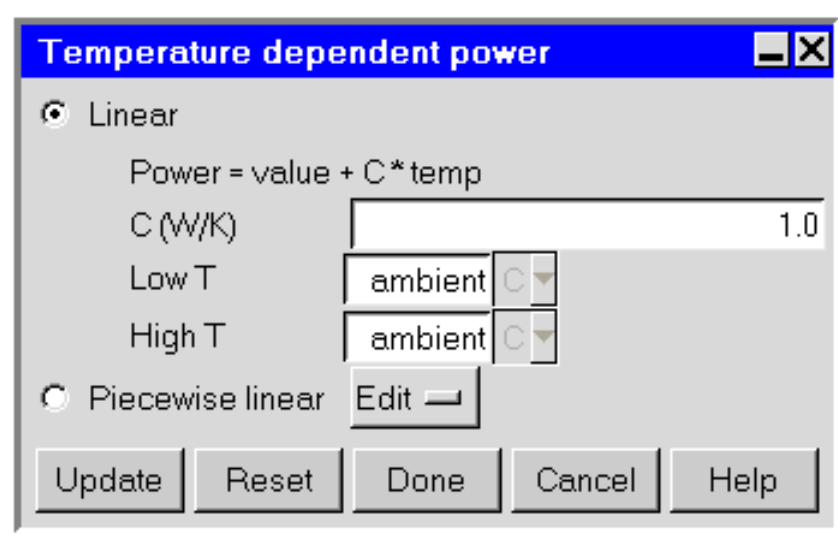


Figure 15.4.3: The Temperature dependent power Panel

temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section ??). If the temperature exceeds the specified value of **High T**, then the power is given by substituting the value of **High T** into the equation at the top of the **Temperature dependent power** panel. If the temperature falls below the specified value of **Low T**, then the power is given by substituting the value of **Low T** into the equation at the top of the **Temperature dependent power** panel.

If you select the **Piecewise linear** option, click **Edit** to open the **Curve specification** panel. To define the temperature dependence of power, specify a list of temperatures and the corresponding power values in the curve specification panel. It is important to give the numbers in pairs, but the spacing between numbers is not important. Click **Accept** when you have finished defining the curve; this will store values and close the **Curve specification** panel. **Airpak** will interpolate the data you provide in the **Curve specification** panel to create a profile for the entire range of temperatures (Figure 15.4.4). If the temperature exceeds the highest temperature specified in the curve, then the power is given by specified power at the highest temperature. Similarly if the temperature drops below the lowest temperature specified in the curve, the power is given by specified power at the lowest temperature.

i Note that piecewise linear option is available only for 2D sources.

i Note that the values entered are actual total powers and not power per unit area.

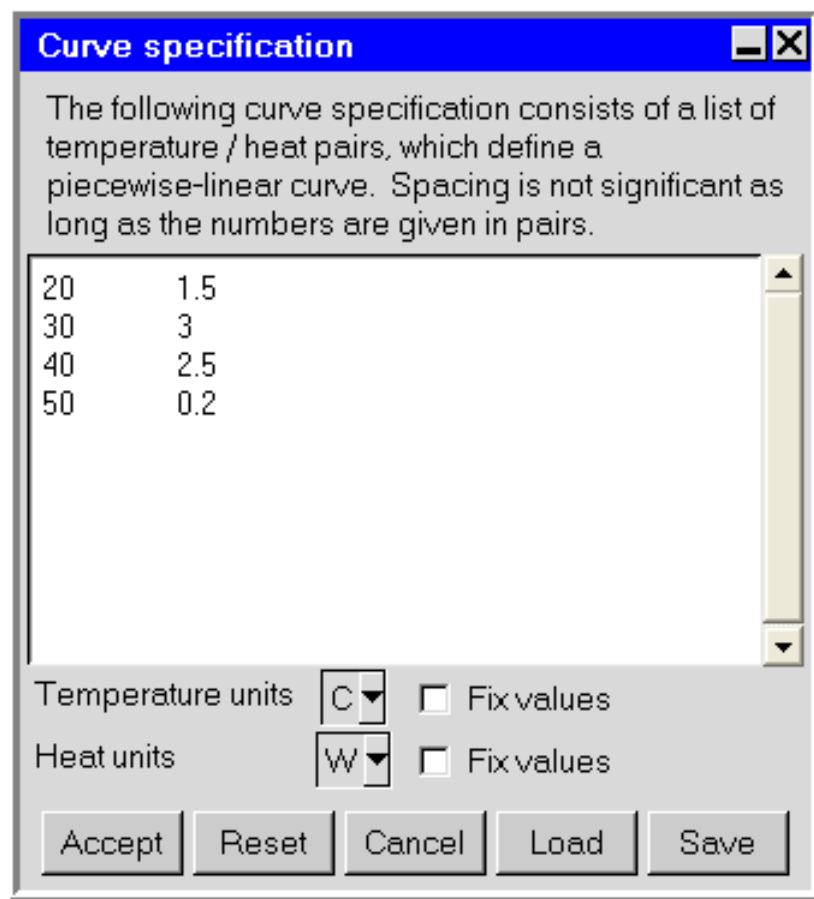


Figure 15.4.4: Curve specification panel

Resistances represent partial obstructions to flow within the room. Resistance geometries include prism, cylinder, and 3D polygon. These types of resistances are three-dimensional and represent flow resistance due to components such as wires, cables, and insulation materials packed within a portion of the room.

Rectangular, circular, inclined, and 2D polygon resistances are 2D shapes and are designed to model planar flow obstructions such as screens, vents, and permeable baffles. See Chapter 10 for more information on 2D resistances.

The effect of any resistance is modeled as a pressure drop through its area or volume. Alternatively, the pressure drop across the resistance can be calculated using either the approach-velocity method or the device-velocity method, both of which require a user-specified velocity loss coefficient. The approach-velocity and device-velocity methods differ from each other only by virtue of a factor called the free area ratio. The calculated pressure drop can be proportional either to the fluid velocity itself, or to the square of the velocity. It is common practice to employ the linear relationship for laminar flow and the quadratic relationship for turbulent flow. In the general case, a combination of the linear and quadratic relationships may more accurately model the pressure drop/volumetric flow curve. **Airpak** also provides a power-law method for calculating the pressure drop through a 3D resistance.



Note that you cannot use the hexahedral mesher if a resistance is placed on an inclined conducting thick partition. The tetrahedral mesher, on the other hand, can be used with all types of resistances.

To configure a 3D resistance in the model, you must specify its geometry (including location and dimensions), the pressure drop model, and the relationship between resistance and velocity. You must also specify the fluid material for the resistance and the total power dissipated by the resistance.

In this chapter, information about the characteristics of a 3D resistance is presented in the following sections:

- Section 16.1: Geometry, Location, and Dimensions
- Section 16.2: Pressure Drop Calculation for a 3D Resistance
- Section 16.3: Adding a Resistance to Your **Airpak** Model

16.1 Geometry, Location, and Dimensions

3D resistance location and dimension parameters vary according to resistance geometries. Resistance geometries include prism, cylinder, and 3D polygon. These geometries are described in Section 7.4.4.

16.2 Pressure Drop Calculation for a 3D Resistance

For a 3D resistance, **Airpak** provides a power-law method for calculating the pressure drop across the resistance:

$$\Delta p = C v^n \quad (16.2-1)$$

where Δp is the pressure drop across the 3D resistance, v is the velocity, and C and n are constants.

Alternatively, **Airpak** can calculate the pressure drop resulting from a resistance either by the approach-velocity method or by the device-velocity method. Because the resistance to the fluid flow due to a volumetric resistance may be different in each of the three coordinate directions, you must provide the loss coefficient and the method to calculate the pressure drop in each direction for a 3D resistance.

The approach-velocity method relates the pressure drop to the fluid velocity:

$$\Delta p = \rho \frac{l_{c1}}{2} v_{app}^n \quad (16.2-2)$$

where l_c is the user-specified loss coefficient (for each coordinate direction in 3D), ρ is the fluid density, and v_{app} is the approach velocity. The approach velocity in Equation 16.2-2 is the component of the approach velocity in the appropriate coordinate direction (x , y , or z) as computed by **Airpak**. The velocity dependence can be linear ($n = 1$), quadratic ($n = 2$), or a combination of linear and quadratic.

The device-velocity method relates the pressure drop induced by the resistance to the fluid velocity:

$$\Delta p = \rho \frac{l_{c2}}{2} v_{dev}^n \quad (16.2-3)$$

where v_{dev} is the device velocity. The velocity dependence can be linear ($n = 1$), quadratic ($n = 2$), or a combination of linear and quadratic.

The difference between the approach-velocity and device-velocity methods is in the velocity used to compute the pressure drop. The device velocity is related to the approach velocity by

$$v_{\text{dev}} = \frac{v_{\text{app}}}{A} \quad (16.2-4)$$

where A is the free area ratio. The free area ratio is the ratio of the area through which the fluid can flow unobstructed to the total planar area of the obstruction.

i Note that the loss coefficient used in the equation for the device velocity is not the same as the loss coefficient used in the equation for the approach velocity. The loss coefficients in Equations 16.2-2 and 16.2-3 are related to the flow regime of the problem:

- For a viscous flow regime (e.g., laminar flow, slow flow, very dense packing), you should select a linear velocity relationship:

$$\Delta p = \frac{\rho}{2} l_{c1} v \quad (16.2-5)$$

- For an inertial flow regime (e.g., turbulent flow), you should select a quadratic velocity relationship:

$$\Delta p = \frac{\rho}{2} l_{c2} v^2 \quad (16.2-6)$$



- For a combination of these two types of flow, you should select a linear+quadratic velocity relationship:

$$\Delta p = \Delta p_{\text{linear}} + \Delta p_{\text{quadratic}} \quad (16.2-7)$$

You can obtain the loss coefficients in several ways:

- experimental measurements
- computational measurements
- from a reference (The loss coefficients for many vent configurations are available in [14].)

16.3 Adding a Resistance to Your Airpak Model

To include a resistance in your **Airpak** model, click on the  button in the **Object** creation toolbar and then click on the  button to open the **Resistances** panel, shown in Figures 16.3.1 and 16.3.2.

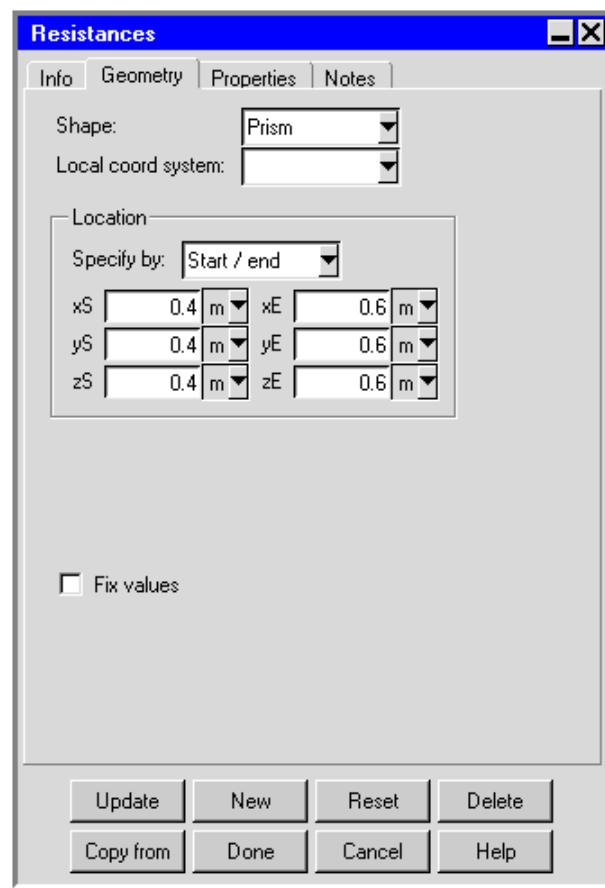


Figure 16.3.1: The Resistances Panel (Geometry Tab)

The procedure for adding a resistance to your **Airpak** model is as follows:

1. Create a resistance. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the resistance, if required. See Section 7.4.1 for details.
3. Change the graphical style of the resistance, if required. See Section 7.4.2 for details.

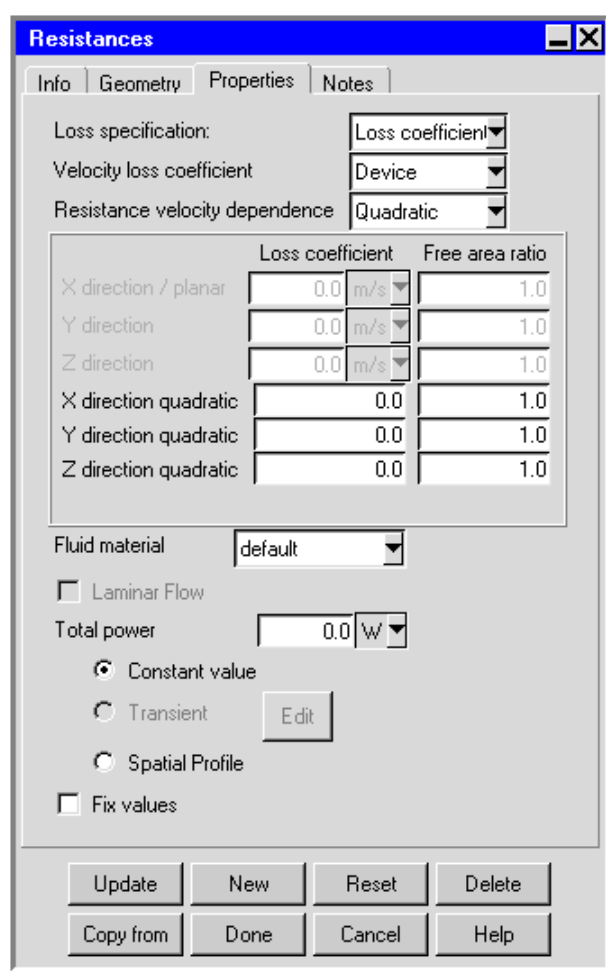


Figure 16.3.2: The Resistances Panel (Properties Tab)

4. In the **Geometry** tab, specify the geometry, position, and size of the resistance. There are three different kinds of geometry available for resistances in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.
5. In the **Properties** tab, specify the characteristics for the vent.
 - (a) Select the **Loss specification** in the drop-down list. The following options are available:
 - To specify the loss coefficient, select **Loss coefficient** and then select the method to be used to calculate the velocity loss coefficient. The following options are available in the **Velocity loss coefficient** drop-down list.

- To use the device-velocity method, select **Device** and select the method to be used to calculate the **Resistance velocity dependence**. There are three options in the drop-down list: **Linear**, **Quadratic**, and **Linear+quadratic**. Finally, specify the linear and/or quadratic **Loss coefficient** and **Free area ratio** for each of the three coordinate directions.
- To use the approach-velocity method, select **Approach** and select the method to be used to calculate the **Resistance velocity dependence**. Finally, specify the linear and/or quadratic **Loss coefficient** for each of the three coordinate directions.
- To calculate the pressure drop using a power-law method, select **Power law**. Specify the **Coefficient** (C in Equation 16.2-1) and the **Exponent** (n in Equation 16.2-1).



You must specify these inputs in SI units.

- To define a piecewise-linear profile for the pressure drop as a function of the speed of the fluid through the resistance in the three coordinate directions, select **Loss curve**. **Airpak** allows you to describe the curve(s) either by positioning a series of points on a graph using the **Resistance curve** graphics display and control window, or by specifying a list of resistance speed/pressure coordinate pairs using the **Curve specification** panel. These options are available in the **Edit** drop-down lists for the **X-direction**, **Y-direction**, and **Z-direction** under **Pressure loss curve data**. For details about using the **Resistance curve** and **Curve specification** panels, see Sections 10.5.1 and 10.5.2.

To load a previously defined curve, click the appropriate **Load** button. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.

To save a curve, click the appropriate **Save** button. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.



Note that the boxes to the right of the **Edit** drop-down lists will be empty if you have not defined a curve for the resistance in a particular direction. Each box will contain the first speed value if you have defined a curve for that direction.

- (b) Specify the **Fluid material** for the resistance. By default, this is specified as **default** for the resistance. This means that the material specified as the **Fluid material** for the resistance is defined under **Default fluid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Fluid material** for the resistance, select a material from the **Fluid material** drop-down list. See Section 7.7 for details on material properties.

- (c) Specify whether the interior of the resistance is to be modeled as a laminar zone by toggling the **Laminar Flow** option. Note that this option is only available when one of the turbulence models has been enabled in the **Basic parameters** panel.
- (d) Specify the total power dissipated by the resistance. There are two options for specifying the total power:

Constant value allows you to specify a constant value for the **Total power**.

Transient allows you to specify the total power as a function of time. This option is available if you have selected **Transient** under **Time variation** in the **Basic parameters** panel. Select **Transient** under **Total power** and enter a value for the **Total power**. To edit the transient parameters for the resistance, click **Edit** next to **Transient**. See Chapter 20 for more details on transient simulations.

Heat exchangers are two-dimensional modeling objects representing the heat exchange with the surrounding air. For a planar heat exchanger in **Airpak**, a lumped-parameter model is used to model a heat exchange element. The heat exchanger allows you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator.

In this chapter, information about the characteristics of heat exchangers is presented in the following sections:

- Section 17.1: Geometry, Locations, and Dimensions
- Section 17.2: Modeling a Planar Heat Exchanger in **Airpak**
- Section 17.3: Adding a Heat Exchanger to Your **Airpak** Model

17.1 Geometry, Location, and Dimensions

Heat exchanger location and dimension parameters vary according to the geometry. Heat exchanger geometries include rectangular, inclined, circular, and 2D polygon. These geometries are described in Section 7.4.4.

17.2 Modeling a Planar Heat Exchanger in **Airpak**

A lumped-parameter model for a heat exchange element (for example, a radiator or condenser), is available in **Airpak**. The heat exchanger allows you to specify both the pressure drop and the heat transfer coefficient as functions of the velocity normal to the heat exchanger.

17.2.1 Modeling the Pressure Loss Through a Heat Exchanger

In the heat exchanger model in **Airpak**, the heat exchanger is considered to be infinitely thin, and the pressure drop through the heat exchanger is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient which you supply. That is, the pressure drop, Δp , varies with the normal component of velocity through the radiator, v , as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \quad (17.2-1)$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient, which can be specified as a constant or as a polynomial function.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^N r_n v^{n-1} \quad (17.2-2)$$

where r_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the resistance.

17.2.2 Modeling the Heat Transfer Through a Heat Exchanger

The heat flux from the heat exchanger to the surrounding fluid is given as

$$q = h(T_{\text{air,d}} - T_{\text{ext}}) \quad (17.2-3)$$

where q is the heat flux, $T_{\text{air,d}}$ is the temperature downstream of the heat exchanger, and T_{ext} is the reference temperature for the liquid. The convective heat transfer coefficient, h , can be specified as a constant or as a polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^N h_n v^n; 0 \leq N \leq 7 \quad (17.2-4)$$

where h_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the resistance in m/s.

Either the actual heat flux (q) or the heat transfer coefficient and heat exchanger temperature ($h, T_{\text{air,d}}$) may be specified. q (either the entered value or the value calculated using Equation 17.2-3) is integrated over the heat exchanger surface area.

17.2.3 Calculating the Heat Transfer Coefficient

To model the thermal behavior of the heat exchanger, you must supply an expression for the heat transfer coefficient, h , as a function of the fluid velocity through the resistance, v . To obtain this expression, consider the heat balance equation:

$$q = \frac{\dot{m} c_p \Delta T}{A} = h(T_{\text{air,d}} - T_{\text{ext}}) \quad (17.2-5)$$

where

q	=	heat flux
\dot{m}	=	fluid mass flow rate
c_p	=	specific heat capacity of fluid
h	=	empirical heat transfer coefficient
T_{ext}	=	external temperature (reference temperature for the liquid)
$T_{\text{air,d}}$	=	temperature downstream from the heat exchanger
A	=	heat exchanger frontal area

Equation 17.2-5 can be rewritten as

$$q = \frac{\dot{m}c_p(T_{\text{air,u}} - T_{\text{air,d}})}{A} = h(T_{\text{air,d}} - T_{\text{ext}}) \quad (17.2-6)$$



where $T_{\text{air,u}}$ is the upstream air temperature. The heat transfer coefficient, h , can therefore be computed as

$$h = \frac{\dot{m}c_p(T_{\text{air,u}} - T_{\text{air,d}})}{A(T_{\text{air,d}} - T_{\text{ext}})} \quad (17.2-7)$$

or, in terms of the fluid velocity,

$$h = \frac{\rho v c_p (T_{\text{air,u}} - T_{\text{air,d}})}{T_{\text{air,d}} - T_{\text{ext}}} \quad (17.2-8)$$

17.3 Adding a Heat Exchanger to Your Airpak Model

To include a heat exchanger in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Heat exchangers panel, shown in Figures 17.3.1 and 17.3.2.

The procedure for adding a heat exchanger to your Airpak model is as follows:

1. Create a heat exchanger. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the heat exchanger, if required. See Section 7.4.1 for details.
3. Change the graphical style of the heat exchanger, if required. See Section 7.4.2 for details.
4. In the **Geometry** tab, specify the geometry, position, and size of the heat exchanger. There are four different kinds of geometry available for heat exchangers in the **Shape** drop-down list. The inputs for these geometries are described in Section 7.4.4.

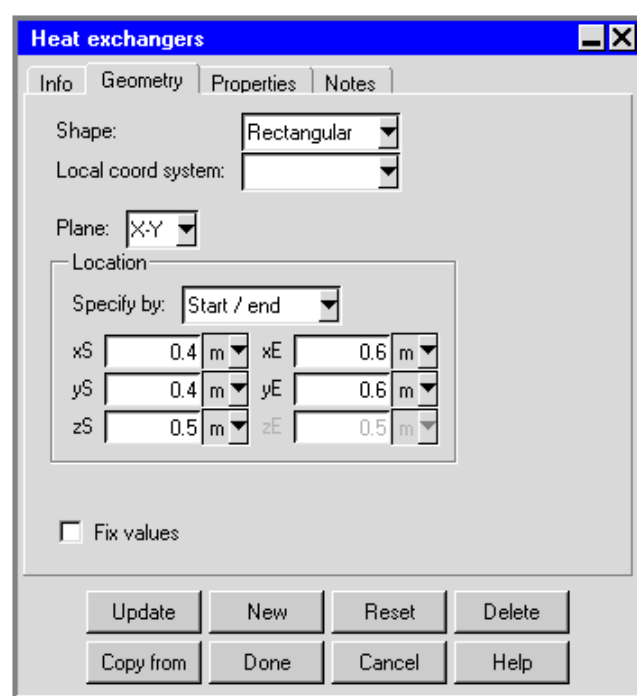


Figure 17.3.1: The Heat exchangers Panel (Geometry Tab)

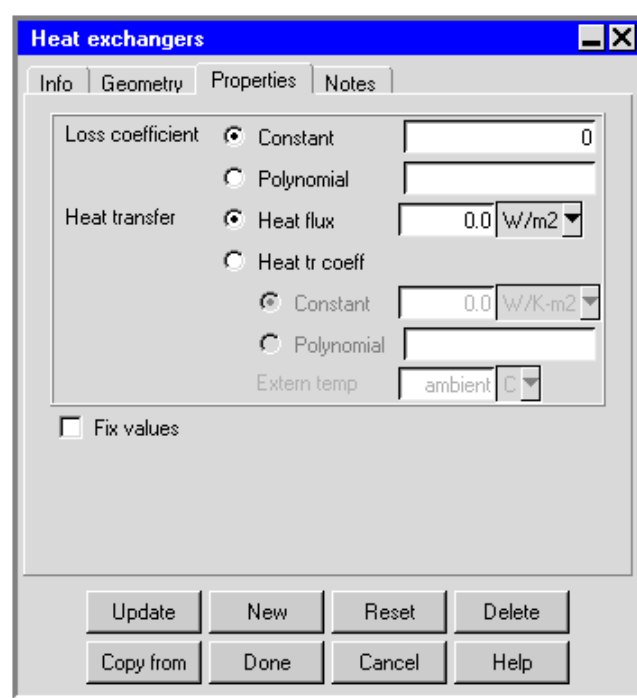


Figure 17.3.2: The Heat exchangers Panel (Properties Tab)

See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object.

5. In the **Properties** tab, specify the **Loss coefficient** for the heat exchanger. To specify a constant loss coefficient, select **Constant** and enter the value in the **Constant** text entry box. To specify a polynomial loss coefficient, select **Polynomial** and enter the coefficients for the polynomial equation (separated by spaces) in the **Polynomial** text entry box. For example, if you have a polynomial equation of the form

$$a + bv + cv^2 + dv^3$$

you would enter

a b c d

in the **Polynomial** text entry box.

i Note that you must use SI units for the polynomial equation.

6. Specify the **Heat transfer** through the heat exchanger. There are two options:

Heat flux specifies a fixed rate of heat transfer from the resistance to the surrounding fluid.

Heat tr coeff specifies a heat transfer coefficient to model the heat input/output of the resistance. You can specify a **Constant** value for the heat transfer coefficient (h in Equation 17.2-3). Alternatively, you can specify a **Polynomial** heat transfer coefficient by entering the coefficients for the polynomial equation (separated by spaces) in the **Polynomial** text entry field.

i Note that you must use SI units for the polynomial equation.

Specify the temperature of the resistance (i.e., $T_{\text{air,d}}$ in Equation 17.2-3) next to **Extern temp**. The value of the **ambient** temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

A hood is a three-dimensional modeling object that consists of a neck, a canopy, a flange, and exhausts. An example of a hood is shown in Figure 18.0.1.

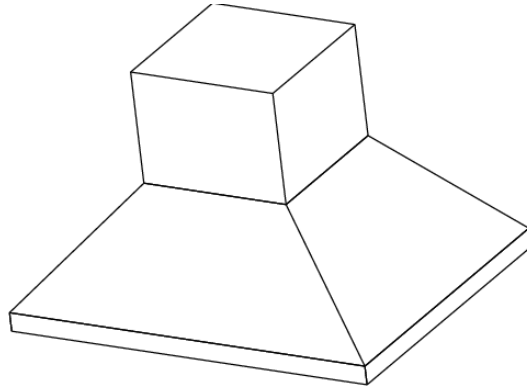


Figure 18.0.1: An Example of a Hood

To configure a hood in the model, you must specify its location and dimensions. You must also specify the flow rate associated with the hood.

In this chapter, information about the characteristics of a hood is presented in the following sections:

- Section 18.1: Location and Dimensions
- Section 18.2: Flow Rate
- Section 18.3: Adding a Hood to Your Airpak Model

18.1 Location and Dimensions

Airpak allows you to specify the position and size of the neck and canopy of the hood. You can also specify the position and size of the exhausts. The hood can have one or two exhausts: Airpak creates two exhausts by default.

You can select whether the flange is vertical (Figure 18.1.1) or horizontal (Figure 18.1.2) on the hood, or you can specify a hood without a flange. If the hood has a flange, you can specify the height of the flange, or you can specify the sizes of its individual parts.

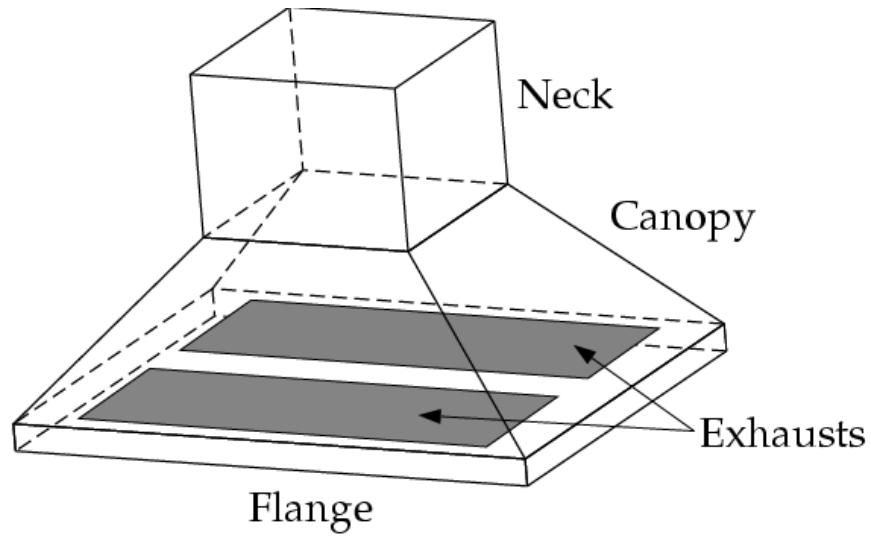


Figure 18.1.1: Hood with Vertical Flange

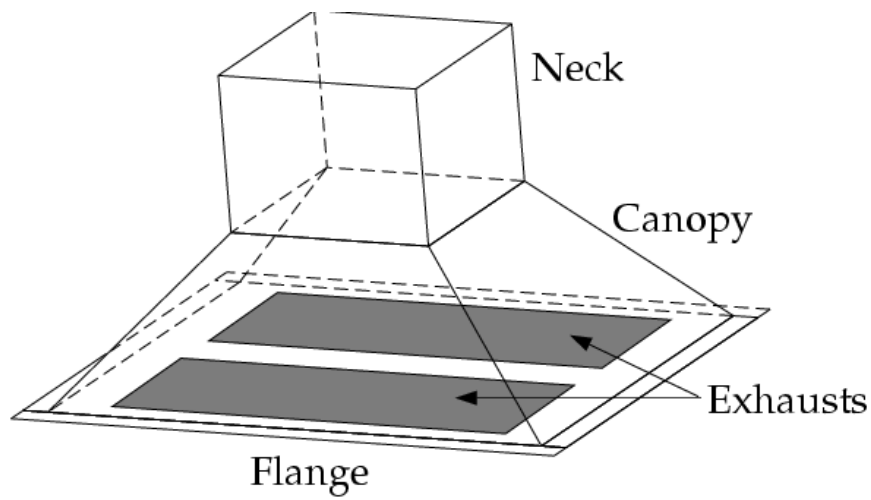


Figure 18.1.2: Hood with Horizontal Flange

18.2 Flow Rate



In **Airpak**, the flow rate for the hood can be specified in terms of total mass flow rate or total volume flow rate. If the hood has two exhausts of equal size, **Airpak** will divide the total mass (or volume) flow rate equally between the two exhausts. If the hood has two exhausts of unequal sizes, **Airpak** will divide the total mass (or volume) flow rate between the two exhausts based on the following equations:

$$\dot{m}_1 = \frac{A_1}{A_1 + A_2} \times \dot{m} \quad (18.2-1)$$

$$\dot{m}_2 = \frac{A_2}{A_1 + A_2} \times \dot{m} \quad (18.2-2)$$

where \dot{m}_1 is the mass (or volume) flow rate for exhaust 1, \dot{m}_2 is the mass (or volume) flow rate for exhaust 2, A_1 is the area of exhaust 1, A_2 is the area of exhaust 2, and \dot{m} is the total mass (or volume) flow rate.

18.3 Adding a Hood to Your Airpak Model

To include a hood in your **Airpak** model, click on the  button in the Object creation toolbar and then click on the  button to open the **Hoods** panel, shown in Figure 18.3.1.

The procedure for adding a hood to your **Airpak** model is as follows:

1. Create a hood. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the hood, if required. See Section 7.4.1 for details.
3. Change the graphical style of the hood, if required. See Section 7.4.2 for details.
4. Specify the position of the hood in the room (see also Section 7.3.7 for details on repositioning an object) and the overall size of the hood (see also Section 7.3.6 for details on resizing an object). You can specify the size and position of the hood in the **Properties** tab in two ways:
 - To specify the location and dimensions of one or more parts of the hood and have **Airpak** adjust the relative location and dimensions of the other parts of the hood:
 - (a) Make sure that **Fix shapes** is selected in the **Properties** tab.
 - (b) If desired, specify the start coordinates (x_S , y_S , z_S) and end coordinates (x_E , y_E , z_E) of the **Neck** of the hood.

Hoods

Info Properties Notes

☒ Fix shapes [Edit details](#)

Neck

xS	-0.2	m	xE	0.2	m
yS	3.0	m	yE	2.7	m
zS	-0.2	m	zE	0.2	m

Canopy

xS	-0.5	m	xE	0.5	m
yS	2.7	m	yE	2.4	m
zS	-0.4	m	zE	0.4	m

Flange type

☒ Vertical ☐ Horizontal ☐ None

Height m

☒ Exhaust 1

xS	-0.4	m	xE	0.4	m
zS	-0.35	m	zE	-0.05	m

☒ Exhaust 2

xS	-0.4	m	xE	0.4	m
zS	0.05	m	zE	0.35	m

Flow rate

☒ Volume m³/s

☐ Mass kg/s

☐ Fix values

Update New Reset Delete

Copy from Done Cancel Help

Figure 18.3.1: The Hoods Panel (Properties Tab)

- (c) If desired, specify the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE) of the Canopy of the hood.
 - (d) If desired, specify the Flange type. There are three options: Vertical, Horizontal, or None. If you select Vertical or Horizontal, you must also specify the Height of the flange.
 - (e) If desired, specify the number of exhausts in the hood by selecting or deselecting Exhaust 1 or Exhaust 2. Specify the start coordinates (xS, zS) and end coordinates (xE, zE) of Exhaust 1 and Exhaust 2, as required.
- To specify the location and dimensions of the individual parts of the hood (without any automatic adjustment of the other parts):
 - (a) Deselect Fix shapes in the Properties tab and click Edit details to open the Hoods shape data panel (Figure 18.3.2).

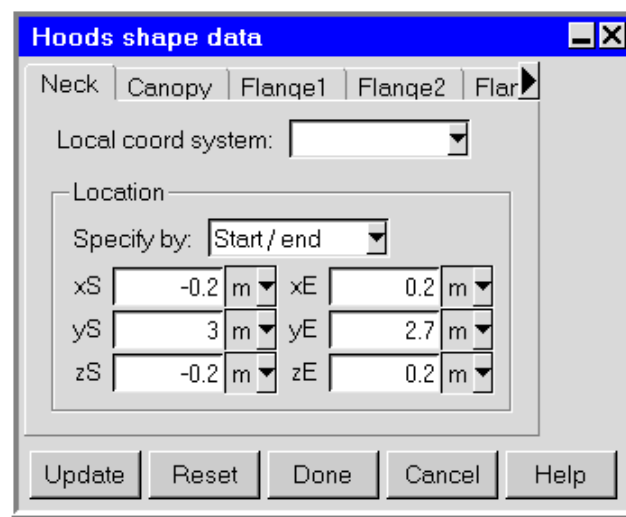


Figure 18.3.2: The Hoods shape data Panel (Neck Tab)

- (b) Select one of the parts of the hood under Section by clicking on the appropriate tab.
- (c) Keep the default Origin of (0, 0, 0) or specify the Origin for the part (if required).
- (d) To specify the neck in the hood, click the Neck tab (Figure 18.3.2). Select Start/end and enter values for the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE) of the neck. Alternatively, you can select Start/length and enter values for the start coordinates (xS, yS, zS) and lengths of the sides (xL, yL, and zL) of the neck.
- (e) To specify the canopy in the hood, click the Canopy tab (Figure 18.3.3). The default canopy is defined as a non-uniform polygon. Specify the plane

in which the base lies (Y-Z, X-Z, or X-Y), the **Height**, and the coordinates of its vertices (low 1, low 2, low 3, low 4, high 1, high 2, high 3, high 4). You can add and remove vertices using the **Add** and **Rem** buttons.

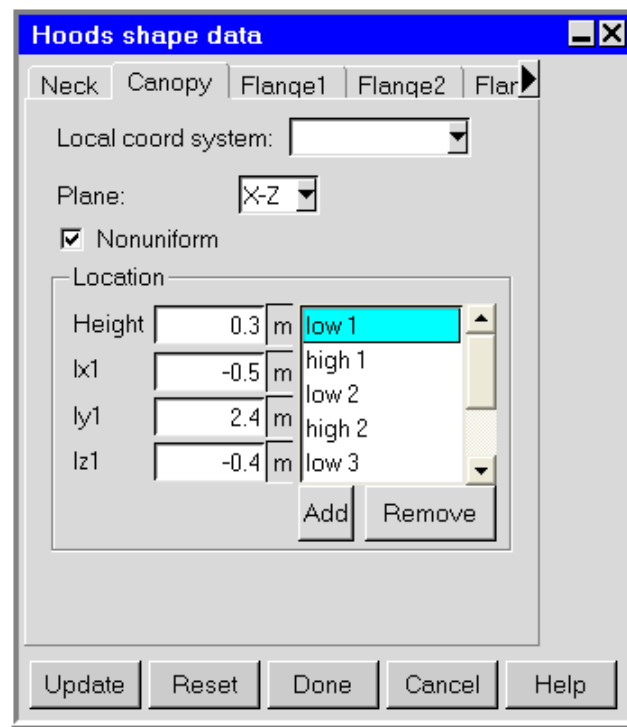


Figure 18.3.3: The Hoods shape data Panel (Canopy Tab)

If you select **None** under **Plane**, the polygonal block will be constructed with zero height and will take the shape of the base of the polygon.

If you require a canopy with a uniform polygonal shape, deselect **Nonuniform** and specify the plane in which the base lies (Y-Z, X-Z, or X-Y), the **Height**, and the coordinates of the vertices (**vert 1**, **vert 2**, **vert 3**, **vert 4**) on the base plane.

- (f) For each flange you want to specify in the hood, click the appropriate **Flange** tab (e.g., **Flange1**).

Select the plane in which the part of the flange lies (Y-Z, X-Z, or X-Y) in the **Plane** drop-down list (Figure 18.3.4). Select **Start/end** and enter values for the start coordinates (**xS**, **yS**, **zS**) and end coordinates (**xE**, **yE**, **zE**) of the part of the flange, or select **Start/length** and enter values for the start coordinates (**xS**, **yS**, **zS**) and lengths of the sides (**xL**, **yL**, and **zL**) of the part of the flange.

- (g) For each exhaust that you want to include in the hood, use the horizontal scroll bar and click the appropriate **Exhaust** tab (e.g., **Exhaust1**).

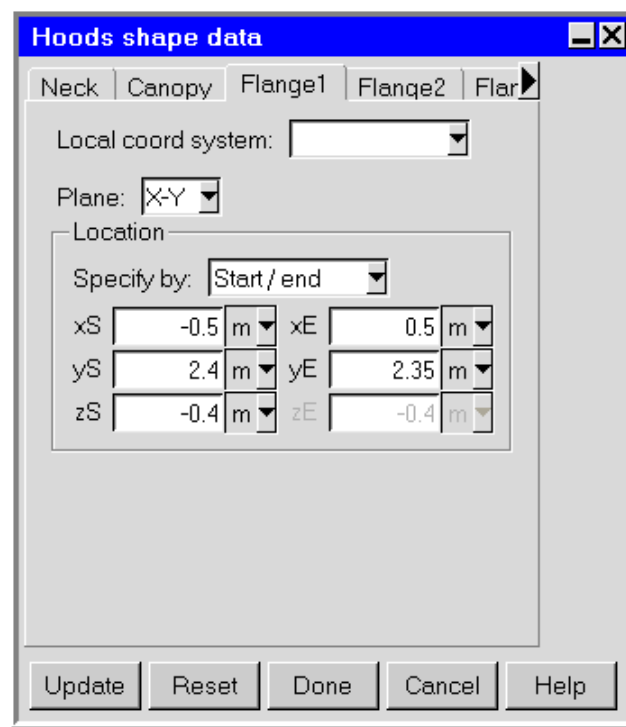


Figure 18.3.4: The Hoods shape data Panel (Flange Tab)

Specify the plane in which the exhaust lies (Y-Z, X-Z, or X-Y) in the Plane drop-down list (Figure 18.3.5).

Select **Start/end** and enter values for the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE) of the exhaust, or select **Start/length** and enter values for the start coordinates (xS, yS, zS) and lengths of the sides (xL, yL, and zL) of the exhaust.

- (h) Click **Done** to save the changes to the hood and close the Hoods shape data panel.
5. Specify the **Flow rate** in the Hoods panel. You can specify a **Volume** flow rate or a **Mass** flow rate.

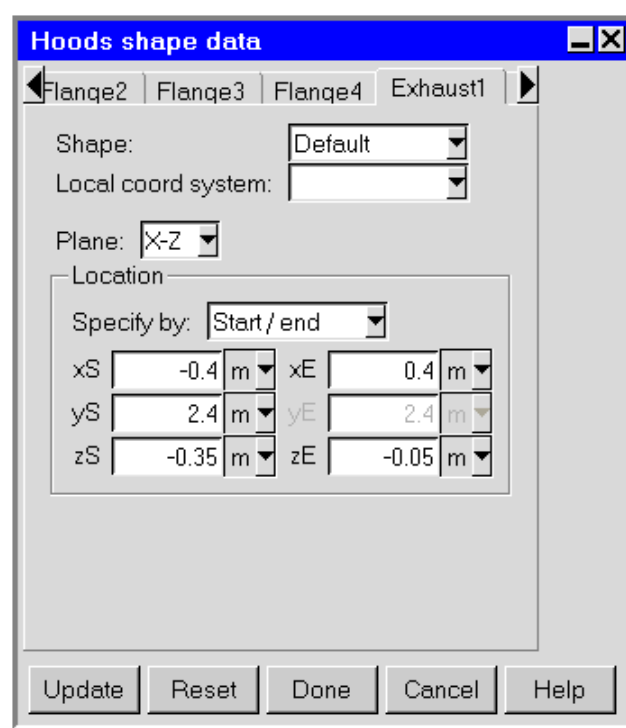


Figure 18.3.5: The Hoods shape data Panel (Exhaust Tab)



Wires are used to construct polygonal ducts of arbitrary cross-section. To construct a duct, a minimum of three wires are necessary. There is no upper limit to the number of wires that can be used. Note that wires are not to be used by themselves; they are only used to construct polygonal ducts. To learn how to construct polygonal ducts, refer to Section 25.3.1.

To configure a wire in the model, the only parameters that you need to specify are the locations of the vertices. A wire can have any number of vertices, and it need not be parallel to any axis.



Polygonal duct macros created using wires must be meshed with the tetrahedral mesher. To learn how to use the tetrahedral mesher, refer to Section 26.3.

19.1 Adding a Wire to Your Airpak Model

To include a wire in your Airpak model, click on the  button in the Object creation toolbar and then click on the  button to open the Wires panel, shown in Figure 19.1.1.

The procedure for adding a wire to your Airpak model is as follows:

1. Create a wire. See Section 7.3.2 for details on creating a new object and Section 7.3.9 for details on copying an existing object.
2. Change the description of the wire, if required. See Section 7.4.1 for details.
3. Change the graphical style of the wire, if required. See Section 7.4.2 for details.
4. Specify the vertices of the wire. There are two ways to modify the position of any vertex on the wire:
 - In the **Geometry** tab, select the name of the vertex (e.g., **vert 1**) in the vertex list and modify its **X**, **Y**, and **Z** coordinate values.
 - Use the <Shift> key and the right mouse button to move the vertex point in the graphics window.

Use the **Add** and **Rem** buttons to add or delete vertices from the wire.

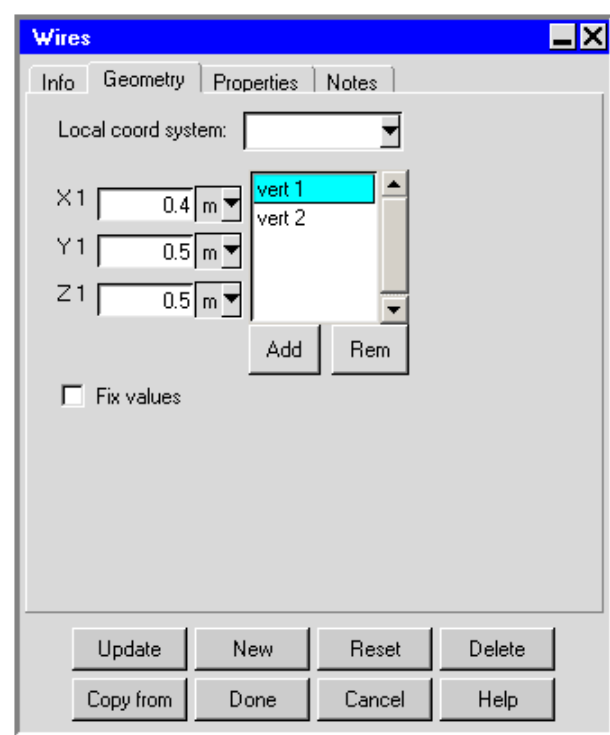


Figure 19.1.1: The Wires Panel (Geometry Tab)

Airpak can solve the equations for conservation of mass, momentum, and energy in time-dependent form. Thus Airpak can be used to simulate a wide variety of time-dependent phenomena, including transient heat conduction and convection, as well as transient species transport.

Activating time dependence is sometimes useful when attempting to solve steady-state problems that tend toward instability (e.g., natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

Airpak uses a fully-implicit time-integration scheme for transient analysis. This chapter provides details about setting up a transient simulation in Airpak and postprocessing the results. See Section 31.6.3 for details about the theory of transient simulations.



The information in this chapter is divided into the following sections:

- Section 20.1: User Inputs for Transient Simulations
- Section 20.2: Specifying Variables as a Function of Time
- Section 20.3: Postprocessing for Transient Simulations

20.1 User Inputs for Transient Simulations

To solve a transient problem, you will follow the procedure outlined below:

1. Enable the **Transient** option in the **Transient setup** tab of the **Basic parameters** panel (Figure 20.1.1). To open the **Basic parameters** panel, double-click on the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Basic parameters**

2. Enter the starting and ending times for the simulation in the **Start** and **End** entry fields.
3. To edit the default transient parameters (if required), click on **Edit parameters**. This will open the **Transient parameters** panel shown in Figure 20.1.2.

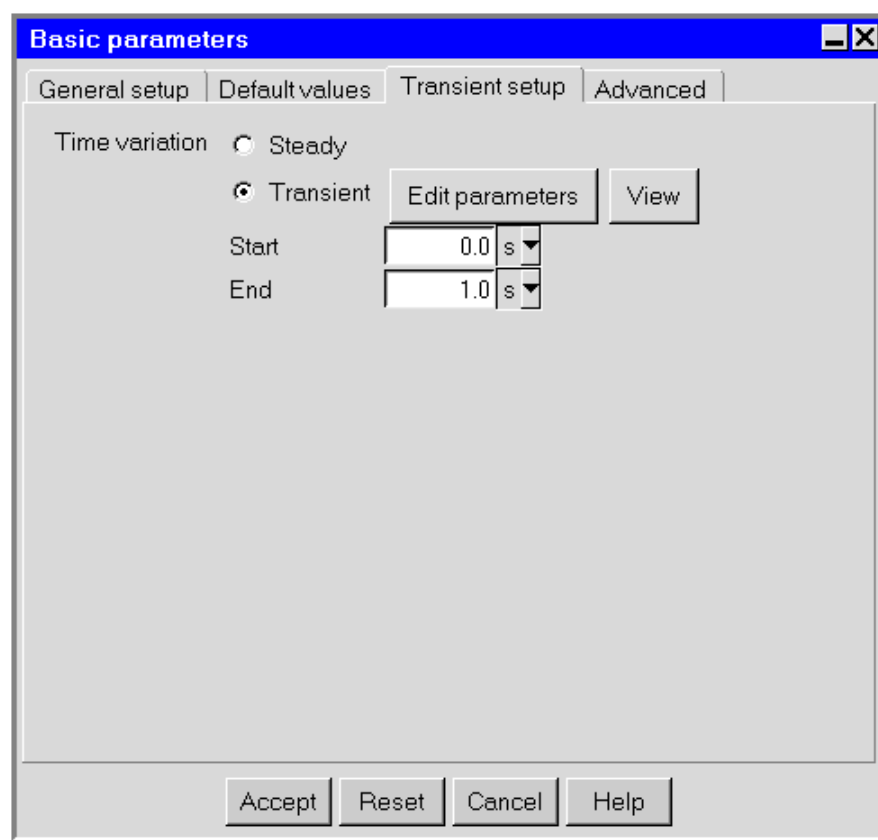


Figure 20.1.1: The Basic parameters Panel (Transient setup Tab)

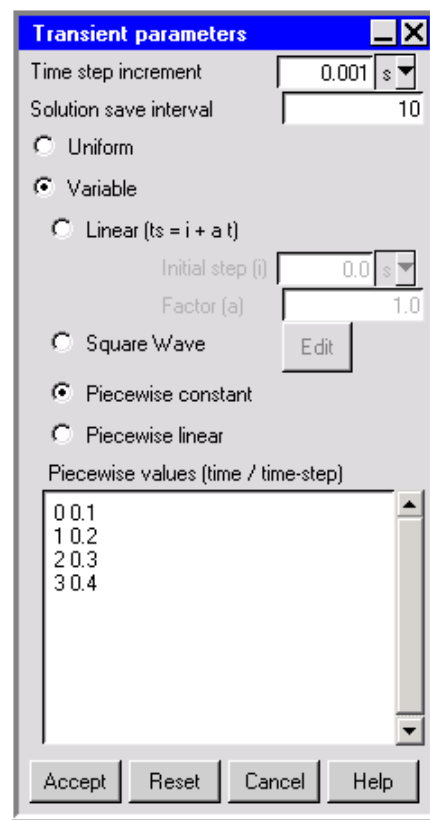


Figure 20.1.2: The Transient parameters Panel

- (a) Specify a value for the **Time step increment**. The default value is 0.001 s. The specified **Time step increment** will be used for all time steps if **Uniform** is selected. It will also be used if **Piecewise linear** is selected and $t_s < t_1$ (see below).
- (b) Specify the **Solution save interval**. This value tells **Airpak** how often during the solution procedure the results should be saved. Every tenth time step is saved by default.



Only steps that have been saved can be used during postprocessing. Large amounts of data can be generated from a transient simulation, so you should decide how much data should be saved for postprocessing. If you are only interested in the solution at the end of the transient simulation, then very little intermediate data need be saved. If you are interested in animating and studying the transient effects in detail, then almost every time step should be saved.

- (c) Specify whether **Airpak** should use a uniform or non-uniform time step.

Uniform specifies a uniform time step. The value of the **Time step increment** will be used for all time steps.

Variable specifies a non-uniform time step. There are three options:

Linear ($ts = i + a t$) specifies a linear variation of the time step with time:

$$\Delta t = \Delta t_0 + at \quad (20.1-1)$$

where Δt is the time step at time t , Δt_0 is the initial time step, and a is a constant. Enter values for the **Initial step (i)** and the **Factor (a)**.

Square Wave specifies a square-wave profile for the time step variation. If the variation of time step required is regular/periodic with time, then this option can be used instead of the **Piecewise constant** option. To specify a square-wave profile, click **Edit** to open the **Square Wave Time-Step Parameters** panel (Figure 20.1.3).

To understand the meaning of the various parameters in the **Square Wave Time-Step Parameters** panel, see Figure 20.1.4.

Piecewise constant specifies a piecewise constant variation of the time step. Enter a list of the time/time-step pairs in the **Piecewise values (time / time-step)** box at the bottom of the **Transient parameters** panel. It is important to give the numbers in pairs, but the spacing between the numbers is not important. An example of the arrangement of the time/time-step pairs is shown below:

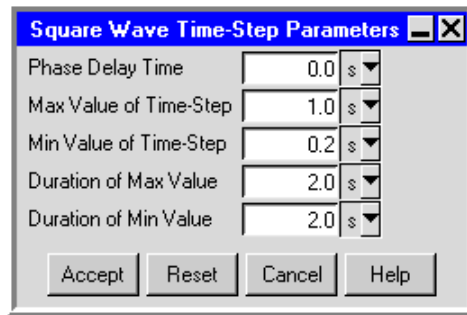


Figure 20.1.3: The Square Wave Time-Step Parameters Panel

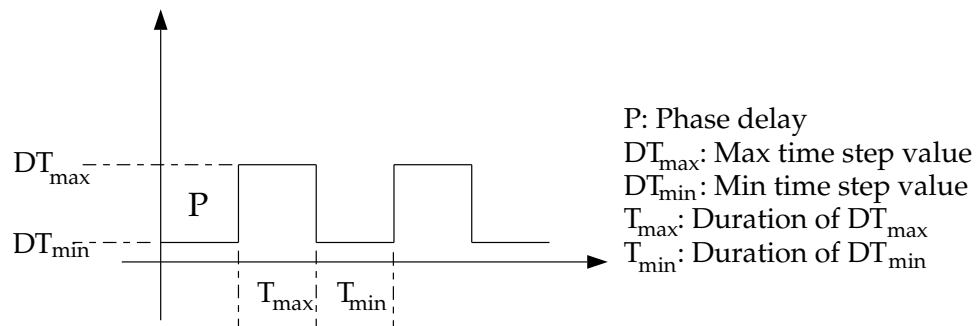


Figure 20.1.4: The Square Wave Inputs for Time Step Variation

$$\begin{array}{ll} t_1 & \Delta t_1 \\ t_2 & \Delta t_2 \\ t_3 & \Delta t_3 \end{array}$$

Airpak will use the time/time-step pairs such that if $t < t_1$, the time step Δt is given by

$$\Delta t = \Delta t_1 \quad (20.1-2)$$

Similarly, if $t_1 \leq t < t_2$,

$$\Delta t = \Delta t_2 \quad (20.1-3)$$

Piecewise linear specifies a piecewise linear variation of the time step. Enter a list of the time/time-step pairs in the **Piecewise values (time / time-step)** box at the bottom of the **Transient parameters** panel. It is important to give the numbers in pairs, but the spacing between the numbers is not important. An example of the arrangement of the time/time-step pairs is shown below:

$$\begin{array}{ll} t_1 & \Delta t_1 \\ t_2 & \Delta t_2 \\ t_3 & \Delta t_3 \end{array}$$

Airpak will use the time/time-step pairs such that if $t_s < t \leq t_1$, the time step Δt is a linear function that varies between Δt_0 and Δt_1 :

$$\Delta t = \Delta t_0 + \left[\frac{\Delta t_1 - \Delta t_0}{t_1 - t_s} \right] (t - t_s) \quad (20.1-4)$$

where t_s is the **Start** time specified in the **Basic parameters** panel, and Δt_0 is the **Time step** increment specified at the top of the **Transient parameters** panel.

Similarly, for $t_1 < t \leq t_2$, Δt is a linear function that varies between Δt_1 and Δt_2 :

$$\Delta t = \Delta t_1 + \left[\frac{\Delta t_2 - \Delta t_1}{t_2 - t_1} \right] (t - t_1) \quad (20.1-5)$$

- (d) Click **Accept** in the **Transient parameters** panel to store the new settings.

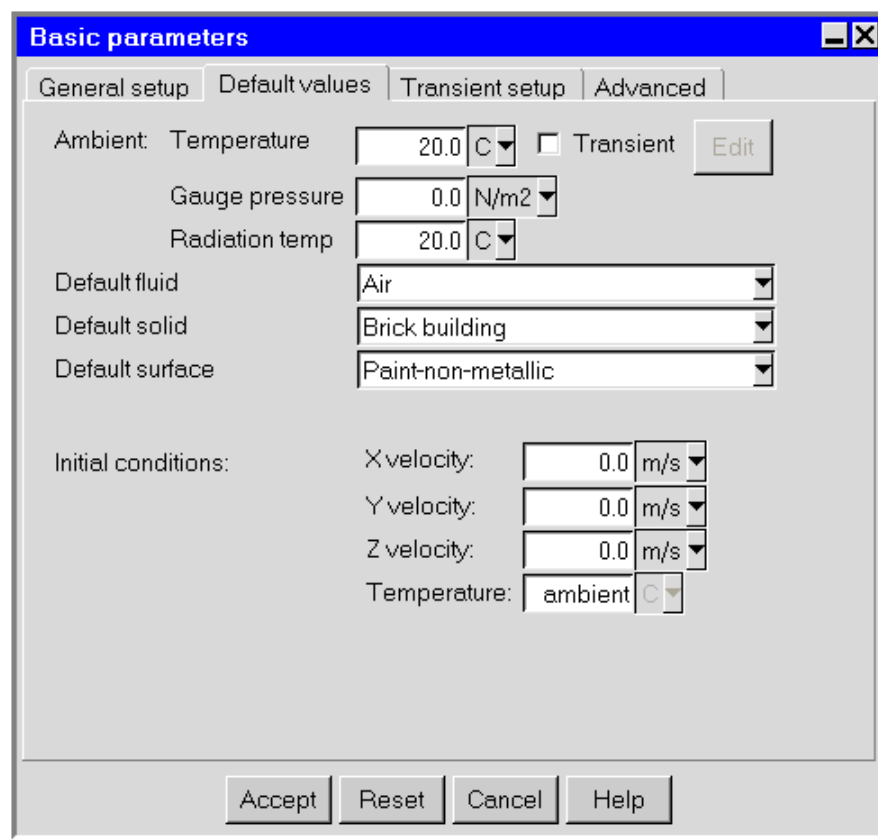


Figure 20.1.5: The Basic parameters Panel (Default values tab)

4. Specify an ambient value of temperature by entering the ambient **Temperature** under **Ambient values** in the **Default values** tab of the **Basic parameters** panel (Figure 20.1.5).
5. To define the variation of ambient temperature as a function of time, select **Transient** next to **Temperature**, and then click **Edit**. This will open the **Transient temperature** panel shown in Figure 20.1.6.

i You must specify all inputs in SI units in the **Transient temperature** panel.

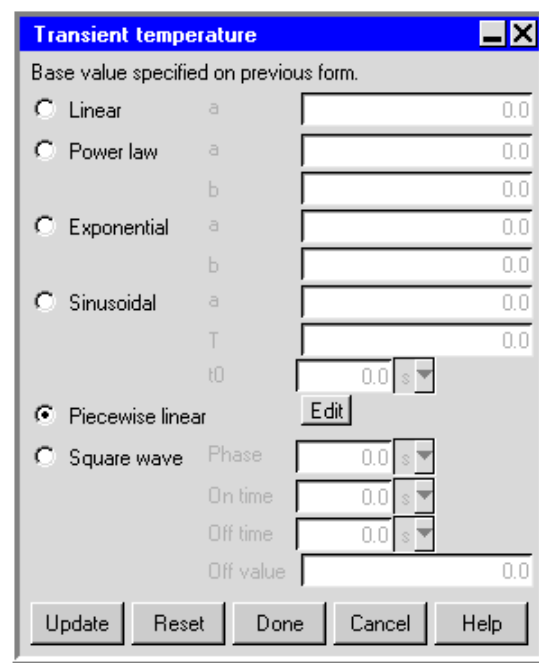


Figure 20.1.6: The Transient temperature Panel

- (a) Select the method for the calculation of the variation of ambient temperature with time. The following options are available, and described in more detail in Section 20.2.

Linear specifies a linear variation of ambient temperature with time. Enter a value for the coefficient a in Equation 20.2-1.

Power law specifies a power law variation of ambient temperature with time. Enter values for the coefficients a and b in Equation 20.2-2.

Exponential specifies an exponential variation of ambient temperature with time. Enter a value for the coefficients a and b in Equation 20.2-3.

Sinusoidal specifies a sinusoidal variation of ambient temperature with time. Enter values for the period T , the phase shift t_0 , and the coefficient a in Equation 20.2-4.

Piecewise linear specifies a piecewise linear variation of ambient temperature with time. There are two ways to define the piecewise linear variation: the **Time/value curve** window and the **Curve specification** panel. These methods are described in Section 20.2.

Square wave specifies a square wave variation of ambient temperature with time. Specify the **Phase**, the **On time**, the **Off time**, and the **Off value** for the square wave, as described in Section 20.2.

- (b) Click **Done** in the **Transient temperature** panel when you have finished specifying the variation of ambient temperature with time.
6. Specify the initial conditions for the fluid in your model in the **Default values** tab of the **Basic parameters** panel (Figure 20.1.5). For a transient simulation, this corresponds to the initial physical state of the fluid. See Section 6.4.8 for more details on setting initial conditions.
 7. Once you have set the desired parameters in the **Basic parameters** panel, click **Accept** to store the new settings.
 8. Specify the transient characteristics of the objects in your **Airpak** model. You can specify transient characteristics for the following objects:
 - **Solid or fluid block:** Specify the total power as a function of time by selecting the **Transient** option in the **Properties** tab of the **Blocks** panel (see Chapter 8) and clicking **Edit**. The **Transient power** panel used to define the transient power of the block is very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the block.
 - **Hollow block:** Specify the species concentrations as a function of time (see Chapter 21 on species transport). Select **Species** in the **Blocks** panel and click the **Edit** button to open the **Species concentrations** panel. Select **Transient** for the desired species and click **Edit** to open the **Transient species** panel. The **Transient species** panel is used to define the transient species concentrations for the block and is very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the block.
 - **Fan:** Specify the transient strength of a characteristic curve fan by turning on **Transient strength** in the **Properties** tab of the **Fans** panel (see Chapter 9) and clicking **Edit**. The **Transient fan strength** panel used to define the transient strength of the fan is similar to the **Transient temperature** panel shown in Figure 20.1.6, except that it contains only the **Piecewise linear** and **Square wave** options.
 - **Vent:** Specify the species concentrations as a function of time (see Chapter 21 for information on species transport). Select **Species** in the **Vents** panel and

click the **Edit** button to open the **Species concentration** panel. Then follow the instructions for the hollow block, described above.

- **Opening:** For a free opening, you can specify the static pressure, temperature, x velocity, y velocity, or z velocity for flow across the opening as a function of time, by turning on the **Transient** option in the **Properties** tab of the **Openings** panel (see Chapter 11) and clicking **Edit**. The **Transient pressure**, **Transient temperature**, **Transient X velocity**, **Transient Y velocity**, and **Transient Z velocity** panels are very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the opening.

i You must specify all inputs in SI units in the **Transient temperature** panel.

You can also specify the species concentrations as a function of time (see Chapter 21 for information on species transport). Select **Species** in the **Openings** panel and click the **Edit** button to open the **Species concentration** panel. Then follow the instructions for the hollow block, described above.

For a recirculation opening, you can specify the start time (**tS**) and the end time (**tE**) of the period when the opening is active.

- **Wall:** You can specify the rate of heat transfer through the wall or the temperature on the outer surface of the wall as a function of time, by turning on the **Transient** option in the **Properties** tab of the **Walls** panel (see Chapter 11) and clicking **Edit**. You can also specify the heat transfer coefficient as a function of time by turning on the **Transient** option in the **Wall external thermal conditions** panel. To open the **Wall external thermal conditions** panel, select **External conditions** in the **Walls** panel and click **Edit**. The **Transient power/area**, **Transient temperature**, and **Transient heat tr coeff** panels are very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the wall.

i You must specify all inputs in SI units in the **Transient temperature** panel.

- **Conducting thick partition:** Specify the total power as a function of time by selecting **Transient** in the **Properties** tab of the **Partitions** panel (see Chapter 14) and clicking **Edit**. The **Transient power** panel used to define the transient power of the partition is very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the partition.
- **Source:** Specify the power as a function of time by turning on the **Transient** option in the **Properties** tab of the **Sources** panel (see Chapter 15) and clicking **Edit**. The **Transient power** panel used to define the transient power of the source

is very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the source.

You can also specify the species concentrations for a 2D source or the mass of a species entering or leaving the room per second for a 3D source, as a function of time (see Chapter 21 for information on species transport). Select **Species** in the **Sources** panel and click the **Edit** button to open the **Species concentrations** panel. Then follow the instructions for the hollow block, described above.

- **Resistance:** Specify the total power as a function of time by selecting **Transient** in the **Properties** tab of the **Resistances** panel (see Chapter 16) and clicking **Edit**. The **Transient power** panel used to define the transient power of the resistance is very similar to the **Transient temperature** panel shown in Figure 20.1.6. You must also specify the start time **tS** and the end time **tE** for the transient simulation for the resistance.

You can also specify the mass of a species entering or leaving the room per second as a function of time (see Chapter 21 for information on species transport). Select **Species** in the **Resistances** panel and click the **Edit** button to open the **Species concentrations** panel. Then follow the instructions for the hollow block, described above.



Note that the start time **tS** and the end time **tE** can be different for different objects.

9. Set the maximum number of iterations to be performed per time step under **Iterations/timestep** in the **Basic settings** panel (Figure 20.1.7). To open this panel, double-click on the **Basic settings** item under the **Solution settings** node in the **Model manager** window. If the **Convergence criteria** are satisfied before this number of iterations is performed, the solution will advance to the next time step. The default value of 20 iterations/time step should be acceptable for most cases. If you find that the solution is not converging at each time step, you can increase the **Iterations/timestep** value and/or decrease the size of the **Time step increment** specified in the **Transient parameters** panel (Figure 20.1.2).
10. Start the calculation using the **Solve** panel. To open this panel, click **Run solution** in the **Solve** menu. Click **Start solution** to start the calculation.

Files will be saved at the frequency specified next to **Solution save interval** in the **Transient parameters** panel (Figure 20.1.2). A separate file is saved for each time step for which data are saved.

20.2 Specifying Variables as a Function of Time

For transient simulations, you can specify variables that vary as a function of time. You can specify different kinds of variations, including linear, power law, exponential,

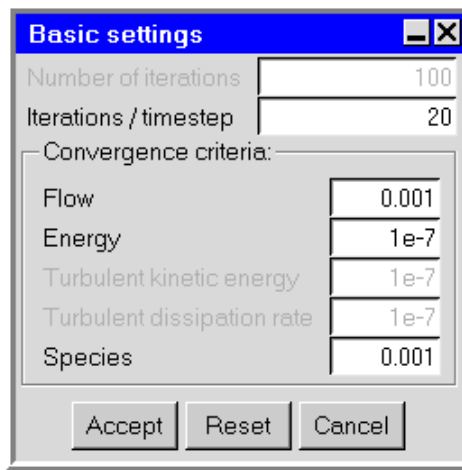


Figure 20.1.7: The Basic settings Panel for a Transient Simulation

sinusoidal, piecewise linear, and square wave:

- linear:

$$s_t = s_0 + at \quad (20.2-1)$$

where t is the time, s_t is the value of the variable at time t , s_0 is the value of the variable at $t = 0$, and a is a constant.

- power law:

$$s_t = s_0 + at^b \quad (20.2-2)$$

where t is the time, s_t is the value of the variable at time t , s_0 is the value of the variable at $t = 0$, and a and b are constants.

- exponential:

$$s_t = s_0 + be^{at} \quad (20.2-3)$$

where t is the time, s_t is the value of the variable at time t , s_0 is the value of the variable at $t = 0$, and a and b are constants.

- sinusoidal:

$$s_t = s_0 + a \sin \left[\frac{2\pi}{T}(t - t_0) \right] \quad (20.2-4)$$

where t is the time, s_t is the value of the variable at time t , s_0 is the value of the variable at $t = 0$, T is the period, t_0 is the phase shift, and a is a constant.

- piecewise linear:

$$s_t = s_0 f(t) \quad (20.2-5)$$

where $f(t)$ is a function of time, s_t is the value of the variable at time t , and s_0 is the value of the variable at $t = 0$. See Section 20.2.2 for information on defining $f(t)$ using the Time/value curve window; see Section 20.2.3 for details on defining $f(t)$ using the Curve specification panel.

- square wave:

$$s_t = s_0 f(t) \quad (20.2-6)$$

where s_t is the value of the variable at time t and s_0 is the value of the variable at $t = 0$. $f(t)$ is a function of time as shown in Figure 20.2.1.

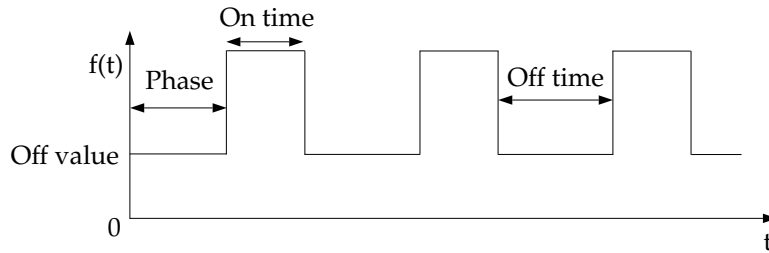


Figure 20.2.1: Definition of a Square Wave

The **Phase** is the time between $t = 0$ and the first peak of the square wave. The **On time** is the time that the square wave is at its peak value. The **Off time** is the time between peak values of the square wave. The **Off value** is the value of the square waves between the peaks of the wave. The peak value is the specified value of the transient quantity (e.g., 293 for Temperature in Figure 20.1.1).

20.2.1 Displaying the Variation of Transient Parameters with Time

After you have specified the transient parameters for your Airpak model (e.g., time step, temperature, power, etc.) you can display the variation of these parameters with time using the Transients panel (Figure 20.2.2). To open the Transients panel, click the View button under Time variation in the Transient setup tab of the Basic parameters panel.

The following functions are available for viewing the variation of transient parameters:

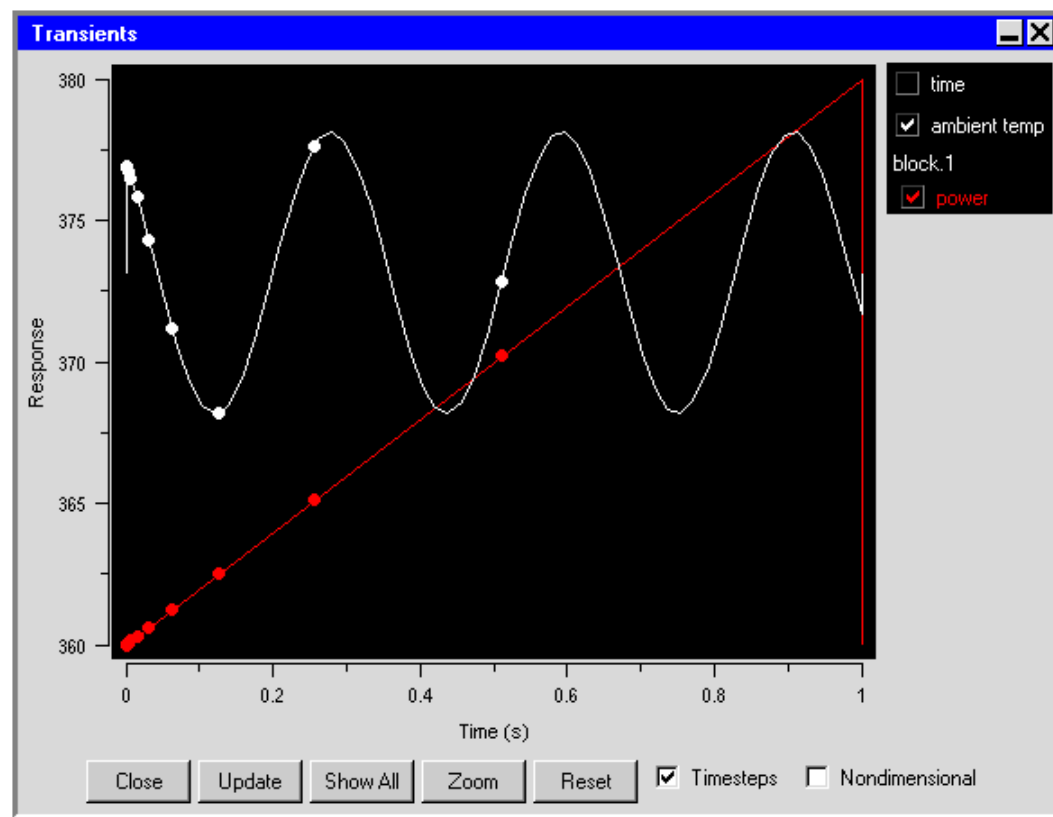


Figure 20.2.2: The Transients Panel

- To display specified variation plots of the transient parameters as a function of time (e.g., **time**, **ambient temp**, **power**), toggle the check boxes in the upper left of the panel .
- To update the variation plots after having made changes in the **Basic parameters** panel, the **Transient parameters** panel, the **Transient temperature** panel, or any similar transient parameter panel, click **Update**.
- To display variation plots of all defined transient parameters, click **Show all**.
- To zoom into an area of the variation plot, click **Zoom**. Position the mouse pointer at a corner of the area to be zoomed, click the left mouse button, drag open a selection box to the desired size, and click the left button again. To cancel a zoom before it is completed, click the right mouse button. To return to the default view, click **Reset**.
- To display the time-step points on the variation plots, turn on the **Timesteps** option.
- To non-dimensionalize the y axis of the variation plot; i.e., divide each parameter value by the initial value of parameter, turn on the **Nondimensional** option.

20.2.2 Using the Time/value curve Window to Specify a Piecewise Linear Variation With Time

You can specify a piecewise linear variation of a variable with time using the Time/value curve graphics display and control window (Figure 20.2.3). To open the Time/value curve window, select **Piecewise linear** in the **Transient temperature** panel (Figure 20.1.6) or in any similar transient parameter panel, and click **Edit**. Select **Graph editor** from the resulting list.

The following functions are available for creating, editing, and viewing a curve:

- To create a new point on the curve, click on the curve with the middle mouse button.
- To move a point on the curve, hold down the middle mouse button while positioned over the point, and move the mouse to the new location of the point.
- To delete a point on the curve, click the right mouse button on the point.
- To zoom into an area of the curve, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the Time/value curve window, with appropriate changes to the axes. After you have zoomed into an area of the model, click on **Full range** to restore the graph to its original axes and scale.

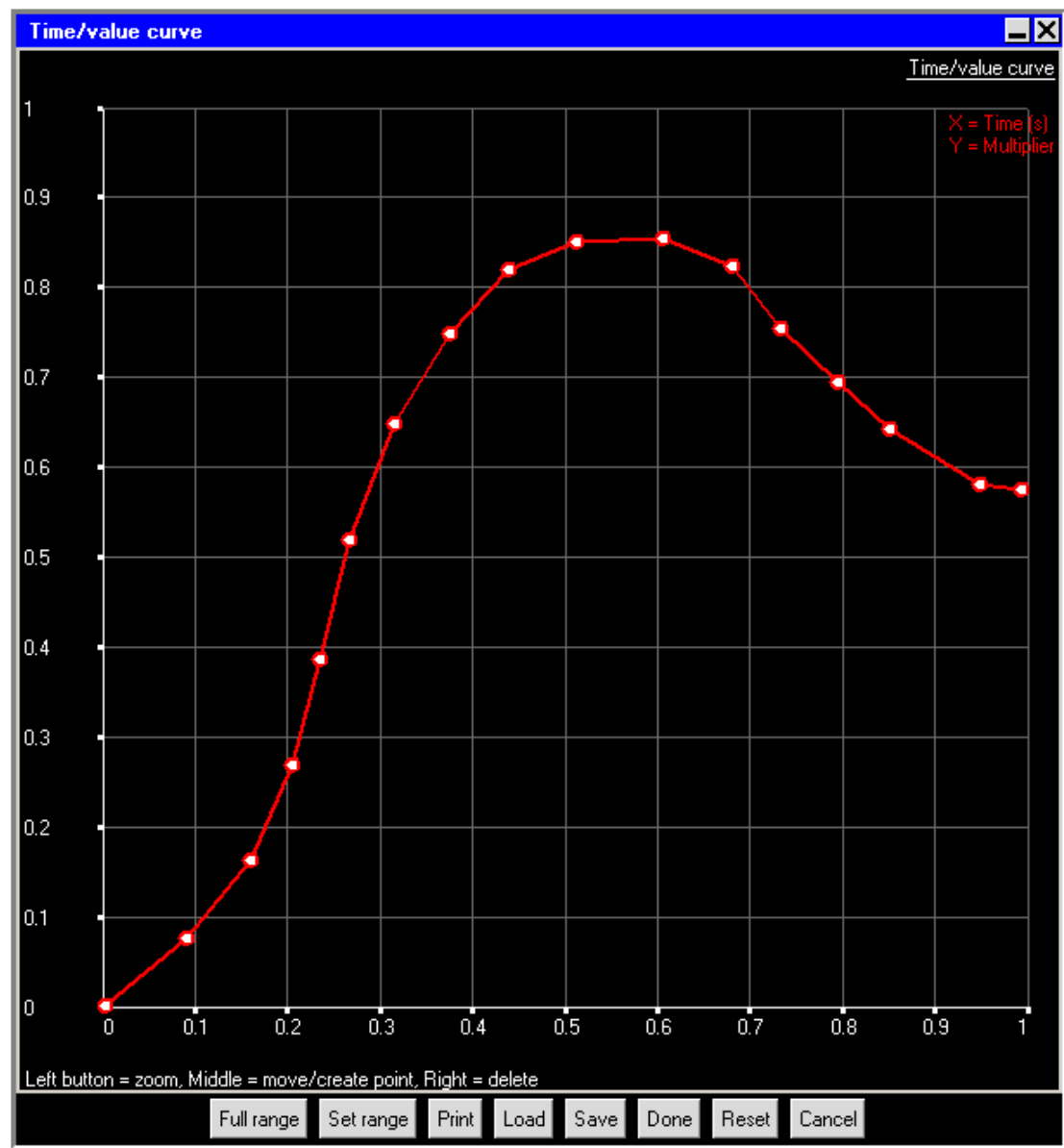


Figure 20.2.3: The Time/value curve Graphics Display and Control Window

- To set the minimum and maximum values for the scales on the axes, click on **Set range**. This will open the **Set range** panel (Figure 20.2.4).

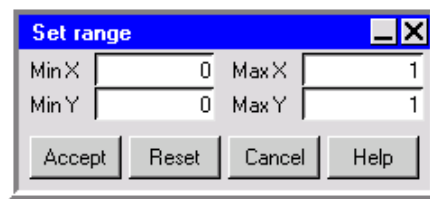


Figure 20.2.4: The Set range Panel

Enter values for Min X, Min Y, Max X, and Max Y and click **Accept**.

- To load a previously defined curve, click on **Load**. This will open the **Load curve** file selection dialog box. Select the file containing the curve data and click **Accept**. See Section 2.1.8 for details on selecting a file.
- To save a curve, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the curve data is to be saved.

You can use the **Print** button to print the curve. See Section 3.5 for details on saving hardcopy files.

Click **Done** when you have finished creating the curve; this will store the curve and close the **Time/value curve** graphics display and control window. Once the curve is defined, you can view the pairs of coordinates defining the curve in the **Curve specification** panel. See Figure 20.2.5 for the pairs of coordinates for the curve shown in Figure 20.2.3.

20.2.3 Using the Curve specification Panel to Specify a Piecewise Linear Variation With Time

You can define a piecewise linear variation of a variable with time using the **Curve specification** panel (Figure 20.2.5). To open the **Curve specification** panel, select **Piecewise linear** in the **Transient temperature** panel (Figure 20.1.6) or in any similar transient parameter panel, and click **Edit**. Select **Text editor** from the resulting list.

To define a curve, specify a list of coordinate pairs in the **Curve specification** panel. It is important to give the numbers in pairs, but the spacing between numbers is not important. Click **Accept** when you have finished entering the pairs of coordinates; this will store the values and close the **Curve specification** panel.

Once the pairs of coordinates have been entered, you can view the piecewise linear curve in the **Time/value curve** graphics display and control window. See Figure 20.2.3 for the piecewise linear curve for the values shown in Figure 20.2.5.

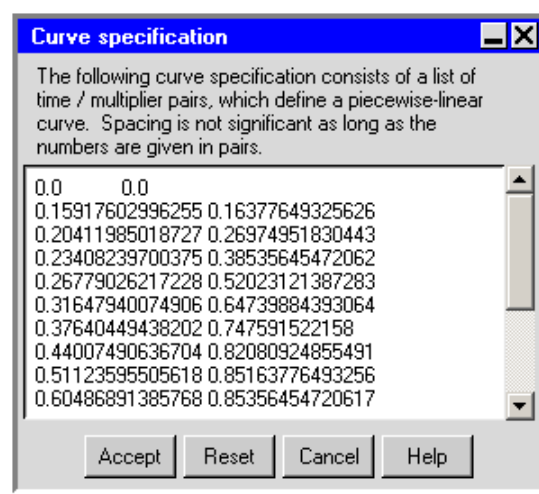



Figure 20.2.5: The Curve specification Panel

20.3 Postprocessing for Transient Simulations

An Airpak solution to a transient problem consists of a set of solutions, each of which describes the state of the model at a specific time step in the process. There are four ways to examine the results of your transient simulation:

- examine the results at a particular time step or value (a “snapshot”)
- create an animation showing how relevant contours and/or vectors change over time
- generate a report for a specified time in the transient simulation
- create a history plot showing the change in a specified variable over time at specified points in the model

20.3.1 Examining Results at a Specified Time

You can examine the results at a particular time step or value (a “snapshot”) using the Post-processing time panel (Figure 20.3.1). To open this panel, select **Transient settings** in the **Post** menu or click on the  button in the **Postprocessing** toolbar.

Post → Transient settings

To examine the results of the transient simulation at a particular time step or value (a snapshot), follow the steps below.

1. Specify the contour or vector data to be displayed in the relevant postprocessing panel (see Chapter 28 for more details on specifying postprocessing objects).

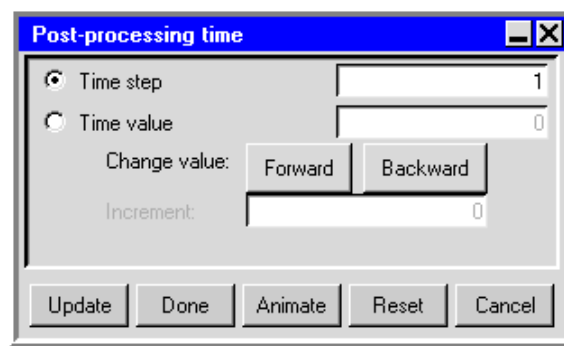


Figure 20.3.1: The Post-processing time Panel

2. Specify either a single time value or a particular time step in the transient simulation to be used for the snapshot. To specify a single time value, select **Time value** in the **Post-processing time** panel (Figure 20.3.1) and enter the value. To specify a particular time step, select **Time step** in the **Post-processing time** panel and enter the time step.

If you specify the time step to be used (for a snapshot analysis), **Airpak** employs the solution results at the specified time step. If you specify a time value that does not exactly correspond to a stored time step, **Airpak** interpolates the snapshot solution between solutions for the time steps before and after the specified value.


3. Click **Update** to display the solution results at the specified time or time step.
4. Click **Animate** to open the **Transient animation** panel. Animating a transient solution using this panel is described in further detail in Section 20.3.3.
5. Click **Forward** or **Backward** to display the solution results at a time value or time step incremented from the specified **Time value** or **Time step**. To display results at the next or previous time value, select **Time value**, input the time value increment (Δt) in the **Increment** field, then click **Forward** or **Backward**. To display results at the next or previous time step, select **Time step** and click **Forward** or **Backward**.

20.3.2 Creating Time-Averaged Results

For a transient problem, you can create and postprocess a new time-averaged solution using the **Time averaged data** panel (Figure 20.3.3). A time-averaged solution will allow you to view the results of a time-dependent problem as if it were steady-state, but without disabling the problem's time dependence.

To open the **Time average** panel, first click on **Time average** in the **Post** menu

Post → **Time average**

or click on the  button in the Object creation toolbar to open the Version selection panel (Figure 20.3.2).

In the Version selection panel, select the set of solution data (e.g., `sample00`) from the list and click **Okay**. Note that you may have to use the scroll bar to locate the desired solution set.

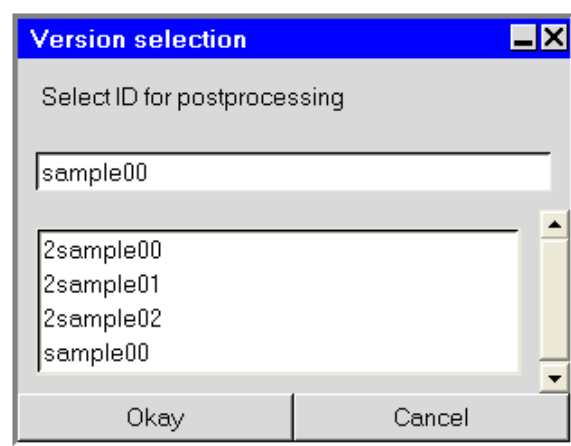


Figure 20.3.2: The Version selection Panel

The **Message** window will report that the solution data have been loaded, and the Time averaged data panel (Figure 20.3.3) will open.

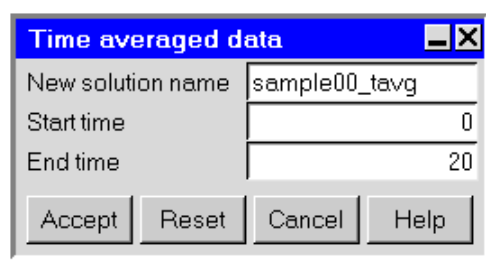


Figure 20.3.3: The Time averaged data Panel

To create a time-averaged solution from your transient solution, use the following procedure:

1. In the Time averaged data panel, specify the **Start time** and **End time** for the averaging and click **Accept**.


The **Message** window will report that the time-averaged data have been written from the original solution set. By default, new solution data will be saved with the suffix `_tavg` appended to the name of the old data set.

2. Open the job with the new solution data (e.g., `sample00_tavg`).

File→Open

3. Examine the time-averaged results (e.g., Post→Object face, Post→Plane cut, etc.).

20.3.3 Creating an Animation

You can create an animation of the results for your transient simulation using the Post-processing time panel (Figure 20.3.1). To open this panel, select **Transient settings** in the Post menu or click on the  button in the Postprocessing toolbar.

Post→Transient settings

To create an animation showing how relevant contours and/or vectors change over time, follow the steps below.

1. Specify the contour or vector data to be displayed in the relevant postprocessing panel (see Chapter 28 for more details on specifying postprocessing objects).
2. Select Time value in the Post-processing time panel (Figure 20.3.1).
3. Click **Animate** to open the Transient animation panel (Figure 20.3.4).

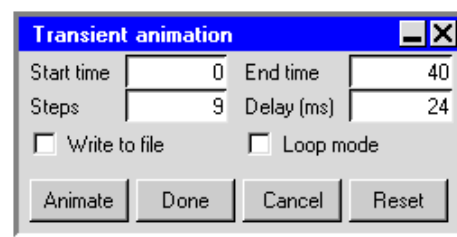


Figure 20.3.4: The Transient animation Panel

4. Specify the **Start time** and **End time** for the animation. By default, the **Start time** is zero and the **End time** is the time corresponding to the solution end time.
5. Specify either the number of steps or the number of “frames” per second.

Steps specifies the number of steps, i.e., the number of frames Airpak should display between the starting frame and the ending frame. Airpak will interpolate smoothly between the starting frame and the ending frame that you define, creating the specified number of frames. Note that the number of steps includes the starting and ending frames.

Delay (ms) specifies the time between each frame in the animation in milliseconds. Note that the delay value is added to the time taken by **Airpak** to create a snapshot for the animation. The time it takes **Airpak** to create a snapshot is dependent on the speed of your system.

6. If you want **Airpak** to play the animation only in the graphics window, and you want the playback to repeat continuously, turn on the **Loop mode** option. To play the animation once through from start to finish, turn off the **Loop mode** option. You can also use the **Loop mode** option if you want to save an animated GIF or FLI file, as described in Section 28.2.5.
7. To save the animation to a file, select the **Write to file** option. See Section 28.2.5 for details on saving an animation. Note that when the **Write to file** option is selected, the **Delay (ms)** field becomes the **Frames/s** field. The **Frames/s** field designates the number of animation frames displayed per second.
8. If you selected the **Write to file** option in the step above, define the region of the graphics window that should be written to the file. See Section 3.5.2 for details about specifying the **Print region**.
9. Click **Animate** to start the animation. To stop the animation during playback, click on the **Interrupt** button in the upper right hand corner of the **Airpak** interface.

20.3.4 Generating a Report

You can generate a report for a specified time in the transient simulation by using the **Define summary report** panel, the **Full report** panel, or the **Define point report** panel.

To open the **Define summary report** panel, click **Summary report** in the **Report** menu.

Report → Summary report

To open the **Full report** panel, click **Full report** in the **Report** menu.

Report → Full report

To open the **Define point report** panel, click **Point report** in the **Report** menu.


Report → Point report

You can generate a report at a specified time or a specified time step in the transient analysis by entering a value for either **Time** or **Step** under **Report time** in the relevant panel. See Chapter 29 for more information on generating reports.

20.3.5 Creating a History Plot

A history plot shows the change in a specified variable over time at specified points in the model. Time is plotted on the x axis and the variable value is plotted on the y axis.

By including a number of points in a single plot, you can compare various locations in the model with respect to the value of a specified variable as a function of time.

You can examine how a variable changes over time at selected points in the model using the History plot panel (Figure 20.3.5). To open this panel, select History plot in the Post menu or click on the  button in the Postprocessing toolbar.

Post→History plot

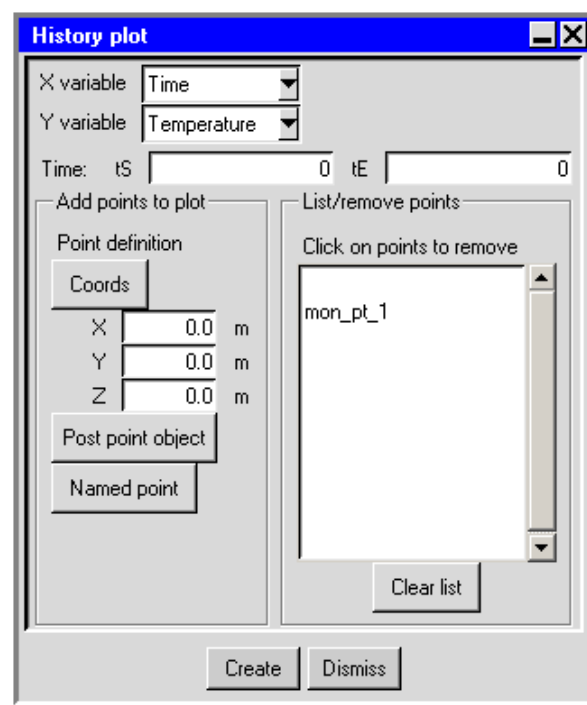


Figure 20.3.5: The History plot Panel

To create a history plot, follow the steps below.

1. Select the **Variable** to be plotted on the vertical axis. **Temperature** is selected by default in the **Variable** list. To change the variable to be displayed, select a new variable from the **Variable** drop-down list. See Chapter 30 for information on variables.
2. Specify the interval of the transient simulation to be examined by entering values for the start time **tS** and the end time **tE**. This interval will be plotted on the horizontal axis.
3. Under **Add points to plot**, specify the location of the point of interest for the history plot. There are three options:

Coords allows you to specify the coordinates of a point. Enter the X, Y, and Z coordinates and click on the **Coords** button. The point will appear in the List/remove points list.

Post point object allows you to select an existing point object from the **Selection** panel (Figure 20.3.6).

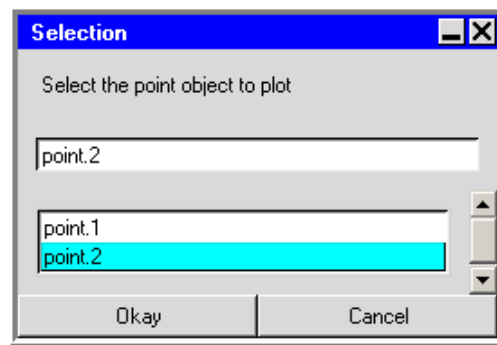


Figure 20.3.6: The Selection Panel

In the **Selection** panel, select the point in the list and click **Okay**. The point will appear in the List/remove points list in the **History plot** panel. See Section 28.2.7 for details on creating a point object.

Named point allows you to select an existing named point from the point drop-down list or create a new named point, as described below.

4. Click **Create** to display the XY time history plot of the selected variable at the point(s) specified. Click **Cancel** to close the **History plot** panel without creating a history plot.

Named Points

Airpak allows you to create named points that can be used to specify the location at which **Airpak** should create a history plot. Named points can also be used to specify the location at which **Airpak** should create a point report (see Section 29.6), define the location of a point object for postprocessing (see Section 28.2.7), and define the location of a point monitor (see Section 27.8).

To select a named point, click on the **Named point** button in the **History plot** panel (Figure 20.3.5) and select a named point from the point drop-down list. The point drop-down list can be used to create a new named point or edit an existing named point. These operations are described below.

The **Named point** panel can be used to view and manage all named points in your **Airpak** model. To open the **Named point** panel (Figure 20.3.7), click on the **Named point** button in the **History plot** panel and select **Edit/create points** in the point drop-down list.

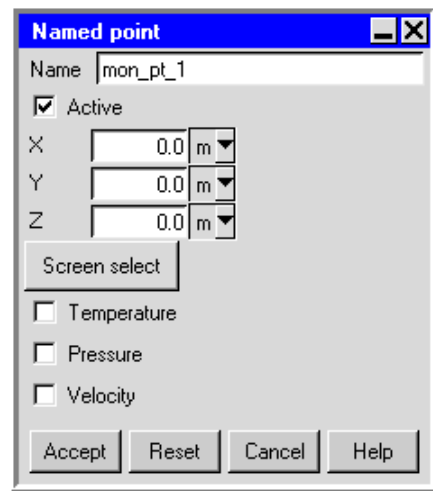


Figure 20.3.7: The Named point Panel

Creating a New Named Point

To create a new named point, click on the **Named point** button in the **History plot** panel and select **Edit/create points** from the point drop-down list. This will open the **Named point** panel (Figure 20.3.7). Enter a name in the **Name** field and then specify the location of the named point in the model. There are two ways to do this:

- Enter the X, Y, and Z coordinates.
- Click the **Screen select** button and then click on a location in the graphics window with your left mouse button to select the location of the named point.

Specify whether **Temperature**, **Pressure**, or **Velocity** will be monitored at the point. Click **Accept** when you have finished creating the named point, and Airpak will close the **Named point** panel.

Editing an Existing Named Point

To edit an existing named point, click on the **Named point** button in the **History plot** panel and select **Edit/create points** in the point drop-down list. Airpak will open the **Named point** panel (Figure 20.3.7). You can edit the name in the **Name** text entry field and then edit the location of the named point in one of two ways:

- Enter the X, Y, and Z coordinates.
- Click the **Screen select** button and then click on a location in the graphics window with the left mouse button to select the location of the named point.

Click **Accept** when you have finished editing the named point, and **Airpak** will close the **Named point** panel.



Note that, if you have used a named point to define the location of a point post-processing object, and you then edit the location of the named point, you will need to click **Update** in the **Named point** panel to update the position of the point postprocessing object in the graphics window.

Deleting Named Points

If there are named points in the **Named point** panel (Figure 20.3.7) that you no longer need, you can easily delete them. To remove specified points, click on the individual point names in the **List/remove points** list. The named point will be permanently removed from the point drop-down list in the **History plot** panel. To remove all named points from the point drop-down lists, click **Clear list** in the **Named point** panel.

Activating and Deactivating Named Points

By default, all named points that you create will be available in your current **Airpak** model. You can temporarily remove a named point from all of the point drop-down lists by turning off the **Active** option for the named point in the **Named point** panel (Figure 20.3.7). You can repeat this for each named point that you want to temporarily remove from your model. When a named point is deactivated, it is simply removed from the point drop-down lists, *not* deleted from **Airpak**. Since it still exists, you can easily add it to the point drop-down lists again by turning the **Active** option back on.

Note that, if you deactivate a named point that is currently being used in your **Airpak** model, **Airpak** will still use the named point. Deactivating a named point only removes it from the point drop-down lists so that it cannot be selected.

Airpak can model species transport with up to twelve species included in the calculation. Airpak models the mixing and transport of species by solving conservation equations describing convection and diffusion for each component species. This chapter describes the species modeling options in Airpak. See Section 31.1.4 for details about the theory of species transport in Airpak.

Information about species transport is provided in the following sections:

- Section 21.1: Overview of Modeling Species Transport
- Section 21.2: User Inputs for Species Transport Simulations
- Section 21.3: Postprocessing for Species Calculations

21.1 Overview of Modeling Species Transport

The basic steps for setting up a problem involving species transport are listed below.

1. Enable species transport, and specify the species to be included in the calculation.
2. Check and/or set the properties of the individual species in the model (e.g., viscosity, specific heat).
3. Set species concentrations for the objects in your model.

To view the species that are available to be included in your model, select the **Fluid** type and the **Gases** sub-type in the material **Edit** panel. (See Section 7.7.2 for details about the material **Edit** panel.)

In many cases, you will not need to modify any physical properties of individual species because Airpak will use the species properties specified in the materials database. Some properties, however, may not be defined as you want them in the database. You may also want to check the database values of other properties to be sure that they are correct for your particular application. The properties of a species can be viewed and/or edited using the **Materials** panel (see Section 7.7.2). You can also create a new material as described in Section 7.7.5.

For each of the species (i.e., fluid materials) in the model, you (or the database) must define the following physical properties:

- volumetric expansion coefficient, which can be a function of temperature
- molecular weight, which is used in the gas law and/or in the calculation of mole-fraction inputs or outputs
- viscosity, which can be a function of temperature
- thermal conductivity and specific heat (in problems involving solution of the energy equation), which can be functions of temperature or (for thermal conductivity only) velocity
- density
- mass diffusion coefficients, which govern the mass diffusion fluxes (Equations [31.1-8](#) and [31.1-9](#)), and can be a function of temperature

Descriptions of these property inputs are provided in Section [7.7.2](#).

21.2 User Inputs for Species Transport Simulations

To solve a problem involving species transport in Airpak, you will follow the procedure outlined below:

1. Enable the calculation of species transport in the Basic parameters panel. To open the Basic parameters panel (Figure 21.2.1), double-click on the Basic parameters item under the Problem setup node in the Model manager window.

Problem setup—>Basic parameters

Enable the calculation of species transport by selecting On next to Species.

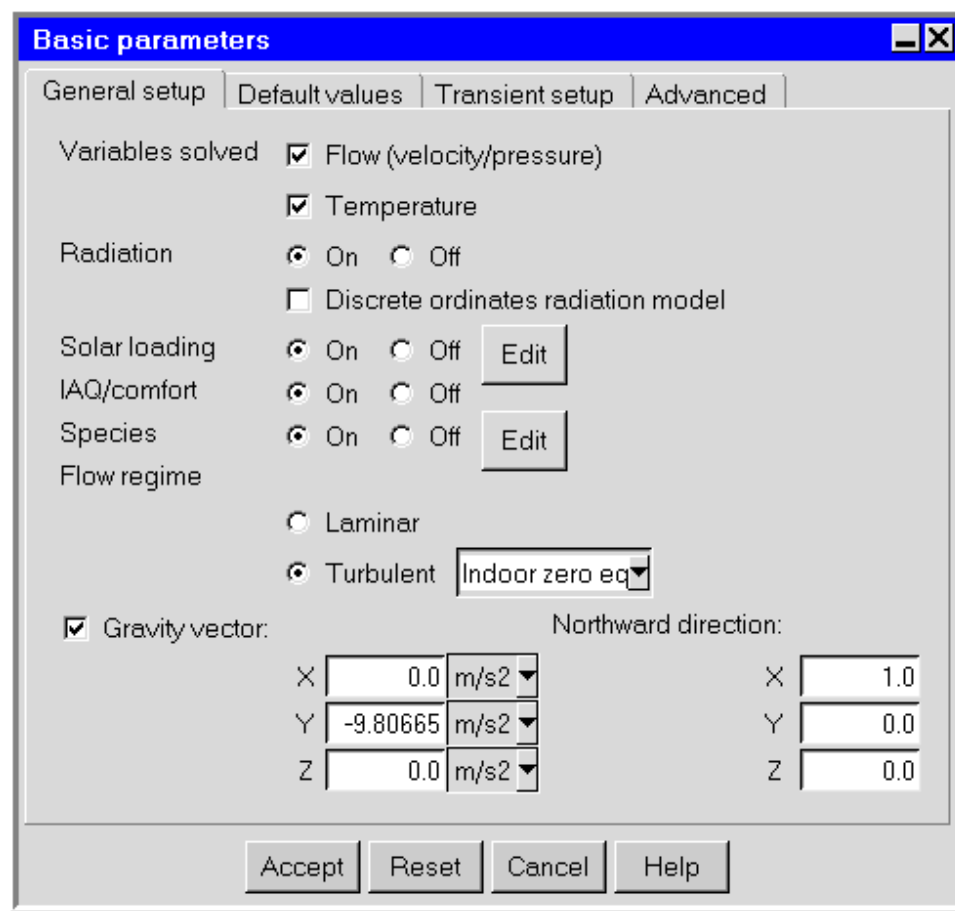


Figure 21.2.1: The Basic parameters Panel (General setup Tab)

2. To define the species to be included in the problem, click Edit next to Species to open the Species definitions panel shown in Figure 21.2.2.



Figure 21.2.2: The Species definitions Panel

- (a) Enter the **Number of species** to be included in the calculation and press the **<Enter>** key on the keyboard. The number of species you enter will then become enabled in the panel.



The maximum number of species you can define is 12.

- (b) Select the species to be included in the calculation from the drop-down lists under **Species** (use the scroll bar if needed). You can also view the properties of a currently selected material, edit the definition of a material, or create a new material using the materials list, as described in Section 7.7.



Note that the most abundant species must appear at the top of the **Species** list.

- (c) Select the concentration units and specify the **Initial concentration** for each species to be included in the calculation.
- Select the concentration units for each species in your calculation from the **fraction** menu to the right of the **Initial concentration** entry field for the species (Figure 21.2.3).



Figure 21.2.3: The fraction Menu

The following options are available:

fraction is the mass fraction of the species

gr/lbm is a humidity ratio (moisture content/dry air content) and is available only for water (h2o)

g/kg is a humidity ratio and is available only for water (h2o)

RH is the relative humidity and is available only for water (h2o)

PPMV is the parts per million by volume (10^6 times the mole fraction) of the species

kg/m3 is the density of the species

- Specify the **Initial concentration** for each species in your calculation.

i Note that you will set initial concentrations explicitly only for the last $N - 1$ species in the panel. The solver will internally convert your inputs to mass fractions for all species and then compute the mass fraction of the most abundant species (the first species in the list) by subtracting the total of the initial mass fractions from 1.

(d) Click **Accept** in the **Species definitions** panel to store the definition of the species to be used in the calculation.

3. Specify the species concentrations for the objects to be included in your **Airpak** model. Note that, for 2D objects, you will set concentrations explicitly only for the last $N - 1$ species for each object. The solver will internally convert your inputs to mass fractions for all species and then compute the mass fraction of the most abundant species (the first species in the list in the **Species definitions** panel shown in Figure 21.2.2) by subtracting the total of the mass fractions for the object from 1.

For 3D objects, you will set volume source parameters explicitly for all N species for each object. For volume sources, each species can have a source, including the most abundant species.

(a) Select the **Species** option in the relevant **Object** panel and click **Edit**. This will open the **Species concentrations** panel.

(b) Enter values for the species concentrations for the object.

- Hollow block, fan, or vent:

- i. Select the check box for each **Species** for which you want to set a concentration in the **Species concentrations** panel (Figure 21.2.4).

i **Airpak** will apply a zero gradient (zero flux) boundary condition for all the species that are not selected in the **Species concentrations** panel.

- ii. Select the concentration units from the **fraction** menu (Figure 21.2.3) and enter a value for the **Concentration** of each **Species** that you have selected.

- Free opening:

- i. Select the check box for each **Species** for which you want to set a concentration in the **Species concentrations** panel (Figure 21.2.5).

i **Airpak** will apply a zero gradient (zero flux) boundary condition for all the species that are not selected in the **Species concentrations** panel.

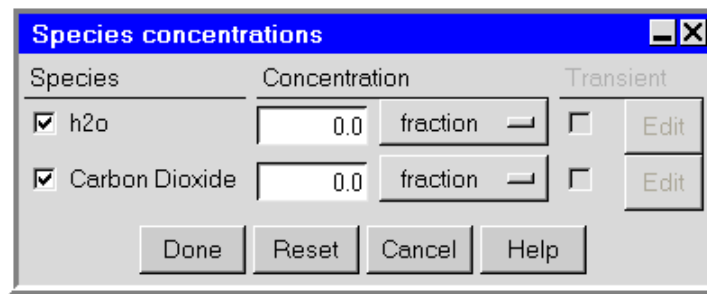


Figure 21.2.4: The Species concentrations Panel for a Hollow Block, Fan, or Vent

- ii. Select the concentration units from the **fraction** menu (Figure 21.2.3) and enter a value for the **Concentration** of each **Species** that you have selected. To specify a uniform concentration for a species, enter a value in the **Concentration** text entry box for the species. To define a spatial profile for the concentration of a species, select **Profile** for the species and click **Edit** to open the **Curve specification** panel (see Section 21.2.1 for details).

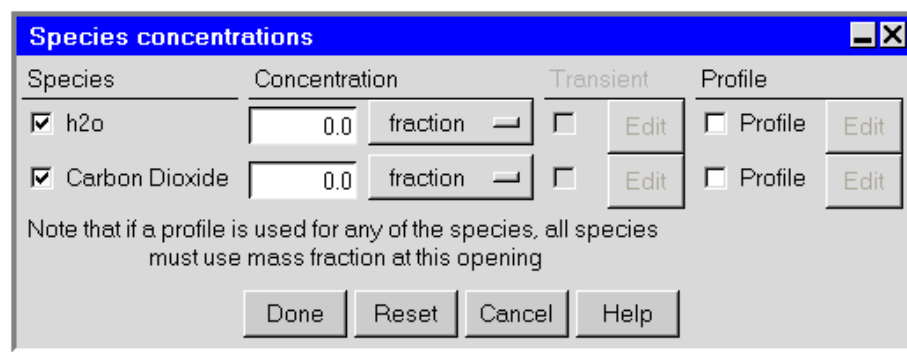


Figure 21.2.5: The Species concentrations Panel for a Free Opening

- 3D resistance:
 - i. Select the check box for each **Species** for which you want to set a volume source in the **Species concentrations** panel (Figure 21.2.6).

i Note that if you deselect a species, **Airpak** will not introduce the species into the room through that resistance.
 - ii. Enter a value for the **Volume source** (the mass of the species entering the room per second) of each **Species** that you want to include in the calculation. Enter a negative value for the species **Volume source** if the species is leaving the room through the resistance.

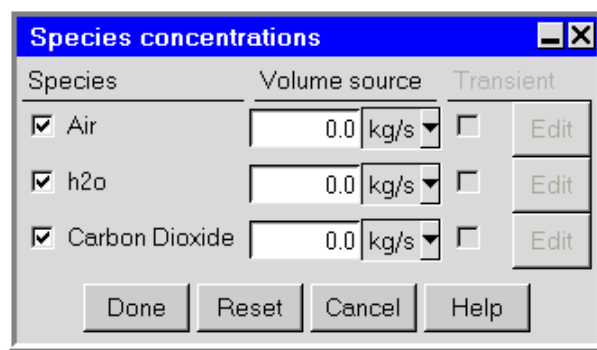


Figure 21.2.6: The Species concentrations Panel for a 3D Resistance

- 2D source:
 - i. Select the check box for each Species for which you want to set a concentration in the Species concentrations panel (Figure 21.2.7).
 - i** Airpak will apply a zero gradient (zero flux) boundary condition for all the species that are not selected in the Species concentrations panel.
 - ii. Select the concentration units from the fraction menu (Figure 21.2.3) and enter a value for the Concentration of each Species that you have selected.

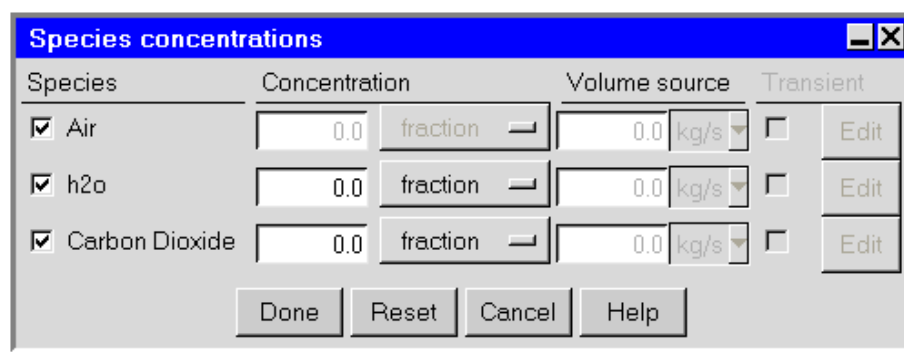


Figure 21.2.7: The Species concentrations Panel for a Source

- 3D source:
 - i. Select the check box for each Species for which you want to set a volume source in the Species concentrations panel (Figure 21.2.7).
 - i** Note that if you deselect a species, Airpak will not introduce the species into the room through that source.

- ii. Enter a value for the **Volume source** (the mass of the species entering the room per second) of each **Species** that you want to include in the calculation. Enter a negative value for the species **Volume source** if the species is leaving the room through the source.
 - (c) To perform a transient calculation for a species, select the check box under **Transient** for the species. To edit the parameters for the transient simulation for the species, click **Edit** under **Transient** for the species. See Chapter 20 for more details on transient simulations.
 - (d) Click **Done** to store the new settings and close the **Species concentrations** panel.
4. If your model contains a recirculation opening, you can specify the increase or decrease of species in the recirculation loop. Select the **Species filter** option in the **Openings** panel and click **Edit**. This will open the **Species filter efficiency** panel shown in Figure 21.2.8.

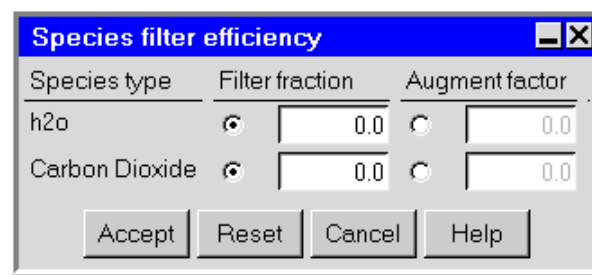


Figure 21.2.8: The Species filter efficiency Panel

For each species in the calculation, there are two options:

Filter fraction is the fraction of the **Species type** that is removed from the model in the recirculation loop.

Augment factor is the factor by which the **Species type** increases in the recirculation loop.

5. Define the rest of your problem in the usual way.
6. If necessary, modify the solution controls for your simulation. These controls are in the **Advanced solver setup** panel. To open the **Advanced solver setup** panel, first click on the **Setup** button in the **Solution** menu.

Solution→**Setup**

This opens the **Solver setup** panel. Next, click on **Advanced defaults** near the bottom of the **Solver setup** panel. This will open the **Advanced solver setup** panel (Figure 21.2.9).

Advanced solver setup

Discretization scheme

Pressure ☐ Standard ☐ Second ☒ Body Force Weighted

Momentum ☒ First ☐ Second

Temperature ☒ First ☐ Second

Turbulent kinetic energy ☒ First ☐ Second

Turbulent dissipation rate ☒ First ☐ Second

h2o ☒ First ☐ Second

Carbon Dioxide ☒ First ☐ Second

Under-relaxation

Pressure	0.3
Momentum	0.7
Temperature	1.0
Viscosity	1.0
Body forces	0.1
Turbulent kinetic energy	0.5
Turbulent dissipation rate	0.5
h2o	1.0
Carbon Dioxide	1.0

☐ Under-relaxation adjustment

Frequency

Linear solver	Type	Termination criterion	Residual reduction tolerance
Pressure	V	0.1	0.1
Momentum	flex	0.1	0.1
Temperature	flex	0.1	0.1
Turbulent kinetic energy	flex	0.1	0.1
Turbulent dissipation rate	flex	0.1	0.1
h2o	flex	0.1	0.1
Carbon Dioxide	flex	0.1	0.1

Precision

Accept Reset Cancel

Figure 21.2.9: The Advanced solver setup Panel for a Species Transport Calculation

- (a) Select a discretization scheme for each of the species in your model. See Section 27.3 for details on the discretization schemes in Airpak.
 - (b) Modify the under-relaxation factors for each of the species in your model, as required. See Section 27.4 for details on setting under-relaxation factors in Airpak.
 - (c) Select a multigrid scheme for each of the species in your model. See Section 27.5 for details on selecting a multigrid scheme in Airpak.
7. Start the calculation using the Solve panel. To open this panel, click on Solve in the Solution menu. Click Accept to start the calculation.

21.2.1 Using the Curve specification Panel to Specify a Spatial Boundary Profile

You can define a spatial boundary profile for the mass fraction of a species at a free opening using the Curve specification panel (Figure 21.2.10). To open the Curve specification panel, select Profile in the Species concentrations panel (Figure 21.2.5) and click Edit.

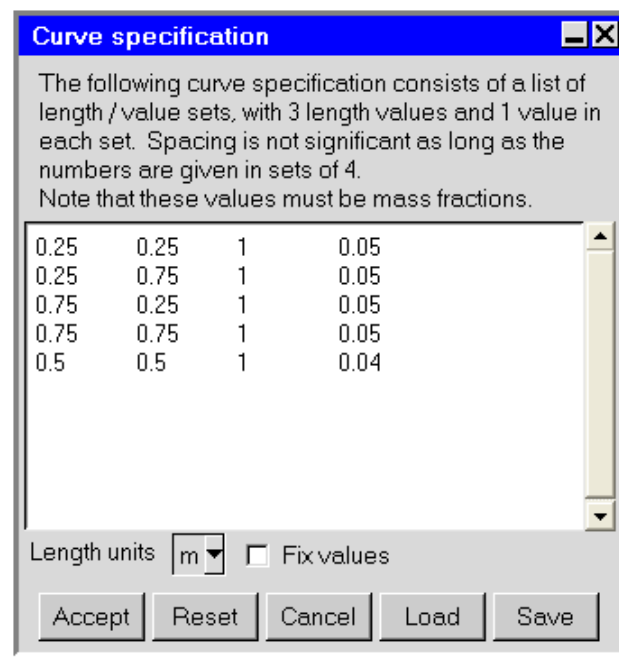


Figure 21.2.10: The Curve specification Panel

To define a profile, specify a list of (x, y, z) coordinates and the corresponding values in the Curve specification panel. For example, the first line in Figure 21.2.10 specifies a mass fraction of 0.05 at (0.25, 0.25, 1). The data in Figure 21.2.10 specify a variation

of mass fractions on the plane $0.25 \leq x \leq 0.75$, $0.25 \leq y \leq 0.75$, $z = 1$. The values in the right-hand column are the mass fractions of the species. Click **Accept** when you have finished defining the profile; this will store the values and close the **Curve specification** panel. **Airpak** will interpolate the data you provide in the **Curve specification** panel to create a profile for the whole boundary.

To load a previously defined profile, click on **Load**. (See Section 28.2.8 for details on saving contour data and using them as a profile.) This will open the **Load curve** file selection dialog box. Select the file containing the profile data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the length units used in the profile data you are loading, you should select the appropriate units in the **Curve specification** panel before you load the profile. If you want to view the data after you have loaded them, using different length units than the default units in the **Curve specification** panel, select the **Fix values** option and then select the appropriate units from the **Length units** list.

To save a profile, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the profile data are to be saved.

21.3 Postprocessing for Species Calculations

Airpak provides postprocessing options for displaying, plotting, and reporting the results of the species calculations. You can generate graphical plots or reports of the following quantities:

- species (mole)
- species (mass)

These variables are contained in the **Variable** and **Value** drop-down lists that appear in the postprocessing and reporting panels. See Chapter 30 for their definitions.

Three models are available in **Airpak** for modeling radiation: the surface-to-surface radiation model, the discrete ordinates (DO) radiation model, and the solar load model. The surface-to-surface model is used by default when you solve for radiation from sources inside the room. Similarly, the DO model is also used to model internal radiation effects. The solar load model is used to model radiation effects from the sun's rays that enter the room through different media.

Heating or cooling of surfaces due to radiation can be included in your model. This chapter describes how to include radiation in your **Airpak** model. See Section 31.4 for details about the theory behind **Airpak**'s radiation models.

Information about radiation modeling is presented in the following sections:

- Section 22.1: Choosing a Radiation Model
- Section 22.2: Using the Surface-to-Surface Radiation Model
- Section 22.3: User Inputs for Radiation Modeling
- Section 22.4: Using the Discrete Ordinates Radiation Model
- Section 22.5: Modeling Solar Radiation Effects

22.1 Choosing a Radiation Model

Typical applications well suited for simulation using radiation heat transfer include the following:

- Surface-to-surface radiant heating or cooling
- Coupled radiation, convection, and/or conduction heat transfer

You should include radiation heat transfer in your simulation when the radiant heat flux, $Q_{\text{rad}} = \sigma(T_{\text{max}}^4 - T_{\text{min}}^4)$, is large compared to the heat transfer rate due to convection or conduction. Typically this will occur at high temperatures where the fourth-order dependence of the radiation heat flux on temperature implies that radiation will dominate. Also, radiation is typically more important for natural convection problems than for forced convection problems of interest in electronics cooling.

If you need to include radiation heat transfer from exterior walls of your model, you can include an external radiation boundary condition for walls.

For problems with symmetry boundaries, the discrete ordinates model should be used to overcome this limitation of the form factor calculation methodology. You may also want to use the discrete ordinates model when your **Airpak** model contains an extremely large number of surfaces and it becomes computationally expensive to perform the form factor calculations. In general, the discrete ordinates model will require more computing time than a calculation using the surface-to-surface model.

The solar load model is a way to account for incident solar radiation by performing a ray tracing shading test for all boundary faces. The solar load model, with its face-by-face shading analysis, is more comprehensive than the solar flux calculator macro, which only calculates the incident solar radiation on a particular surface. See Section 25.1.1 for details about the solar flux macro.

22.2 Using the Surface-to-Surface Radiation Model

The surface-to-surface radiation model in **Airpak** provides an economical way to account for radiation effects in most applications. The surface-to-surface model uses form factors (also known as view factors) that are calculated for surfaces of **Airpak** objects. The method of computing form factors in **Airpak** does not account specifically for symmetry boundaries when **Airpak** computes form factors to or from surfaces that are adjacent to symmetry boundaries.

22.2.1 Radiation Modeling for Objects

Airpak allows you to specify that any or all objects exchange radiative energy with other objects in the model or with a specified remote temperature if you are using the surface-to-surface radiation model. For radiation, the heat transfer rate is defined as

$$q = \sigma e F (T_{\text{surface}}^4 - T_{\text{remote}}^4) \quad (22.2-1)$$

where T_{surface} is the temperature of the object surface, T_{remote} is the temperature of the surface to which the object radiates heat, σ is the Stefan-Boltzmann constant, F is a view factor specifying the fraction of radiant energy that is intercepted by the surface of the object, and e is the emissivity of the surface of the object (defined as part of the properties of the surface material specified for the object).

If you specify an object as radiating energy to and from a specified temperature, you must specify a remote temperature and an appropriate view factor.



If you specify an object as radiating energy to other objects in the model, **Airpak** automatically calculates the view factors and computes the radiative heat flux based on calculated object temperatures. If the sum of the view factors for a radiating object is less than 1,

Airpak will define the remainder to radiate to the radiation temperature (Radiation temp) specified under Ambient values in the Basic parameters panel (see Section [6.4.6](#)).

22.3 User Inputs for Radiation Modeling

To solve a problem involving the surface-to-surface radiation model in Airpak (the default radiation model), you will follow the procedure outlined below:

1. Enable the calculation of radiation in the **General setup** tab of the **Basic parameters** panel. To open the **Basic parameters** panel (Figure 22.3.1), double-click the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

 Problem setup →  Basic parameters

Enable the calculation of radiation by selecting **On** next to **Radiation**. Note that **On** is selected by default, but if you do not specify any surfaces to be included in the radiation calculation, it will be ignored and radiation will not be computed. If you have defined some radiating surfaces in your model, you can turn off the calculation of radiation by selecting **Off** next to **Radiation** in the **Basic parameters** panel.

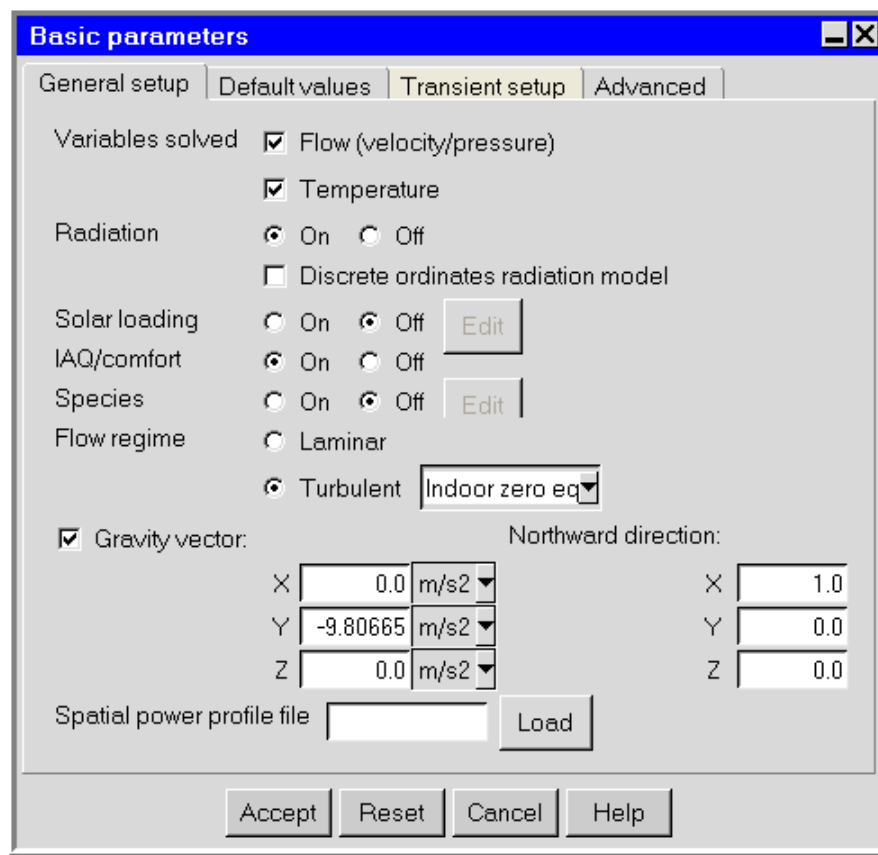


Figure 22.3.1: The Basic parameters Panel (General setup Tab)

2. Specify the objects to be included in the radiation calculation. You can do this in two ways:

- Select the radiation option in each object panel (i.e., the Walls panel, the Blocks panel, etc.) and specify the radiation parameters for each object or surface (described below). The view factors can then be calculated in one of two ways:
 - Open the **Form factors** panel and click **Compute**. You can also display, save, and edit the view factors, as described in Section 22.3.2.
 - If you do not compute the view factors before you start the calculation, **Airpak** will compute them for you at the start of the calculation.

If you have already calculated the view factors for your model and you do not want **Airpak** to recalculate them, select **Don't recompute** in the **Form factors** panel (Figure 22.3.4).

- Specify the objects in the **Form factors** panel (described in Section 22.3.2).
3. Define the rest of your problem in the usual way.
 4. Start the calculation using the **Solve** panel. To open this panel, click on **Start solution** in the **Solve** menu.

If you have enabled radiation on surfaces in your model, you can turn off the calculation of radiation by selecting the **Disable radiation calculations** option in the **Solve** panel.

22.3.1 User Inputs for Specification of Radiation in Individual Object Panels

The calculation of radiation for blocks, 2D sources, person objects, partitions, and walls objects can be specified in individual **Object** panels in **Airpak**. You can choose the object surfaces to radiate either to a fixed temperature or to other surfaces in the model.

The procedure for specifying radiation in your model using individual object panels is described below.

1. Turn on radiation for the object. You will do this in different ways for different kinds of objects.
 - **Block:** You can specify radiation for the whole block or for individual sides of the block.
 - For the whole block, select the **Radiation** option in the **Blocks** panel. Click **Edit** to access the **Radiation specification** panel (Figure 22.3.2).
 - For individual sides of the block, select the **Individual sides** option in the **Blocks** panel. Click **Edit** to open the **Individual side specification** panel. Select **Radiation properties** to access the radiation properties, which are the same as those shown in Figure 22.3.2.

- 2D source: Select the Radiation option in the Sources panel. Click Edit to access the Radiation specification panel (Figure 22.3.2).
- Person: Select the Radiation option in the Person panel. Click Edit to access the Radiation specification panel (Figure 22.3.2).
- Wall: Select the Inner surface radiation option in the Walls panel and click Edit to access the Radiation specification panel (Figure 22.3.2).
- Partition: Select the Low side or High side option under Side specification in the Partitions panel and click Edit to open the Low (or High) side surface properties panel. Select Radiation to access the radiation properties, which are the same as those shown in Figure 22.3.2.

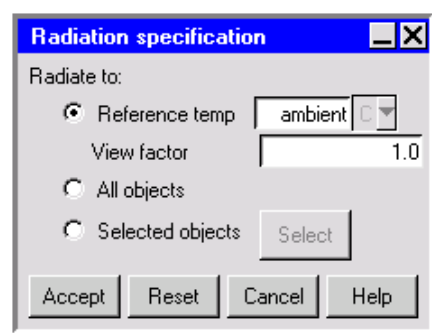


Figure 22.3.2: The Radiation specification Panel

2. Specify the nature of the surface(s) with which the object or surface exchanges radiation by selecting one of the options below in the Radiation specification panel.

Reference temp specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Reference temp** and a **View factor**. The default view factor is 1.0 and the value of the **ambient** temperature is defined under **Ambient values** in the **Basic parameters** panel (see Section 6.4.6).

All objects specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. Airpak computes the radiative heat flux to and from the object or surface based on the calculated temperatures and view factors (with respect to the object or surface) for each object or surface.

Selected objects specifies that the object or surface exchanges radiation only with selected objects and surfaces in the model. Click **Select** to open the **Radiation object selection** panel (Figure 22.3.3). This panel contains a list of available objects and surfaces.

The **Radiation object selection** panel lists all objects and surfaces currently defined as subject to radiative heat transfer. To select one or more objects

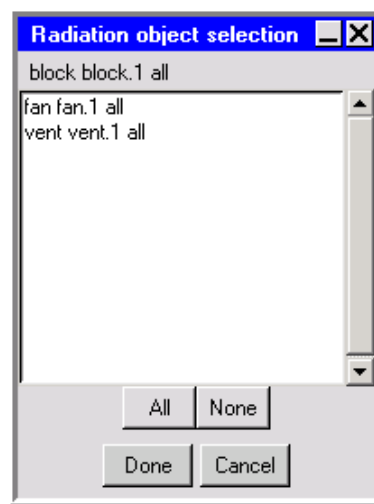


Figure 22.3.3: The Radiation object selection Panel

or surfaces from the list, left-click the name(s) of the object(s) and surface(s) and click **Done**.


Click **Accept** in the **Radiation specification** panel when you have finished the specification.

3. Specify the material to be used for the currently selected side of the object or surface. This material defines the roughness and emissivity of the object or surface, and is specified in different ways for different kinds of objects.
 - **Block:** You can specify material properties for the whole block or for individual sides of the block.
 - For the whole block, specify the material under **Surface material** in the **Blocks** panel.
 - For individual sides of the block, select the **Individual sides** option in the **Blocks** panel. Click **Edit** to open the **Individual side specification** panel and specify the **Material**.
 - **2D source:** Specify the material next to **Material** in the **Sources** panel.
 - **Person:** Specify the material under **Surf material** in the **Person** panel.
 - **Wall:** Specify the material next to **Internal material** for a non-zero-thickness wall in the **Walls** panel. If the wall has zero thickness, specify the **External material** (which is used for both the inside and the outside of a zero-thickness wall).
 - **Partition:** Select the **Low side** or **High side** option under **Side specification** in the **Partitions** panel. Click **Edit** to open the **Low (or High) side surface properties** panel and specify the **Material**.

By default, the material is specified as **default** for blocks, 2D sources, partitions, and walls. This means that the material specified on the object or surface is defined under **Default surface** in the **Problem setup** panel (see Section 6.4.7). To change the material for an object or surface, select a material from the material drop-down list. See Section 7.7 for details on material properties.

4. Calculate the view factors using the **Form factors** panel, as described below.

22.3.2 User Inputs for Specification of Radiation Using the Form factors Panel

The calculation of radiation objects can be specified using the **Form factors** panel. To open the **Form factors** panel (Figure 22.3.4), select **Radiation** in the **Model** menu or click the  button in the **Model and solve** toolbar.

Model → Radiation

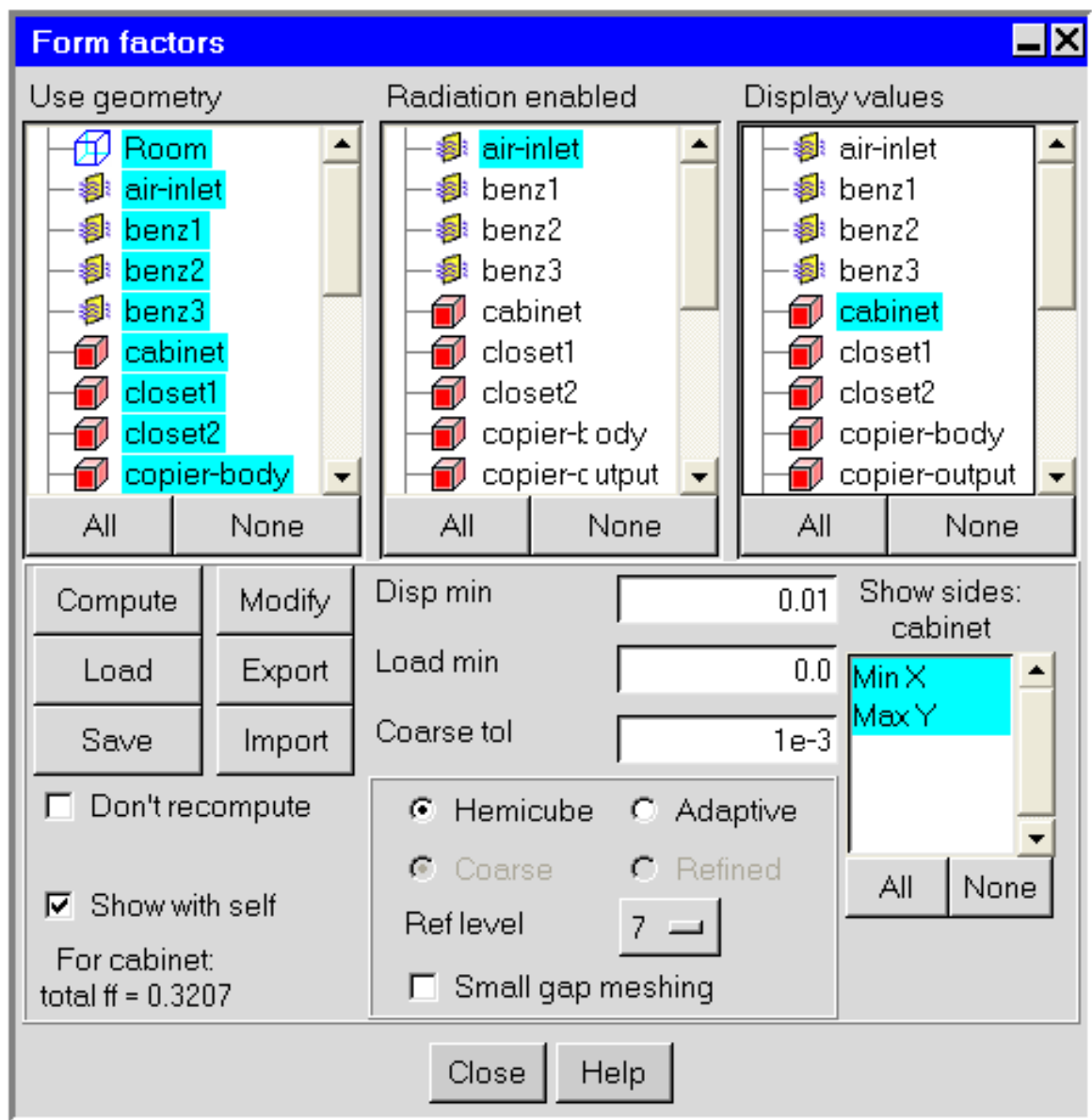


Figure 22.3.4: The Form factors Panel

Specifying the Objects to Include in the Radiation Calculation

All objects can receive radiant energy regardless of their ability to radiate energy themselves. To specify that an object will receive radiant energy from other objects in the model, click on the name of the object in the **Use geometry** list using the left mouse button. Any object that has been specified to take part in the view factor calculation (i.e., receive radiant energy) will be highlighted in blue in the **Use geometry** list. To include all objects in the **Use geometry** list in the radiation calculation, click **All** at the bottom of the list. To deselect an object in the **Use geometry** list, click on the name of the object in the **Use geometry** list using the left mouse button. To deselect all objects in the **Use geometry** list, click **None** at the bottom of the list.



If any sides of the room have been specified as wall, opening, or vent objects, you will be able to include radiation to these sides. For those sides of the room that have the **Default** specification (i.e., an adiabatic wall), radiant energy from other objects will be lost to these sides using a partial enclosure model. View factor calculations will be disabled for these walls, which will instead be treated like a black body with the default **Radiation temperature** specified in the **Basic parameters** panel.

All the objects that can radiate energy are listed under **Radiation enabled**. To specify that an object will radiate, click on the name of the object in the **Radiation enabled** list using the left mouse button. Any object that has been specified to radiate to at least one other object or surface will be highlighted in blue in the **Radiation enabled** list. To include all objects in the **Radiation enabled** list in the radiation calculation, click **All** at the bottom of the list. To deselect an object in the **Radiation enabled** list, click on the name of the object in the **Radiation enabled** list using the left mouse button. To deselect all objects in the **Radiation enabled** list, click **None** at the bottom of the list.

Computing View Factors

When you have selected the objects you want to participate in the radiation model, click **Compute** to compute the view factors. The computation of the view factors may take a few seconds to a few minutes, depending on the size of the model.

To calculate the view factors, **Airpak** first creates a coarse mesh. It then discards the volume mesh and coarsens the surface mesh. The **Coarse tol** option allows you to specify the angular tolerance to be used during coarsening; i.e., it specifies the minimum angle between adjacent facets of the surface mesh for which coarsening will not happen. A small value allows higher accuracy, but **Airpak** will take more time to compute the mesh. A value of 1 (degree) is usually sufficient. To disable coarsening completely, enter a value of -1 .

There are two methods available in **Airpak** for calculating the view factors: the hemicube method and the adaptive method. The hemicube method uses a differential area-to-area method and calculates the view factors on a row-by-row basis. The view factors calculated from the differential areas are summed to provide the view factor for the whole surface. This method originated from the use of the radiosity approach in the field of computer graphics [8].

To use the hemicube method to compute the view factors, select **Hemicube** in the **Form factors** panel. You can choose the refinement level to be used to calculate the view factors. To change the refinement level, click on the number to the right of **Ref level** in the **Form factors** panel (Figure 22.3.4) and select a new number from the drop-down list. There are seven refinement levels available in **Airpak**. A higher level allows higher accuracy, but **Airpak** will take more time to compute the mesh. It is recommended that you use the hemicube method for large complex models, because it is faster than the adaptive method for these types of models.

The adaptive method calculates the view factors on a pair-by-pair basis using a variety of algorithms (analytic or Gauss quadrature) that are chosen adaptively depending on the proximity of the surfaces. To maintain accuracy, the order of the quadrature increases the closer the faces are together. For surfaces that are very close to each other, the analytic method is used. **Airpak** determines the method to use by performing a visibility calculation. The Gaussian quadrature method is used if none of the rays from a surface are blocked by the other surface. If some of the rays are blocked by the other surface, then either a Monte Carlo integration method or a quasi-Monte Carlo integration method is used.

Two options are available in the **Form factors** panel for the **Adaptive** method: **Coarse** and **Refined**. If you select the **Coarse** option, 16 visibility samples will be taken from each surface. If you select the **Refined** option, 64 visibility samples will be taken from each surface. The calculation of the view factors will be significantly faster if you select the **Coarse** option, but the **Refined** option provides higher accuracy. It is recommended that you use the adaptive method for simple models, because it is faster than the hemicube method for these types of models.

If you make a change to your model and you do not want to recompute the view factors when you calculate a solution, select **Don't recompute** in the **Form factors** panel.

Displaying View Factors

To view the computed view factors for an object, click on the name of the object in the **Display values** list. The name of the selected object will be displayed under **Show sides**, and the sides of the object that will participate in the radiation calculation will be listed in the **Show sides** list. **Airpak** displays arrows in the graphics window indicating which objects the selected object will radiate to, the names of the sides that are radiated to, and the values of the view factors. An example is shown in Figure 22.3.5, where a partition

will radiate to two blocks.

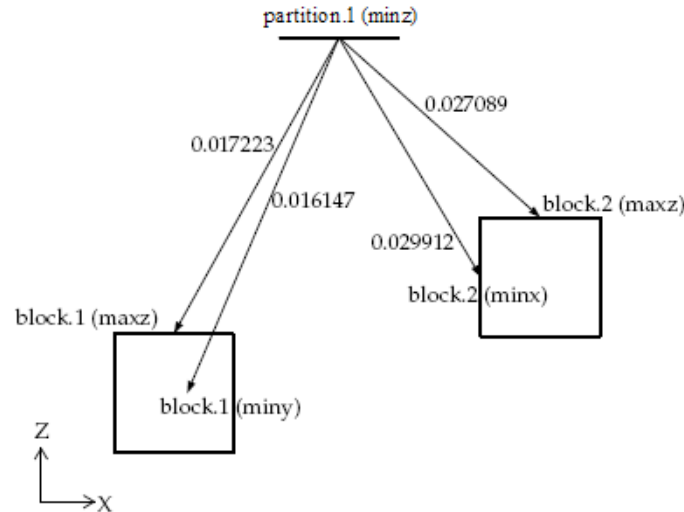


Figure 22.3.5: A Partition Radiates to Two Blocks

To display the view factors for all objects in the **Display values** list, click **All** at the bottom of the **Display values** list. To remove all the view factors from the graphics window, click **None** at the bottom of the **Display values** list. It is recommended that you select just one or two objects at a time in the **Display values** list; otherwise, the display of view factors in the graphics window will become cluttered. You can also display the view factors for just one side of an object by first selecting the object in the **Display values** list and then selecting the side of the object in the **Show sides** list. To display the view factors for all the sides in the **Show sides** list, click **All** at the bottom of the **Show sides** list. To remove these view factors from the graphics window, click **None** at the bottom of the **Show sides** list. If you select a heat sink object in the **Display values** list, **Airpak** will display the view factors for the radiation between the heat sink object and the other objects in the model, and also the view factors for the radiation between the parts of the heat sink object. To remove the view factors for the radiation between the parts of the heat sink object from the graphics window, deselect **Show with self** in the **Form factors** panel (Figure 22.3.4). To redisplay these view factors, select **Show with self**.

Another way to control the number of view factors displayed in the graphics window is to specify a value for **Disp min** or **Load min**. View factors that are smaller than the specified **Load min** value will not be loaded into the **Airpak** model, and so cannot be viewed or used in the calculation. View factors that are smaller than the specified **Disp min** value will not be displayed in the graphics window, but they will be loaded into the model and used in the computation.

Saving the Computed View Factors

To save the computed view factors, click **Save**. Airpak will save the view factors to a file. To reload the view factors, click the **Load** button. If you click **Compute**, Airpak will automatically load the view factors after they have been computed.

To export the view factors to an ASCII file, click the **Export** button. You can reuse the ASCII file in other Airpak jobs, and you can edit the file and add view factors to it using a text editor. To import an ASCII file, click the **Import** button and select the file to be imported.

Modifying the Computed View Factors

If you want to modify the view factors, click **Modify**. This will open the Modify form factors panel (Figure 22.3.6).

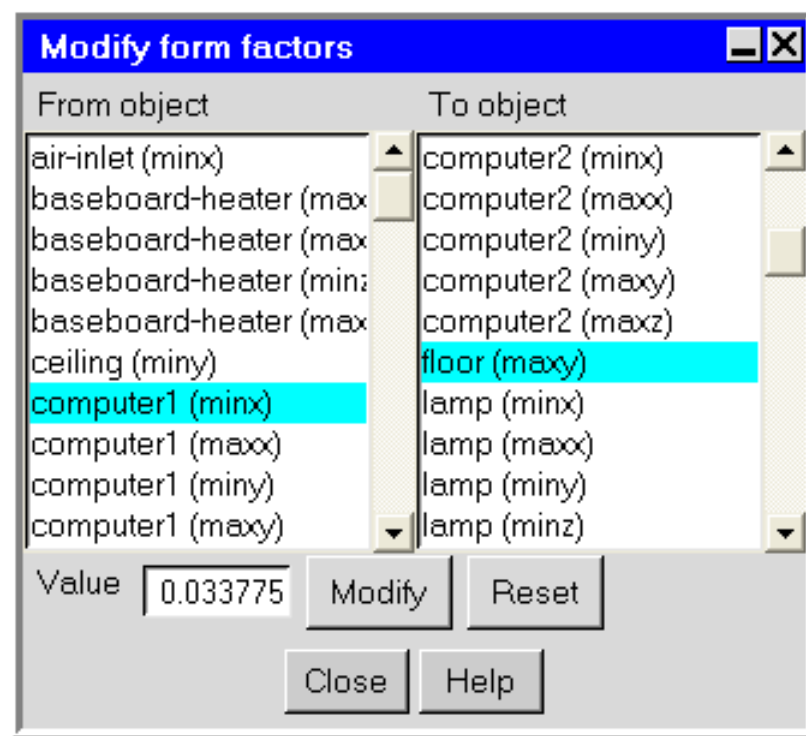


Figure 22.3.6: The Modify form factors Panel



You can modify any of the computed view factors by selecting the name of the surface which is radiating in the **From object** list and the name of the surface which is being radiated to in the **To object** list. Define the new view factor by entering a value in the **Value** text entry box and then click **Modify**. Modified view factors are shown in pink in

the graphics window; the computed view factors are shown in blue. Click **Reset** to reset the modified value back to the computed value.

22.4 Using the Discrete Ordinates Radiation Model

To solve a problem involving the discrete ordinates radiation model in **Airpak**, you will follow the procedure outlined below:

1. Enable the calculation of radiation in the **General setup** tab of the **Basic parameters** panel. To open the **Basic parameters** panel (Figure 22.3.1), double-click the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Basic parameters**

Enable the calculation of radiation by first turning **On Radiation** and then selecting **Discrete ordinates radiation model**.

2. Specify the surface material to be used for each object in your model. This material defines the roughness and emissivity of the surface of the object.

By default, the material is specified as **default** for blocks, 2D sources, partitions, and walls. This means that the material specified on the surface of the object is defined under **Default surface** in the **Default values** tab of the **Basic parameters** panel (see Section 6.4.7). To change the surface material for an object, select a material from the material drop-down list. See Section 7.7 for details on material properties.

3. Define the rest of your problem in the usual way.
4. Start the calculation using the **Solve** panel. To open this panel, click on **Start solution** in the **Solve** menu. Click **Accept** to start the calculation.

If you have enabled radiation in your model, you can turn off the calculation of radiation by selecting the **Disable radiation calculations** option in the **Solve** panel.

Airpak will solve the transport equations for the discrete ordinates model once every five iterations in order to reduce the impact on the available computational resources. **Airpak** will display the convergence history for the discrete ordinates equations (**do-intensity**) in the **Monitor** graphics display and control window (see Section 27.12.2).



22.5 Modeling Solar Radiation Effects

Airpak's solar load model allows you to include the effects of direct solar illumination as well as diffuse solar radiation. Given the model geometry and pertinent solar information such as terrestrial location, date, and time, the model performs a ray tracing shading test for all boundary surfaces. This approach includes a two-band (visible and infrared) spectral model for solar illumination. The transparent material model computes the absorptivity and transmissivity as functions of incident angle.

22.5.1 User Inputs for the Solar Load Model

To use the solar load model, follow the procedure outlined below.

1. Enable the calculation of radiation in the **General setup** tab of the **Basic parameters** panel. To open the **Basic parameters** panel (Figure 22.3.1), double-click the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

 **Problem setup** →  **Basic parameters**

In the **Basic parameters** panel, turn on the **Solar loading** option and click **Edit** to open the **Solar Load Model parameters** panel (Figure 22.5.1).

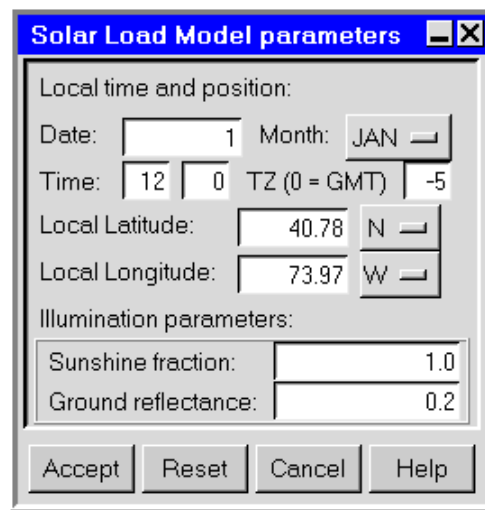


Figure 22.5.1: The Solar Load Model parameters Panel

2. In the **Solar Load Model parameters** panel, specify the following parameters under **Local time and position**:
 - (a) Specify a value for the **Date** and select the month from the **Month** menu.
 - (b) Specify the local time at the desired location in the two fields to the right of **Time**. The time is based on a 24-hour clock, thus acceptable values range from 0 h 0 min (12:00 a.m.) to 23 h 59.99 min (11:59.99 p.m.). Values entered in the first text-entry field (hour) must be integral, but values entered in the second text-entry field (minute) can be integral or fractional. For example, if the local time was 12:01:30 a.m., you would enter 0 for the hour and 1.5 for the minute. If the local time was 4:17 p.m., you would enter 16 for the hour and 17 for the minute.
 - (c) (optional) Specify the local time zone of the desired location using the offset value to the right of the **Time** entry fields. The options are: Greenwich Mean

Time (GMT), Atlantic Standard Time (AST), Eastern Standard Time (EST), Central Standard Time (CST), Mountain Standard Time (MST), and Pacific Standard Time (PST). The time zone is used to approximate the longitude (see below) of the location if you are unsure of the coordinates.

- (d) Specify the **Local Latitude** of the desired location. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator. Select the hemisphere (N or S) from the menu to the right of the **Local Longitude** entry field.
- (e) Specify the **Local Longitude** of the desired location. The longitude is approximated if you specify the local time zone, but you can enter a more precise value if you know it. Any value you enter here will take precedence over the time zone. Values may range from 0° to 180° . Select the hemisphere (W or E) from the menu to the right of the **Local Longitude** entry field.

3. Under **Illumination parameters**, specify the following parameters:

- (a) Specify the **Sunshine fraction**, which is a factor between 0 and 1 used to account for the effects of clouds which may reduce the direct solar irradiation. Clear sky is modeled by setting the value equal to 1 and complete cloud cover is modeled by setting the value equal to 0. Partial cloud cover is modeled by setting the value to be between 0 and 1. The default value is 1.0.
- (b) Specify the **Ground reflectance**, which is a parameter that is used in determining the contributions of reflected solar radiation from ground surfaces. Reflected solar radiation from ground surfaces is a function of the direct normal irradiation, the time of the year, the tilt angle of the surface, and the ground reflectance. It is treated as part of the total diffuse solar irradiation. Ground reflectance values can vary depending on the ground surface (i.e., concrete, grass, rock, gravel, asphalt). The default value is 0.2.

4. Specify the solar classification of all room objects in each **Object** panel. For each object, select one of the following options next to **Solar behavior**:

Insulator indicates that the surface will not be considered in the solar radiation modeling.

Opaque indicates that the surface will not allow solar radiation to pass through it.

Transparent indicates that the surface will allow a portion of the solar radiation to pass through it.

5. Specify the surface material to be used for each object or surface. This surface material defines the roughness, emissivity, and the absorption and transmission parameters of the surface. If the material is opaque, you will specify the absorptivities in the visible and infrared portions of the spectrum. If the material is transparent, you will specify the visible transmittance and shading coefficient.

By default, the material is specified as **default** for blocks, 2D sources, partitions, and walls. This means that the material specified on the object or surface is defined under **Default surface** in the **Default values** tab of the **Basic parameters** panel (see Section 6.4.7). To change the material for an object or surface, select a material from the material drop-down list. See Section 7.7 for details on material properties.

6. Define the rest of the problem in the usual way.

Airpak uses the Dynamic-Q optimization method to solve constrained-design-optimization problems. See Section 31.5 for details about the theory behind Airpak's optimization method.

Information about solving design-optimization problems using Airpak is presented in the following sections:

- Section 23.1: When to Use Optimization
- Section 23.2: User Inputs for Optimization

23.1 When to Use Optimization

Design optimization can be used for a variety of CFD problems. Some examples include the following:

- Placement of airflow supplies
- Placement of airflow returns
- Optimizing supply temperatures
- Optimizing system airflow rates
- Optimizing floor tile percentage open in data centers
- Optimizing exhaust airflow rates

Airpak's optimization method is particularly suited for problems with a large number of design variables and constraint functions. The maximum number of design variables allowed is 50 and the maximum number of constraint functions is 10.


23.2 User Inputs for Optimization

To solve an optimization problem in **Airpak**, you will follow the procedure outlined below.

1. Define all the required design variables in the appropriate input fields in the same way you would define parameters. For more information on how to define parameters see Section 24.2.



Note that boolean parameters and fan failure cannot be defined as design variables for optimization problems.

2. To enable the optimization, select **Optimization** in the **Setup** tab of the **Parameters and optimization** panel. To open the **Parameters and optimization** panel (Figure 23.2.1), select **Run optimization** in the **Solve** menu or click the  button in the **Model and solve** toolbar.

Solve → Run optimization

3. Define the range of design variables.
 - (a) Select the design variable from the list in the **Design variables** tab of the **Parameters and optimization** panel (Figure 23.2.2). Expanding a design variable's tree node displays objects and properties associated with the variable.
 - (b) Specify the starting value of the variable in the **Base value** text box.
 - (c) Specify the minimum and maximum values the variable can take in the **Min value constraint** and **Max value constraint** text boxes.
 - (d) If the design variable can only take values in multiples of a number, select **Allow only multiples** and specify the lowest multiple in the provided text box. For example, the fin count for a heatsink can be specified only in multiples of 1.

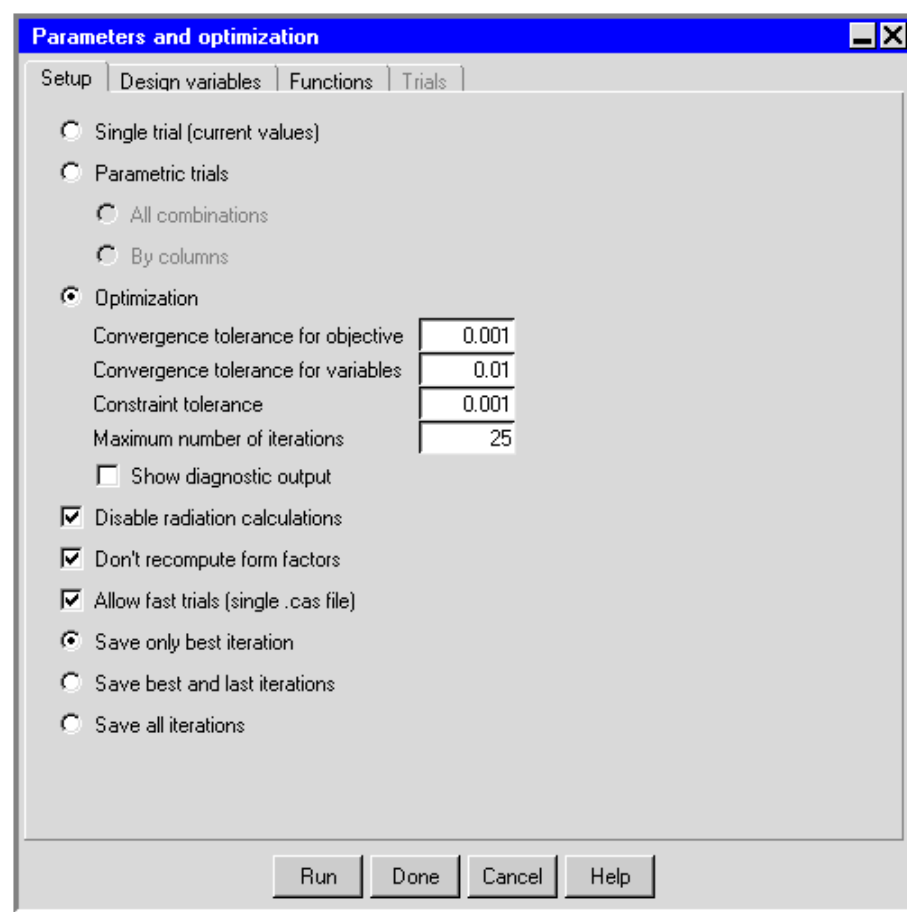


Figure 23.2.1: The Parameters and optimization Panel (Setup Tab)

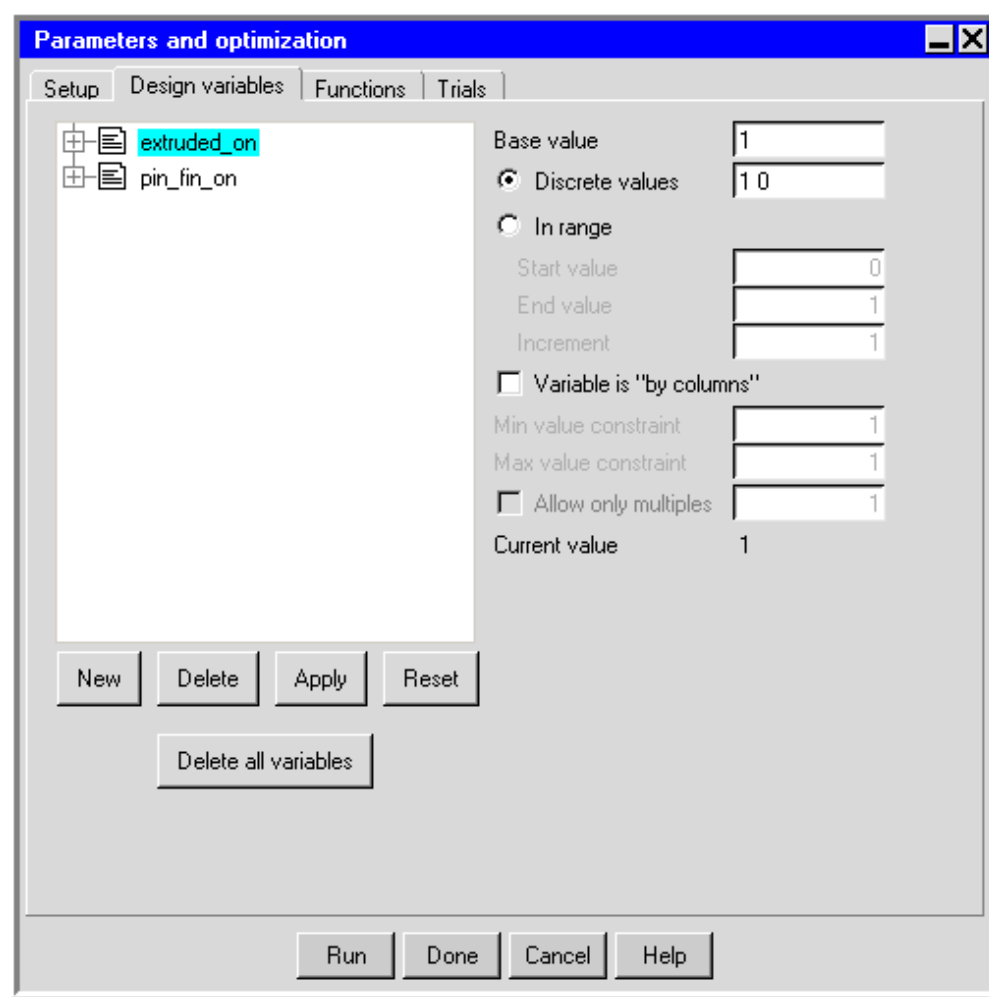


Figure 23.2.2: The Parameters and optimization Panel (Design variables Tab)

- Define functions in the Functions tab of the Parameters and optimization panel (Figure 23.2.3). These functions will be set as objective or constraint functions.

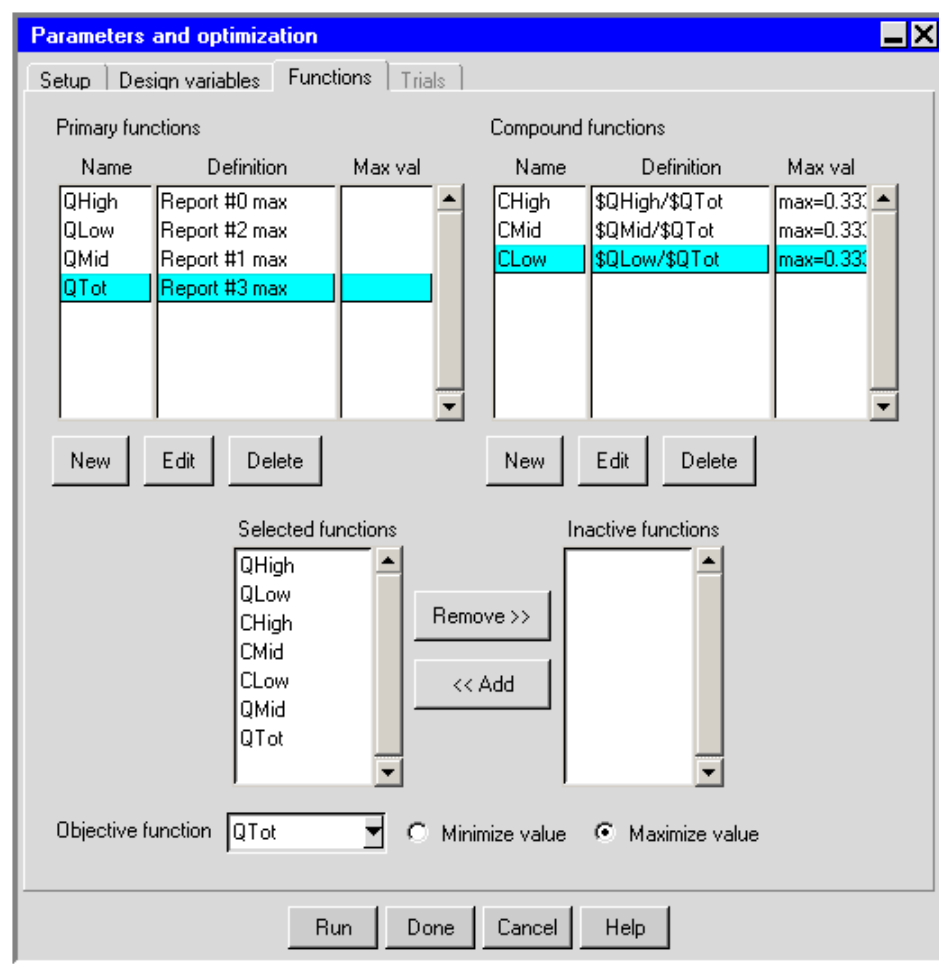


Figure 23.2.3: The Parameters and optimization Panel (Functions Tab)

Functions are classified as **Primary functions** and **Compound functions**. **Primary functions** can be defined using design variables and accessible solution data. **Compound functions** are functions of **Primary functions**. Compound functions cannot be created without creating **Primary functions**.

- (a) To create a primary function, click the **New** button under **Primary functions**. This will open the Define primary function panel (Figure 23.2.4).

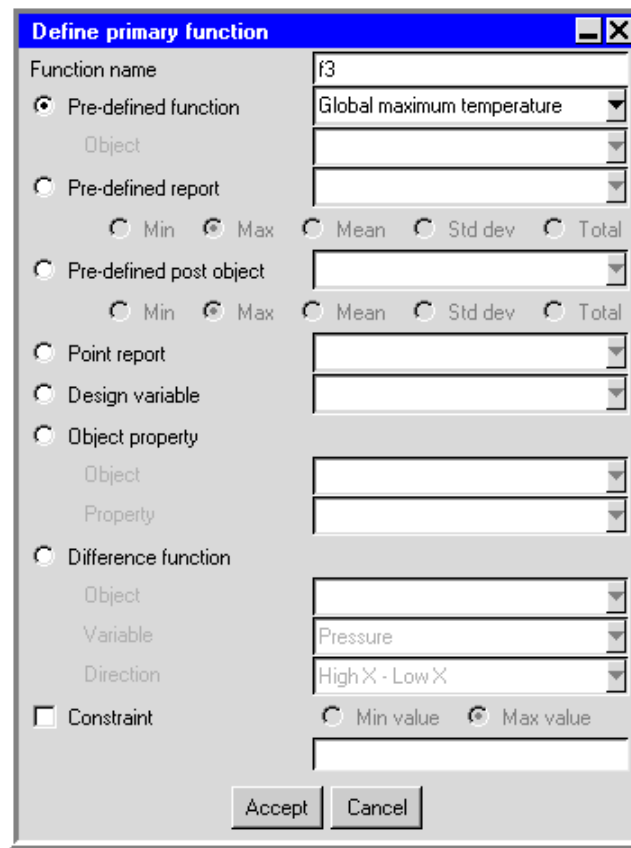


Figure 23.2.4: The Define primary function Panel

- i. Primary functions can be defined in different ways by selecting one of the following options in the Define primary function panel:

Pre-defined function allows you to define a primary function by selecting one of the following Airpak pre-defined functions from the drop-down list: Global maximum temperature, Maximum temperature of objects, and Mean temperature of objects (see Figure 23.2.5).

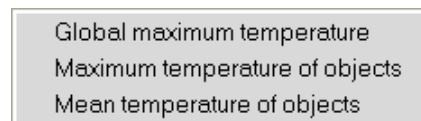


Figure 23.2.5: The Pre-defined function drop-down list

Pre-defined report allows you to define a primary function by selecting one of the user pre-defined summary reports from the drop-down list, and by selecting one of the following functions corresponding to the report: **Min**, **Max**, **Mean**, **St dev**, and **Total**. See Section 27.10 for more information on defining summary reports.

Pre-defined post objects allows you to define a primary function by selecting one of the user pre-defined post objects from the drop-down list (e.g., object faces and plane cuts), and by selecting one of the following functions corresponding to the object: **Min**, **Max**, **Mean**, **St dev**, and **Total**. See Section 27.9 for more information on defining post objects.

Point report allows you to define a primary function by selecting one of the user defined point reports from the drop-down list. See Section 28.2.7 for more information on defining point reports.

Design variable allows you to define a primary function by selecting one of the design variables from the drop-down list. See Section 24.5 for more information on defining design variables.

Object property allows you to define a primary function by selecting an object from the **Object** drop-down list, and by selecting an appropriate object property from the **Property** drop-down list. Only certain object properties (e.g., geometry, power, and radiation parameters) can be used to define primary functions.

Difference function allows you to define a primary function using differences of mean values of post-processing variables across two opposing faces of an object. To use the difference function, select an **Object**, **Variable**, and **Direction** from the drop-down lists. The function value is calculated by subtracting the mean values of a post-processing variable computed across two sides of an object (e.g., **High X** and **Low X**). For example, this function can be used to determine the pressure drop across a heatsink by selecting the difference in the pressures at the leading and trailing sides of the heatsink.

- ii. If this is a constraint function, check the **Constraint** box, click **Min value** or **Max value**, and enter the constraint value.
- (b) To create a compound function, click the **New** button under **Compound functions**. This will open the **Define compound function** panel (Figure 23.2.6). In the **Define compound function** panel. Specify the **Function name** and the **Definition** of the compound function. To define the compound function, the appropriate primary functions should be used. In the **Definition** box, the primary function name must be preceded by a \$ sign.
- (c) Specify an objective function.

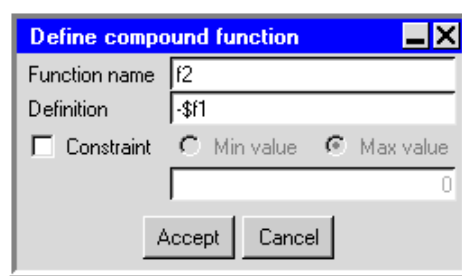


Figure 23.2.6: The Define compound function Panel

- i. Select one of the previously defined functions from the **Objective function** drop-down list, in the **Parameters and optimization** panel (Figure 23.2.3).
- ii. To maximize the objective function, select **Maximize value**, otherwise, the objective function will be minimized by default.

i Note that if a function is neither a constraint function nor an objective function, the function value is merely reported with the optimization results.

5. Choose which functions to make inactive or active.
 - (a) To make a function inactive, choose the function under **Selected functions** in the **Parameters and optimization** panel (Figure 23.2.3) and place the function in the **Inactive functions** list by clicking the **Remove** button. Note that placing the function in the **Inactive functions** list does not delete the function. Functions placed in the **Inactive functions** list are not reported with the optimization results.
 - (b) To activate a function, choose the function under **Inactive functions** and click the **Add** button.

i To delete a function, select the function under **Primary functions** or **Compound functions** and click the **Delete** button.

6. Start the optimization process by clicking the **Run** button in the **Parameters and optimization** panel.

When **Airpak** starts performing calculations, the **Optimization run** window will open and **Airpak** will display all the function values, design variables, and the running times for each optimization iteration. In addition, the function values and design variables are plotted versus iteration number. The plots are displayed on the left hand side of the **Optimization run** window (Figure 23.2.7).

The graphics window is fully interactive. For example, at any time during the calculation, you can zoom in on a specific portion of the graph. This graphics window is similar to the **Monitor** graphics display (see Section 27.12.2).

Other options available in this window include

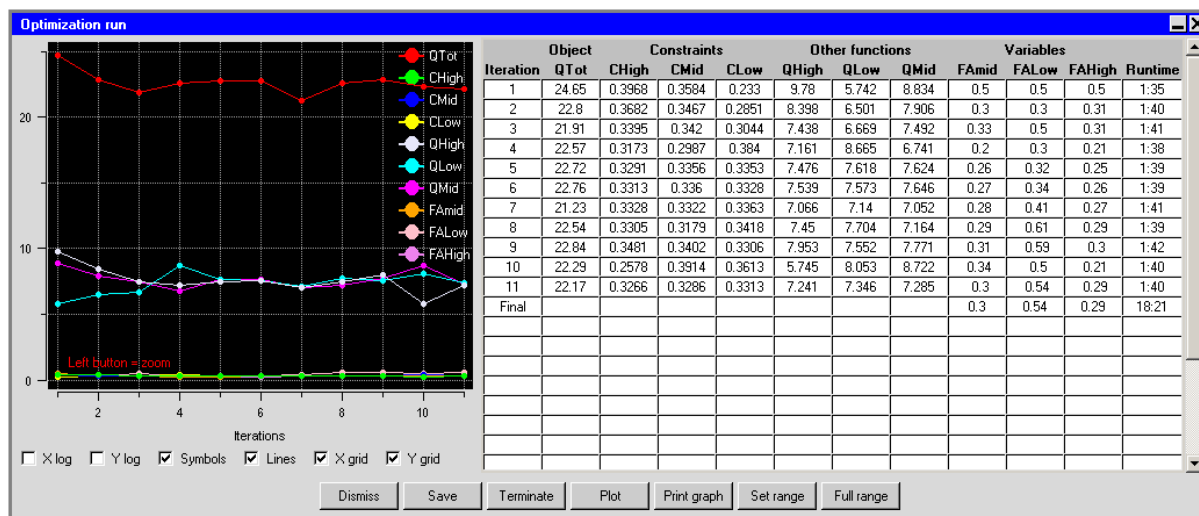


Figure 23.2.7: The Optimization run Panel

- turning off variation of certain function or variable values as a function of optimization iteration number, and
- plotting of design variables.

To turn off the display of certain variation plots of functions or variables as functions of iteration number (e.g., objective function, design variables), click the legend line showing the function or variable. To display the variable again, click the appropriate legend line.

To plot a design variable against a function, click the **Plot** button in the **Optimization run** window. From **Selection** panels, select the x axis and y axis variables. The plots show the variation of the design variables against the selected function for each constant value of the remaining variables. For more information on plotting and reporting of design variables see Section 24.9.

Airpak allows you to parameterize your model, so that you can determine the effect of various object sizes or other characteristics on the solution.

Information about parameterization is divided into the following sections:

- Section [24.1](#): Overview of Parameterization
- Section [24.2](#): Defining a Parameter in an Input Field
- Section [24.3](#): Defining Check Box Parameters
- Section [24.4](#): Defining Radio Button Parameters (Option Parameters)
- Section [24.5](#): Defining a Parameter (Design Variable) Using the Parameters and optimization Panel
- Section [24.6](#): Deleting Parameters
- Section [24.7](#): Defining Trials
- Section [24.8](#): Running Trials
- Section [24.9](#): Function Reporting and Plotting

24.1 Overview of Parameterization

A design process requires the evaluation of a number of available options to determine the most appropriate case that meets the primary design requirement (e.g., smallest size of the room, lowest fan speed that cools the given system to the specified temperature, smallest vent size that meets the system requirements). All of these design scenarios require you to change different parameters and calculate multiple solutions to optimize the design. By studying the results of these multiple calculations, you can identify the trends that affect the performance of the system and you can optimize the design of your model.

Airpak provides a convenient way to specify the range of values for geometric sizes and locations, boundary conditions (e.g., fan curves, vent loss coefficients), and material properties in a single model building exercise. Airpak then instructs the solver to run the different trials that you selected. This process saves the time required to build or specify each model separately and run each of the trials yourself in a sequential manner.

Parameters in **Airpak** are numeric or string constants that you can use in your model instead of actual numeric or string inputs, so that you can easily change their values to model different designs. For example, if you wanted to assign the flow rate for a fan to be the numeric value 0.01, you can specify a parameter called **flowrate** and assign a value of 0.01 to this parameter. You can assign more than one value to a parameter, to define trial calculations that you want to perform for your model. Each trial represents a combination of parameter values, so it is possible to perform multiple trials for your model. In addition, different design scenarios can be tested parametrically by parameterizing check-box and radio-button options. For example, the effect of changing the type of a diffuser assembly in a design, from a round ceiling diffuser to a linear slot diffuser, can be tested by performing two trials, made by turning on/off appropriate diffusers. Similarly, the effect of turbulence can be tested by setting up a parameter for **Flow regime** radio buttons in the **Basic parameters** panel.

There are four ways to define a parameter in **Airpak**:

- Enter a parameter in an input field for an object, material, problem setup default, or solver default (Section 24.2).
- Enter a check box parameter for an object, material, or problem setup default (Section 24.3).
- Enter a radio button parameter (**Option Parameter**) for a problem setup default (Section 24.4) or solver default.
- Define a parameter in the **Parameters and optimization** panel (Section 24.5).



Note that parameters can be used for all inputs for all objects, but they cannot be used for macros.

Once the parameters have been defined, you can redefine the initial value and specify additional values using the **Parameters and optimization** panel (Section 24.5).

You can use the parameters to define the trial calculations that you want to perform for your model (Section 24.7), and finally, you can instruct **Airpak** to run the trials (Section 24.8) and to plot the results (Section 24.9).

24.2 Defining a Parameter in an Input Field

A parameter can be defined in an input field for any object, material, problem setup defaults, and solver defaults. For example, if you want to specify a parameter for the volume flow rate of a fan, you can define a parameter called **flowrate** in the **Volume** text entry field next to the **Fixed**, under **Flow rate**, in the **Fans** panel, as shown below.

Fan type: ☒ Fixed flow ☐ Characteristic curve

☐ Exhaust
☒ Intake

Fluid temp: ambient

Flow: ☒ Normal ☐ Given

☒ Vector X: 0 Y: 0 Z: 0
☐ Angles A: 0 T: 0

Flow rate

☒ Volume 0.0 m3/s
☐ Mass 0.0 kg/s

Swirl: ☐ Magnitude 0.0
☒ RPM 0.0

☐ Species Edit
☐ Turbulence Edit

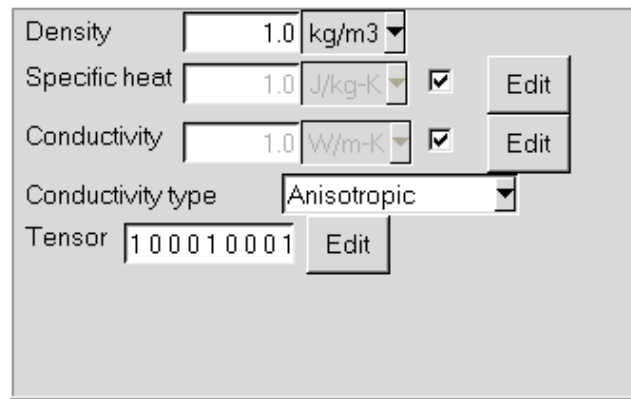
Solar behavior: ☒ Opaque ☐ Transparent

☐ Fix values

You can specify any combination of alphanumeric characters and symbols (except spaces) for the parameter name.



When you specify a parameter in the input field for an object or material, you must always preface the parameter name with the \$ sign. For instance, in the example above, the input for the volume flow rate for the fan is `$flowrate`. If you specify a parameter for a scaling factor when defining the anisotropic conductivity of a material, you must enclose the scaling factor parameter in {} in the **Tensor** field in the **Materials** panel. An example is shown below.



If the parameter you define is related to the geometry of an object or the properties of a material, **Airpak** will ask you to enter a value for the parameter when you click **Update** or **Done** in the **Object** panel or the **Materials** panel. For example, if you specify a parameter for the radius of the fan (**\$radius**), **Airpak** will ask you to provide an initial value for the radius in the **Param value** panel (Figure 24.2.1).

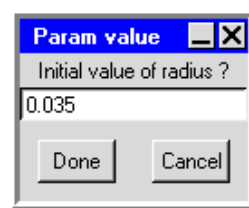


Figure 24.2.1: The **Param value** Panel

When you click **Done** in the **Param value** panel, **Airpak** will update the display of the object in the graphics window using the specified initial value. You can define more than one value for a parameter using the **Parameters and optimization** panel (see Section 24.5).

If the parameter you specified is related to the physical characteristics of the object (excluding the geometry), **Airpak** will set the initial value of the parameter to be zero. You will then be prompted to enter an initial value for the parameter upon opening the **Parameters and optimization** panel. You can redefine the initial value and specify additional values using the **Parameters and optimization** panel (see Section 24.5).

Simple arithmetic operations such as addition, division, and multiplication can be used with a combination of parameters and/or numeric inputs. For example, you can specify the volume flow rate for the fan to be equal to

$(\$a * (\$flowrate) / \$b + 0.1)$



If you use an arithmetic expression to define a parameter, the expression must be enclosed in parentheses.

i Note that, by default, integer arithmetic is performed for division if both the divisor and dividend are entered as integers. If you want floating point division to be performed, then either the divisor or dividend should be entered as a floating point number. For example, $\$a/10$ will give a quotient of 0 if $\$a$ is defined as 1, while $\$a/10.0$ will give a quotient of 0.1 if $\$a$ is 1.

Note that the definition of a parameter is independent of units; i.e., parameters are purely numeric inputs. In the above example for the volume flow rate of a fan, **Airpak** will calculate the expression

$$(\$a * (\$flowrate)) / \$b + 0.1$$

by substituting the values of the parameters into the expression and then the unit (e.g., ft³/s) will be added to the final value of the expression.

To allow you to most efficiently define common parameters for copies of an object, **Airpak** will automatically copy the parameters that you have defined for an object when you copy the object. The copied parameters will then be used for all copies of the object, rather than using the current parameter values. Parameters used for geometric entries will automatically be summed to the values of the corresponding geometric entries that you will request for each copy of the object. For example, if you define the geometric xE coordinate of an object with a parameter $\$a$ and then make copies of the object that are translated in the x direction, then the xE field for the copies will be

$$\$a + d$$

where d is the prescribed offset of the copy of the object.

24.3 Defining Check Box Parameters

A parameter can be defined to turn on/off a check box parametrically. To define a check box parameter, right-click any check-box option in the **Object**, **Materials**, or **Basic parameters** panel. For example, to define the existence or non-existence of an object, position the cursor directly over the **Active** checkbox in the Info tab of the Object panel and right-click the **Active** check box to open the **Active parameter** panel (Figure 24.3.1).

There are two ways to define the **Active** parameter:

- Specify the variable name that will be set to be equal to the object's name. In the **Active parameter** panel, select **Option is ON if variable is equal to this object's name**, and enter the **Variable name**. **Airpak** will ask you to provide an initial value for the variable in the **Param value** panel (Figure 24.2.1). The name of the object or some other name must be entered, enclosed within quotation marks.
- Specify an expression and the numeric value of the expression. In the **Active parameter** panel, select **Option is ON if expression is equal to a specified value**, and enter

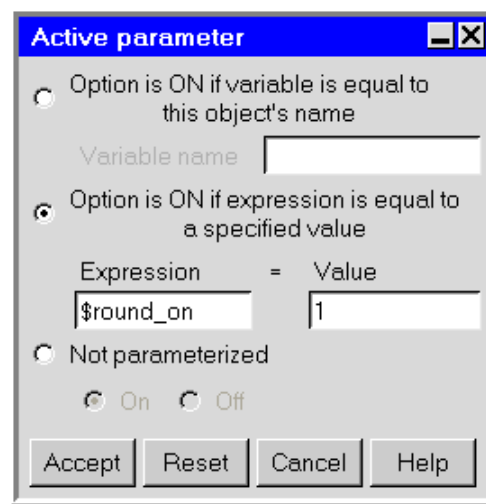


Figure 24.3.1: The Active parameter Panel

the Expression and the Value. Also, provide an initial value for the variable in the Param value panel (Figure 24.2.1).

i Note that the variable name and the expression must be preceded by a \$ sign.

To turn off the parameterization for a check box, first select **Not Parametrized**, then select **On**.

Examples

1. Consider a model with two air supply diffuser assemblies (round ceiling and linear slot). To selectively turn on/off the activity of the diffusers, you can define the active parameters as follows:
 - (a) Set the parameter values for the round ceiling diffuser to 0 (not active) and 1 (active). Open the **Active parameter** panel for the `diffuser.1`, and select **Option is ON if expression is equal to a specified value**. In the **Expression** entry box enter `$round_on`, and in the **Value** entry box enter 1, as shown in Figure 24.3.2. Airpak will ask you to provide an initial value for the variable in the **Param value** panel. Enter 1 and click **Done**. To set another value of the parameter to 0, for which the diffuser assembly will not be **Active**, you will use **Parameters and optimization** panel (see Section 24.5).
 - (b) Set the parameter values for the linear slot diffuser assembly to "diffuser.2" (active) or "off" (not active). As described above, open the **Active parameter** panel for the `diffuser.2`, and select **Option is ON if variable is equal to this**

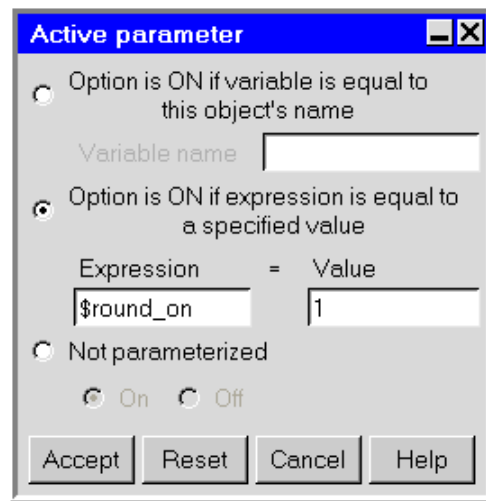


Figure 24.3.2: The Active parameter Panel for the Round Ceiling Diffuser

object's name. In the Variable name entry box enter `$linear_on`, as shown in Figure 24.3.3. Airpak will ask you to provide an initial value for the variable in the Param value panel. Enter "diffuser.2" and click Done. To set another value of the parameter to "off", for which the diffuser assembly will not be Active, you will use Parameters and optimization panel (see Section 24.5).

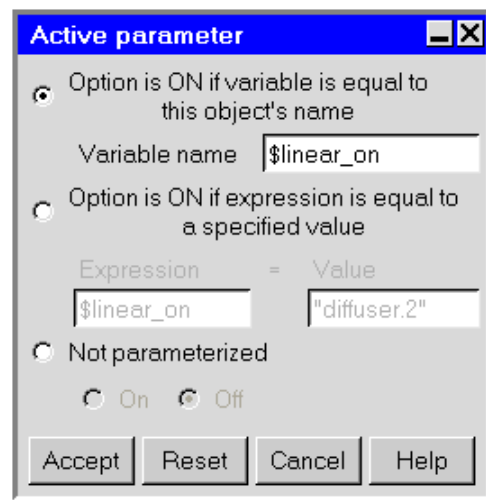


Figure 24.3.3: The Active parameter Panel for the Linear Slot Diffuser

2. This example will demonstrate the use of boolean expressions. Consider a model with five partitions in which you would like to replace one partition with a resistance in each of five trials by selectively turning on/off the activity of partitions and

resistances. Five resistances need to be defined with the same locations as the partitions. You can define the **active** parameters as follows:

- (a) For each partition, respectively, define the following boolean expressions in the **Active parameter** panel, under **Expression**, after selecting **Option is ON** if expression is equal to a specified value: ($\$resistance!=1$), ($\$resistance!=2$), ($\$resistance!=3$), ($\$resistance!=4$), and ($\$resistance!=5$). Under **Value**, enter 1 for every partition, which means logical **true** (every other number is **false**). After you specified the expression for the first partition, **Airpak** will ask you to provide an initial value for the variable in the **Param value** panel. Enter 1 and click **Done**.
- (b) For each resistance, respectively, define the following boolean expressions in the **Active parameter** panel, under **Expression**, after selecting **Option is ON** if expression is equal to a specified value: ($\$resistance==1$), ($\$resistance==2$), ($\$resistance==3$), ($\$resistance==4$), and ($\$resistance==5$). Under **Value**, enter 1 for every resistance. To set values of the parameter to 2, 3, 4, and 5 to determine which vent will be active in each trial, you will use **Parameters and optimization** panel (see Section 24.5).



Note that a check box parameter in the **Basic parameters** panel can only be defined using an expression and a numeric value of the expression.

24.4 Defining Radio Button Parameters (Option Parameters)

A parameter can be defined to select a radio-button option, parametrically, from the **Basic parameters**, **Parallel settings**, or **Advanced solver setup** panel. To define an **Option parameter**, right-click any radio-button option in the **Basic parameters**, **Parallel settings**, or **Advanced solver setup** panel. For example, to define the **Flow regime** parametrically, right-click the **Laminar** or **Turbulent** radio button, next to **Flow regime**, in the **Basic parameters** panel. This will open the **Option parameter** panel (Figure 24.4.1).

In this panel, specify the expression that represents the name of the parameter (e.g., `flow_regime`). The value of the expression will determine the option. The default value of the parameter is 0. The options, along with the values that enable them, are listed in the panel (Figure 24.4.1). For example, the default value of the `flow_regime` parameter is 0, which enables **Laminar** flow. To set another value of the parameter to 1, which enables **Turbulent** flow option, you will use **Parameters and optimization** panel (see Section 24.5).



Note that the expression must be preceded by a \$ sign.



If the value of the specified expression is not equal to any of the predefined values listed in the **Option parameter** panel, then the default option is used for that value.

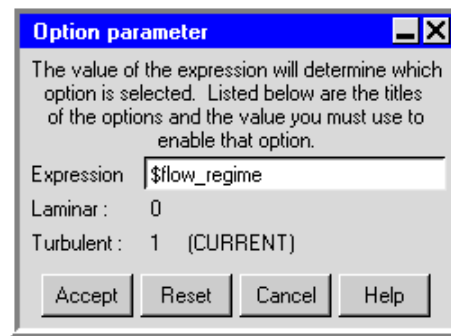



Figure 24.4.1: The Option parameter Panel

24.5 Defining a Parameter (Design Variable) Using the Parameters and optimization Panel

You can define additional new parameters, also called design variables, or edit existing parameters using the **Parameters and optimization panel** (Figures 24.5.1, 24.5.2). To open this panel, click the  button in the **Model and solve toolbar**.

In the **Design variables** tab, the **Parameters and optimization panel** displays all the parameter names that are currently defined, along with their associated values (Figure 24.5.2).

To define a new parameter, click the **New** button in the **Design variables** tab, and enter the name of the parameter in the **New variable name?** text entry field in the **Query** dialog box. Airpak will add the new parameter to the list in the **Parameters and optimization panel**. You can specify any combination of alphanumeric characters and symbols (except spaces) for the parameter name. Enter the value(s) of the parameter. There are two ways of specifying values for parameters:

- Select **Discrete**, and in the text entry field enter multiple values for a parameter separated by commas or spaces.
- Select **In range**, and enter **Start value**, **End value**, and **Increment**.

Click **Apply** to update the list of available trials corresponding to the new set of parameters (see Section 24.7).

To edit previously defined parameters, enter the new value for the parameter in the corresponding **Base value** field. You can then enter multiple values for a parameter using one of the above two methods.

Opening up a design variable's tree displays the object(s) and properties associated with the variable.

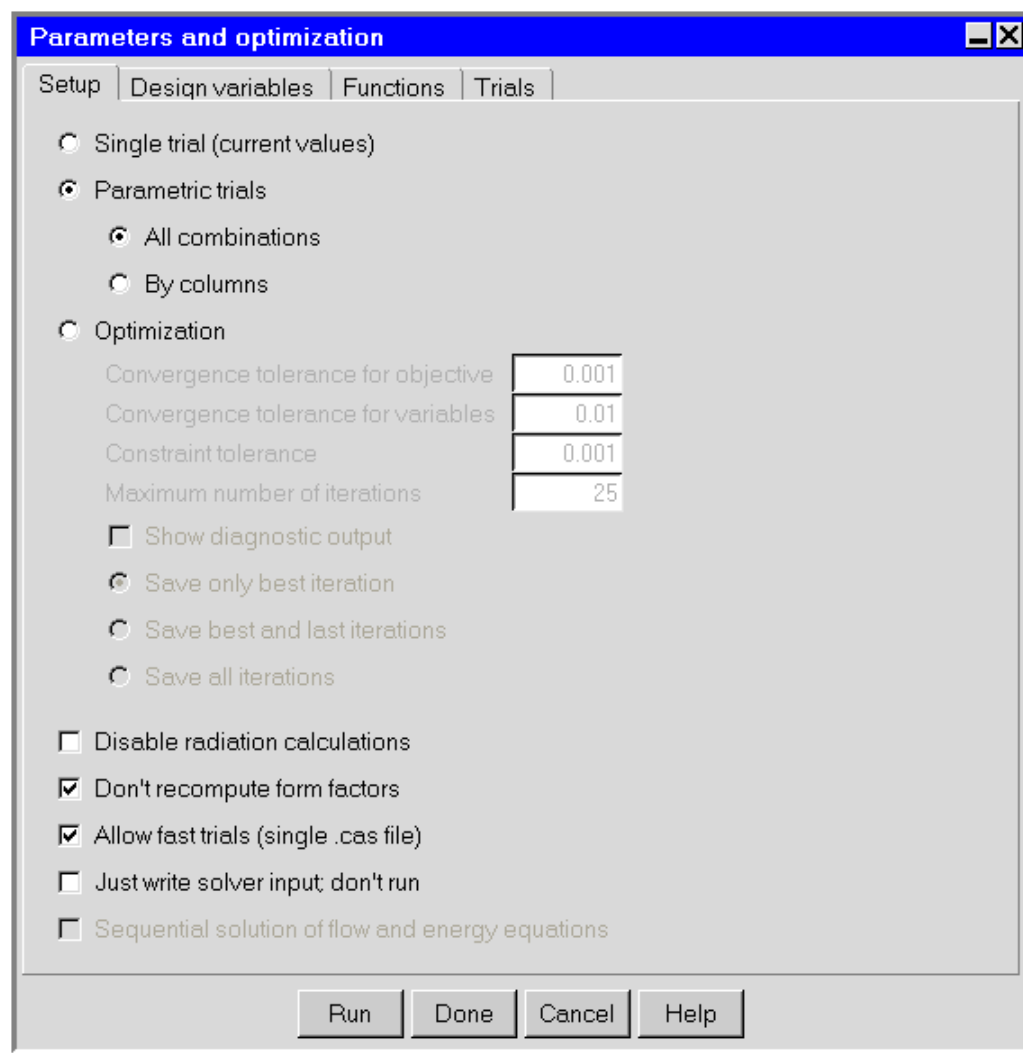


Figure 24.5.1: The Parameters and optimization Panel (Setup Tab)



Note that you can use a parameter in the definition of another parameter's values, but you must precede the parameter name with a \$ sign in the **Base value** field of the Parameters and optimization panel.

24.6 Deleting Parameters

If you have parameters defined in your model that you no longer need, you can delete these parameters from your model. To delete a parameter, select the parameter in the Design variables tab of the Parameters and optimization panel (Figure 24.5.2), and click the Delete button.

Airpak will delete the parameter only if it is not being used anywhere in your model.

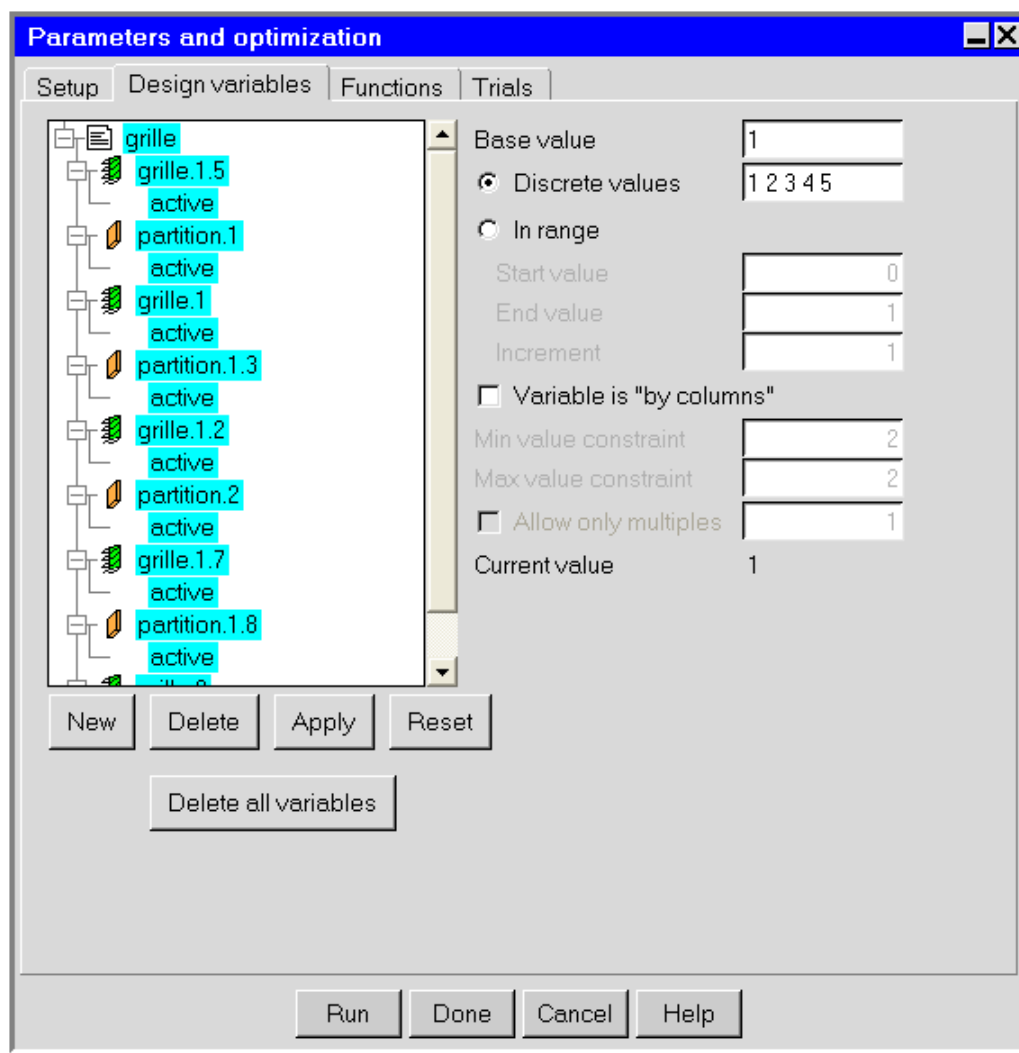


Figure 24.5.2: The Parameters and optimization Panel (Design variables Tab)

To delete all parameters (design variables) and replace all their uses with their present values, click the Delete all variables button.

24.7 Defining Trials

Parameters in Airpak are used to define the trial calculations that you want to perform for your model. Each trial represents a combination of parameter values. Airpak calculates possible combinations from the list of parameters specified in the Design variables tab of the Parameters and optimization panel and lists them in the Trials tab of the panel (Figure 24.7.1). Alternatively, you can select Define trials in the Solve menu.

Solve→Define trials

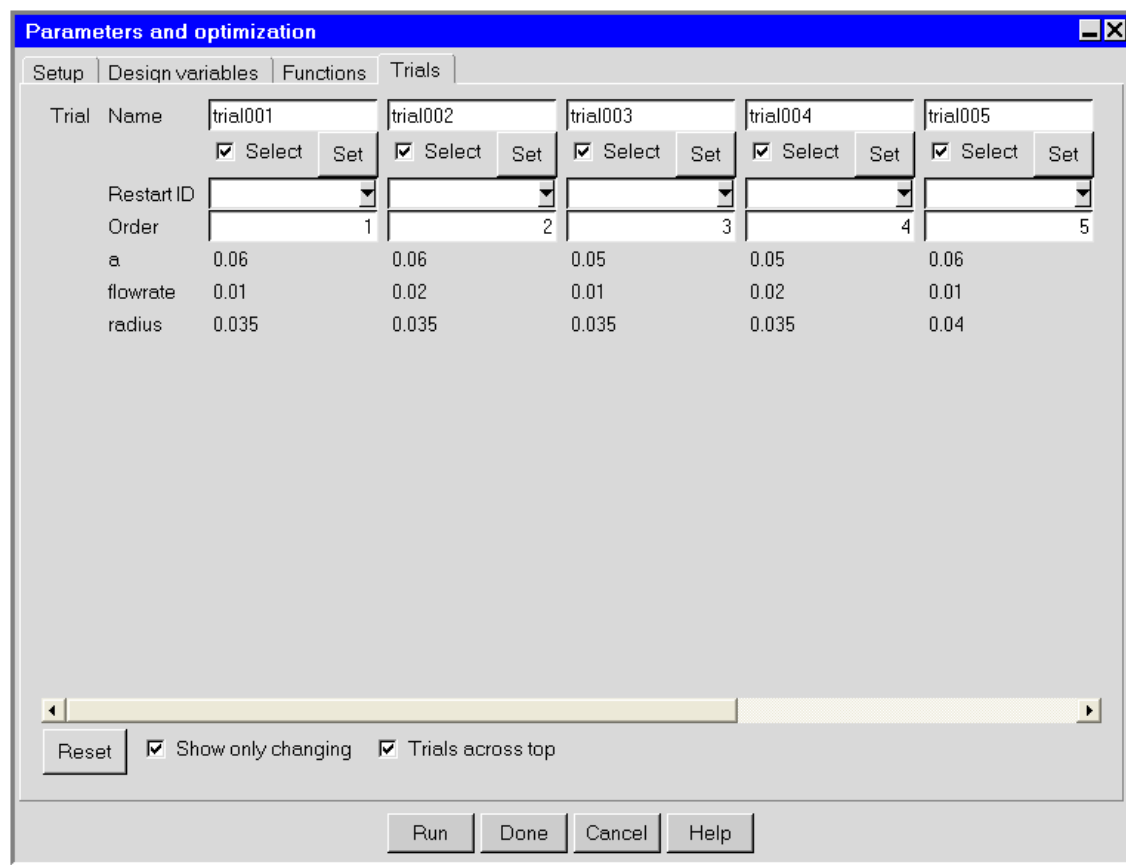


Figure 24.7.1: The Parameters and optimization Panel (Trials Tab)

There are two ways of listing possible combinations from the list of parameters specified in the **Design variables** tab of the **Parameters and optimization** panel: listing all combinations and listing "by column". To choose one of the two options, select **Parametric trials** in the **Setup** tab of the **Parameters and optimization** panel (Figure 24.5.1).

All combinations lists all possible combinations from the list of parameters specified in the **Design variables** tab of the **Parameters and optimization** panel.

By Column controls the number of trial combinations, by selecting **Variable** is "by columns" for certain parameters in the **Design variables** tab. If this option is selected for some of the parameters, then the number of trial combinations will be the number of all possible combinations of parameters that are not defined "by column" (i.e., parameters for which the option **Variable** is "by columns" is not selected). For example, if you define the parameters p , u , and v that can take the following values:

$$\begin{aligned}
 p &= 10, 20, 30 \\
 u &= 1, 2, 3, 4 \\
 v &= 100, 200
 \end{aligned}
 \tag{24.7-1}$$

then

if p is defined "by column", there would be $4 * 2 = 8$ trials;
 if u is defined "by column", there would be $3 * 2 = 6$ trials;
 if v is defined "by column", there would be $3 * 4 = 12$ trials,
 if both p and u are defined "by column", there would be 2 trials;
 if both p and v are defined "by column", there would be 4 trials;
 if both u and v are defined "by column", there would be 3 trials.

If a parameter that had been defined "by column" has fewer defined values than the combination of other parameters, **Airpak** will use the last value specified for that parameter until all trials have been defined.

24.7.1 Selecting Trials

Select the trials that you want to run from the list of possible trials in the **Trials** tab of the **Parameters and optimization** panel (Figure 24.7.1). Turn the **Select** option on or off for a trial in the list to toggle its selection state. Click on **Set** above a trial to make it the first one to be run by **Airpak**. Note that as you click **Set** for a particular trial, **Airpak** will update the graphics window to reflect the geometry that is defined by the values of the parameters used in that trial. This allows you to inspect the effect of changes to the geometry of objects in the model corresponding to each trial. To further change the order in which the trials are run, you can enter the appropriate values in the text boxes next to **Order**.

To speed up convergence of subsequent trials, you can run any trial (except for the first one) using the solution from a previous trial as an initial condition. To do so, select the **Trial Name** corresponding to the solution you want to use for restart from the **Restart ID** drop-down list of the new trial (e.g., Figure 24.7.2).

24.8 Running Trials

Once you have selected the trials that you want to run, you will instruct **Airpak** to run the trials.

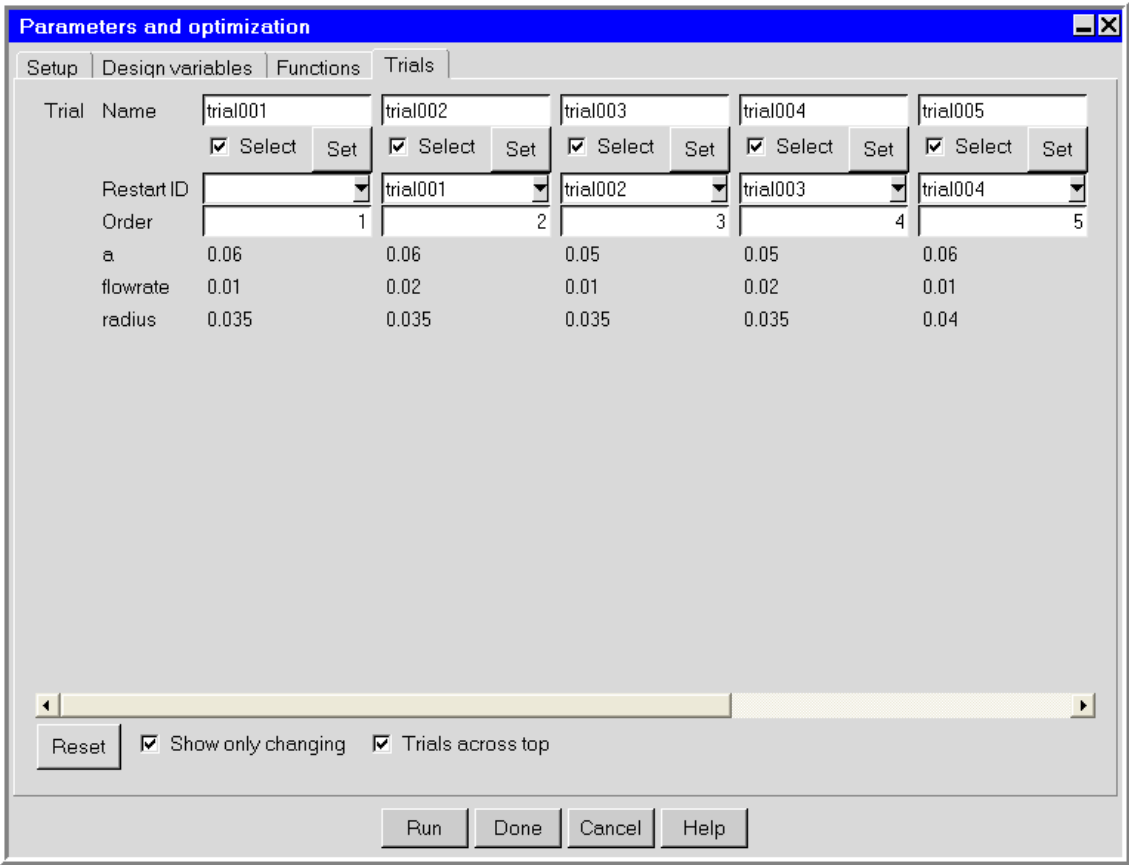


Figure 24.7.2: The Parameters and optimization Panel Displaying Restart ID Options (Trials Tab)

24.8.1 Running a Single Trial

To run a single trial in your Airpak model, follow the steps below.

1. Select **Single trial (current values)** in the **Setup** tab of the **Parameters and optimization** panel (Figure 24.5.1).
2. (optional) To request that Airpak generate one or more function values after the run, define the appropriate function in the **Functions** tab of the **Parameters and optimization** panel, as described in Chapter 23.
3. Set the solver controls for the trial. You will use the **Solve** panel to set the solver controls. (See Section 27.11 for more details on setting the solver controls.) To open the **Solve** panel, select **Run solution** in the **Solve** menu.

Solve → **Run solution**

- (a) To view information about the trial that will be performed, click the **Trial info** button in the **Solve** panel. Airpak will list information about the trial in the **Message** window and also open the **Parameters and optimization** panel if it is not already displayed.
- (b) If you want Airpak to generate a report when the trial is complete, select **Write report when finished** in the **Solve** panel. See Section 27.10 for details on defining a report to be generated when the calculation is complete.
- (c) Click **Start solution** in the **Solve** panel to start the calculation.



You can also start the solution from the **Parameters and optimization** panel by clicking the **Accept** button. This will open the **Start solution** panel. To start the solution click **Yes**.

Airpak will set the values of the parameters to those specified for the trial selected in the **Trials** tab of the **Parameters and optimization** panel (Figure 24.7.1), and then perform the calculation. When the calculation is finished, Airpak will save the results of the simulation and generate a function report (if requested) and display it on the screen. It will also generate a summary report (if requested). You can then perform postprocessing in the usual way (as described in Chapter 28) and you can view the results of the reports that Airpak has generated, as described in Section 29.5.

You can also stop the solver during the trial calculation using the **Terminate** button in the **Monitor** window (see Section 27.12.2 for details). The message **Solution (projectname) is finished** appears in the **Message** window when the solver is done. Note that the **Monitor** windows will remain open until you close them by clicking **Done**.

24.8.2 Running Multiple Trials

To run multiple trials for your **Airpak** model, follow the steps below.

1. Select **Parametric trials** in the **Setup** tab of the **Parameters and optimization** panel and select one of the two options: **All combinations** or **By columns**. For more details on defining trials see Section 24.7.
2. Select the trials to be run in the **Trials** tab of the **Parameters and optimization** panel (Figure 24.7.1), as described in Section 24.7.1. At this point, you may want to toggle the naming scheme of the trials, by clicking on **Reset**. The **Trial naming** dialog will open, and you will be able to choose between **Numbered** (the default naming scheme, see, for example Figure 24.7.2), or **Value**, a naming scheme that uses values of the parameters in each trial name.
3. (optional) To request that **Airpak** generate one or more function values after the run, define the appropriate function in the **Functions** tab of the **Parameters and optimization** panel, as described in Chapter 23.
4. Set the solver controls for the trials. You will use the **Solve** panel to set the solver controls. (See Section 27.11 for more details on setting the solver controls.) To open the **Solve** panel (Figure 24.8.1), select **Run solution** in the **Solve** menu.

Solve→**Run solution**

- (a) Select the **Perform multiple trials** option in the **Solve** panel. You can view information about the trials that will be performed if you click the **Trial info** button in the **Solve** panel. **Airpak** will list trial information in the **Message** window, including the number of trials to be performed, which variables are used, the values specified for each variable, and the values of the variables for the trials to be performed.
- (b) If you are running **Airpak** on a multiprocessor machine, you can specify the number of trials that **Airpak** should run simultaneously on your machine in the **Num concurrent** text entry field in the **Solve** panel. For example, if you enter 2 in the **Num concurrent** field, **Airpak** will run two trials simultaneously. When one of the trials has finished, **Airpak** will start the third trial, and so on.



Note that the number of trials that **Airpak** can run simultaneously is limited by the number of available **Airpak** licenses.

- (c) If you have chosen to submit multiple trials to a script file, you may want to turn on the **Launch from script** option. If this option is enabled, **Airpak** will automatically launch the script file after all of the case files have been written for the different trials. This option will be useful in times when you lock the screen after instructing **Airpak** to write a script file for running multiple trials.

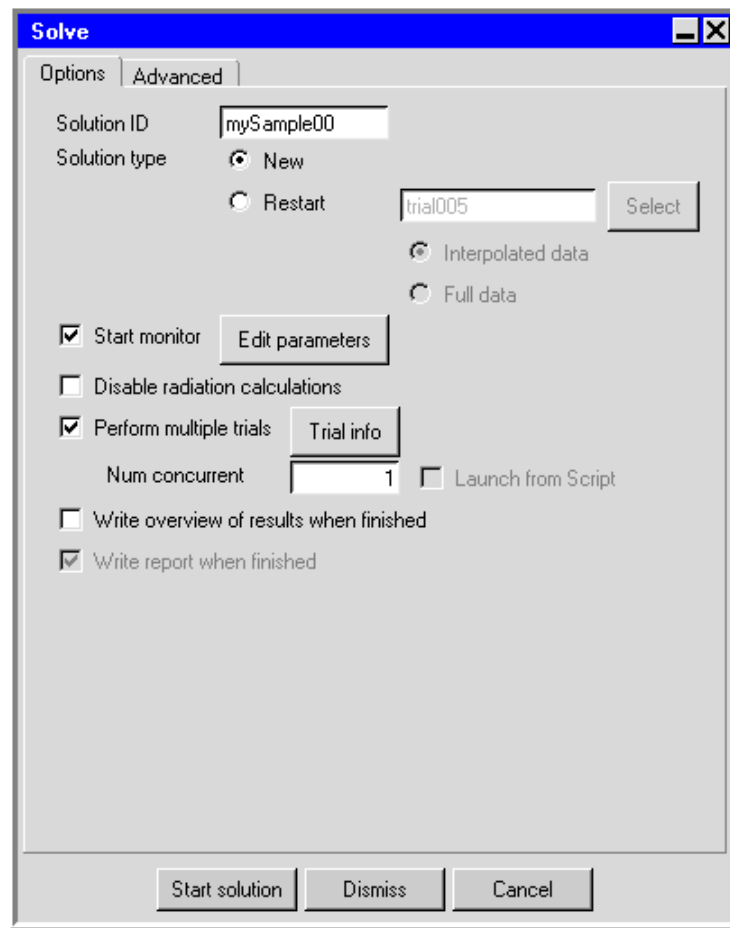


Figure 24.8.1: The Solve Panel

- (d) If the trials do not involve changes to the model geometry, enable the **Use fast trials if possible** option in the **Advanced** tab of the **Solve** panel. This will reduce the time taken per trial and also reduce the memory storage space because only a single case file will be written. If this option is disabled, a separate mesh will be created and a separate case file will be written for every trial regardless of whether the geometry has changed or not.
- (e) If you want **Airpak** to generate a report when each trial is complete, select **Write report when finished** in the **Options** tab of the **Solve** panel. See Section 27.10 for details on defining a report to be generated when the calculation is complete.
- (f) Click **Start solution** in the **Solve** panel to start the calculation.



You can also start the solution from the **Parameters and optimization** panel by clicking the **Accept** button. This will open the **Start solution** panel. To start the solution click **Yes**.

If you entered 1 next to **Num concurrent** in the **Solve** panel, **Airpak** will set the values of the parameters to those specified for the first trial selected in the **Trials** tab of the **Parameters and optimization** panel (Figure 24.7.1), and then start the calculation, using a solution ID of `trial000`. When the calculation is finished, **Airpak** will save the results of the simulation and generate a report or a function value (if requested). **Airpak** will then set the values of the parameters to those specified for the second trial in the **Trials** tab of the **Parameters and optimization** panel, and start a new calculation. When this calculation is complete, **Airpak** will save the results of the simulation using a new solution ID (`trial001`), and generate a report. This process of setting the values of the parameters, performing the calculation, saving the results, and generating reports is repeated until all the selected trials have been completed.

If you have specified that **Airpak** should run two or more calculations simultaneously, **Airpak** will set the values of the parameters to those specified for the first trial selected in the **Trials** tab of the **Parameters and optimization** panel (Figure 24.7.1), and then start the calculation, using a solution ID of `trial000`, **Airpak** will then set the values of the parameters to those specified for the second trial selected in the **Trials** tab of the **Parameters and optimization** panel, and start a second calculation with a new solution ID (`trial001`), and so on. When the first calculation is finished, **Airpak** will save the results of the simulation and generate a report (if requested). **Airpak** will then set the values of the parameters to those specified for the next trial selected in the **Trials** tab of the **Parameters and optimization** panel, and start a new calculation. When the second calculation is complete, **Airpak** will save the results of the simulation, generate a report (if requested), and start the next trial, and so on. This process of setting the values of the parameters, performing the calculation, saving the results, and generating reports is repeated until all the selected trials have been completed.

You can then perform postprocessing in the usual way, specifying which trial results you want to examine by selecting the appropriate solution ID. See Section 28.4 for details. You can also view the results of the reports that **Airpak** has generated, as described in Section 29.5.

You can stop the solver during the trial calculations using the **Terminate** button in the **Monitor** window (see Section 27.12.2 for details). All trial runs are terminated, and the message **Solution (projectname) is finished** appears in the **Message** window when the solver is done. Note that the **Monitor** windows will remain open until you close them by clicking **Done**.

24.9 Function Reporting and Plotting

As **Airpak** starts performing the trials, the **Parametric trials** panel will open and **Airpak** will display all the function values defined a priori, as well as parameters and running times for each trial (Figure 24.9.1).

The function values reported in the **Parametric trials** panel can be plotted against the

Parametric trials							
Trial	Functions				Variables		Runtime
	Kloss	bDh	delDt	Vel	Thickness	Length	
tr_Length_0_025_Thickness_0_001	0.942	0.025	0.001	26.67	0.001	0.025	1:37:37
tr_Length_0_025_Thickness_0_004	1.222	0.025	0.004	26.67	0.004	0.025	2:12
tr_Length_0_025_Thickness_0_016	1.03	0.025	0.016	26.67	0.016	0.025	2:06
tr_Length_0_025_Thickness_0_030	1.007	0.025	0.03	26.67	0.03	0.025	12:47
tr_Length_0_025_Thickness_0_5	0.9465	0.025	0.5	26.67	0.5	0.025	2:11
tr_Length_035_Thickness_0_001	0.9406	0.035	0.001	26.67	0.001	0.035	2:01
tr_Length_035_Thickness_0_004	1.258	0.035	0.004	26.67	0.004	0.035	6:21
tr_Length_035_Thickness_0_016	1.054	0.035	0.016	26.67	0.016	0.035	1:57
tr_Length_035_Thickness_0_030	1.021	0.035	0.03	26.67	0.03	0.035	16:13
tr_Length_035_Thickness_0_5	0.9449	0.035	0.5	26.67	0.5	0.035	2:00
tr_Length_0_05_Thickness_0_004	1.311	0.05	0.004	26.67	0.004	0.05	9:15
tr_Length_0_05_Thickness_0_016	1.094	0.05	0.016	26.67	0.016	0.05	5:40
tr_Length_0_05_Thickness_0_030	1.051	0.05	0.03	26.67	0.03	0.05	1:54
tr_Length_0_05_Thickness_0_001	0.9445	0.05	0.001	26.67	0.001	0.05	1:50
tr_Length_0_05_Thickness_0_5	0.9538	0.05	0.5	26.67	0.5	0.05	2:00
Total							2:46:05

Figure 24.9.1: The Parametric trials Panel

parameters. To create plots of functions versus parameters, follow the steps below:

1. If the Parametric trials panel is not already opened, select **Show optimization/param results** from the **Report** menu to open the **Parametric trials** panel.

Report → Show optimization/param results

2. In the **Parametric trials** panel (Figure 24.9.1), click the **Plot** button to open the **Selection** panel (Figure 24.9.2). In the **Selection** panel, select the design variable for the x axis (e.g., **Thickness**), and click **Okay**.

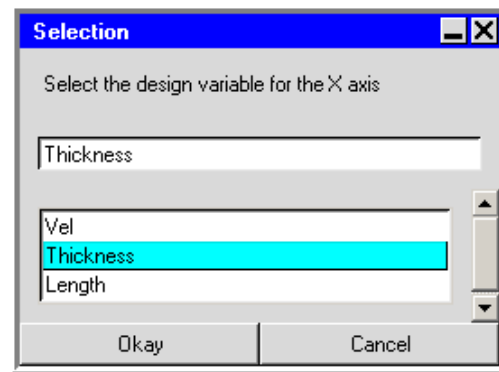


Figure 24.9.2: The Select Panel (x axis selection)

3. In the **Selection** panel (Figure 24.9.3), select the function for the y axis (e.g., **Kloss**) and click **Accept**. This will display the plot (e.g., **Kloss vs Thickness**), as shown in Figure 24.9.4. For more information on modifying the range and appearance of the plot, printing, saving, and reloading plot data, see Section 28.3.2.

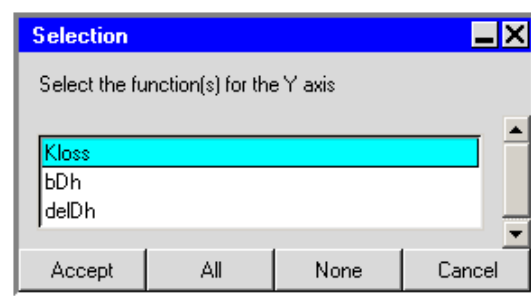


Figure 24.9.3: The Select Panel (y axis selection)

4. To display another plot, click the **Plot** button in the **Parametric trials** panel again, and repeat steps 2 and 3, to make x and y axis selections. You can repeat this step multiple times, which will allow you to view multiple plots at the same time.

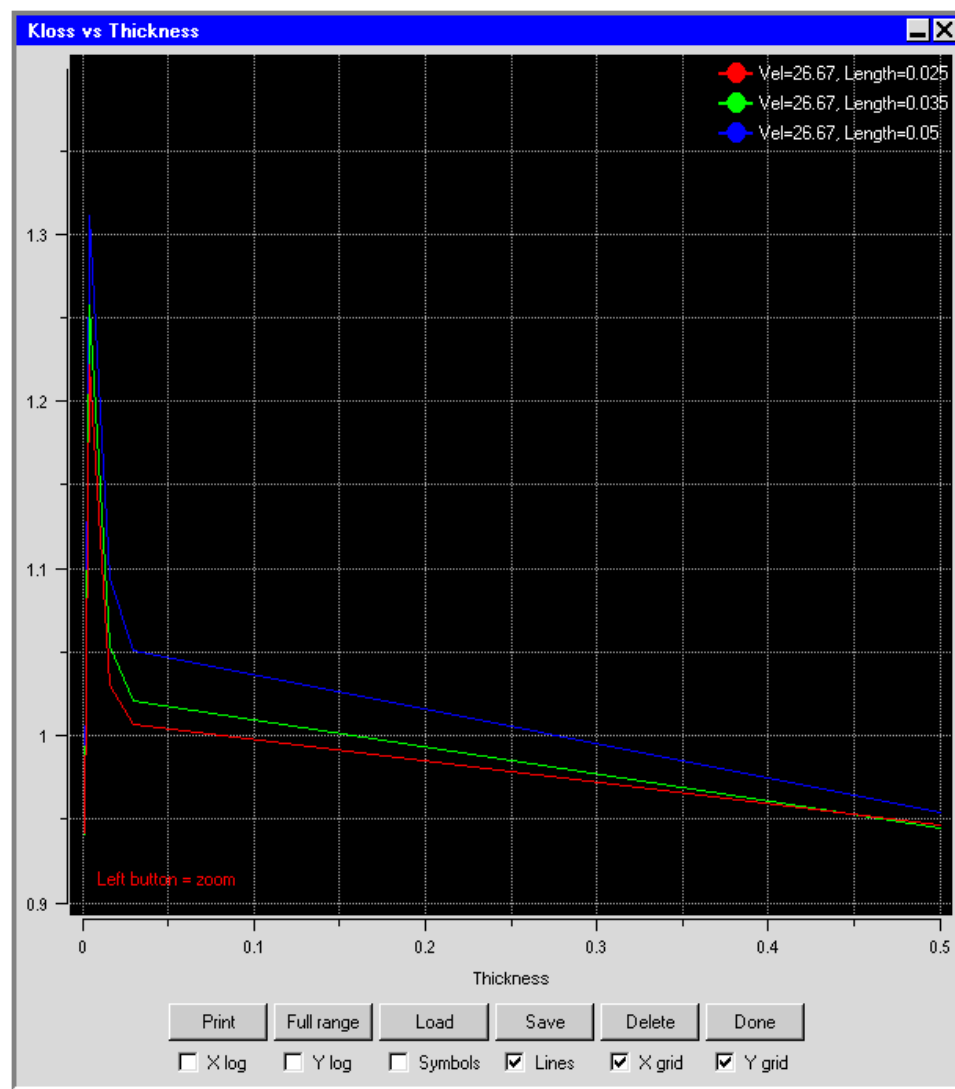
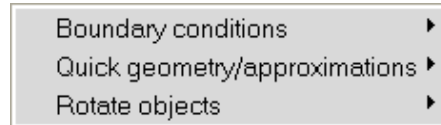


Figure 24.9.4: Plot

Macros can help streamline the process of building your **Airpak** model. There are three general types of macros you can choose from that are accessible from the **Macros** menu.



This chapter describes the macros available in **Airpak** and how you can use them. Information in this chapter is organized into the following sections:

- Section [25.1](#): Boundary Condition Macros
- Section [25.3](#): Geometry Macros
- Section [25.4](#): Object Rotation Macros

25.1 Boundary Condition Macros

The following boundary condition-related macros are described in the following sections:

- Section [25.1.1](#): Solar Flux Boundary Condition
- Section [25.1.2](#): Atmospheric Boundary Layers
- Section [25.2](#): Diffuser Macros

25.1.1 Solar Flux Boundary Condition

The solar flux macro allows you to account for incident solar radiation, given a model geometry and pertinent solar information such as terrestrial location, date, and time. Once **Airpak** has calculated a value for the heat flux resulting from incident solar radiation, you can then specify this value as a boundary condition either for a wall object or for the individual sides of a block object in your **Airpak** model. See Section [7.5](#) for more information about walls and blocks.

Specifying a Solar Flux Boundary Condition for Your Airpak Model

To include a solar flux boundary condition in your Airpak model, select **Solar Flux** and then **Boundary conditions** in the **Macros** menu. This will open the **Solar Flux Calculator** panel, shown in Figure 25.1.1.

Macros→Boundary conditions→Solar Flux

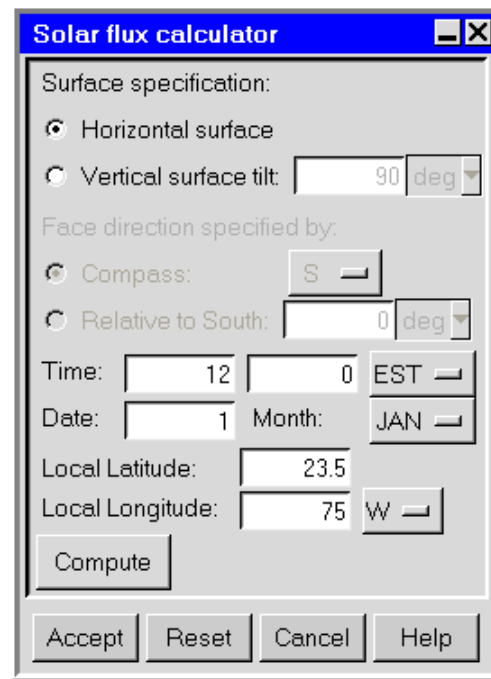


Figure 25.1.1: The Solar Flux Calculator Panel

The procedure for specifying a solar flux boundary condition for your model is as follows:

1. Specify the incline of the model surface relative to the ground under **Surface specification**. There are two options:

Horizontal surface specifies a surface that is parallel to the ground.

Vertical surface tilt allows you to specify the incline, or tilt, of the surface relative to the ground. Values greater than 0° up to and including 90° may be specified.

2. (vertical or tilted surfaces only) Specify the direction the surface is facing under **Face direction specified by**. There are two options:

Compass allows you to select one of the eight cardinal directions from the **Compass** menu.

Relative to South allows you to specify the surface face direction in degrees relative to South. Values typically range from -180° to 180° , where 0° is South, 90° is West, -180° or 180° is North, and -90° is East.

3. Specify the local time at the desired location in the two fields to the right of **Time**. The time is based on a 24-hour clock, thus acceptable values range from 0 h 0 min (12:00 a.m.) to 23 h 59.99 min (11:59.99 p.m.). Values entered in the first text-entry field (hour) must be integral, but values entered in the second text-entry field (minute) can be integral or fractional. For example, if the local time was 12:01:30 a.m., you would enter 0 for the hour and 1.5 for the minute. If the local time was 4:17 p.m., you would enter 16 for the hour and 17 for the minute.
4. (optional) Specify the local time zone of the desired location using the menu to the right of the **Time** text-entry fields. The options are: Greenwich Mean Time (GMT), Atlantic Standard Time (AST), Eastern Standard Time (EST), Central Standard Time (CST), Mountain Standard Time (MST), and Pacific Standard Time (PST). The time zone is used to approximate the longitude (see below) of the location if you are unsure of the coordinates.
5. Specify the **Date** and **Month** of the year.
 - (a) Specify a value for the **Date**.
 - (b) Select the month from the **Month** menu.
6. Specify the **Local Latitude** of the desired location. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator.
7. Specify the **Local Longitude** of the desired location. The longitude is approximated if you specify the local time zone, but you can enter a more precise value if you know it. Any value you enter here will take precedence over the time zone. Values may range from 0° to 180° . Select the hemisphere (W or E) from the menu to the right of the **Local Longitude** text-entry field.
8. Click **Compute** to accept the parameters entered into the **Solar Flux Calculator** panel and calculate the solar irradiation flux boundary condition for your **Airpak** model. The **Message** window will display the following information:
 - Local sunrise and sunset times
 - Apparent solar time
 - Direct (ED) and normal (EDN) components of the solar irradiation flux on the surface in W/m^2
 - Diffuse sky radiation (Ed) in W/m^2
 - Total short-wave irradiance ($E_t = E_D + E_d$) in W/m^2

For more information about these quantities and how they are calculated, see the ASHRAE Fundamentals Handbook [1].

i If you click **Compute** in the **Solar Flux Calculator** panel after specifying an improper value for the date, time, latitude, or longitude, the **Message** window will inform you that a parameter is wrong. To fix an error, re-open the **Solar Flux Calculator** panel, adjust the parameter to an acceptable value, and click **Compute**.

9. Specify the calculated heat flux as a boundary condition on the desired **Airpak** wall or block object. This step uses a wall object as an example.

Model→**Walls**

- (a) Create or select the desired wall object (see Section 13.7).
- (b) In the **Walls** panel, specify an **Outside heat flux** equal to the total short-wave irradiance computed by the solar flux macro.

For example, at 7:00 a.m. on April 2 at 43.6° north latitude and 73.2° west longitude, the direct component of the solar irradiation flux on a vertical wall facing west is calculated to be 0 W/m², and the diffuse sky radiation is calculated to be about 89 W/m². The reason that the direct component is 0 is that the outer surface of this particular wall is pointed away from the sun at 7:00 a.m.

At 4:00 p.m. on the same day, the direct component of the solar irradiation flux on the wall is calculated to be about 610 W/m² because the wall is now facing the sun. Combined with the calculated diffuse sky radiation of 532 W/m², the total short-wave irradiance on the wall is about 1142 W/m².

25.1.2 Atmospheric Boundary Layers

The atmospheric boundary layer macro allows you to characterize external wind flows by the wind direction, wind speed, height above the ground, and a description of the local terrain. Using these parameters, **Airpak** can estimate the mean wind velocity at any height h using the following equations:

$$U(h) = \begin{cases} U_{\text{met}} \left(\frac{d_{\text{met}}}{H_{\text{met}}} \right)^{a_{\text{met}}} \left(\frac{h}{d} \right)^a & h < d \\ U_{\text{met}} \left(\frac{d_{\text{met}}}{H_{\text{met}}} \right)^{a_{\text{met}}} & h \geq d \end{cases} \quad (25.1-1)$$

where

- U_{met} = wind speed measured from a nearby meteorological station
- H_{met} = anemometer height
- a_{met} = terrain factor for the meteorological station
- d_{met} = boundary layer thickness at the meteorological station
- a = terrain factor for desired location
- d = boundary layer thickness at desired location

In Equation 25.1-1, note that $U(h)$ will remain constant at any height h equal to or greater than the thickness of the boundary layer d .

The anemometer height is typically 10 m above ground level, with $a_{\text{met}} = 0.14$ and $d_{\text{met}} = 270$ m, depending on the category of terrain surrounding the meteorological station. For the atmospheric boundary layer macro, you can characterize the local terrain as one of the four types listed in Table 25.1.2 [1], or you can specify the terrain parameters yourself using your own knowledge or experience.

Table 25.1.1: Terrain Factor and Boundary Layer Thickness for Four Categories of Land

Terrain Category	Description	a	d (m)
1	Large city centers, in which at least 50% of buildings are higher than 21 m over a distance of at least 2000 m upwind.	0.33	460
2	Urban, suburban, and wooded areas, and other areas with closely spaced obstructions compared to or larger than single-family dwellings (over a distance of at least 2000 m upwind).	0.22	370
3	Open terrain with scattered obstacles generally less than 10 m high.	0.14	270
4	Flat, unobstructed areas exposed to wind flowing over a large body of water (no more than 500 m inland).	0.10	210

If one of the two-equation turbulence models (standard k - ϵ or RNG k - ϵ) is being used, the following boundary conditions will be applied to the atmospheric boundary layer:

- turbulent kinetic energy (constant)

$$k = \frac{(U^*)^2}{\sqrt{C_\mu}} \quad (25.1-2)$$

- turbulent dissipation rate (varies with elevation)

$$\epsilon = \frac{(U^*)^3}{\kappa h} \quad (25.1-3)$$

where $C_\mu = 0.09$, $\kappa = 0.42$ (von Kármán's constant), and

$$U^* = \frac{\kappa U_{\text{met}}}{\ln\left(\frac{H_{\text{met}}}{s}\right)} \quad (25.1-4)$$

where s is the roughness height.

From the above information, **Airpak** creates a rectangular opening with profiles to define the wind velocity at various heights above the ground. The points in the profiles are closer together nearer to the ground and more separated farther from the ground.

Once you have created the opening object, you may replace the turbulence boundary conditions set by the macro with actual measured data if you are using one of the two-equation turbulence models (standard k - ϵ or RNG k - ϵ). See Section 11.4.1 for details about turbulence profiles.

Adding an Atmospheric Boundary Layer to Your Airpak Model

Before you add an atmospheric boundary layer to your **Airpak** model, you will need to specify the northward direction for your model in the **Basic parameters** panel. To open the **Basic parameters** panel, double-click on the **Basic parameters** item under the **Problem setup** node in the **Model manager** window.

Problem setup—>Basic parameters

Under **Northward direction**, specify values for (X, Y, Z) to define the northward direction for your model. See Section 6.4.4 for more information about specifying the northward direction.

To include an atmospheric boundary layer in your **Airpak** model, select **Boundary conditions** and then **Atmospheric boundary layer** in the **Macros** menu. This will open the **Atmospheric boundary layer** panel, shown in Figure 25.1.2.

Macros—>Boundary conditions—>Atmospheric boundary layer

The procedure for adding an atmospheric boundary layer to your model is as follows:

1. Specify the position and size of the boundary layer.
 - (a) Specify the **Plane** in which the boundary layer lies (Y-Z, X-Z, or X-Y).
 - (b) Select **Start/end** and enter values for the start coordinates (xS, yS, zS) and end coordinates (xE, yE, zE, as appropriate) of the boundary layer, or select **Start/length** and enter values for the start coordinates (xS, yS, zS) and the lengths of the sides (xL, yL, zL, as appropriate) of the boundary layer.
2. Specify the wind direction under **Wind direction specified by**.

There are two options:

Atmospheric boundary layer

Open plane dimensions:

Plane: **Y-Z**

Location

Specify by: **Start / end**

xS	-5.0	m	xE	5.0	m
yS	0.0	m	yE	3.0	m
zS	-5.0	m	zE	5.0	m

Wind direction specified by:

☒ Compass: **N**

☐ Relative to North: **0** deg

Wind speed: **5.0** m/s

Anemometer height: **10.0** m

Exponent (met): **0.14**

Boundary layer thickness (met): **270** m

Local terrain type:

Large city centers

Urban, suburban, wooded areas

Scattered obstacles on open terrain

Flat, unobstructed areas

Exponent: **0.14**

Boundary layer thickness: **270.0** m

Surface roughness height: **0.3** m

Profile density: **10**

Profile direction: Y ☒ Z ☐

☐ Create internal part

Name: **blayer.1**

Save Load

Accept Reset Cancel Help

Figure 25.1.2: The Atmospheric boundary layer Panel

Compass allows you to select one of the eight cardinal directions from the **Compass** menu. The direction you select indicates from which direction the wind is blowing.

Relative to North allows you to specify the wind direction in degrees relative to the northward direction you specified in the **Basic parameters** panel (see Section 6.4.4 for details). Values typically range from -180° to 180° , where 0° is North, 90° is East, -180° or 180° is South, and -90° is West. The value you enter indicates from which direction the wind is blowing.

Note that the wind direction you specify indicates from where the wind is blowing (e.g., from the North to the South). For example, if you define the positive x axis as the northward direction in the **Basic parameters** panel and specify a wind direction of North in the **Atmospheric boundary layer** panel, the x -velocity magnitude will be negative, which indicates that the wind is blowing in the direction of the negative x axis.

3. Specify the **Wind speed** measured at the meteorological station (U_{met} in Equation 25.1-1).
4. Specify the **Anemometer height** (H_{met} in Equation 25.1-1). Typically, the wind speed is measured at 10 m above ground level.
5. (optional) Specify the **Exponent (met)**, which is the terrain factor for the meteorological station (a_{met} in Equation 25.1-1). The default value of a_{met} is 0.14, but you may enter your own value for the terrain factor based on your own knowledge or experience.
6. (optional) Specify the **Boundary layer thickness (met)** for the meteorological station (d_{met} in Equation 25.1-1). The default value of d_{met} is 270 m, but you may enter your own value for the boundary layer thickness based on your own knowledge or experience.
7. Select the **Local terrain type**. The four options (**Large city centers**; **Urban, suburban, wooded areas**; **Scattered obstacles on open terrain**; and **Flat, unobstructed areas**) are described in Table 25.1.2.
8. (optional) Specify the **Exponent**, which is the terrain factor for the desired location (a in Equation 25.1-1). If your problem is not accurately modeled with one of the four pre-defined terrain types, you can enter your own value for the terrain factor based on your own knowledge or experience.
9. (optional) Specify the **Boundary layer thickness** for the desired location (d in Equation 25.1-1). If your problem is not accurately modeled with one of the four pre-defined terrain types, you can enter your own value for the boundary layer thickness based on your own knowledge or experience.

10. (two-equation turbulence models only) Specify the **Surface roughness height** for the problem (s in Equation 25.1-4). By default, this value is set to 0.3 m. If your problem is not accurately described by this value, you can enter your own value for the height of the roughness sublayer based on your own knowledge or experience.
11. Specify the **Profile density**, which is the number of points that will be included in the velocity profile for the atmospheric boundary layer.
12. Specify the **Profile direction**. This is the direction in the **Airpak** model (X, Y, or Z, as available) in which the variation of height (h in Equation 25.1-1) is taking place, and should be perpendicular to the specified northward direction. See Section 6.4.4 for more information about specifying the northward direction.
13. Specify whether you want the boundary layer to be available as one part or as an individual object.
 - If you want the boundary layer to be available as one part in your **Airpak** model, select **Create internal part** and specify a name in the **Name** text entry field. The default name is **blayer.n**, where **n** is the next sequential number among numbered boundary layers.
 - If you want the boundary layer to be available as an opening object, deselect **Create internal part**. **Airpak** will create an opening with profiles to define the wind velocity at various heights above the ground.
14. If you want to save the current boundary layer specifications to be reused in your current model or to be used in a different model, click the **Save** button to open the **Macro information** dialog box (which is the same as the **File selection** dialog box described in Section 2.1.8). **Airpak** will save the specifications in a subdirectory named **blayer_info**.

To load boundary layer specifications that you have saved in this manner, click the **Load** button and select the boundary layer specifications to be loaded.
15. Click **Accept** to accept the parameters entered into the **Atmospheric boundary conditions** panel and create the boundary layer.

Modifying an Existing Atmospheric Boundary Layer

After the boundary layer has been created and positioned in your **Airpak** model, it cannot be modified using the **Atmospheric boundary layer** panel. There are two ways to modify an existing boundary layer:

- If you selected the **Create internal part** option in the **Atmospheric boundary layer** panel, you can edit the boundary layer in the same way that you edit a part (see Section 7.8.5).

- If you did not select the **Create internal part** option in the **Atmospheric boundary layer** panel, you must select and modify the opening object that makes up the boundary layer.

25.2 Diffuser Macros

For simulations of room airflow, it is important to accurately model the behavior of air inlet diffusers and thus predict the penetration and spread of the air in the room. This performance aspect of air inlet diffusers is referred to as the throw of the diffuser, which is defined as the distance downstream of the diffuser where the jet centerline velocity equals a specified terminal velocity. Due to the large scale differences between the dimensions of the room and the important small geometrical features of diffusers, it is infeasible computationally to explicitly model the detailed geometry of diffusers. Simply neglecting the geometrical details of the diffuser usually leads to airflow simulations that deliver the correct amount of air into the room, but fail to achieve the proper jet penetration. However, simplified diffuser models that account for diffuser performance or the velocity field in the vicinity of the diffuser exit can be used to predict the penetration and spread of the air jet into the room. **Airpak** includes diffuser macros to help you accurately model several different types of diffusers with simplified boundary conditions.

Information about modeling diffusers in **Airpak** is presented in the following sections:

- Section [25.2.1](#): Diffuser Modeling Methods
- Section [25.2.2](#): Diffuser Types
- Section [25.2.3](#): Steps for Adding a Diffuser to Your **Airpak** Model
- Section [25.2.5](#): Specifying Airflow Performance Data
- Section [25.2.4](#): Specifying Geometry, Position, and Size
- Section [25.2.6](#): Specifying the Extrusion and Flow Directions
- Section [25.2.7](#): Additional Inputs for Specific Types of Diffusers
- Section [25.2.8](#): Specifying Supply-Air and Room-Air Conditions
- Section [25.2.9](#): Specifying the Modeling Method

25.2.1 Diffuser Modeling Methods

The two primary methods for modeling diffusers with simplified boundary conditions [\[5\]](#) are the momentum method and the box method. These two methods are described below.

Momentum Method

The momentum method assumes that the airflow from a particular diffuser can be predicted using the isothermal axisymmetric jet formula

$$\frac{u_m}{u_0} = K_1 \frac{\sqrt{A_0}}{x} \quad (25.2-1)$$

where u_m is the centerline or maximum jet velocity at a distance x from the diffuser, u_0 is the initial jet velocity, A_0 is the effective area, and K_1 is the empirically-determined centerline velocity decay constant. K_1 may be a function of many factors, including diffuser type, Reynolds number, jet confinement, or possible deflections [1]. In airflow simulations, the volumetric flow rate, Q , is specified and diffuser performance data provide information on the throw x for a specific terminal velocity u_m and a given flow rate Q . Equation 25.2-1 can then be rearranged to solve for the initial jet velocity:

$$u_0 = \frac{u_m^2 x^2}{K_1^2 Q} \quad (25.2-2)$$

where $Q = u_0 A_0$.

The initial jet velocity, u_0 , may be larger than the velocity, u , that is calculated by simply using the volumetric flow rate and the geometric area that the diffuser occupies (A_{geom}):

$$u = \frac{Q}{A_{\text{geom}}} \quad (25.2-3)$$

To properly model the diffuser performance, the simulation needs to ensure that the airflow from the diffuser enters the room with momentum corresponding to velocity u_0 instead of momentum corresponding to velocity u . Typically, the diffuser is modeled with a volumetric flow rate Q and a cross-sectional area A_{geom} . A momentum source that accounts for u_0 being the initial jet velocity is added to the diffuser to reproduce diffuser performance in the simulation.

The same approach is used for linear jets; however, the implementation in **Airpak** for ceiling diffusers (circular, square, and vortex) maintains the radial or lateral jet behavior of the diffuser by modeling the circumferential distance of the diffuser and extruding the direction normal to the ceiling until the modeled flow area equals the computed effective area.

Box Method

Although the momentum method works well for most types of diffusers, entrainment effects of discrete air jets from nozzle, linear slot, or valve diffusers are more accurately modeled using the box method.

The box method does not explicitly model the jet behavior in the immediate vicinity of the air inlet diffuser; instead, it specifies flow boundary conditions on the sides of a rectangular region adjacent to the air inlet diffuser. For best accuracy with the box method, velocity profiles from the inlet diffuser manufacturer or from measured data should be used.

The box method in **Airpak** uses flat velocity profiles to account for entrainment effects, but you can improve the accuracy by supplying measured velocity profiles when you are editing the objects created by the diffuser macro

25.2.2 Diffuser Types

The following diffuser types are available in **Airpak**:

- Grille
- Ceiling
- Displacement
- Slot
- Nozzle
- Valve
- Vortex

Section 25.2.3 describes the steps for defining a diffuser in **Airpak**. See Figure 25.2.1 for general information about inputs for each type of diffuser. Details about the inputs for each type of diffuser are provided in Sections 25.2.5–25.2.9.

25.2.3 Steps for Adding a Diffuser to Your Airpak Model

To include a diffuser in your **Airpak** model, select **Boundary conditions** and then **Diffuser** in the **Macros** menu. This will open the **Diffuser** panel, shown for the different types of diffusers in Figures 25.2.2–25.2.14.

Macros→Boundary conditions→Diffuser

The procedure for adding a diffuser to your model is as follows:

1. Specify the **Diffuser type** by selecting **Grille**, **Ceiling**, **Displacement**, **Slot**, **Nozzle**, **Valve**, or **Vortex**. The panel will change (as shown in Figures 25.2.2–25.2.14) depending on your selection of **Diffuser type**. Note that each **Diffuser type** panel has two tabs: **Geometry** and **Properties**.

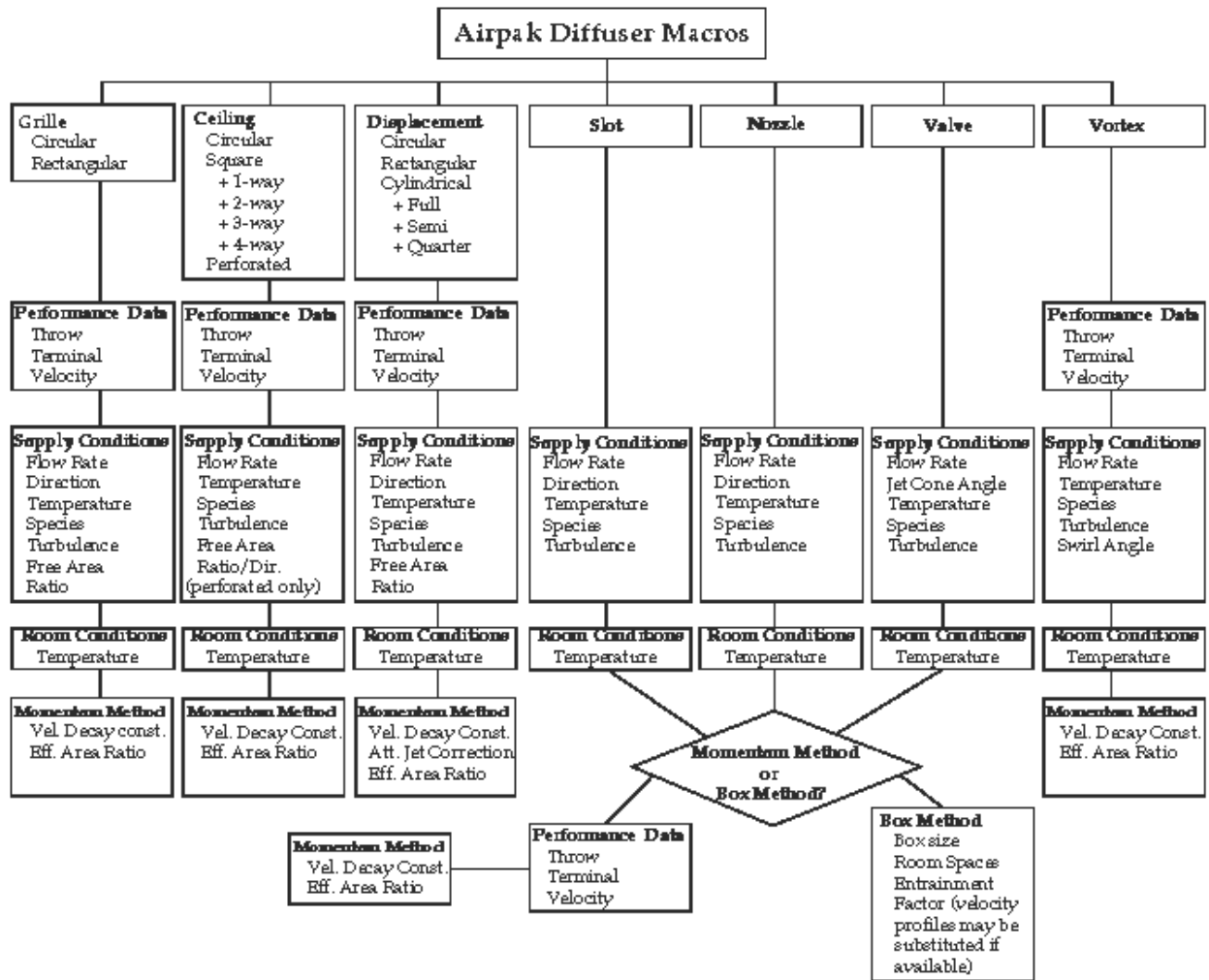
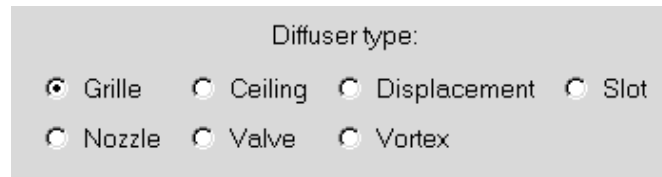


Figure 25.2.1: Diffuser Macro Flow Chart



2. Specify the geometry, position, and size of the diffuser. See Section 25.2.4 for details.
3. Specify the airflow performance data. See Section 25.2.5 for details.
4. Specify the extrusion direction and, if required, the flow direction for the diffuser. See Section 25.2.6 for details.
5. Specify any additional inputs that may be required for the diffuser. See Section 25.2.7 for details.
6. Specify the supply-air and room-air conditions. See Section 25.2.8 for details.
7. Select the method of modeling the diffuser, and specify the related parameters. See Section 25.2.9 for details.
8. Specify whether you want the diffuser to be available as one part or as individual objects:
 - If you want the diffuser to be available as one part in your **Airpak** model, select **Create internal part** and specify a name in the **Name** text entry field. The default name is **diffuser.n**, where **n** is the next sequential number among numbered diffusers.
 - If you want the diffuser to be available as individual block, source, fan, or opening objects, deselect **Create internal part**. The individual **Airpak** objects that comprise the diffuser model will have the default name of **type.n**, where **type** is the type of diffuser and **n** is the next sequential number among numbered **Airpak** objects. For example, if you create a rectangular grille diffuser using the momentum method, **Airpak** will create a fan object called **grille.1** and a source object called **grille.2**.
9. If you want to save the current diffuser specifications to be reused in your current model or to be used in a different model, click the **Save** button to open the **Macro information** dialog box (which is the same as the **File selection** dialog box described in Section 2.1.8). **Airpak** will save the specifications in a subdirectory named **diffuser_info**.

To load diffuser specifications that you have saved in this manner, click the **Load** button and select the diffuser specifications to be loaded.
10. Click **Accept** to accept the parameters entered into the **Diffuser** panel and create the diffuser.

i Objects created by the diffuser macro have hidden, performance-related information associated with the spatial location as defined in the **Diffuser** panel, and thus are not appropriate for copying or translating operations. To create several diffusers of similar geometry and performance, save the diffuser setup information and reuse it if you want to create a diffuser. The same restriction applies to running the diffuser at a different flow rate. Simply delete the previous diffuser and use the diffuser macro to create a new one.

i Objects created with the diffuser macro cannot be parameterized.

Modifying an Existing Diffuser

After the diffuser has been created and positioned in your **Airpak** model, it cannot be modified using the **Diffuser** panel. There are two ways to modify an existing diffuser:

- If you selected the **Create internal part** option in the **Diffuser** panel, you can edit the diffuser in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create internal part** option in the **Diffuser** panel, you must select and modify the object or objects that make up the diffuser.

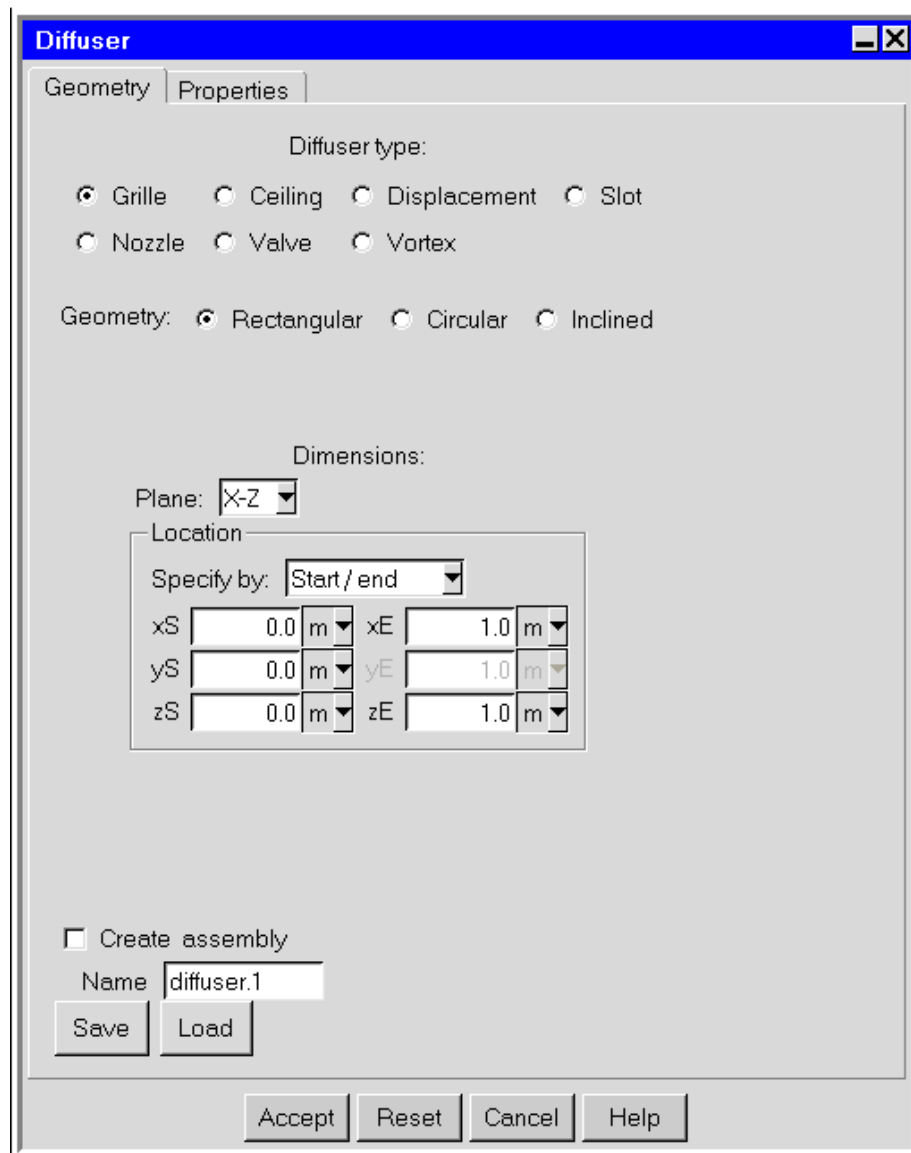


Figure 25.2.2: The Grille Diffuser Panel (Geometry Tab)

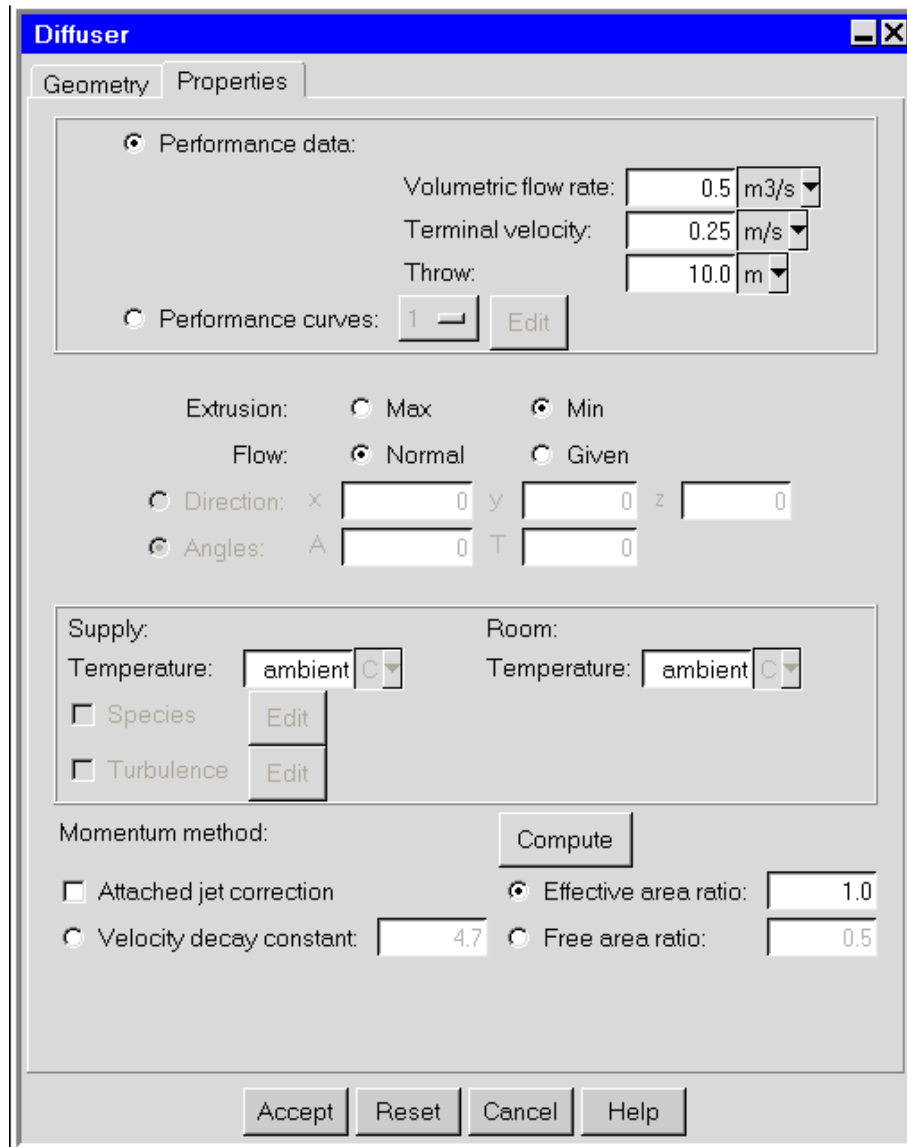


Figure 25.2.3: The Grille Diffuser Panel (Properties Tab)

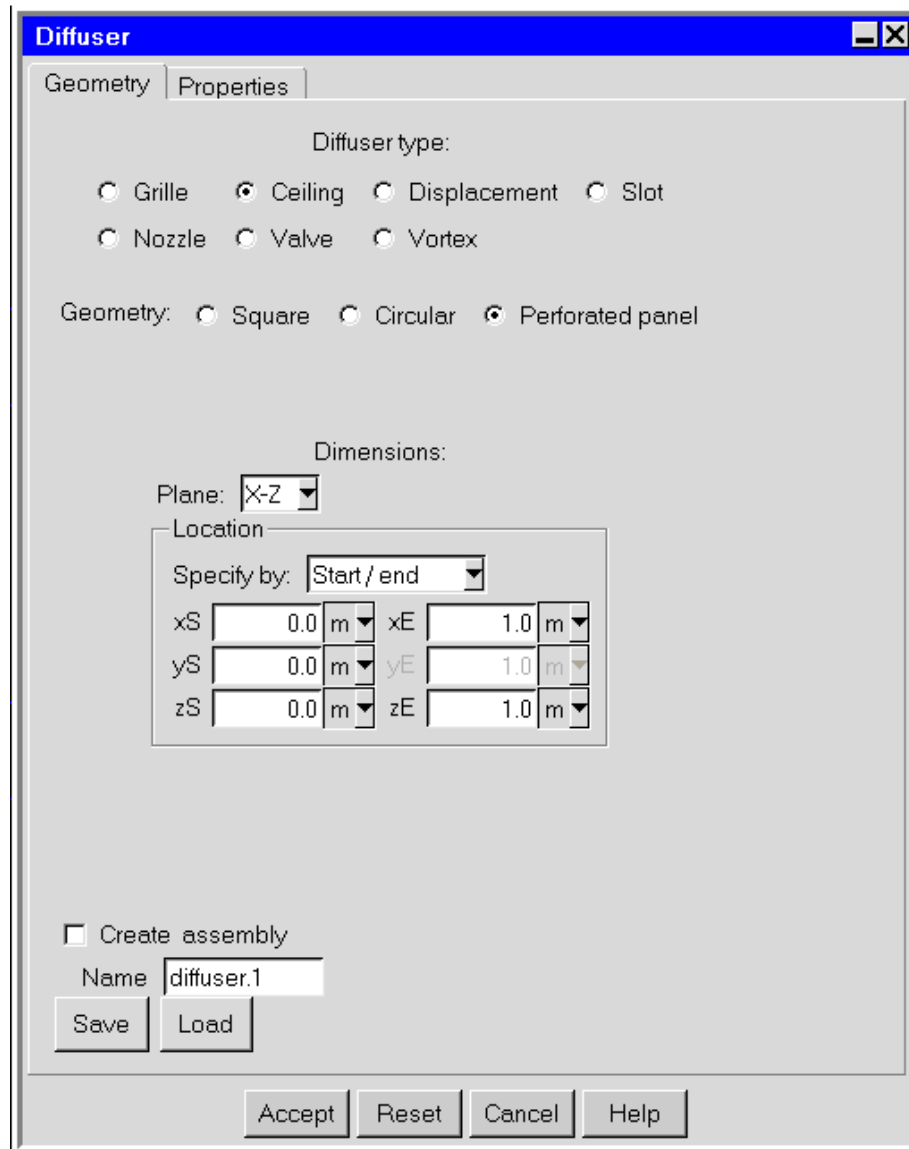


Figure 25.2.4: The Ceiling Diffuser Panel (Geometry Tab)

The screenshot shows the 'Diffuser' dialog box with the 'Properties' tab selected. The 'Performance data' section is active, showing 'Volumetric flow rate' at 0.5 m3/s, 'Terminal velocity' at 0.25 m/s, and 'Throw' at 10.0 m. Below this, 'Performance curves' shows a selection of '1' and an 'Edit' button. The 'Extrusion' section has 'Max' and 'Min' radio buttons, with 'Min' selected. The 'Flow' section has 'Normal' and 'Given' radio buttons, with 'Normal' selected. The 'Direction' section has 'x', 'y', and 'z' input fields, all set to 0. The 'Angles' section has 'A' and 'T' input fields, both set to 0. The 'Supply' and 'Room' sections each have a 'Temperature' dropdown set to 'ambient' and an 'Edit' button. The 'Momentum method' section has a 'Compute' button and two radio buttons: 'Velocity decay constant' (set to 4.7) and 'Effective area ratio' (set to 1.0). The 'Free area ratio' is set to 0.5. At the bottom are 'Accept', 'Reset', 'Cancel', and 'Help' buttons.

Diffuser

Geometry Properties

☒ Performance data:

Volumetric flow rate: 0.5 m3/s

Terminal velocity: 0.25 m/s

Throw: 10.0 m

☐ Performance curves: 1 Edit

Extrusion: ☐ Max ☒ Min

Flow: ☒ Normal ☐ Given

☐ Direction: x 0 y 0 z 0

☒ Angles: A 0 T 0

Supply: Temperature: ambient Edit

Room: Temperature: ambient Edit

☐ Species Edit

☐ Turbulence Edit

Momentum method: Compute

☐ Velocity decay constant: 4.7 ☒ Effective area ratio: 1.0

☐ Free area ratio: 0.5

Accept Reset Cancel Help

Figure 25.2.5: The Ceiling Diffuser Panel (Properties Tab)

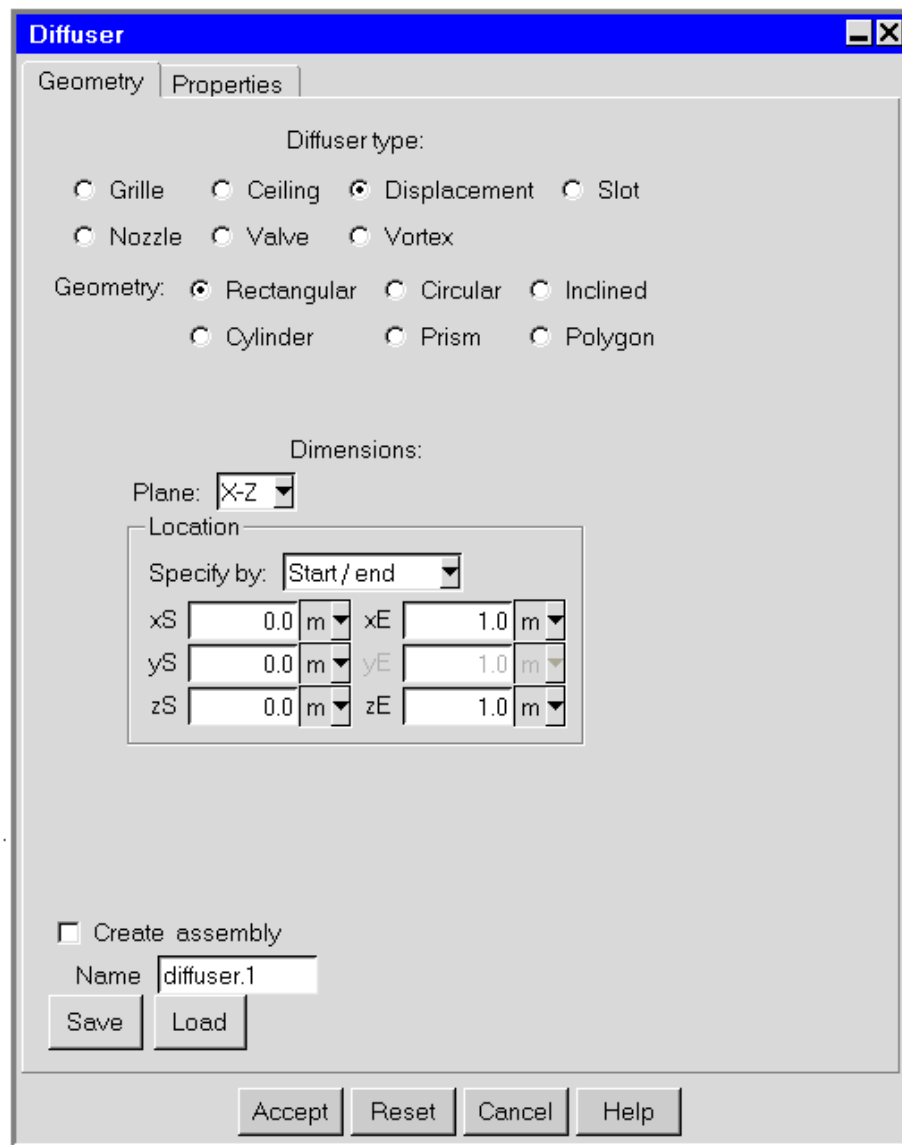


Figure 25.2.6: The Displacement Diffuser Panel (Geometry Tab)

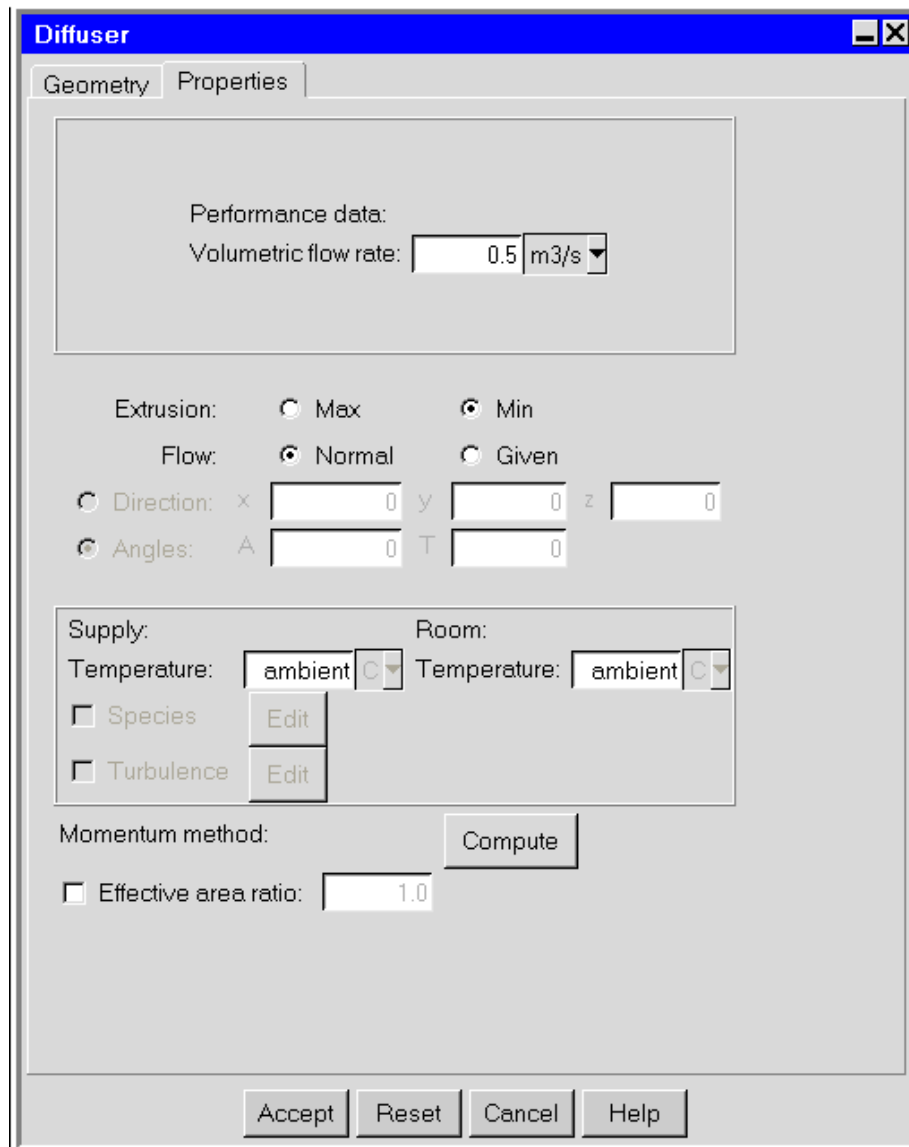


Figure 25.2.7: The Displacement Diffuser Panel (Properties Tab)

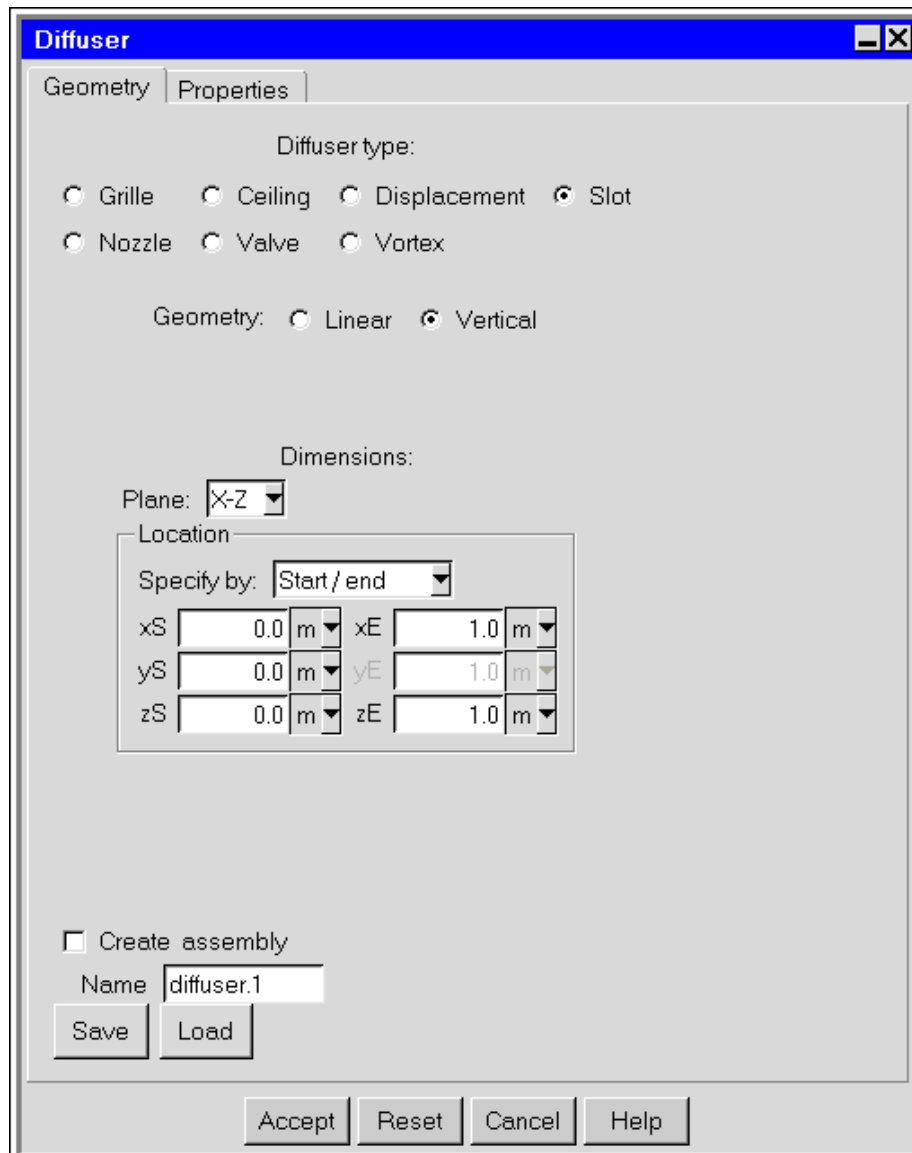


Figure 25.2.8: The Slot Diffuser Panel (Geometry Tab)

Diffuser

Geometry Properties

☒ Performance data:

Volumetric flow rate: 0.5 m³/s

Terminal velocity: 0.25 m/s

Throw: 10.0 m

☐ Performance curves: 1 Edit

Extrusion: ☐ Max ☒ Min

Flow: ☒ Normal ☐ Given

☐ Direction: x 0 y 0 z 0

☒ Angles: A 0 T 0

Supply: Temperature: ambient C

☐ Species Edit

☐ Turbulence Edit

Room: Temperature: ambient C

☒ Momentum method: Compute

☐ Velocity decay constant: 4.9

☒ Effective area ratio: 1.0

☐ Variable constant

☐ Box method: Edit

Accept Reset Cancel Help

Figure 25.2.9: The Slot Diffuser Panel (Properties Tab)

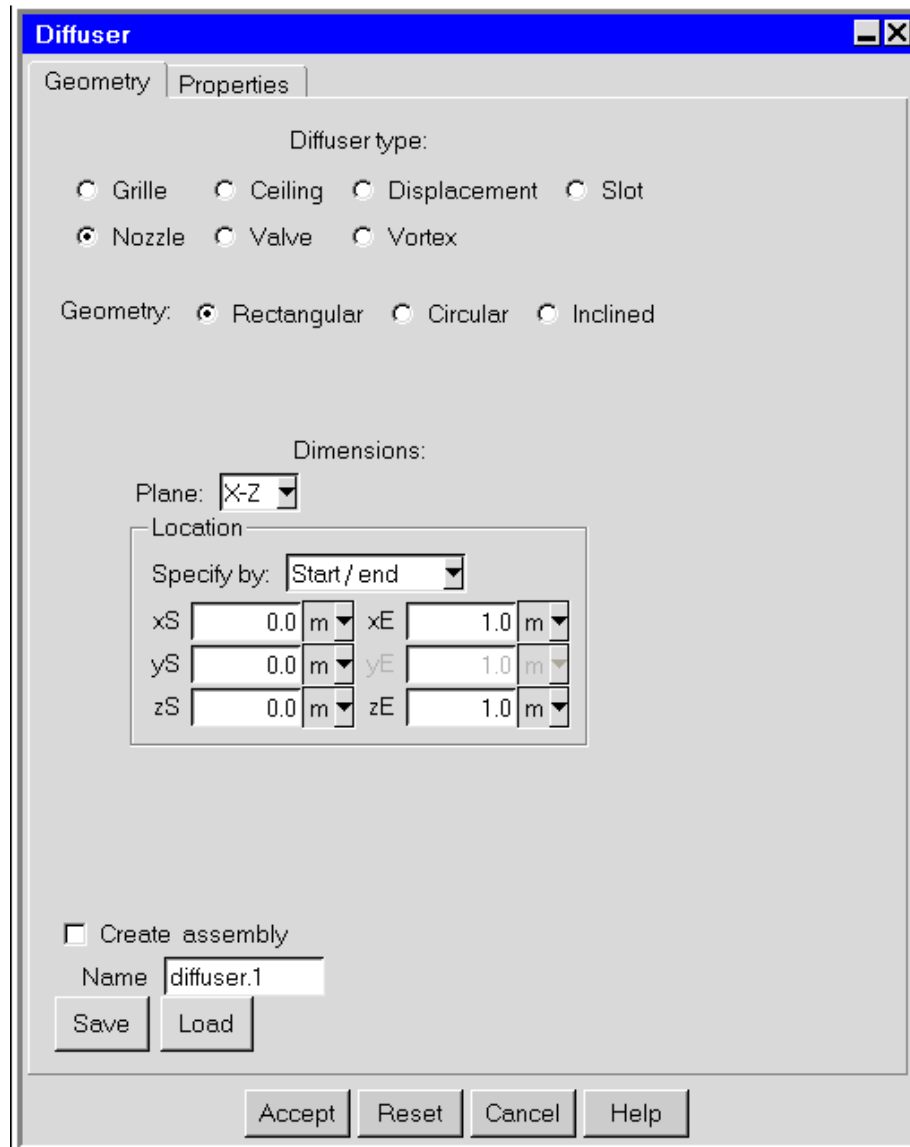


Figure 25.2.10: The Nozzle Diffuser Panel (Geometry Tab)

Diffuser [minimize] [maximize] [close]

Geometry Properties

☒ Performance data:

Volumetric flow rate:

Terminal velocity:

Throw:

☐ Performance curves:

Extrusion: ☐ Max ☒ Min

Flow: ☒ Normal ☐ Given

☐ Direction: x y z

☒ Angles: A T

Supply: Room:

Temperature: Temperature:

☐ Species

☐ Turbulence

☒ Momentum method:

☐ Velocity decay constant: ☒ Effective area ratio:

☐ Box method:

Figure 25.2.11: The Nozzle Diffuser Panel (Properties Tab)

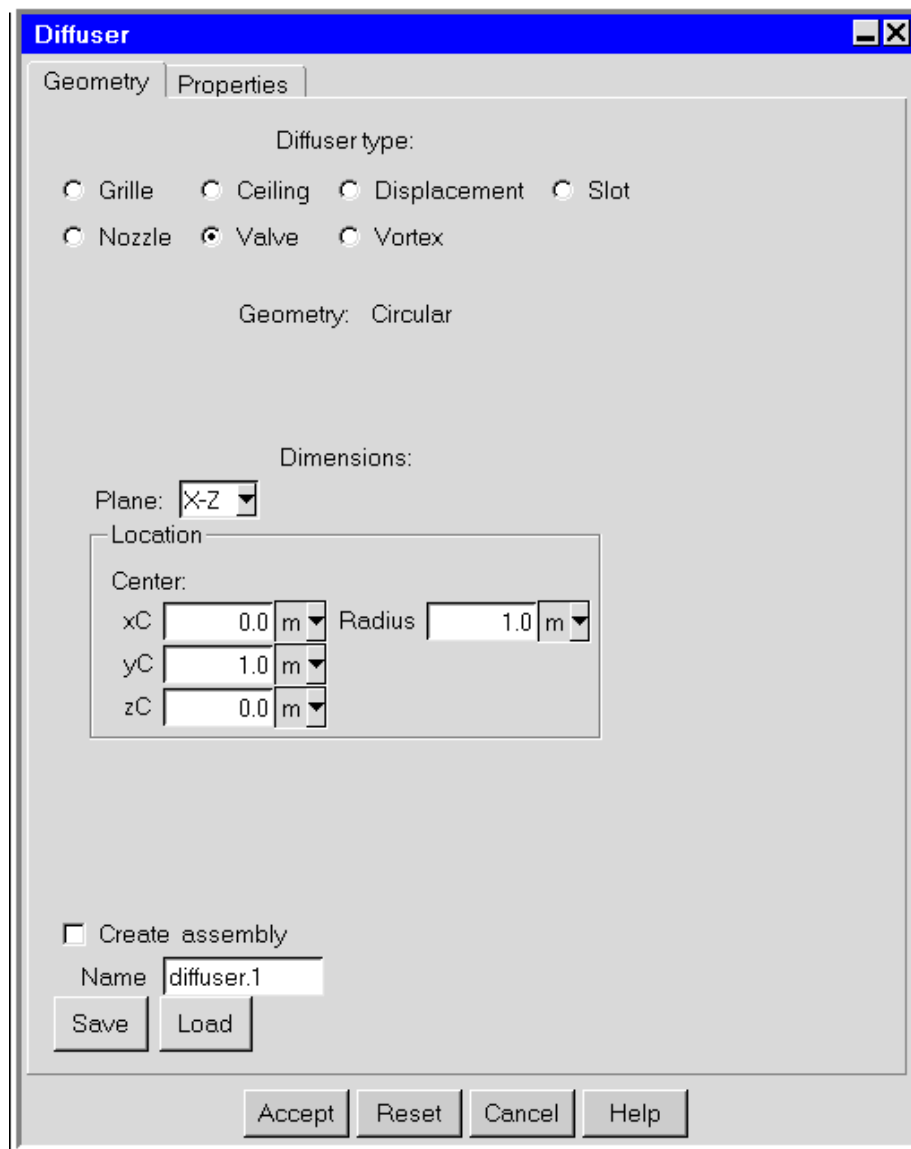


Figure 25.2.12: The Valve Diffuser Panel (Geometry Tab)

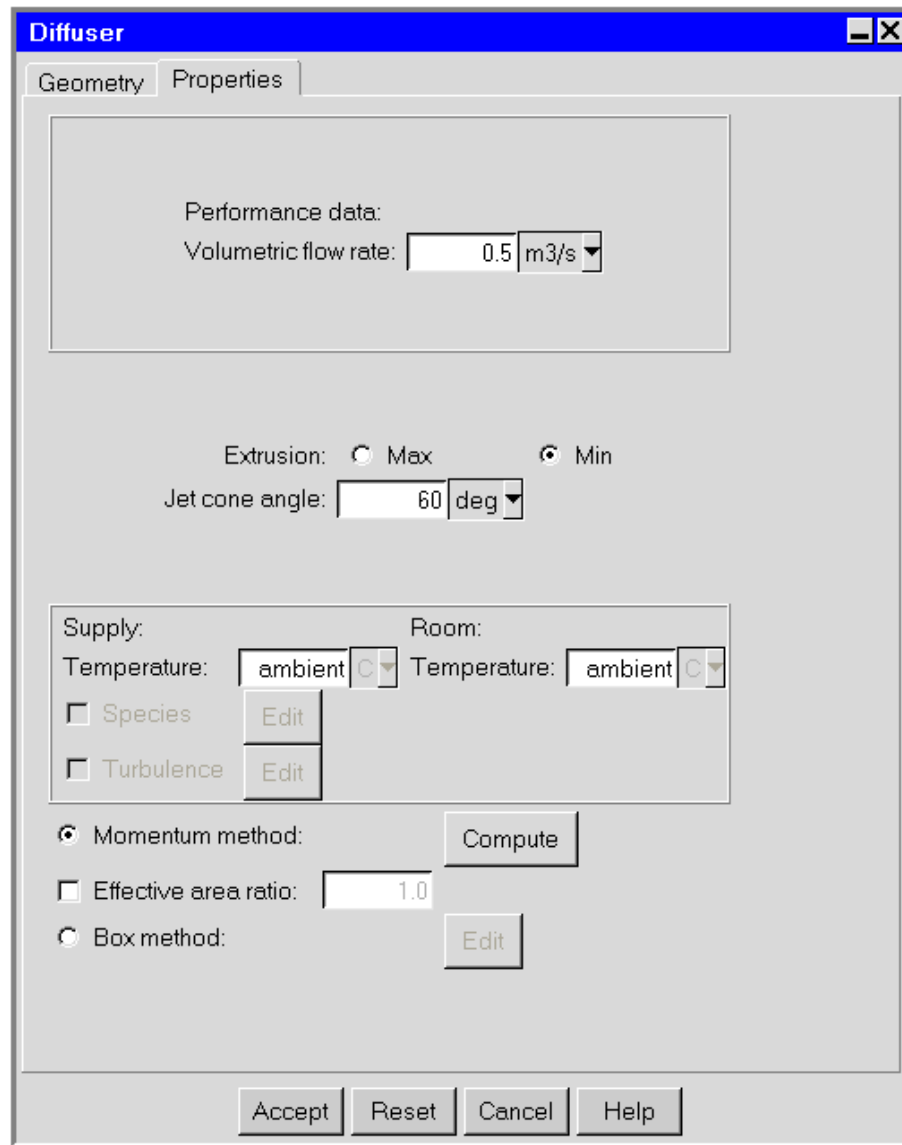


Figure 25.2.13: The Valve Diffuser Panel (Properties Tab)

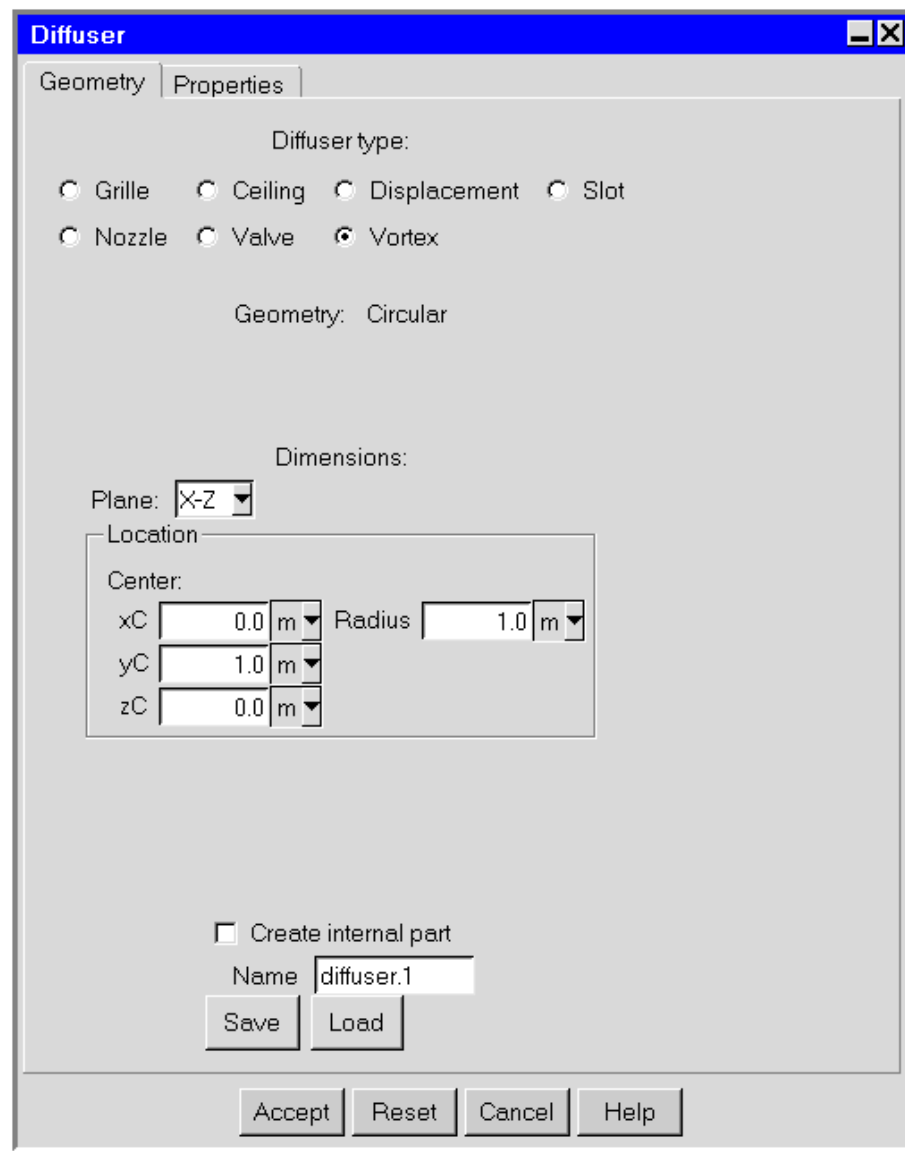


Figure 25.2.14: The Vortex Diffuser Panel (Geometry Tab)

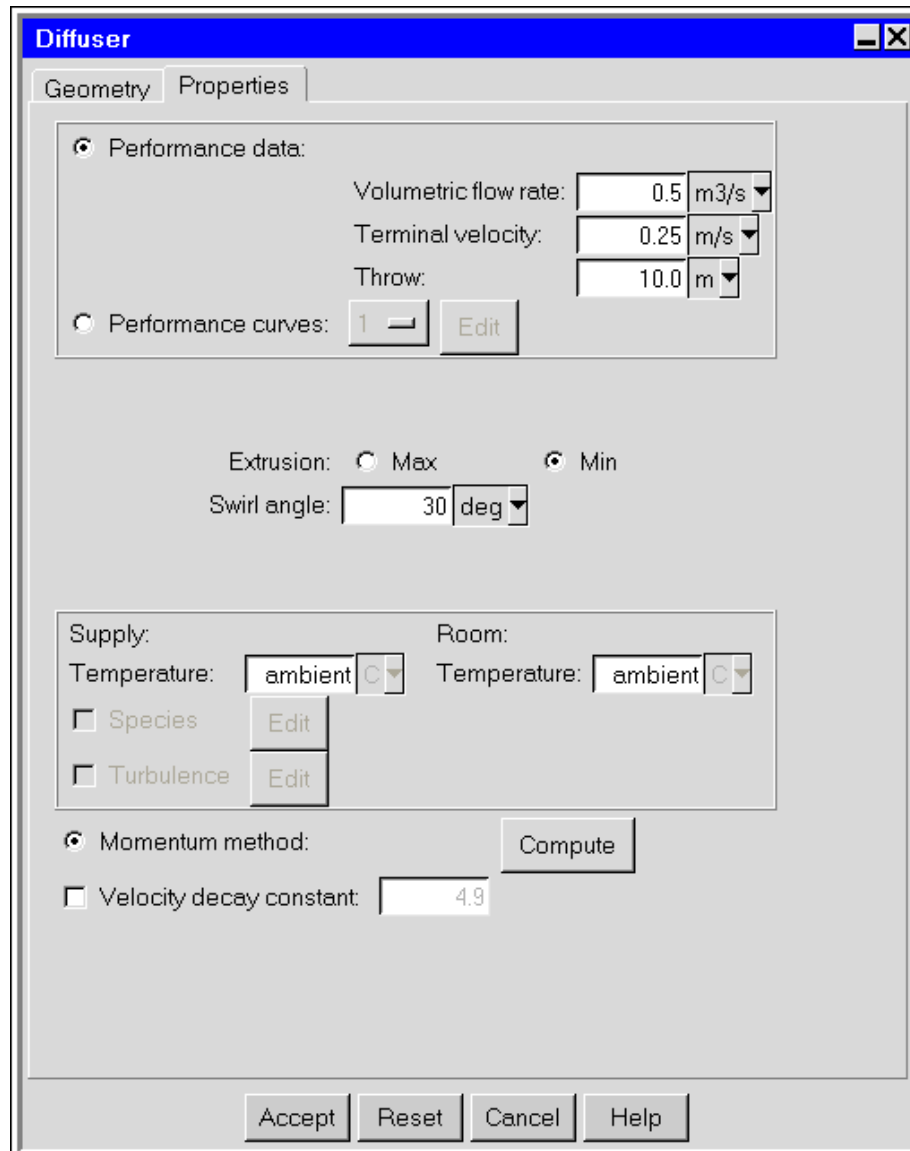


Figure 25.2.15: The Vortex Diffuser Panel (Properties Tab)

25.2.4 Geometry, Position, and Size

The geometry types that are available will depend on which type of diffuser you are using. See Figures 25.2.2–25.2.14 and below for details. The inputs for the standard **Airpak** geometries are described in Section 7.4.4. See Section 7.3.6 for details on resizing an object and Section 7.3.7 for details on repositioning an object. one

i Note that or the diffuser macro, there are two additional geometry types that you can specify for a specific type of diffuser: square geometry and perforated panel geometry. The inputs for both square and perforated panel geometry are similar to those of standard rectangular geometry. See Section 25.2.4 for details.

Geometry: ☐ Square ☒ Circular ☐ Perforated panel

Dimensions:

Plane:

Location

Center:

xC m m m

yC m

zC m

Radius m

Information about the availability of each geometry type and how **Airpak** uses the information you provide for it is provided below.

Rectangular Geometry

Rectangular geometry is available for grille, displacement, and nozzle diffusers. Depending on the modeling method you select (momentum or box), specifying rectangular geometry will instruct **Airpak** to create different objects to represent your diffuser in the model.

Momentum Method

If you select the momentum method, **Airpak** will create a 3D source object (four-sided top and bottom) with a rectangular fan object on the supply side. The height of the source object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1), which is defined as

$$A_0 = \left(\frac{K_1 Q}{u_m x} \right)^2 \quad (25.2-4)$$

where K_1 is the velocity decay coefficient, Q is the volumetric flow rate, u_m is the terminal velocity, and x is the throw.

Note that for rectangular displacement diffusers, the 3D source object will be created only if you specify an effective area ratio of less than 1. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Box Method

If you select the box method (nozzle diffusers only), **Airpak** will create a hollow prism block object with rectangular openings on five sides of the block. The opening on the supply side will be the intake, and the four openings perpendicular to the supply side will be the exhaust. See Section 25.2.1 for more information about the box method, and Section 25.2.9 for information about inputs for it.

Circular Geometry

Circular geometry is available for grille, ceiling, displacement, nozzle, valve, and vortex diffusers. Depending on the modeling method you select (momentum or box), specifying circular geometry will instruct **Airpak** to create different objects to represent your diffuser in the model.

Momentum Method

If you select the momentum method, **Airpak** will create different objects depending on the type of diffuser you selected:

- Grille, displacement, or nozzle diffuser: **Airpak** will create a cylindrical source object with a circular fan object on the supply side. The height of the source object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Note that for circular displacement diffusers, the cylindrical source object will be created only if you specify an effective area ratio of less than 1.

- Ceiling or vortex diffuser: **Airpak** will create a cylindrical fan object. The height of the cylindrical fan object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).
- Valve diffuser: **Airpak** will create a circular opening object. If you have specified an effective area ratio of less than 1, **Airpak** will also create a circular partition object. The radius of the partition object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Box Method

If you select the box method (nozzle or valve diffusers only), **Airpak** will create a cylindrical block object, a cylindrical fan object, and a circular opening object. The fan object will occupy the same space as the block object, with the block object acting as the support structure. The opening object will be on the supply side of the block object. See Section 25.2.1 for more information about the box method, and Section 25.2.9 for information about inputs for it.

Inclined Geometry

Inclined geometry is available for grille, displacement, and nozzle diffusers, for which you must use the momentum method. Specifying inclined geometry will instruct **Airpak** to create a 3D source object (four-sided top and bottom) with an inclined rectangular fan object on the supply side. The height of the source object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Note that for inclined displacement diffusers, the 3D source object will be created only if you specify an effective area ratio of less than 1. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Cylinder Geometry

Cylinder geometry is available for displacement diffusers, for which you must use the momentum method. Depending on the number of active corners you specify for the diffuser, specifying cylinder geometry will instruct **Airpak** to create different objects to represent your diffuser in the model. See Section 25.2.7 for information about specifying active corners. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Four Active Corners

If you specify the maximum of four active corners, **Airpak** will create a 3D fan object with a 16-sided top and bottom. If you have specified an effective area ratio of less than 1, **Airpak** will also create a 3D source object (four-sided top and bottom) on each of the 16 equatorial sides of the fan. The height of the source objects is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Less Than Four Active Corners

If you specify one, two, or three active corners, **Airpak** will create objects to represent one, two, or three quarters of a cylinder. For each corner selected, **Airpak** will create a

3D “pie piece” block object with four inclined rectangular fan objects on the four outer sides of the block. If you have specified an effective area ratio of less than 1, **Airpak** will also create a 3D source object (four-sided top and bottom) on the four outer sides of the block. The height of the source objects is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Prism Geometry

Prism geometry is available for displacement diffusers, for which you must use the momentum method. Specifying prism geometry will instruct **Airpak** to create a prism block object with a rectangular fan object on the supply side. If you have specified an effective area ratio of less than 1, **Airpak** will also create a 3D source object (four-sided top and bottom) on the supply side. See Section 25.2.7 for information about specifying the supply side for prism geometry. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Polygon Geometry

Polygon geometry is available for displacement diffusers, for which you must use the momentum method. Specifying polygon geometry will instruct **Airpak** to create a 3D fan object with a top and bottom that have the number of vertices specified by you. If you have specified an effective area ratio of less than 1, **Airpak** will also create a 3D source object (four-sided top and bottom) on each equatorial side of the fan. The height of the source objects is determined by the effective area of the diffuser (A_0 in Equation 25.2-1). See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Perforated Panel Geometry

Perforated panel geometry is available for ceiling diffusers, for which you must use the momentum method. Specifying perforated panel geometry will instruct **Airpak** to create a 3D source object (four-sided top and bottom) with a rectangular fan object on the supply side. The height of the prism source object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1). See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Square Geometry

Square geometry is available for ceiling diffusers, for which you must use the momentum method. Depending on the number of active sides you specify for the diffuser, specifying square geometry will instruct **Airpak** to create different objects to represent your diffuser in the model. See Section 25.2.7 for information about specifying active sides. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Four Active Sides

If you specify the maximum of four active sides, **Airpak** will create a 3D polygon fan object. The height of the polygon fan object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Less Than Four Active Sides

If you specify one, two, or three active sides, **Airpak** will create a prism block object with rectangular intake fan objects on the specified active sides of the block object. The fan objects will be perpendicular to the supply side of the block.

Linear Geometry

Linear geometry is available for slot diffusers. Depending on the modeling method you select (momentum or box) and number of active sides you specify for the diffuser, specifying linear geometry will instruct **Airpak** to create different objects to represent your diffuser in the model. See Section 25.2.7 for information about specifying active sides. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Momentum Method with Four Active Sides

If you select the momentum method and specify the maximum of four active sides, **Airpak** will create a 3D polygon fan object. The height of the polygon fan object is determined by the effective area of the diffuser (A_0 in Equation 25.2-1).

Momentum Method with Less Than Four Active Sides

If you select the momentum method and specify one, two, or three active sides, **Airpak** will create a prism block object with rectangular intake fan objects on the specified active sides of the block object. The fan objects will be perpendicular to the supply side of the block.

Box Method

If you select the box method, **Airpak** will create a hollow prism block object with rectangular openings on up to five sides of the block. The number of openings will depend on the number of active sides that you specify. The opening on the supply side will be the exhaust, and the other openings perpendicular to the supply side will be the intake. See Section 25.2.1 for more information about the box method, and Section 25.2.9 for information about inputs for it.

Vertical Geometry

Vertical geometry is available for slot diffusers. Depending on the modeling method you select (momentum or box), specifying vertical geometry will instruct **Airpak** to create different objects to represent your diffuser in the model.

Momentum Method

If you select the momentum method, **Airpak** will create a 3D source object (four-sided top and bottom) with a rectangular fan object on the supply side. The height of the source object is determined by the effective area of the diffuser, A_0 , which is defined in Equation 25.2-4. The air flow will be delivered normal or at a specified angle to the surface on which the diffuser is located. See Section 25.2.1 for more information about the momentum method, and Section 25.2.9 for information about inputs for it.

Box Method

If you select the box method, **Airpak** will create a hollow prism block object with rectangular openings on five sides of the block. The opening on the supply side will be the intake, and the four openings perpendicular to the supply side will be the exhaust. See Section 25.2.1 for more information about the box method, and Section 25.2.9 for information about inputs for it.

clearpage

25.2.5 Specifying Airflow Performance Data

There are two ways you can specify performance data for the diffuser:

- Specify data for a single point.
- Specify a curve or curves.

☒ Performance data:

 Volumetric flow rate: 0.5 m³/s

 Terminal velocity: 0.25 m/s

 Throw: 10.0 m

☐ Performance curves: 1 Edit



For displacement and valve diffusers, only the first method is available.

Specifying Performance Data for a Single Point

To specify performance data for a single point, select **Performance data** in the **Diffuser** panel and specify the following values:

- **Volumetric flow rate** of air through the diffuser (Q in Equation 25.2-2)
- **Terminal velocity** (u_m in Equation 25.2-2) that corresponds to the air jet centerline velocity value for which the throw is specified
- **Throw distance** of the air jet from the diffuser (x in Equation 25.2-2) at which the air jet centerline velocity equals the specified terminal velocity

Note that, for displacement and valve diffusers, only the volumetric flow rate is specified.

Specifying Performance Curves

To specify performance data for a series of points, select **Performance curves** in the **Diffuser** panel. For grille, ceiling, slot, nozzle, and vortex diffusers, you can define up to three curves using the **Performance data** panel (Figure 25.2.16). As noted above, this specification method is not available for displacement and valve diffusers.

The procedure for defining a performance curve is as follows:

1. Specify the number of the curve (1, 2, or 3) in the drop-down list above the **Edit** button.
2. Click **Edit** to open the **Performance data** panel.
3. Specify a list of values in the **Performance data** panel. A curve specification consists of a list of volumetric flow rate/throw sets, with one volumetric flow rate value and one throw value in each set. It is important to give the numbers in sets of two, but the spacing between numbers is not important. The sets must be entered in order of increasing volumetric flow rate so that the data can be interpolated when required.
4. Specify the **Terminal velocity** that corresponds to the air jet centerline velocity value for which the throw is desired. The throw distance of an air jet is a function of the volumetric flow rate for a particular terminal velocity.
5. Select the appropriate **Volumetric flow rate units** for your listed values from the unit definition list.
6. Select the appropriate **Throw units** for your listed values from the unit definition list.

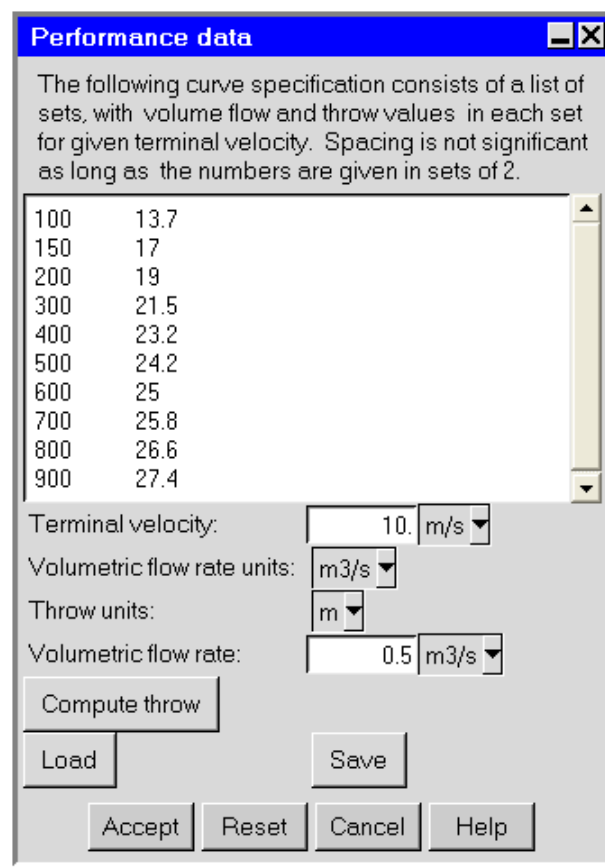


Figure 25.2.16: The Performance data Panel

7. Click **Accept**. This will store the values and close the Performance data panel. Airpak will interpolate the data you provided for the volumetric flow rate and the throw to create a profile for the diffuser.
8. Click **Edit** in the Diffuser panel to re-open the Performance data panel.
9. In the Performance data panel, specify the **Volumetric flow rate** for the diffuser.
10. Click **Compute throw** to compute the throw distance value for the diffuser based on the performance curve. In the Diffuser panel, a set of **Performance data** will be updated (**Volumetric flow rate**, **Terminal velocity**, and **Throw**). The Message window will also print this information.

Loading a Previously Defined Curve

To load a previously defined profile, click on **Load**. (See Section 28.2.8 for details on saving contour data and using them as a profile.) This will open the Load curve file

selection dialog box. Select the file containing the profile data and click **Accept**. See Section 2.1.8 for details on selecting a file. If you know the units used in the profile data you are loading, you should select the appropriate units in the **Performance data** panel before you load the profile.

Saving a Curve

To save a profile, click on **Save**. This will open the **Save curve** dialog box, in which you can specify the filename and directory to which the profile data are to be saved.

25.2.6 Specifying the Extrusion and Flow Directions

Specifying the Extrusion Direction

For all diffuser types, you will need to specify the direction of the **Extrusion**. For 3D diffusers, the height, or extrusion, is usually calculated using the effective area (A_0 in Equation 25.2-1). The extrusion direction is the direction in which the **Airpak** objects are extended. Depending on the location of the diffuser in the room, you need to make sure that the diffuser is extruded into the room volume and not out of the room volume.

The image shows a dialog box for specifying extrusion and flow directions. It contains the following controls:

- Extrusion:** Two radio buttons, **Max** and **Min**. **Min** is selected.
- Flow:** Two radio buttons, **Normal** and **Given**. **Normal** is selected.
- Direction:** A section with three input fields for **x**, **y**, and **z** coordinates. Each field has a small icon to its left (a circle with a dot for x and y, and a circle with a cross for z) and a value of **0** is entered in each field.
- Angles:** A section with two input fields for **A** and **T** angles. Each field has a small icon to its left (a circle with a dot for A, and a circle with a cross for T) and a value of **0** is entered in each field.

There are two options for the **Extrusion** direction:

Max specifies that the extrusion points in the direction of increasing coordinate value.

For example, if you have positioned a grille diffuser on a surface on the x - y plane with the positive z direction pointing into the room, you should select **Max**.

Min specifies that the extrusion points in the direction of decreasing coordinate value.

For example, if you have positioned a grille diffuser on a surface on the x - z plane with the negative y direction pointing into the room, you should select **Min**.

Note that you will not specify the extrusion direction in this way for a cylindrical displacement diffuser. See Section 25.2.7 for details.

Specifying the Flow Direction

If you are modeling grille, slot, nozzle, or certain types of ceiling (perforated panel geometry) or displacement (rectangular, circular, or inclined geometry) diffusers, you will need to specify the **Flow** direction. There are two options:

- If the air flows into the room normal to the diffuser, select **Normal**.
- To specify the flow angle of the fluid entering the room through the diffuser, select **Given**. There are two specification methods:

Direction allows you to specify the direction vector (x, y, z) for the flow. Only the direction of the vector is used by **Airpak**; the magnitude is ignored.

Angles allows you to specify the direction of the flow entering the room in terms of angles relative to the inward-pointing normal direction. You can specify two angles, **A** (α) and **T** (θ), to account for incoming flow not aligned with the normal direction. If the normal inward vector points in the positive direction, then α and θ must have values in the range $-90^\circ < \text{angle} < 90^\circ$, with $\alpha=\theta=0^\circ$ representing flow in the positive normal direction. If the normal inward vector points in the negative direction, then α and θ must have values in the range $90^\circ < \text{angle} < 270^\circ$, with $\alpha=\theta=180^\circ$ representing flow in the negative normal direction. Since the diffuser may be oriented in any one of three planes (x - y , y - z , or x - z), the roles of α and θ will vary depending on the orientation of the plane of the diffuser.

If you are modeling valve, vortex, or certain types of ceiling (square geometry) or displacement (cylindrical or prism geometry) diffusers, you will use other methods to specify the flow direction. See Section 25.2.7 for details. If you are modeling polygon displacement diffusers or circular ceiling diffusers, the flow direction will be normal to the diffuser.

25.2.7 Additional Inputs for Specific Types of Diffusers

For specific types of diffusers, there are additional inputs that are required in the Diffuser panel. These special inputs are described below.

Specifying Active Corners for a Cylindrical Displacement Diffuser

If you have selected cylindrical geometry for a displacement diffuser, you must specify the active **Corners** of the diffuser. You must specify a minimum of one active corner and may specify up to four. Depending on the plane you selected for the diffuser, the four corner options (+ +, - +, - -, + -) will represent the four quadrants of a complete cylinder with its center at the location that you have specified.

Corners:

☒ ++ ☒ -+ ☒ -- ☒ +- ☐ - -

— +, — —, and + — options represent the remaining three quadrants in a counter-clockwise direction (upper left, lower left, lower right), respectively.

Specifying the Supply Side for a Prism Displacement Diffuser

If you have selected prism geometry for a displacement diffuser, you must specify the Side from which air will be supplied to the room (Xmin, Xmax, Ymin, Ymax, Zmin, or Zmax). Xmax, Ymax, and Zmax refer to the sides of the diffuser facing in the positive coordinate directions. Xmin, Ymin, and Zmin refer to the sides of the diffuser facing in the negative coordinate directions.

Side:

☒ Xmin ☐ Xmax ☐ Ymin ☐ Ymax ☐ Zmin ☐ Zmax

Specifying Active Sides for Ceiling and Slot Diffusers

If you have selected square geometry for a ceiling diffuser or linear geometry for a slot diffuser, you must specify the **Active sides** of the diffuser. You must specify a minimum of one active side and may specify up to four. Depending on the plane you selected for the diffuser, four different sides of the diffuser (Xmin, Xmax, Ymin, Ymax, Zmin, or Zmax) will be available for you to define as active sides. Xmax, Ymax, and Zmax refer to the sides of the diffuser facing in the positive coordinate directions. Xmin, Ymin, and Zmin refer to the sides of the diffuser facing in the negative coordinate directions.

Active sides:

☒ Xmin ☒ Xmax ☐ Ymin ☐ Ymax ☒ Zmin ☒ Zmax

For example, if you have specified an x - z plane, the Ymin and Ymax options will not be relevant, and you must choose from one to four sides from Xmin, Xmax, Zmin, and Zmax.

Special Inputs for Valve and Vortex Diffusers

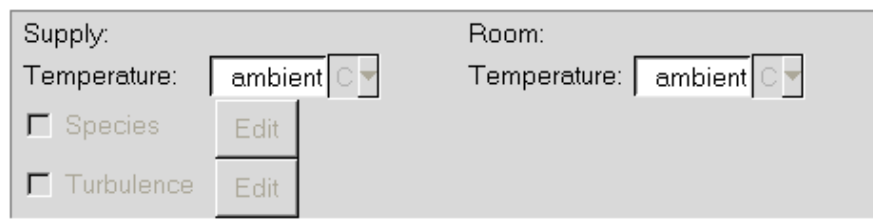
If you have selected a valve diffuser, you will need to specify the **Jet cone angle** instead of the flow direction.

If you have selected a vortex diffuser, you will need to specify the **Swirl angle** instead of the flow direction.

25.2.8 Specifying Supply-Air and Room-Air Conditions

Specifying Supply-Air and Room-Air Temperatures

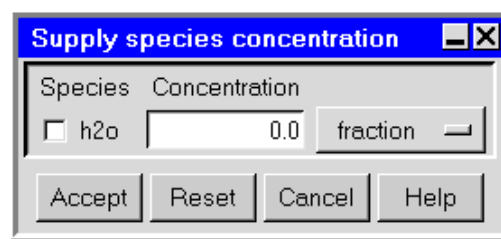
For all diffuser types, Airpak calculates a local Archimedes number (Ar) to give an indication of the relative importance of buoyant forces. For Airpak to calculate the local value of Ar , you will need to estimate the temperatures of both the supply air and the room air. Specify the temperature of the supply air in the **Temperature** field under **Supply**, and the temperature of the room air in the **Temperature** field under **Room**.



The screenshot shows a panel with two main sections: 'Supply' and 'Room'. Each section has a 'Temperature' field with a dropdown menu currently set to 'ambient'. Below the 'Supply' section, there are two checkboxes: 'Species' and 'Turbulence', each followed by an 'Edit' button.

Specifying Supply-Air Species Parameters

If you have enabled the **Species** option in the **Problem setup** panel, you can specify species parameters for the diffuser supply air using the **Supply species concentration** panel (Figure 25.2.17). To open the **Supply species concentration** panel, select the **Species** option in the **Diffuser** panel and click **Edit**.



The screenshot shows a dialog box titled 'Supply species concentration'. It has a table with two columns: 'Species' and 'Concentration'. The first row shows 'h2o' in the 'Species' column and '0.0' in the 'Concentration' column. To the right of the 'Concentration' field is a dropdown menu set to 'fraction'. At the bottom of the dialog are four buttons: 'Accept', 'Reset', 'Cancel', and 'Help'.

Figure 25.2.17: The Supply species concentration Panel

See Chapter 21 for details on modeling species transport.

Specifying Supply-Air Turbulence Parameters

If you selected the two-equation or RNG k - ϵ model in the **Basic parameters** panel, you can specify turbulence parameters for the diffuser supply air using the **Turbulence equation parameters** panel (Figure 25.2.18). To open the **Turbulence equation parameters** panel, select **Turbulence** in the **Diffuser** panel and click **Edit**.

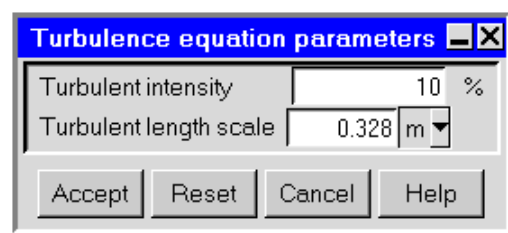


Figure 25.2.18: The Turbulence equation parameters Panel

Specify the **Turbulent intensity** as a percentage, and the **Turbulent length scale** in the appropriate units of length. See Section 6.4.3 for details on determining turbulence parameters.

25.2.9 Specifying the Modeling Method

For all diffuser types, you will need to select the method of modeling the diffuser. See Section 25.2.1 for more information about diffuser modeling methods.

There are two methods of modeling diffusers in **Airpak**.

Momentum method specifies the momentum method for modeling the diffuser. This option is available for all diffuser types, and is recommended for grille, ceiling, displacement, and vortex diffusers. See Section 25.2.4 for details about the types of **Airpak** objects that are created when you choose the momentum method for specific diffusers. See below for information about input options for the momentum method.

Box method specifies the box method for modeling the diffuser. This option is available and recommended for slot, nozzle, and valve diffusers. See Section 25.2.4 for details about the types of **Airpak** objects that are created when you choose the box method for specific diffusers. See below for information about input options for the box method.

Inputs for the Momentum Method

The default approach in the momentum method is to use jet theory and published correlations to determine the effective area. You can modify this approach in several ways:

- Specify that corrections be applied
- Specify parameters that are used in the correlations for some diffusers
- Specify aspects of the diffuser performance that you want to control yourself

Momentum method:

☐ Attached jet correction

☒ Velocity decay constant:

☐ Effective area ratio:

☐ Free area ratio:

The options for controlling the momentum method are as follows:

Attached jet correction specifies a correction factor for the default value of K_1 . This option is available for grille diffusers and accounts for jets from high side-wall diffusers attached to the ceiling.

Velocity decay constant directly specifies a value for K_1 that overrides the default value of the velocity decay constant. This option is available for grille, ceiling, slot, nozzle, and vortex diffusers.

Variable constant calculates a value for K_1 based on the estimated discharge velocity. This option is available only for slot diffusers with linear geometry.

Effective area ratio directly specifies the ratio of the effective area (A_0 in Equation 25.2-1) to the total area of the diffuser. This option is available for grille, displacement, slot, nozzle, and valve diffusers. Specifying the effective area ratio, which can be supplied by the diffuser manufacturer or obtained from your own diffuser performance measurements, allows you to bypass calculating the effective area based on jet theory and correlations.

Free area ratio directly specifies the free area ratio, which is a parameter in correlations for the velocity decay constant K_1 for grille and perforated-panel ceiling diffusers.

The **Compute** button instructs **Airpak** to calculate values for various parameters and display them in the **Message** window. These parameters include the discharge velocity, effective area, Archimedes number, and others that depend on the momentum method options you have selected.

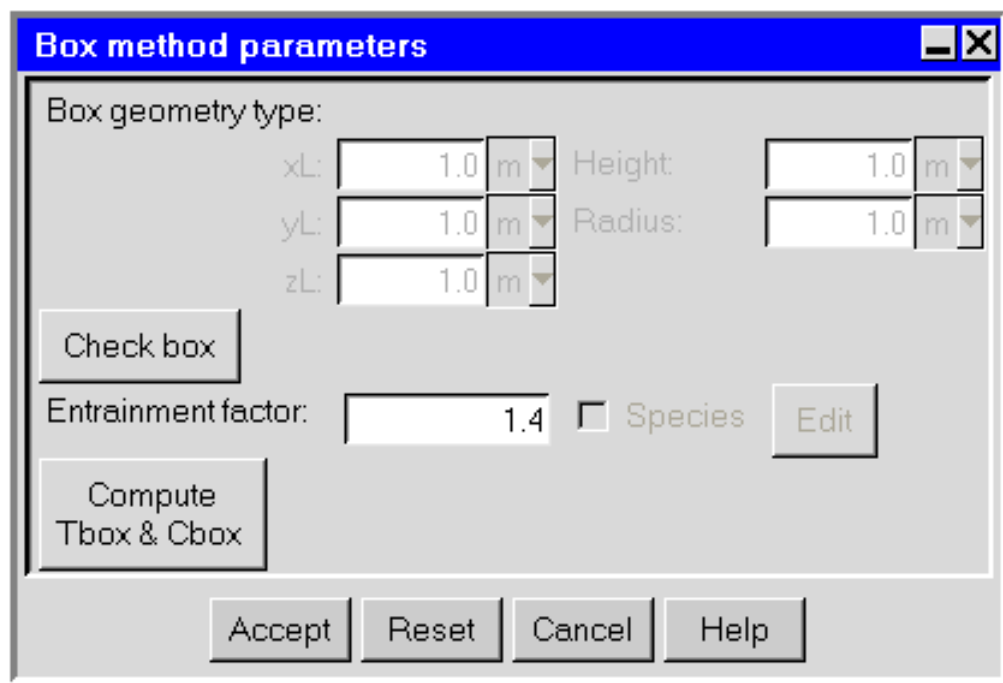


Figure 25.2.19: The Box method parameters Panel

Inputs for the Box Method

You can specify parameters for the box method using the **Box method parameters** panel (Figure 25.2.19). To open the **Box method parameters** panel for a slot, nozzle, or valve diffuser, select the **Box method** option in the **Diffuser** panel and click **Edit**.

The procedure for specifying box method parameters for your diffuser is as follows:

1. For rectangular diffusers, enter values for the lengths of the sides (x_L , y_L , z_L) of the box under **Box geometry type**. For circular diffusers, enter values for the **Height** and **Radius** of the “box”.
2. Click the **Check box** button to check whether the box dimensions are large enough to adequately model the diffuser.

i The Archimedes number (Ar) cannot be 0 if you are using the box method. If you get an **Error** dialog box, click **OK** and adjust either the room temperature or the supply air temperature so that they are different values. See Section 25.2.8 for details.

The **Message** window will inform you if your box dimensions are too small or too large. Once you have specified adequate box dimensions, click **Check box** and the **Message** window will display **Check OK**.

3. Specify the **Entrainment factor**, which is the ratio of the volumetric flow rate of the air jet relative to the volumetric flow rate of air exiting the diffuser.
4. If you have enabled the **Species** option in the **Problem setup** panel, you can specify species parameters for the diffuser exhaust air using the **Exhaust species concentration** panel (Figure 25.2.20). To open the **Exhaust species concentration** panel, select **Species** in the **Box method parameters** panel and click **Edit**.

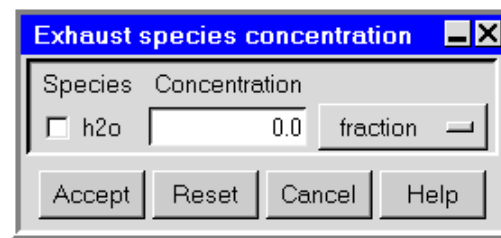


Figure 25.2.20: The Exhaust species concentration Panel

See Chapter 21 for details on modeling species transport.

5. Click the **Compute Tbox & Cbox** button to compute the temperature and, if required, the mass fraction of species that exit the box. This accounts for mixing between the supply air and the entrained room air. The **Message** window will display values for **Tbox** (temperature) and **Cbox** (mass fraction).
6. Click **Accept** in the **Box method parameters** panel.

25.3 Geometry Macros

The following geometry-related macros are described in this section:

- Section 25.3.1: Polygonal Ducts
- Section 25.3.3: Closed Box
- Section 25.3.4: 1/4 Polygonal Cylinder
- Section 25.3.5: Cylinder Plates
- Section 25.3.6: Cylindrical Enclosure
- Section 25.3.7: Polygonal Circle
- Section 25.3.8: Polygonal Cylinder
- Section 25.3.9: Hemisphere

25.3.1 Polygonal Ducts

Polygonal ducts are created using wire objects (described in Chapter 19). At least three wires are necessary to construct a polygonal duct. The number of walls (sides) of the duct is equal to the number of wires used in its construction. The macro uses the non-uniform polygonal block object with a zero height to construct the walls (sides) of the duct from the wires.

25.3.2 Adding a Polygonal Duct to Your Airpak Model

To include a polygonal duct in your Airpak model, select **Quick geometry/approximations** and then **Polygonal ducts** in the **Macros** menu. This will open the **Polygonal ducts** panel, shown in Figure 25.3.1.

Macros→Quick geometry/approximations→Polygonal ducts

Once you have created the wires that will be used to construct the polygonal duct, the procedure for adding a polygonal duct to your model is as follows:

1. Select the wires to be used to create the polygonal duct in the **Available** list. The names of the wires that will be used will be displayed in the **Used** list.
2. If you want to close the duct at the starting end, select **Close starting end**.
3. If you want to close the duct at the finishing end, select **Close finishing end**.
4. If you want the polygonal duct to be available as an assembly in your Airpak model, select **Create assembly**. If you want the polygonal duct to be available as individual objects, deselect **Create assembly**. The individual block elements that comprise the walls (sides) of the polygonal duct will be created with the default name **duct.n**, where **n** is the next sequential number among numbered block objects in the duct.
5. If you have created an assembly and want to change the name of the polygonal duct, enter a new name in the **Name** text entry field. The default name is **polyduct.n**, where **n** is the next sequential number among numbered polygonal ducts.
6. Click **Accept** to accept the parameters entered into the **Polygonal ducts** panel and create the polygonal duct.

Modifying an Existing Polygonal Duct

After the polygonal duct has been created and positioned in your Airpak model, it cannot be modified using the **Polygonal ducts** panel. There are two ways to modify an existing polygonal duct:

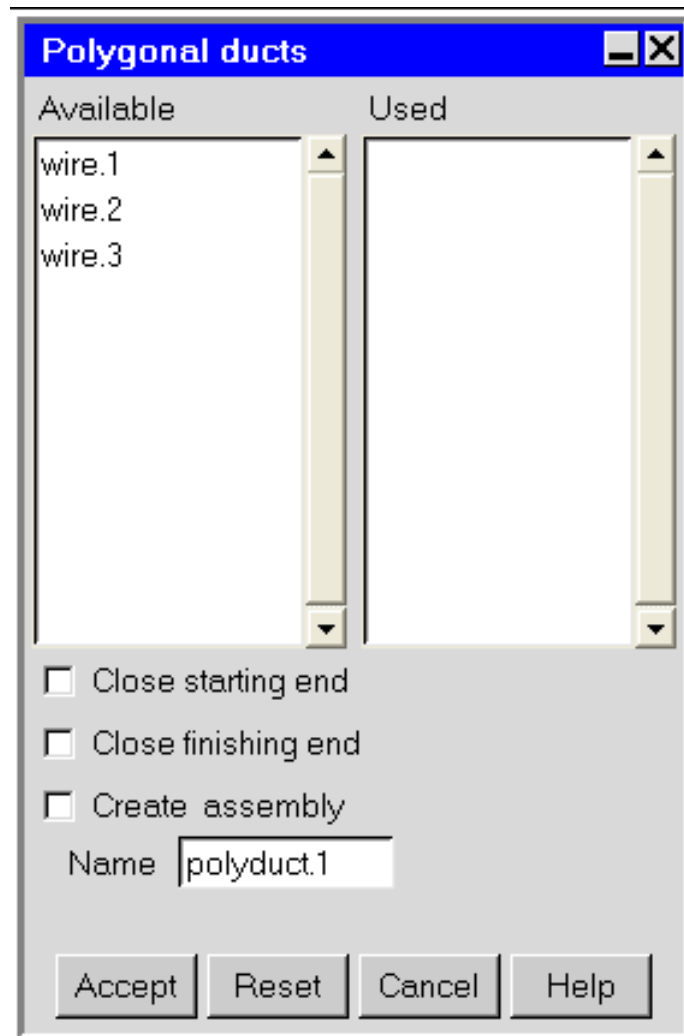


Figure 25.3.1: The Polygonal ducts Panel

- If you selected the **Create assembly** option in the **Polygonal ducts** panel, you can edit the polygonal duct in the same way that you edit an assembly (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Polygonal ducts** panel, you must individually select and modify the polygonal duct's block objects.

25.3.3 Closed Box

The closed box macro allows you to enclose spaces in your **Airpak** model and assign properties to the sides of the enclosure. Two types of enclosures can be created using the closed box macro:

- A box that encloses the whole room with six **Airpak** wall objects
- A box that encloses part of the room with six **Airpak** partition objects

The closed box macro can be used to reduce the number of steps required to create the six individual sides of an enclosure.

Adding a Closed Box to Your Airpak Model

To include a closed box in your **Airpak** model, select **Quick geometry/approximations** and then **Closed box** in the **Macros** menu. This will open the **Closed Box** panel, shown in Figure 25.3.2.

Macros—>Quick geometry/approximations—>Closed Box

The procedure for adding a closed box to your model is as follows:

1. Select **Start/end** and enter values for the start coordinates (**xS**, **yS**, **zS**) and end coordinates (**xE**, **yE**, **zE**) of the box, or select **Start/length** and enter values for the start coordinates (**xS**, **yS**, **zS**) and the lengths of the sides (**xL**, **yL**, **zL**) of the box.
2. Specify whether the box is **Internal** (encloses part of the room) or **External** (encloses the whole room) next to **Box type**.
3. If the thickness of the box sides is to be non-zero, select **Thick walls**. For zero-thickness box sides, deselect **Thick walls**.
4. (non-zero-thickness box sides only) Specify the **Thickness** of the box sides. When the thickness of the sides of an internal box is non-zero, the heat flow through the sides is based on the conductivity of the sides. For non-zero-thickness sides on an external box, the heat flow is computed based on the conductivity of the sides, the computed temperature on the inner surfaces of the sides, and the conditions applied at the outer surfaces of the sides.

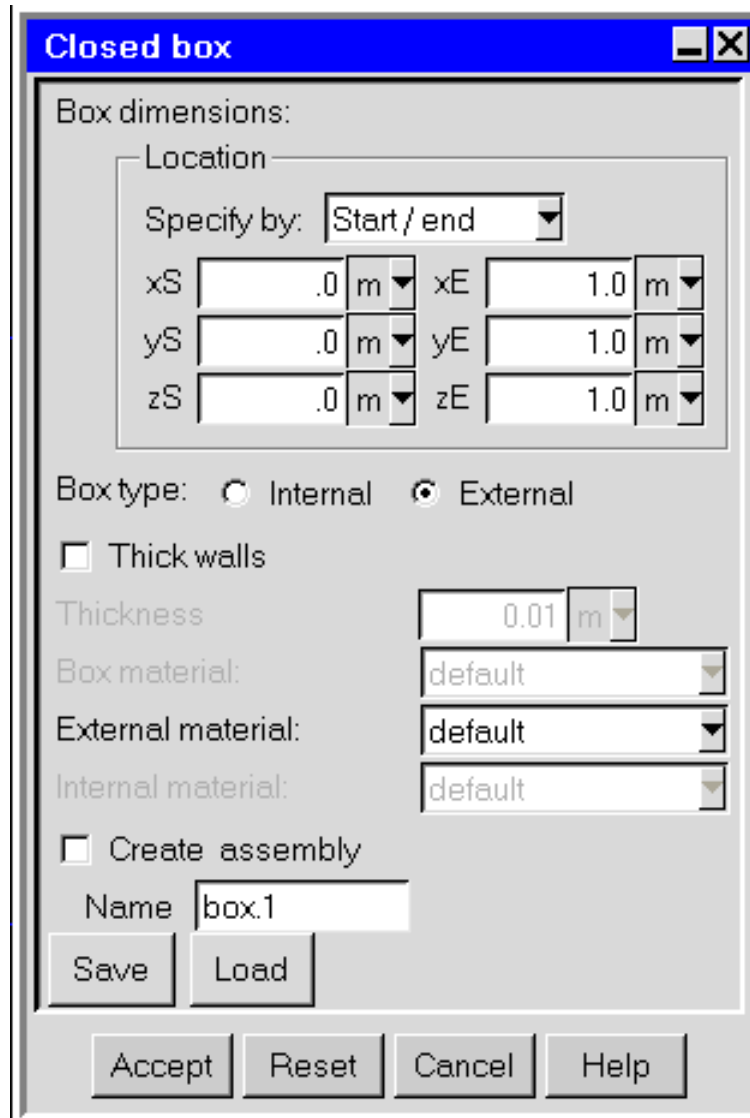


Figure 25.3.2: The Closed Box Panel

If the thickness is non-zero, each side is expanded inward or outward from the plane of the side. If the thickness is positive, the sides are expanded to the specified thickness in the positive direction of the coordinate axis that is normal to the plane of each side. If the thickness is specified with a negative sign, the expansion is in the negative direction of the coordinate axis normal to the plane of each side.

5. (non-zero-thickness box sides only) If you specify a non-zero thickness for the sides of the box, you must also specify the **Box material** for the sides. The selection of **default** means that the material specified as the **Box material** is defined under **Default solid** in the **Basic parameters** panel (see Section 6.4.7). To change the **Box material** for the sides, select a material from the **Box material** drop-down list. See Section 7.7 for details on material properties.
6. (external box sides only) Specify the **External material** for the sides of the box. The selection of **default** means that the material specified as the **External material** for the sides is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **External material** for the sides, select a material from the **External material** drop-down list. The surface roughness and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.
7. (external, non-zero-thickness box sides only) If you specify a non-zero thickness for external box sides, you also need to specify the **Internal material** for the sides. The selection of **default** means that the material specified as the **Internal material** for the sides is defined under **Default surface** in the **Basic parameters** panel (see Section 6.4.7). To change the **Internal material** for the sides, select a material from the **Internal material** drop-down list. The surface roughness and emissivity are defined as part of the surface material parameters. You can edit these values if you select **Edit definition** in the materials list. See Section 7.7 for details on material properties.
8. Specify whether you want the box to be available as one part or as individual objects.
 - If you want the box to be available as one part in your **Airpak** model, select **Create assembly** and specify a name in the **Name** text entry field. The default name is **box.n**, where **n** is the next sequential number among numbered boxes.
 - If you want the box to be available as individual wall or partition objects, deselect **Create assembly**. **Airpak** will create six partition objects for an **Internal** box (one for each side of the box) or six wall objects for an **External** box (to represent the floor, ceiling, and sides of the room).
9. If you want to save the current box specifications to be reused in your current model or to be used in a different model, click the **Save** button to open the **Macro information** dialog box (which is the same as the **File selection** dialog box described in Section 2.1.8). **Airpak** will save the specifications in a subdirectory named **box_info**.

To load box specifications that you have saved in this manner, click the **Load** button and select the box specifications to be loaded.

10. Click **Accept** to accept the parameters entered into the **Closed Box** panel and create the box.

Modifying an Existing Closed Box

After the box has been created and positioned in your **Airpak** model, it cannot be modified using the **Closed Box** panel. There are two ways to modify an existing box:

- If you selected the **Create assembly** option in the **Closed Box** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Closed Box** panel, you must individually select and modify the wall or partition objects that make up the box.

25.3.4 1/4 Polygonal Cylinder

The 1/4 polygonal cylinder macro can be used to create a 1/4 polygonal representation of a cylinder for the following object types:

- Blocks
- Sources
- Resistances

Facets for the total number to be used are specified, and the number must be an even number.

Adding a Polygonal Cylinder to Your Airpak Model

To include a 1/4 polygonal cylinder in your **Airpak** model, select **Quick geometry/approximations** and then **1/4 Polygonal cylinder** in the **Macros** menu. This will open the **Polygonal cylinder** panel, shown in Figure 25.3.3.

Macros → Quick geometry/approximations → 1/4 Polygonal cylinder

The procedure for adding a 1/4 polygonal cylinder to your model is as follows:

1. Specify the **Dimensions** of the polygonal enclosure. Enter the values of the start coordinates (x_C , y_C , z_C), **Radius** and the **Length** of the cylinder.

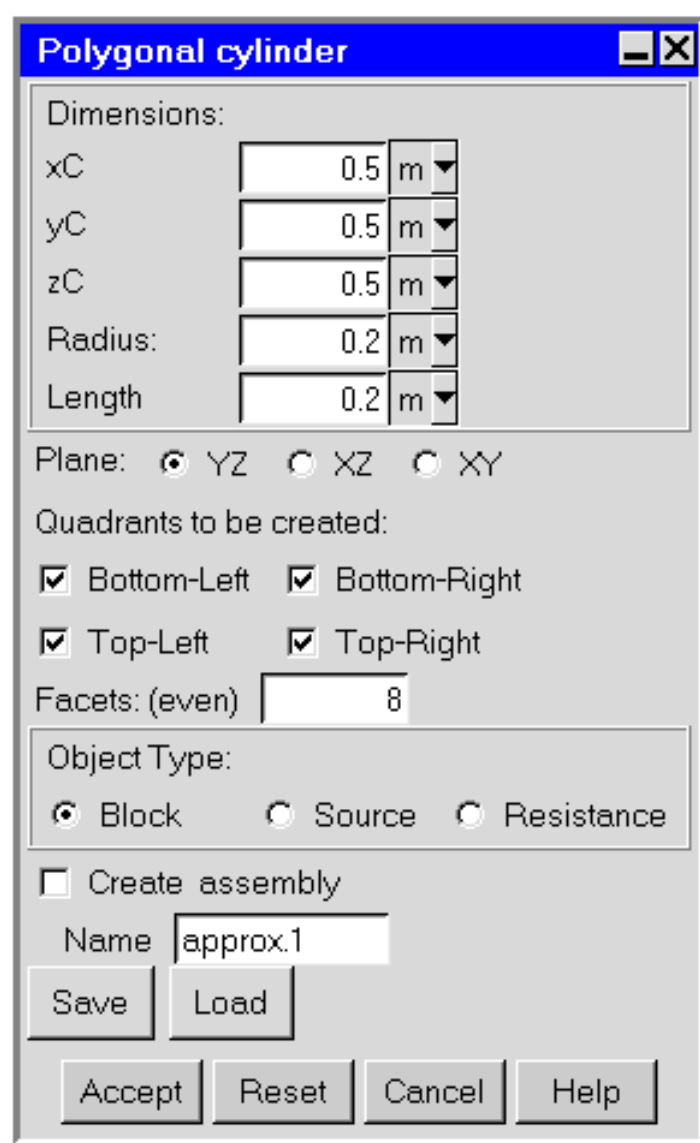


Figure 25.3.3: The 1/4 Polygonal cylinder Panel

2. Select the **Plane** of the polygonal cylinder (YZ, XZ, or XY).
3. Select the **Quadrants to be created**. You can choose to create any or all of the four quadrants of the cylinder. All four quadrants are created as separate geometries.
4. Specify the total number of **Facets** as an even number.
5. Select the **Object Type** that will be created (Block, Source, or Resistance).

Modifying an Existing 1/4 Polygonal Cylinder

After the 1/4 polygonal cylinder has been created and positioned in your **Airpak** model, it cannot be modified using the **Polygonal cylinder** panel. There are two ways to modify an existing polygonal cylinder:

- If you selected the **Create assembly** option in the **Polygonal cylinder** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Polygonal cylinder** panel, you must individually select and modify the wall or partition objects that make up the polygonal cylinder.

25.3.5 Cylinder Plate

The cylinder plate macro allows you to create a cylindrical representation of a plate in your **Airpak** model by faceting the surface of a cylinder with thin conducting plates. The facets are specified per quarter and must be even in number.

Adding a Cylinder Plate to Your Airpak Model

To include a cylindrical plate in your **Airpak** model, select **Quick geometry/approximations** and then **Cylinder plates** in the **Macros** menu. This will open the **Cylinder plates** panel, shown in Figure 25.3.4.

Macros → Quick geometry/approximations → Cylinder plates

The procedure for adding a cylinder plate to your model is as follows:

1. Specify the **Dimensions** of the cylinder plate. Enter the values of the start coordinates (xC, yC, zC), the **Radius** of the circle, and the **Length** of the plate.
2. Specify whether the polygon will be created **Inside** (polygon touches the edge of the bounding circle) or **Outside** (polygon is tangent to the edge of the inner circle).
3. Select the **Axis** of the polygon (X, Y, or Z).
4. Specify the **Facets per Quarter** as an even number.

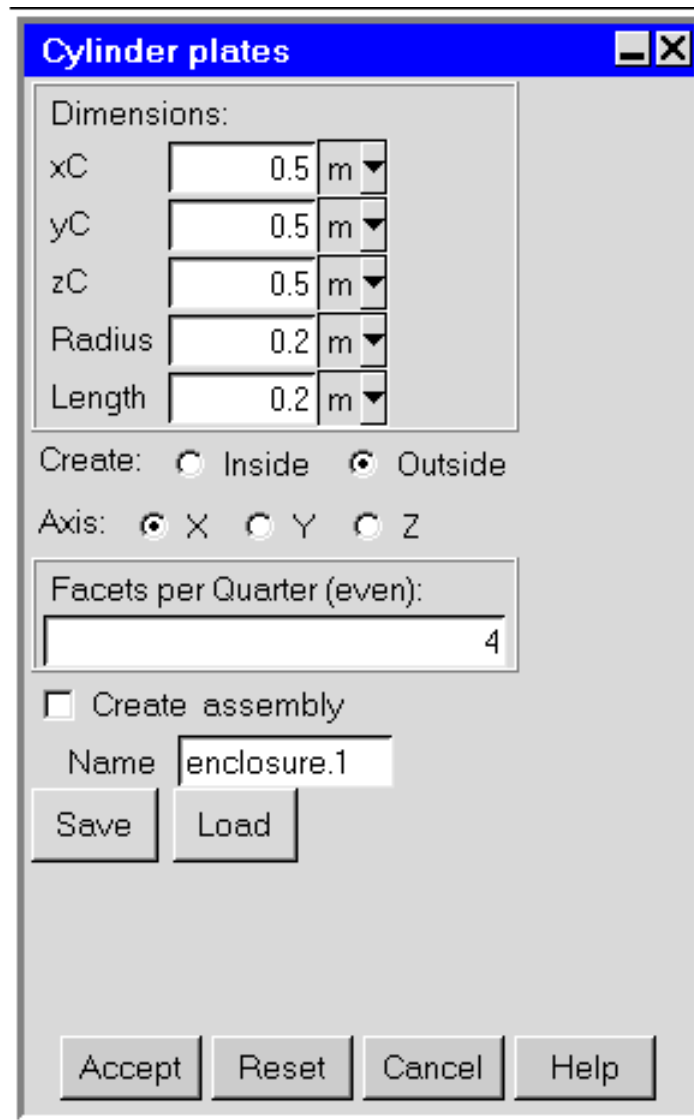


Figure 25.3.4: The Cylinder plates Panel

Modifying an Existing Cylinder Plate

After the cylinder plate has been created and positioned in your **Airpak** model, it cannot be modified using the **Cylinder Plate** panel. There are two ways to modify an existing box:

- If you selected the **Create assembly** option in the **Cylinder plates** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Cylinder plates** panel, you must individually select and modify the wall or partition objects that make up the cylinder plate.

25.3.6 Cylindrical Enclosure

This macro allows you to create a cylindrical enclosure for your **Airpak** model. This enclosure consists of four hollow polygonal blocks that change the shape of the room into a cylinder. Walls are then used to facet the sides of the polygonal blocks in contact with the fluid and the remaining sections of the room. Facets are per quarter, and must be even in number.

Adding a Cylindrical Enclosure to Your Airpak Model

To include a cylindrical enclosure in your **Airpak** model, select **Quick geometry/approximations** and then **Cylindrical enclosure** in the **Macros** menu. This will open the **Cylindrical enclosure** panel, shown in Figure 25.3.5.

Macros—>Quick geometry/approximations—>Cylindrical enclosure

The procedure for adding a cylindrical enclosure to your model is as follows:

1. Specify the **Dimensions** of the cylindrical enclosure. Enter the values of the start coordinates (x_C , y_C , z_C), the **Radius** of the circle, and the **Length** of the plate.
2. Specify whether the polygon will be created **Inside** (polygon touches the edge of the bounding circle) or **Outside** (polygon is tangent to the edge of the inner circle).
3. Specify whether the polygon will be sized to the dimension of the room or not by selecting **Yes** or **No**, respectively, for **Size Room**.
4. Select the **Axis** of the polygon (**X**, **Y**, or **Z**).
5. Specify the **Facets per Quarter** as an even number.

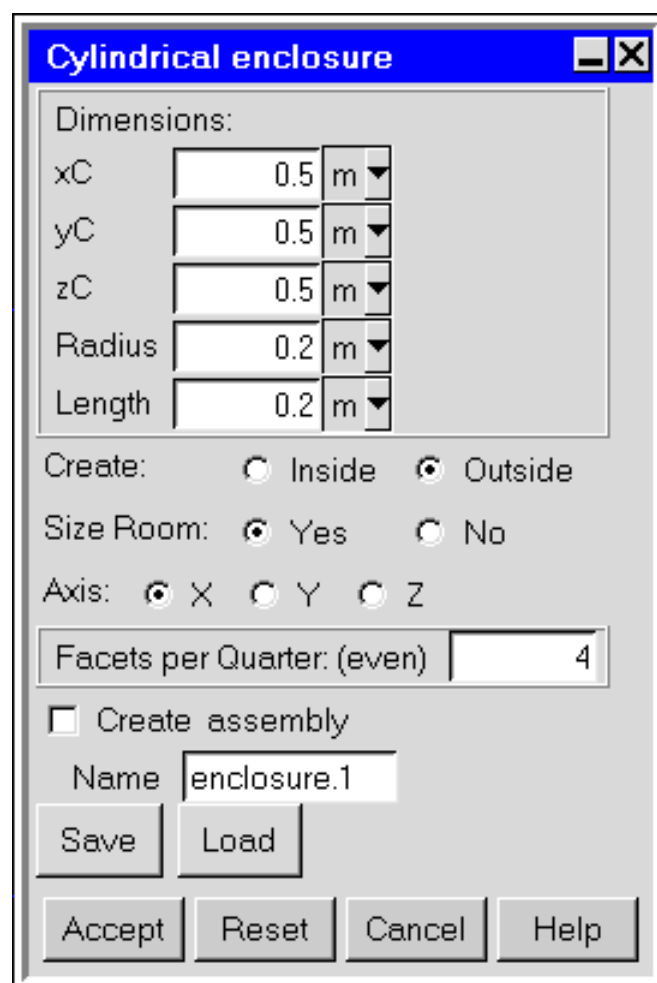


Figure 25.3.5: The Cylindrical enclosure Panel

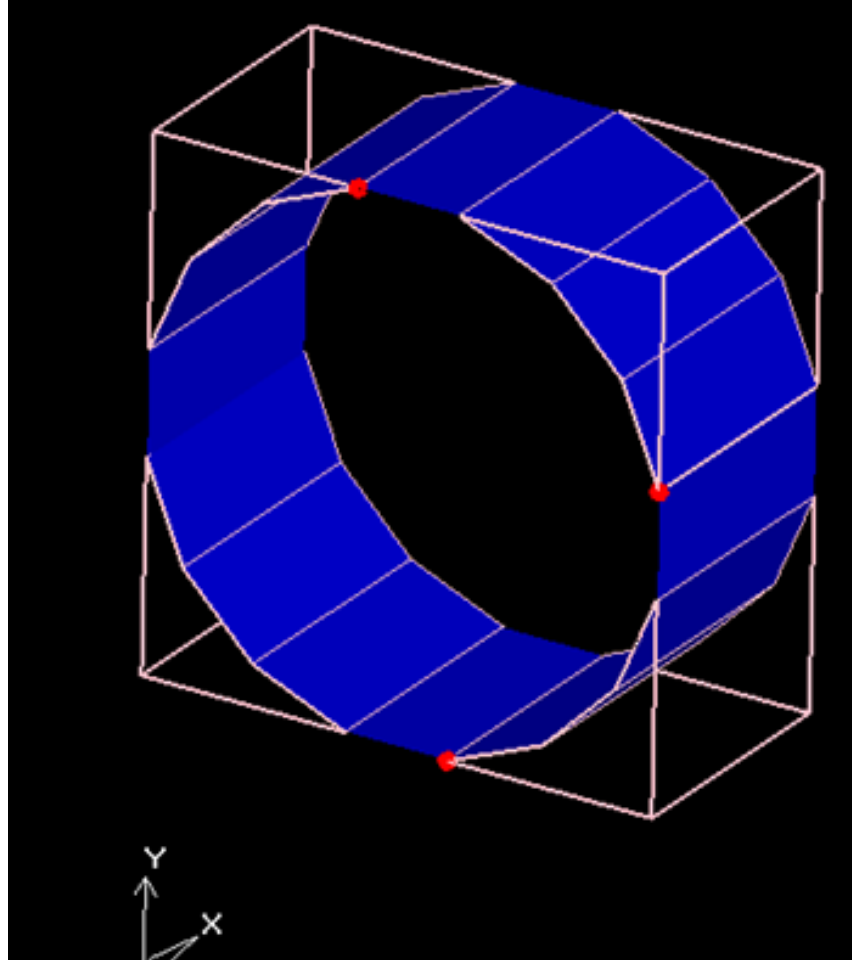


Figure 25.3.6: A Cylindrical Enclosure with 16 Facets and 4 Corners

Modifying an Existing Cylindrical Enclosure

After the polygonal enclosure has been created and positioned in your **Airpak** model, it cannot be modified using the **Cylindrical enclosure** panel. There are two ways to modify an existing enclosure:

- If you selected the **Create assembly** option in the **Polygonal enclosure** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Cylindrical enclosure** panel, you must individually select and modify the wall or partition objects that make up the polygonal enclosure.

25.3.7 Polygonal Circle

The polygonal circle macro allows you to create polygonal circle approximations for your **Airpak** model. The following types of faceted 2D objects can be created:

- fans
- walls
- vents
- partitions
- openings
- sources

The total number of faces (must be even) are specified in the setup.

Adding a Polygonal Circle to Your Airpak Model

To include a polygonal circle in your **Airpak** model, select **Quick geometry/approximations** and then **Polygonal circle** in the **Macros** menu. This will open the **Polygonal circle** panel, shown in Figure 25.3.7.

Macros→Quick geometry/approximations→Polygonal circle

The procedure for adding a polygonal circle to your model is as follows:

1. Specify the **Dimensions** of the polygonal enclosure. Enter the values of the start coordinates (**xC**, **yC**, **zC**), and the **Radius** of the circle.

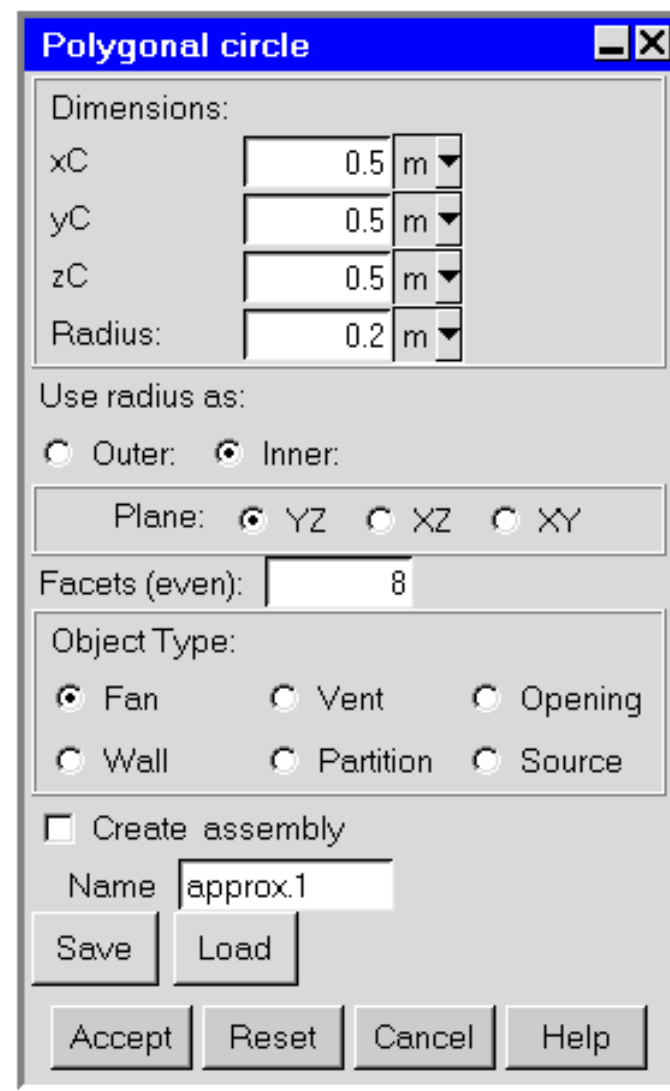


Figure 25.3.7: The Polygonal circle Panel

2. Specify whether the radius will be used as the **Outer** surface (polygon is tangent to the edge of the inner circle) or the **Inner** surface (polygon touches the edge of the bounding circle).
3. Select the **Plane** of the polygonal circle (YZ, XZ, or XY).
4. Specify the **Facets per Quarter** as an even number.
5. Select the **Object Type** that will be created (Fan, Vent. Opening, Wall, Partition, or Source).

Modifying an Existing Polygonal Circle

After the polygonal circle has been created and positioned in your **Airpak** model, it cannot be modified using the **Polygonal circle** panel. There are two ways to modify an existing polygonal circle:

- If you selected the **Create assembly** option in the **Polygonal circle** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Polygonal circle** panel, you must individually select and modify the wall or partition objects that make up the polygonal circle.

25.3.8 Polygonal Cylinder

The polygonal cylinder macro can be used to create a polygonal representation of a cylinder for the following object types:

- Blocks
- Sources
- Resistances

Facets for the total number to be used are specified, and the number must be an even number.

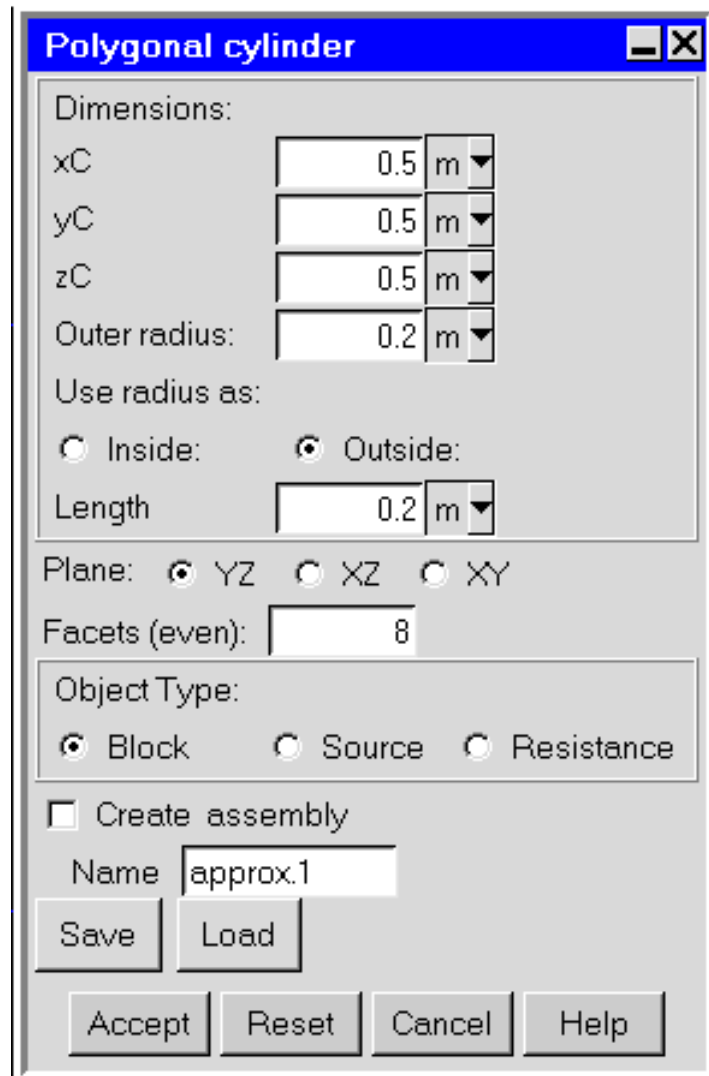


Figure 25.3.8: The Polygonal cylinder Panel

Adding a Polygonal Cylinder to Your Airpak Model

To include a polygonal cylinder in your Airpak model, select **Quick geometry/approximations** and then **Polygonal cylinder** in the **Macros** menu. This will open the **Polygonal cylinder** panel, shown in Figure 25.3.8.

Macros→Quick geometry/approximations→Polygonal cylinder

The procedure for adding a polygonal cylinder to your model is as follows:

1. Specify the **Dimensions** of the polygonal enclosure. Enter the values of the start coordinates (**xC**, **yC**, **zC**), and the **Outer radius** of the circle.
2. Specify whether the radius will be used as the **Outside** surface (polygon is tangent to the edge of the inner circle) or the **Inside** surface (polygon touches the edge of the bounding circle).
3. Specify the **Length** of the cylinder.
4. Select the **Plane** of the polygonal cylinder (**YZ**, **XZ**, or **XY**).
5. Specify the total number of **Facets** as an even number.
6. Select the **Object Type** that will be created (**Block**, **Source**, or **Resistance**).

Modifying an Existing Polygonal Cylinder

After the polygonal cylinder has been created and positioned in your Airpak model, it cannot be modified using the **Polygonal cylinder** panel. There are two ways to modify an existing polygonal cylinder:

- If you selected the **Create assembly** option in the **Polygonal cylinder** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Polygonal cylinder** panel, you must individually select and modify the wall or partition objects that make up the polygonal cylinder.

25.3.9 Hemisphere

The hemisphere macro can be used to create a layered representation of a hemisphere comprised of stacked non-uniform cylinders. You have the ability to use any of the meshing options with this macro, as opposed to using true hemisphere geometry which would restrict usage to only the tetrahedral mesher.

Adding a Hemisphere to Your Airpak Model

To include a hemisphere in your Airpak model, select **Quick geometry/approximations** and then **Hemisphere** in the **Macros** menu. This will open the **Hemisphere** panel, shown in Figure 25.3.9.

Macros → Quick geometry/approximations → Hemisphere

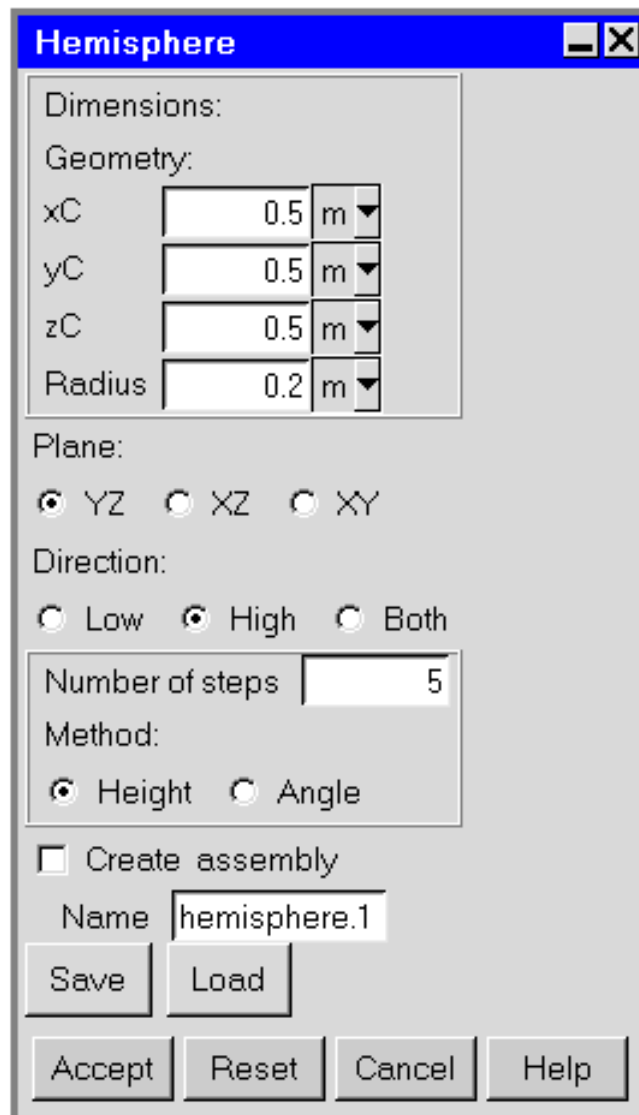


Figure 25.3.9: The Hemisphere Panel

The procedure for adding a hemisphere to your model is as follows:

1. Specify the **Dimensions** of the polygonal enclosure. Enter the values of the start

coordinates (x_C , y_C , z_C), and the **Radius** of the circle.

2. Specify whether the radius will be used as the **Outside** surface (polygon is tangent to the edge of the inner circle) or the **Inside** surface (polygon touches the edge of the bounding circle).
3. Select the **Plane** of the hemisphere (**YZ**, **XZ**, or **XY**).
4. Select the **Direction** of the hemisphere (**Low**, **High**, or **Both**). The **Direction** refers to whether the hemisphere will be created advancing out from the high side of the specified plane that contains the center of the hemisphere, or from the low side of that specified plane.
5. Specify the **Number of steps** which is the number of non-uniform cylinder layers to use in forming the hemisphere.
6. Select the **Method** (**Height** or **Angle**) to be used to create the hemisphere. The **Height** option will create layer which are uniform in height. Alternatively, the **Angle** option will create layers that vary in height but represent uniform increments of angle to the center of the hemisphere for each layer.

Modifying an Existing Hemisphere

After the hemisphere has been created and positioned in your **Airpak** model, it cannot be modified using the **Hemisphere** panel. There are two ways to modify an hemisphere:

- If you selected the **Create assembly** option in the **Hemisphere** panel, you can edit the box in the same way that you edit a part (see Section 7.8.5).
- If you did not select the **Create assembly** option in the **Hemisphere** panel, you must individually select and modify the wall or partition objects that make up the hemisphere.

25.4 Object Rotation Macros

The following object rotation macros are described in this section:

- Section 25.4.1: Rotating Plates
- Section 25.4.2: Rotating Prismatic Blocks
- Section 25.4.3: Rotating Polygonal Blocks
- Section 25.4.4: Rotating Groups of Prismatic Blocks

25.4.1 Rotating Individual Plates

This macro allows you to create a polygonal block that represents an individual rotated plate in your **Airpak** model. You can rotate an individual plate about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis. Where multiple plates are selected, the centroid/edge that you specify is for the individual plate, and not for the bounding box of the combined selection. **Airpak** then generates an inclined plane to represent the rotated plate.

Adding a Rotated Individual Plate to Your Airpak Model

To rotate an individual plate and add it to your **Airpak** model, select **Rotate objects** and then **Individual plates** in the **Macros** menu. This will open the **Rotate prism plates** panel, shown in Figure 25.4.1.

Macros—>Rotate objects—>Individual plates

The procedure for rotating an individual plate and adding it to your model is as follows:

1. Select the **Axis** of rotation (**X**, **Y**, or **Z**).
2. Specify the **Angle** of rotation.
3. Select **Pivot about** location of the plate as either a given point (**Point**), its centroid (**Centroid**), or a particular edge through an arbitrary angle and given coordinate (**Vertex**).
4. Specify the centroid coordinate of the plate (**x0** and **y0**).
5. Specify whether to use the list of all **Available** partitions in the model for the rotation operation, or **Used** which is a list of selected partitions.
6. Click **Accept**.

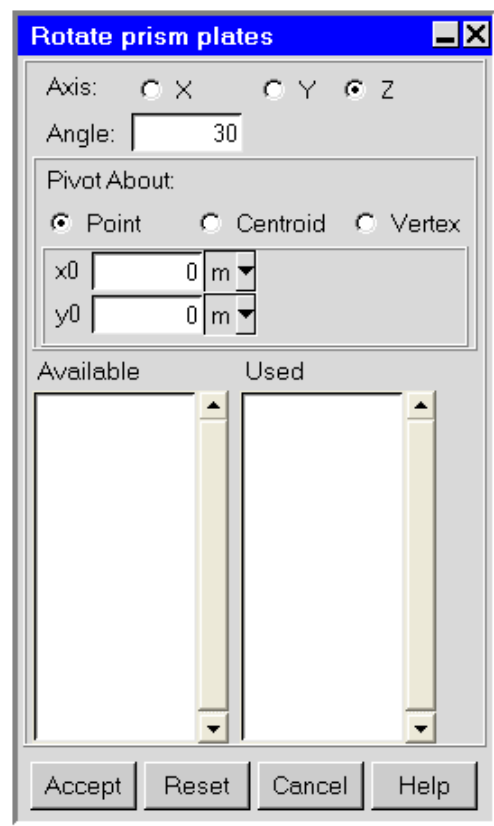


Figure 25.4.1: The Rotate prism plates Panel

25.4.2 Rotating Prismatic Blocks

This macro allows you to create a polygonal block that represents an individual rotated prismatic block in your Airpak model. You can rotate a prism block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis. Where multiple plates are selected, the centroid/edge that you specify is for the individual prism block, and not for the bounding box of the combined selection. Airpak generates a polygonal block that is the representation of the rotated prism block.

Adding a Rotated Prismatic Block to Your Airpak Model

To rotate an individual prism block and add it to your Airpak model, select Rotate objects and then /Individual Prism blocks in the Macros menu. This will open the Rotate prism blocks panel, shown in Figure 25.4.2.

Macros—>Rotate objects—>Individual prism blocks

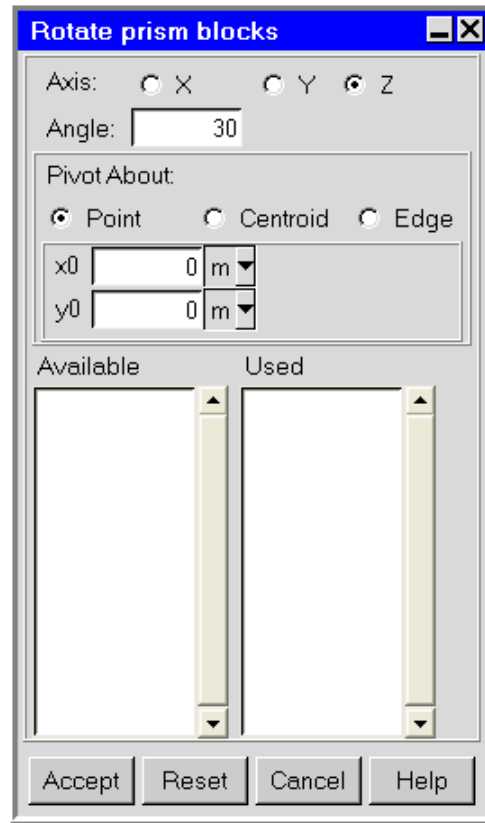


Figure 25.4.2: The Rotate prism blocks Panel

The procedure for rotating a prism block and adding it to your model is as follows:

1. Select the **Axis** of rotation (X, Y, or Z).
2. Specify the **Angle** of rotation.
3. Select **Pivot about** method as either a given point (**Point**), its centroid (**Centroid**), or a particular edge through an arbitrary angle and given coordinate (**Vertex**).
4. Specify the centroid coordinate (x0 and y0).
5. Specify whether to use the list of all **Available** partitions in the model for the rotation operation, or **Used** which is a list of selected partitions.
6. Click **Accept**.

25.4.3 Rotating Polygonal Blocks

This macro allows you to create a polygonal block that represents an individual rotated polygonal block in your **Airpak** model. You can rotate a polygonal block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis. Where multiple plates are selected, the centroid/edge that you specify is for the individual polygonal block, and not for the bounding box of the combined selection. **Airpak** generates a polygonal block that is the representation of the rotated prism block.

Adding a Rotated Polygonal Block to Your Airpak Model

To rotate an individual polygonal block and add it to your **Airpak** model, select **Rotate** objects and then **Individual polygonal blocks** in the **Macros** menu. This will open the **Rotate polygonal blocks** panel, shown in Figure 25.4.3.

Macros—>Rotate objects—>Individual polygonal blocks

The procedure for rotating a polygonal block and adding it to your model is as follows:

1. Specify the **Angle** of rotation.
2. Select **Pivot about** coordinate (guixP, yP, and zP).
3. Specify whether to use the list of all **Available** partitions in the model for the rotation operation, or **Used** which is a list of selected partitions.
4. Click **Accept**.

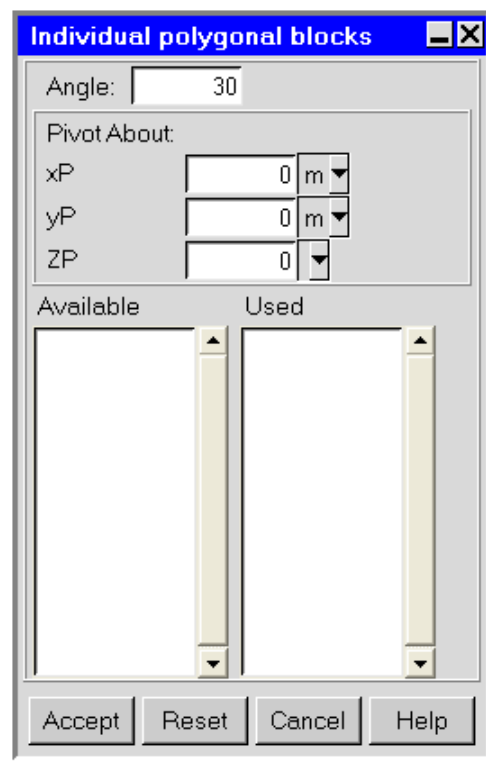


Figure 25.4.3: The Rotate polygonal blocks Panel

25.4.4 Rotating Groups of Prismatic Blocks

This macro allows you to create a polygonal block that represents an individual rotated polygonal block in your Airpak model. You can rotate a polygonal block about either a given point, its centroid, or a particular edge through an arbitrary angle and given coordinate axis. Where multiple plates are selected, the centroid/edge that you specify is for the individual polygonal block, and not for the bounding box of the combined selection. Airpak generates a polygonal block that is the representation of the rotated prism block.

Adding a Rotated Group of Prism Blocks to Your Airpak Model

To rotate a group of prismatic blocks and add it to your Airpak model, select **Rotate objects** and then **Groups of prism blocks** in the **Macros** menu. This will open the **Rotate group of prism blocks** panel, shown in Figure 25.4.4.

Macros—>Rotate objects—>Groups of prism blocks

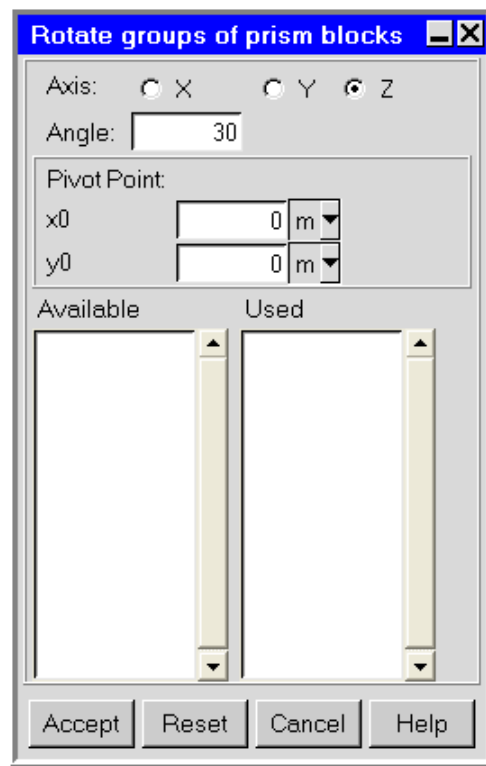


Figure 25.4.4: The Rotate groups of prism blocks Panel

The procedure for rotating a group of prism blocks and adding it to your model is as follows:

1. Select the **Axis** of rotation (X, Y, or Z).
2. Specify the **Angle** of rotation.
3. Select the **Pivot Point** coordinate (x0 and y0).
4. Specify whether to use the list of all **Available** partitions in the model for the rotation operation, or **Used** which is a list of selected partitions.
5. Click **Accept**.

This chapter provides information about generating a computational mesh for your **Airpak** model.

- Section 26.1: Overview
- Section 26.2: Mesh Quality and Type
- Section 26.3: Guidelines for Mesh Generation
- Section 26.4: Creating a Minimum-Count Mesh
- Section 26.5: Refining the Mesh Globally
- Section 26.6: Refining the Mesh Locally
- Section 26.7: Controlling the Meshing Order for Objects
- Section 26.8: Non-Conformal Meshing Procedures for Assemblies
- Section 26.9: Displaying the Mesh
- Section 26.10: Checking the Mesh
- Section 26.11: Exporting a Mesh
- Section 26.12: Loading an Existing Mesh

26.1 Overview

Once you have finished designing your model, you need to generate the computational mesh that is used as the basis of the solution procedure. The mesh consists of discrete elements located throughout the computational domain. Within each element, **Airpak** solves the equations that govern the flow and heat transfer in the room.

A good computational mesh is an essential ingredient for a successful and accurate solution. If the overall mesh is too coarse, the resulting solution may be inaccurate. If the overall mesh is too fine, the computational cost may become prohibitive. In summary, the cost and accuracy of the solution are directly dependent on the quality of the mesh.

Airpak automates the mesh generation procedure, but allows you to customize the meshing parameters in order to refine the mesh and optimize trade-offs between computational

cost and solution accuracy. You can apply such modified parameters at a global level (affecting the entire computational domain) or to specific modeling objects. This flexibility provides you with an efficient mesh generation process that you can influence as much (or as little) as you want.

The meshing procedure that **Airpak** follows is based on a set of rules that govern how each type of object is to be meshed. **Airpak** operates on a “cocooning” methodology whereby each object is meshed individually, as tightly as your specifications permit, in order to resolve the physics of the solution optimally. In every case, optimal resolution of the physics depends on the specific problem to be solved. The mesh elements are smaller near objects (as shown in Figure 26.1.1), to take into account thermal and velocity gradients that are often present near the boundaries of an object. By contrast, the open spaces between objects are meshed with large elements, to minimize computational costs.

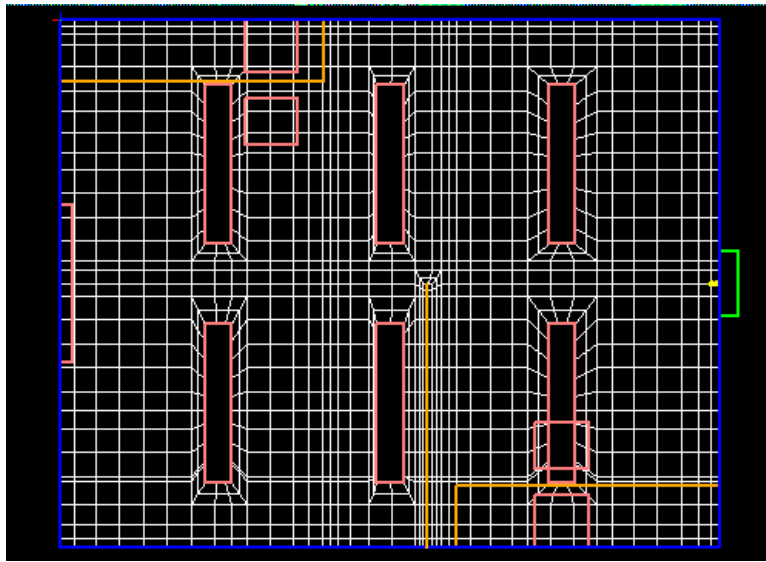


Figure 26.1.1: Mesh with Small Elements Near Objects and Large Elements in Open Spaces

26.2 Mesh Quality and Type

26.2.1 Mesh Quality

As mentioned in Section 26.1, the quality of the mesh is one of the most critical aspects of a CFD model. A good mesh is essential for a good solution. A good mesh requires proper resolution, smoothness, low skewness, and an appropriate number of elements. The main requirements can be summarized as follows:

- The mesh must be fine near objects where the gradients of temperature and velocity may be very large (e.g., heated blocks or partitions, room walls with nearby objects). See Figure 26.1.1 for an example.
- The expansion ratio from one mesh element to the next should be kept in the range between 2 and 5, although in some critical areas a lower value might be better. The mesh in Figure 26.1.1 shows good expansion ratios.
- An equilateral element (cube or equilateral tetrahedron) is optimal. Since it is generally not possible to have only optimal elements, you should instead focus on maintaining a low aspect ratio and regular (not skewed) shape for each element. This will reduce the number of long, thin elements and the number of distorted elements, both of which can decrease accuracy and destabilize the solution. Figure 26.2.1 shows examples of elements with low and high skew.

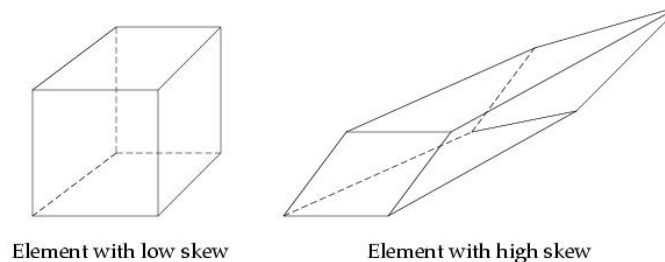


Figure 26.2.1: Elements with Low and High Skew

- It is also possible to have non-conformal meshing in a certain region of the model to improve grid quality and/or to reduce the mesh count. A bounding box can be applied to a certain region, and the mesh inside this region need not match the mesh outside the region. One disadvantage of this method is that currently, particle traces cannot cross the boundaries of a non-conformal interface. See Section 28.2.10 for more information on particle traces.
- For an efficient calculation, the mesh should be coarser in areas where the gradients of velocity and temperature are small. Since there are no small changes in flow

behavior to be captured, it would be wasteful to have a fine mesh in such regions. The cost of the calculation will be directly proportional to the number of elements in the mesh, so it is best to concentrate the elements where you need them, and reduce the number of elements elsewhere.

26.2.2 Hexahedral, Tetrahedral and Hex-Dominant Meshes

There are three types of meshers available in **Airpak**: hexahedral, tetrahedral and hex-dominant. The hexahedral unstructured mesher (the default) is appropriate for most applications. For geometrically complicated models that include, for example, spherical or ellipsoidal objects, the tetrahedral mesher will generally produce a better mesh than the hexahedral mesher. In fact, the tetrahedral mesher is required if your model includes ellipsoids, elliptical cylinders, or polygonal ducts. The hex-dominant mesher can be used for automatic meshing of CAD geometries. Hex-dominant is a robust and highly automated unstructured mesh generator that can handle grids of virtually unlimited size and complexity, consisting mostly of hexahedral elements but including triangular, tetrahedral or pyramidal cells. The hex-dominant mesher uses advanced meshing algorithms to allow the most appropriate cell type to be used to generate body-fitted meshes for the most general CAD geometries. Additionally, an **Airpak** hexahedral mesh can be generated as Cartesian or unstructured. The hexahedral Cartesian mesher can create better quality cells for some simple problems, but may not approximate geometry that is either curved or not aligned with the model axes, as well as the unstructured hexahedral mesher. The Cartesian mesher will turn off O-grid type meshes around objects, and will use stair-stepping to approximate inclined and curved faces. It is generally recommended to use the **Hexa unstructured** mesher (the default option). The same controls that are used for the automatic unstructured hexahedral mesher (the default mesher) will be available for you to create a Cartesian mesh. Additionally, the controls for the hexahedral and hex-dominant meshers are the same. Hence, for information on the procedures for hex-dominant meshing, refer to the section on hexahedral meshing, Section 26.3.1. See Section 26.3.2 for information on the procedures for tetrahedral meshing.

26.3 Guidelines for Mesh Generation

For best results, mesh generation should be an iterative procedure. Guidelines for approaching mesh generation are presented in this section, and specific instructions for the procedures mentioned here are provided in the sections that follow. The hexahedral meshing procedure, which should be used for most applications, is presented in Section 26.3.1. These procedures are valid for the hex-dominant mesher as well. Procedures for using the optional tetrahedral mesher are presented in Section 26.3.2.



The tetrahedral mesher should be used only when you are meshing very complex geometric shapes (i.e., when there are objects present in the model that cannot be

meshed with the hexahedral mesher, or if the hexahedral mesher fails to produce an acceptable grid). Furthermore, the use of the tetrahedral mesher with many objects that have very small dimensions (such as partitions with a small thickness) can lead to very large meshes. Objects for which the tetrahedral mesher must be used include ellipsoids, elliptical cylinders, and polygonal ducts.

26.3.1 Hexahedral Meshing Procedure

The recommended approach to hexahedral mesh generation is as follows:

1. Generate a mesh using **Airpak's** default parameters for a coarse mesh (see Section 26.4.1), using the default **Hexa unstructured** mesher. The resulting mesh (called the minimum-count mesh) contains the minimum number of elements required to adequately represent the model geometry and satisfy the default meshing rules.

You can compute an approximate solution on this initial mesh to determine quickly if the calculation runs properly and the results seem reasonable, before proceeding to refine the mesh and compute a more accurate solution. This initial calculation will also allow you to estimate the total computation cost.

2. Generate a refined mesh.
 - (a) Set the **Max X size**, **Max Y size**, and **Max Z size** values to about 1/20 of the room dimensions in the corresponding directions.
 - (b) Select the **Normal** mesh resolution.
 - (c) Generate the mesh.

See Section 26.5.1 for details.

3. Examine the mesh, using plane cuts and diagnostics (as described in Sections 26.9 and 26.10), to see if it satisfies the following requirements:
 - The number of elements between solid faces should be at least 2.
 - There should be at least 4 or 5 elements on each flow object (openings, vents, resistances, fans).
 - The mesh quality should satisfy the requirements outlined in Section 26.2.
4. If the mesh does not satisfy these requirements, use the object-specific meshing controls to refine the mesh locally and improve the mesh quality. Suggestions are listed below:
 - Refine the mesh around objects where the temperature and velocity gradients are expected to be high (e.g., heated blocks and partitions, objects blocking or diverting the flow, fans).

- If the element count on the face of an object is low, use the **X**, **Y**, or **Z count** (or the corresponding parameter) to change it.
- Use high and low element size and ratio to refine the mesh around objects. You can also use the inward/outward size and ratio to refine the mesh.

See Section 26.6 for details.

Additional options for improving the mesh are as follows:

- If your model includes a small number of objects, and the element count or spacing on all of them is low, you can use the **Init height** option to turn on O-grid cocooning (Hexa unstructured mesher only) and place graded elements on the faces of all objects. This option should be used *only* when the number of objects is not too large; otherwise, the mesh count can become very large. See Section 26.5.1 for details.
- In some cases, setting the **Max O-grid height** to a small enough value relative to the object size will yield better mesh quality by localizing the cocooning action of the mesher. Note that the value of **Max O-grid height** should be higher than that of **Init height**, and that a value of 0 for **Max O-grid height** leaves the O-grid height unbounded. See Section 26.5.1 for details.
- Aligning faces that are nearly aligned (see Figure 26.3.1) will generally reduce the overall mesh size and result in better convergence (due to the improved aspect ratio of the elements). You can align faces by modifying the coordinates of the appropriate object(s), or by using the alignment tools (see Section 2.1.3).

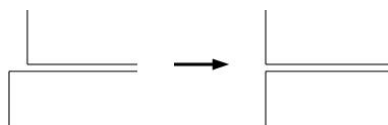


Figure 26.3.1: Aligning Nearly-Aligned Faces

- Rectangular objects are generally easier to mesh than circular objects. In some cases, you may be able to improve the mesh quality without affecting the solution by replacing circular objects (fans, openings, etc.) with rectangular objects of the same type. For models with circular objects you may also want to consider using the tetrahedral mesher (described in Section 26.3.2) instead of the hexahedral mesher.
- You can use fluid blocks to refine or otherwise improve the mesh locally. Unless you change their material properties, fluid blocks do not alter the flow or temperature distribution inside the room because they are, by default, made of air, which is the material inside the rest of the room as well.

5. Calculate a solution on the refined mesh.

6. For optimal accuracy, refine the mesh further, calculate a solution, and compare it with the solution for the previous mesh. Repeat until the solution is grid-independent (i.e., until it no longer changes with each new mesh).

26.3.2 Tetrahedral Meshing Procedure

As mentioned in Section 26.3, the tetrahedral mesher should be used only when you are meshing very complex geometric shapes, i.e., when there are objects present in the model that cannot be meshed with the hexahedral mesher, or if the hexahedral mesher fails to produce an acceptable grid.

The recommended approach to tetrahedral mesh generation is as follows:

1. Generate a coarse mesh by turning off the **Max tetra size** criterion (see Section 26.4.2 for details). The resulting mesh (called the minimum-count mesh) contains the minimum number of elements required to adequately represent the model geometry and satisfy the default meshing rules.

You can compute an approximate solution on this initial mesh to determine quickly if the calculation runs properly and the results seem reasonable, before proceeding to refine the mesh and compute a more accurate solution. This initial calculation will also allow you to estimate the total computation cost.

2. Generate a refined mesh.
 - (a) Turn on the **Max tetra size** option and specify the desired maximum length of an element's side. A typical value is about 1/20 of the smallest room dimension.
 - (b) Generate the mesh.

See Section 26.5.2 for details.

3. Examine the mesh, using plane cuts and diagnostics (as described in Sections 26.9 and 26.10), to see if it satisfies the following requirements:
 - The number of elements between solid faces should be at least 2.
 - There should be at least 4 or 5 elements on each flow object (openings, vents, resistances, fans).
 - The mesh quality should satisfy the requirements outlined in Section 26.2.
4. If the mesh does not satisfy these requirements, use the object-specific meshing controls to refine the mesh locally and improve the mesh quality. Suggestions are listed below:
 - Refine the mesh around objects where the temperature and velocity gradients are expected to be high (e.g., heated blocks and partitions, objects blocking or diverting the flow, fans).

- Use high and low element size to refine the mesh around objects. You can also use the inward/outward size to refine the mesh.

See Section 26.6 for details.

Additional options for improving the mesh are as follows:

- Reduce the **Max tetra size** if many of the elements are too large. This option should be used with caution, since it can cause the mesh count to become very large.
- Aligning faces that are nearly aligned (see Figure 26.3.1) will generally reduce the overall mesh size and result in better convergence (due to the low aspect ratio of the elements). You can align faces by modifying the coordinates of the appropriate object(s).
- You can use fluid blocks to refine or otherwise improve the mesh locally. Unless you change their material properties, fluid blocks do not alter the flow or temperature distribution inside the room because they are, by default, made of air, which is the material inside the rest of the room as well.


5. Calculate a solution on the refined mesh.
6. For optimal accuracy, refine the mesh further, calculate a solution, and compare it with the solution for the previous mesh. Repeat until the solution is grid-independent (i.e., until it no longer changes with each new mesh).

26.4 Creating a Minimum-Count Mesh

As discussed in Sections 26.3.1 and 26.3.2, the minimum-count mesh is useful for computing an initial solution. Airpak generates a minimum-count mesh using the minimum number of elements necessary to represent the geometry and satisfy the default meshing rules for the objects in your model.

26.4.1 Creating a Minimum-Count Hexahedral Mesh

The steps you should follow to create a minimum-count hexahedral mesh for your model are listed below:

1. Open the **Mesh control panel** (Figure 26.4.1) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve toolbar**.
Model—>Generate mesh
2. Click on the **Generate** tab (the default when the panel is opened for the first time) to access the controls for mesh generation. Keep the default selection of **Hexa unstructured** for the **Mesh type**.

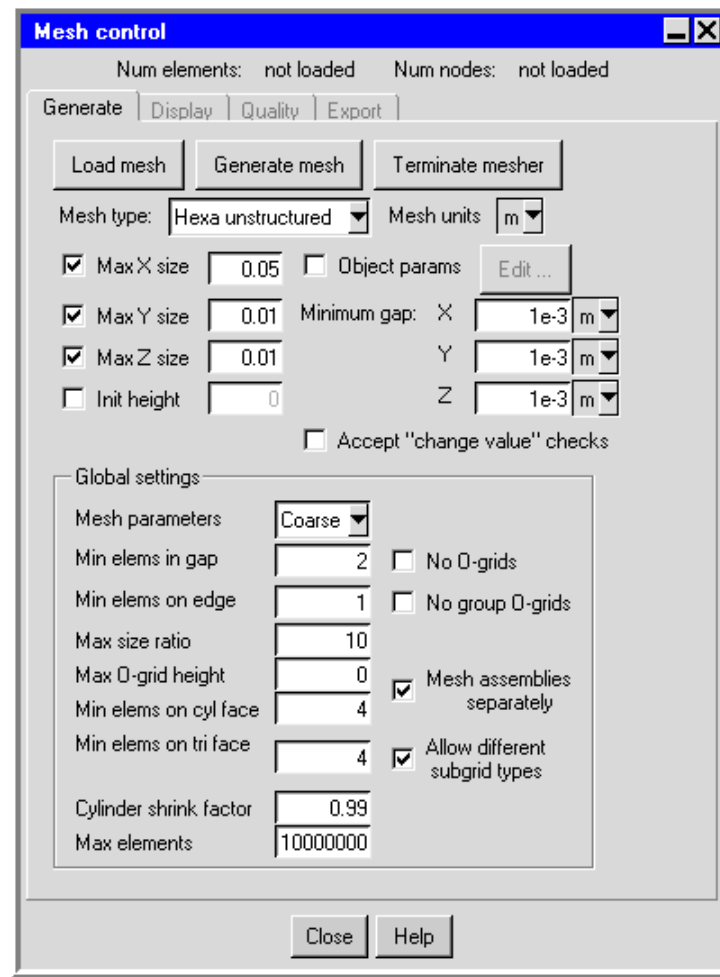



Figure 26.4.1: The Mesh control Panel for the Hexahedral Mesher

3. Select **Coarse** in the **Mesh parameters** drop-down list to update the panel with the default settings for a coarse mesh.
4. Click **Generate mesh**. **Airpak** will generate the minimum-count mesh, which you can view as described in Section 26.9. If you want to stop the meshing process before it is complete, click **Terminate mesher**.

As discussed in Section 26.3.1, you will generally want to refine the minimum-count mesh after computing an initial solution. **Airpak** provides several parameters for refining the mesh globally (i.e., throughout the entire computational domain) and locally (i.e., on or near individual objects in the model). See Sections 26.5 and 26.6 for details.

26.4.2 Creating a Minimum-Count Tetrahedral Mesh

The steps you should follow to create a minimum-count tetrahedral mesh for your model are listed below:

1. Open the Mesh control panel (Figure 26.4.2) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model** and solve toolbar.

Model—>Generate mesh

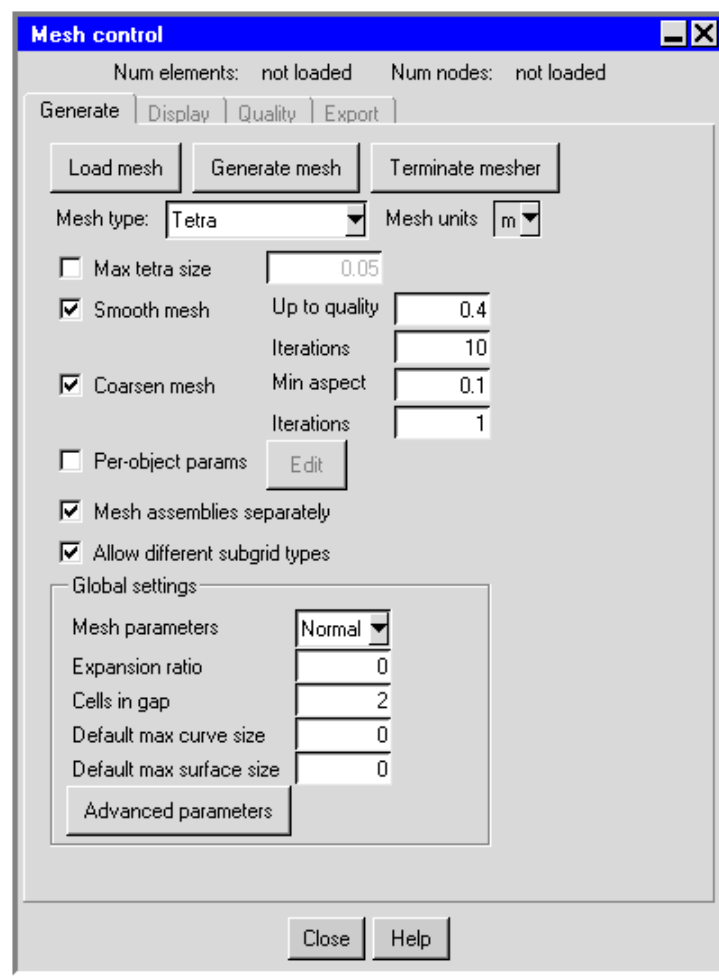


Figure 26.4.2: The Mesh control Panel for the Tetrahedral Mesher

2. Click on the **Generate** tab (the default when the panel is opened for the first time) to access the controls for mesh generation.
3. Select **Tetra** in the **Mesh type** drop-down list.

4. Turn off the **Max tetra size** option.
5. Click **Generate mesh**. **Airpak** will generate the minimum-count mesh, which you can view as described in Section 26.9. If you want to stop the meshing process before it is complete, click **Terminate mesher**.


As discussed in Section 26.3.2, you will generally want to refine the minimum-count mesh after computing an initial solution. **Airpak** provides several parameters for refining the mesh globally (i.e., throughout the entire computational domain) and locally (i.e., on or near individual objects in the model). See Sections 26.5 and 26.6 for details.

26.5 Refining the Mesh Globally

As discussed in Sections 26.3.1 and 26.3.2, you will generally want to refine the minimum-count mesh after computing an initial solution. The steps you should follow to refine the mesh globally (i.e., throughout the entire computational domain) are presented here for both hexahedral and tetrahedral meshes. See Section 26.6 for information about refining the mesh on or near individual objects in the model.

26.5.1 Global Refinement for a Hexahedral Mesh

The procedure for globally refining a hexahedral mesh is as follows:

1. Open the **Mesh control panel** (Figure 26.4.1) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.
Model—>Generate mesh
2. Click on the **Generate** tab to access the controls for mesh generation.
3. Select **Hexa unstructured** (the default mesher) in the **Mesh type** drop-down list.
4. Turn on the **Max X size**, **Max Y size**, and **Max Z size** specifications, and set each one to the desired maximum element length in each direction. Typical values are about 1/20 of the room dimensions in the corresponding directions.
5. (*only* for models with a small number of objects) To increase the number of elements on or near all objects, and reduce the mesh spacing on or near all objects, turn on the **Init height** option and specify a value.

The **Init height** specifies the maximum height of the first element layer generated on the surface of any modeling object (e.g., PCB, block, or fan). By default, **Airpak** uses its own internal rules to determine the height of the first element on the surface of an object (the initial height).

i This option should be used *only* when the number of objects is not too large; otherwise, the mesh count can become very large.

6. Next to **Minimum gap**, specify the minimum distances separating objects in your model in the X, Y, and Z coordinate directions.

This specification is used by **Airpak** whenever the distance between two objects is less than this value, but greater than the model's zero tolerance. If the minimum gap specified for any of the three coordinate directions is more than the size of the smallest object in that coordinate direction, **Airpak** will display a **Minimum separation** warning. You can let **Airpak** determine the appropriate minimum gap by clicking the **Change value and mesh** in the **Minimum separation** warning panel. You can choose to let **Airpak** automatically set appropriate minimum gap values without using the **Minimum separation** warning panel by selecting **Accept "change value"** checks in the **Mesh control** panel.

7. In the **Mesh parameters** drop-down list, select **Normal** to update the panel with the default settings for a normal, non-coarse mesh. If you want, you can modify the default settings, which are defined as follows:

Min elems in gap specifies the minimum number of elements between adjacent objects.

Min elems on edge specifies the minimum number of elements on each edge of each object.

Max size ratio specifies the maximum ratio of the sizes of adjacent elements (for the whole model).

Max O-grid height (Hexa unstructured mesher only) specifies the distance of the cocooning from the surface of the object (see Figure 26.5.1).

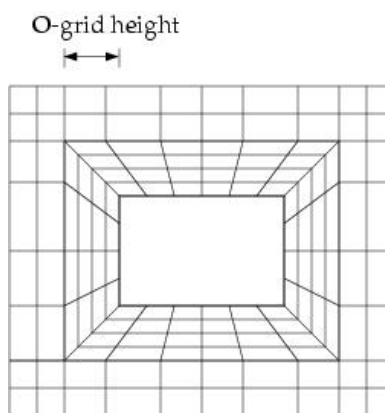


Figure 26.5.1: O-Grid Height

Min elems on cyl face specifies the minimum number of elements on the circular face of a cylindrical object (e.g., a cylindrical block). If you consider the mesh around the cylinder to be box-like, this is the minimum number of intervals on each edge of the box.

Min elems on tri face specifies the minimum number of elements on the triangular face of an object (e.g., a polygonal block with a triangular face). If you consider the mesh around the triangle to be box-like, this is the minimum number of intervals on each edge of the box.

Cylinder shrink factor specifies the factor of adjustment for diameters of cylindrical shapes in cases where the cylinder touches other surfaces tangentially at its side surface.

Max elements specifies the maximum number of elements in the mesh.

No O-grids (Hexa unstructured mesher only) indicates whether or not objects will have O-grids around them. This option is off by default, indicating that **Airpak** will put O-grids around all objects, including those that contain other objects.

No group O-grids (Hexa unstructured mesher only) indicates whether or not objects with other objects inside them (e.g., an enclosure that has blocks inside it) will have O-grids around them. This option is off by default, indicating that **Airpak** will put O-grids around all objects, including those that contain other objects.


Mesh assemblies separately indicates whether or not **Airpak** should generate a non-conformal mesh for the assemblies that have the **Mesh separately** option turned on. This option allows you to turn off (or on) all defined non-conformal meshing for the entire model. See Section 26.8 for more information on non-conformal meshing.

Allow different subgrid types indicates whether or not **Airpak** should generate a mesh type for the assemblies different from the global mesh type, as specified under **Mesh type** in the **Meshing** tab of the **Assemblies** panel (Section 7.8.2). This option allows you to turn off (or on) different mesh types for all assemblies for the entire model.

8. Click **Generate mesh**. **Airpak** will generate the globally refined mesh, which you can view as described in Section 26.9. If you want to stop the meshing process before it is complete, click **Terminate mesher**.

26.5.2 Global Refinement for a Tetrahedral Mesh

The procedure for globally refining a tetrahedral mesh is as follows:

1. Open the **Mesh control** panel (Figure 26.4.2) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.

Model → Generate mesh

2. Select **Generate** at the top of the panel to access the controls for mesh generation.
3. Select **Tetra** in the **Mesh type** drop-down list.
4. To increase the overall number of elements, turn on the **Max tetra size** specification and set it to the desired maximum element length. A typical value is about 1/20 of the smallest room dimension.
5. For most cases, the remaining parameters in the **Mesh control** panel will not need to be changed. If you want, however, you can modify the default settings, which are defined as follows:

Smooth mesh activates a smoothing algorithm to improve the quality of the mesh generated by the tetrahedral mesher. This option is on by default, and you should generally *not* turn it off. When the smoothing algorithm is used, you can specify the quality (i.e., aspect ratio) desired in the **Up to quality** field. You can also change the maximum number of **Iterations** allowed to reach the desired quality. If **Airpak** does not reach the specified quality for all elements after the specified number of smoothing iterations have been performed, it will stop.

Coarsen mesh activates a coarsening algorithm that decreases the mesh count in the domain. This option is on by default, and you should generally *not* turn it off. When the coarsening algorithm is used, elements that are not needed are merged with adjacent elements. The **Min aspect** parameter determines which elements are candidates for being coarsened. If an element has an aspect ratio less than or equal to the specified **Min aspect**, the coarsening algorithm will attempt to merge it with an adjacent element. You can also change the maximum number of **Iterations** allowed to reach the desired minimum aspect ratio. If **Airpak** does not reach the specified minimum aspect ratio for all elements after the specified number of smoothing iterations have been performed, it will stop.

Per-object params enables local mesh refinement (see Section 26.6).

Mesh assemblies separately indicates whether or not **Airpak** should generate a non-conformal mesh for the assemblies that have the **Mesh separately** option turned on. This option allows you to turn off (or on) all defined non-conformal meshing for the entire model. See Section 26.8 for more information on non-conformal meshing.

Allow different subgrid types indicates whether or not **Airpak** should generate a mesh type for the assemblies different from the global mesh type, as specified under **Mesh type** in the **Meshing** tab of the **Assemblies** panel (Section 7.8.2). This option allows you to turn off (or on) different mesh types for all assemblies for the entire model.

Expansion ratio specifies the rate at which the tetrahedra in the mesh grow as they move away from the surfaces of objects. The size of the tetras in the fluid, $h_{t,f}$, at a distance d from a surface is defined as $h_{t,f} = h_{t,s} + (r - 1)d$, where $h_{t,s}$ is the size of the tetras on the object surface and r is the expansion ratio.

Cells in gap specifies the minimum number of elements between adjacent objects.

Default max curve size specifies the maximum size of an element on a curve. The default value of $1e10$ leaves the maximum size unlimited.

Default max surface size specifies the maximum size of an element on a surface. The default value of $1e10$ leaves the maximum size unlimited.

6. Additional meshing parameters (which you will generally not need to modify) are in the **Tetra params** panel (Figure 26.5.2), which you can open by clicking **Advanced parameters** in the Mesh control panel. These parameters are defined below.

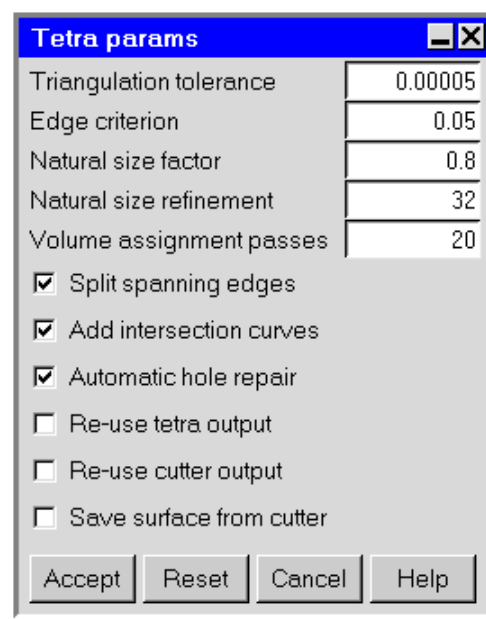


Figure 26.5.2: The Tetra params Panel

Triangulation tolerance specifies how closely the mesh matches the model geometry at boundaries of objects. For each surface or curve, a box is created around the object. The maximum dimension of the box is found and is multiplied by the **Triangulation tolerance** to give a local tolerance for the triangulation (placement of triangular element faces on the geometry) of the surface or curve.

The surface or curve is evaluated on a grid of points such that the piecewise linear approximation to the surface or curve deviates from the actual geometry

by an amount less than the local tolerance. A **Triangulation tolerance** greater than 0.01 can result in poor approximations to the geometry, and a value less than 10^{-5} will use too much memory.

Edge criterion controls the placement of mesh elements near object boundaries.

Natural size factor specifies the smallest element size, which is equal to the **Natural size factor** multiplied by the smallest feature length in the model.

Natural size refinement specifies the minimum number of nodes on a curved edge if no local minimum size applies. A value of 16, for example, means that a circle will be represented by 16 line segments.

Volume assignment passes specifies the number of times that the mesher will pass through the model dividing it up into smaller parts to be meshed.

Split spanning edges specifies that any edge in the mesh that has two nodes on surface elements will be split with a mid-node, along with the attached tetras, ensuring that there are no tetras with all four nodes on surfaces.

Add intersection curves specifies that the curve at the intersection of two objects should be computed explicitly. This is especially important for objects with curved shapes (e.g., a cylinder intersecting with another cylinder) because such an intersection curve has a somewhat complex shape. For the intersection between two rectangular objects, on the other hand, the intersection curve is a straight line, so no special computation is required.

Automatic hole repair (used only if you already have a mesh) instructs the mesher to try to fix problems automatically during the meshing process. If the mesher fails, turn this option on and try again.

Re-use tetra output (used only if you already have a mesh) uses the previous mesh as a starting point when a second attempt is made to mesh the model.

Re-use cutter output (used only if you already have a mesh) uses the previous subdivision of the model into smaller parts to be meshed, rather than dividing it up again.

Save surface from cutter generates only the surface mesh, not the volume mesh. The resulting mesh cannot be used in your **Airpak** simulation; it is only for diagnostic purposes if you are having trouble generating a mesh.

Click **Accept** to save any changes to your settings and close the **Tetra params** panel.

7. In the **Mesh control** panel, click **Generate mesh**. **Airpak** will generate the globally refined mesh, which you can view as described in Section 26.9. If you want to stop the meshing process before it is complete, click **Terminate mesher**.


26.6 Refining the Mesh Locally

As discussed in Sections 26.3.1 and 26.3.2, you will generally want to refine the minimum-count mesh after computing an initial solution. Steps for refining the mesh locally (i.e.,

on or near individual objects in the model) are presented in this section. See Section 26.5 for information about refining the mesh throughout the entire computational domain.

26.6.1 General Procedure

The general procedure for performing local mesh refinement is the same for both hexahedral and tetrahedral meshes; only the object-specific parameters differ between mesh types. The steps you should follow to refine the mesh locally are listed below:

1. Open the **Mesh control panel** (Figure 26.4.1 or 26.4.2) by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve toolbar**.
Model—>Generate mesh
2. Click on the **Generate** tab at the top of the panel to access the controls for mesh generation.
3. Select the appropriate **Mesh type** (**Hexa unstructured**, **Hexa cartesian**, **Tetra**, or **Mesher-HD**) in the drop-down list.
4. Enable the **Object params** option (note that this option is called **Per-object params** if you selected **Tetra** for the **Mesh type**) and click the **Edit** button beside it to open the **Per-object meshing parameters panel** (Figure 26.6.1), which allows you to define meshing parameters specific to each object in the model.

If you want to turn off the use of object-specific meshing parameters you can click the **Disable all** button.

5. To define meshing parameters for an object, select the object in the tree in the **Per-object meshing parameters panel**, and enable the **Use per-object parameters** option. This will allow you to set the object's meshing parameters. The parameters for each type of object are described in Sections 26.6.4–26.6.10. To specify a meshing parameter, enable the option, and then enter the desired value under **Requested**. (If you have already generated a mesh, the values for the current mesh will appear under **Actual**.)
6. After you have set all of the desired object-specific meshing parameters, click **Done** in the **Per-object meshing parameters panel** and then return to the **Mesh control panel** and click **Generate mesh**. **Airpak** will generate the locally refined mesh, which you can view as described in Section 26.9. If you want to stop the meshing process before it is complete, click **Terminate mesher**.

26.6.2 Definitions of Object-Specific Meshing Parameters

The following types of object-specific meshing parameters are used in **Airpak**:

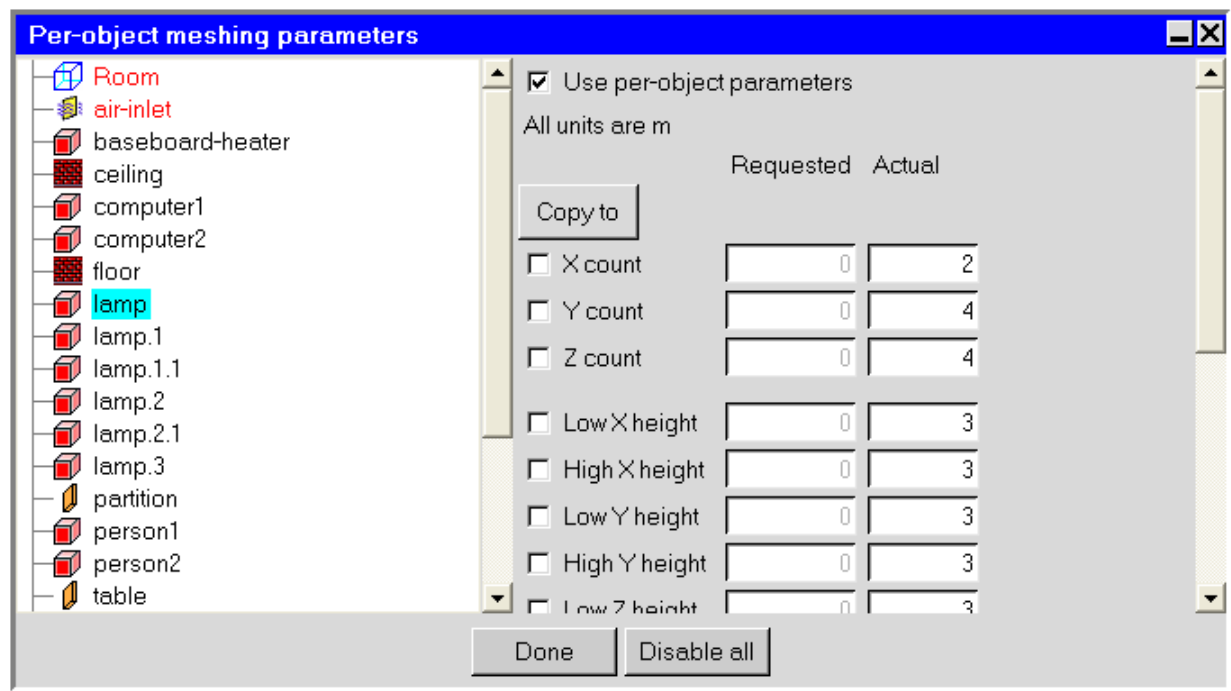


Figure 26.6.1: The Per-object meshing parameters Panel

- The *element count* is the number of divisions into which an edge is subdivided (i.e., the number of elements that lie along the edge). Counts can be associated with the edges of an object.
- The *initial element height* is the height of the first element on the surface of an object in the direction normal to that surface, pointing outside of the object.
- The *inward height* is the height of the first element on the surface of an object in the direction normal to that surface, pointing inside of the object.
- The *element height ratio* is the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the object, on the outside of the object. Together with the initial element height, this parameter controls mesh grading and density in the region near the surface of a modeling object, on the outside of the object.
- The *inward ratio* is the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the object, on the inside of the object. Together with the initial element height, this parameter controls mesh grading and density in the region near the surface of a modeling object, on the inside of the object.

- Planar surfaces and objects are considered to have *high* and *low* ends. Such surfaces and objects (unless they are inclined) are always aligned with a coordinate plane and normal to one of the coordinate directions. If, for example, a surface lies in the x - y plane, the z axis is normal to the plane. The high side of a plane or object is the side for which the normal points in the direction of increasing coordinate value. The low side is the side for which the normal points in the direction of decreasing coordinate value. Figure 26.6.2 shows an example of high- and low-side surface definitions for two surfaces aligned with the x - z and y - z planes.

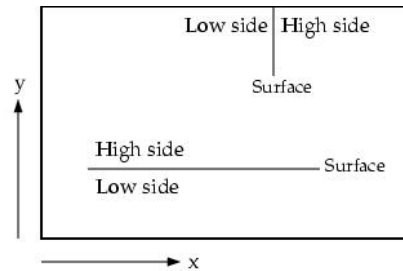


Figure 26.6.2: High- and Low-Side Surface Definition

Note that you will specify meshing parameters for each separate object in an assembly, accessed by expanding the assembly tree node in the **Model manager** window. By contrast, you will specify parameters for separate meshing of each assembly that you create, if you want to use non-conformal meshing. See Section 26.2 for more information about non-conformal meshing, and Section 26.8 for information on the procedure to set up non-conformal meshing around assemblies.

26.6.3 Defining Meshing Parameters for Multiple Objects

If you have two or more objects of the same type and shape (e.g., two prism blocks), you can assign object-specific meshing parameters to all of them at the same time in the **Per-object mesh parameters** panel. To accomplish this, hold down the <Control> key, and click the left mouse button on the name of each of the objects for which you want to use the same meshing parameters. When you then set the desired meshing parameters, they will apply to all selected objects.

Note that you can define the same meshing parameters only for objects of the same type *and* shape. You *cannot*, for example, define the same meshing parameters for a prism block and a cylinder block. Similarly, you cannot define the same meshing parameters for a circular fan and a circular vent.

26.6.4 Meshing Parameters for Rooms

The meshing parameters for a room, shown in Figure 26.6.3 for a hexahedral mesh, are as follows:

☐ Use per-object parameters

All units are m

	Requested	Actual
<input type="checkbox"/> Low X height	0	0.1
<input type="checkbox"/> High X height	0	0.1
<input type="checkbox"/> Low Y height	0	0.25
<input type="checkbox"/> High Y height	0	0.25
<input type="checkbox"/> Low Z height	0	0.356
<input type="checkbox"/> High Z height	0	0.356
<input type="checkbox"/> Low X ratio	2	2
<input type="checkbox"/> High X ratio	2	2
<input type="checkbox"/> Low Y ratio	2	2
<input type="checkbox"/> High Y ratio	2	2
<input type="checkbox"/> Low Z ratio	2	2
<input type="checkbox"/> High Z ratio	2	2

Figure 26.6.3: The Meshing Parameters Available for a Room

- For both hexahedral and tetrahedral meshes:
 - Initial element heights in the inward direction from each of the six surfaces that comprise the room boundaries (Low X height, Low Y height, Low Z height, High X height, High Y height, High Z height). See Section 26.6.2 for definitions of the high and low sides.
- For hexahedral meshes only:
 - Element height ratios in the inward direction from each of the six surfaces that comprise the room boundaries (Low X ratio, Low Y ratio, Low Z ratio, High X ratio, High Y ratio, High Z ratio). See Section 26.6.2 for an explanation of the element height ratio.

26.6.5 Meshing Parameters for Blocks

The meshing parameters for a block are described separately for each block shape.

Prism Blocks

The meshing parameters for a prism block, shown in Figure 26.6.4 for a hexahedral mesh, are as follows:

☐ Use per-object parameters

All units are m

	Requested	Actual
<input type="checkbox"/> X count	0	2
<input type="checkbox"/> Y count	0	25
<input type="checkbox"/> Z count	0	13
<input type="checkbox"/> Low X height	0	0.012
<input type="checkbox"/> High X height	0	0.012
<input type="checkbox"/> Low Y height	0	0.1
<input type="checkbox"/> High Y height	0	0.1
<input type="checkbox"/> Low Z height	0	0.1
<input type="checkbox"/> High Z height	0	0.1
<input type="checkbox"/> Low X ratio	2	2
<input type="checkbox"/> High X ratio	2	2
<input type="checkbox"/> Low Y ratio	2	2
<input type="checkbox"/> High Y ratio	2	2
<input type="checkbox"/> Low Z ratio	2	2
<input type="checkbox"/> High Z ratio	2	2
<input type="checkbox"/> Inward height	0	0.712
<input type="checkbox"/> Inward ratio	2	2

Figure 26.6.4: The Meshing Parameters Available for a Block

- For both hexahedral and tetrahedral meshes:
 - Initial element heights in the outward direction from each of the six surfaces of the block (Low X height, Low Y height, Low Z height, High X height, High Y height, High Z height). See Section 26.6.2 for definitions of the high and low sides.

- For hexahedral meshes only:
 - Number of elements in each direction on the block's edges (**X count**, **Y count**, **Z count**).
 - Element height ratios in the outward direction from each of the six surfaces of the block (**Low X ratio**, **Low Y ratio**, **Low Z ratio**, **High X ratio**, **High Y ratio**, **High Z ratio**). See Section 26.6.2 for an explanation of the element height ratio.
 - Initial element height in the inward direction from the surfaces of the block (**Inward height**).
 - Initial element height ratio in the inward direction from the surfaces of the block (**Inward ratio**).

Cylinder Blocks

The meshing parameters for a cylinder block are as follows:

- For both hexahedral and tetrahedral meshes:
 - Initial element heights in the outward direction from the bottom circular surface (**Low end height**) and the top circular surface (**High end height**).
 - Initial element height in the outward direction from the block's cylindrical side surface (**Side height**).
- For hexahedral meshes only:
 - Number of elements along the diameter of the circular surfaces (**Diameter count**) and along the side of the cylinder (**Side count**).
 - Initial element height in the inward direction from the cylindrical side surface (**Inward height**).
 - Element height ratios in the outward direction from the bottom circular surface (**Low end ratio**) and the top circular surface (**High end ratio**).
 - Element height ratios in the outward direction from the block's cylindrical side surface (**Side ratio**) and in the inward direction from the cylindrical side surface (**Inward ratio**).

3D Polygon Blocks

The meshing parameters for a 3D polygon block are as follows:

- Number of elements along each side of the block (**Side 1 count**, **Side 2 count**, etc.) and along the height of the block (**Height count**).

- Initial element heights in the outward direction from each side of the block (**Side 1 height**, **Side 2 height**, etc.) and from the bottom and top surfaces of the block (**Bottom height**, **Top height**).
- Element height ratios in the outward direction from each side of the block (**Side 1 ratio**, **Side 2 ratio**, etc.) and from the bottom and top surfaces of the block (**Bottom ratio**, **Top ratio**).

Ellipsoid Blocks

The meshing parameters for an ellipsoid block are as follows:

- For tetrahedral meshes only:
 - Initial element heights in the outward direction from the curved surface of the block (**Curved surface size**).
 - Initial element heights in the outward direction from the flat surfaces of the ellipsoid block that appear if you deactivate one or more octants of the block (**Flat (internal) surface size**).



Note that ellipsoidal blocks cannot be meshed with the hexahedral mesher.

Elliptical Cylinder Blocks

The meshing parameters for an elliptical cylinder block are as follows:

- Initial element heights in the outward direction from the bottom elliptical surface (**Low end height**) and the top elliptical surface (**High end height**).
- Initial element height in the outward direction from the block's cylindrical side surface (**Side height**).
- Element height ratios in the outward direction from the bottom elliptical surface (**Low end ratio**) and the top elliptical surface (**High end ratio**).
- Element height ratio in the outward direction from the block's cylindrical side surface (**Side ratio**).



It is recommended that you use the tetrahedral mesher when your model contains elliptical cylinder blocks.

26.6.6 Meshing Parameters for Fans

The meshing parameters for a cylinder or 3D polygon fan are the same as for a cylinder or 3D polygon block. See Section 26.6.5 for details.

The meshing parameters for a rectangular, circular, 2D polygon, or inclined fan are the same as for a rectangular, circular, 2D polygon, or inclined vent. See Section 26.6.7 for details.

26.6.7 Meshing Parameters for Vents

The meshing parameters for a vent are described separately for each vent shape.

Circular Vents

The meshing parameters for a circular vent, shown in Figure 26.6.5 for a hexahedral mesh, are as follows:

☐ Use per-object parameters

All units are m

	Requested	Actual
<input type="checkbox"/> Diameter count	0	
<input type="checkbox"/> Low end height	0	
<input type="checkbox"/> High end height	0	
<input type="checkbox"/> Inward height	0	
<input type="checkbox"/> Low end ratio	2	
<input type="checkbox"/> High end ratio	2	
<input type="checkbox"/> Inward ratio	2	

Figure 26.6.5: The Meshing Parameters Available for a Circular Vent

- For hexahedral meshes only:
 - Number of elements along the diameter of the vent (Diameter count).

- Initial element heights in the direction normal to the vent (**Low end height**, **High end height**). See Section 26.6.2 for definitions of the high and low sides.
If the vent is on an external wall, only the appropriate high or low side facing the interior of the room is applicable.
- Initial element heights in the inward direction (**Inward height**), measured from the outer radius.
- Element height ratios in the direction normal to the vent (**Low end ratio**, **High end ratio**). See Section 26.6.2 for definitions of the high and low sides.
If the vent is on an external wall, only the appropriate high or low side facing the interior of the room is applicable.
- Element height ratios in the inward direction (**Inward ratio**), measured from the outer radius.
- For tetrahedral meshes only:
 - Initial element height of the first element next to the vent (**Element height**).

Rectangular Vents

The meshing parameters for a rectangular vent are as follows:

- For hexahedral meshes only:
 - Number of elements along the edges of the vent (**X count**, **Y count**; **Y count**, **Z count**; or **X count**, **Z count**; depending on the plane in which the vent lies).
 - Initial element heights in the direction normal to the vent (**Low end height**, **High end height**). See Section 26.6.2 for definitions of the high and low sides.
If the vent is on an external wall, only the appropriate high or low side facing the interior of the room is applicable.
 - Initial element height in the inward direction (**Inward height**) for both coordinate directions on the surface of the vent.
 - Element height ratios in the direction normal to the vent (**Low end ratio**, **High end ratio**). See Section 26.6.2 for definitions of the high and low sides.
If the vent is on an external wall, only the appropriate high or low side facing the interior of the room is applicable.
 - Element height ratios in the inward direction (**Inward ratio**) for both coordinate directions on the surface of the vent.
- i** Only two of the three parameters (number of elements, inward height, and inward ratio) can be specified for any coordinate direction. The third parameter is determined from the other two.

- For tetrahedral meshes only:
 - Initial element height of the first element next to the vent (**Tetra size**).

Inclined Vents

The meshing parameters for an inclined vent are as follows:

- For hexahedral meshes only:
 - Number of elements in the coordinate direction along which the vent is inclined (**X count**, **Y count**, or **Z count**), and in the other two directions (**Y-Z count**, **X-Z count**, or **X-Y count**).
 - Initial element heights in the direction normal to the vent (**Low end height**, **High end height**). See Section 26.6.2 for definitions of the high and low sides.
 - Element height ratios in the direction normal to the vent (**Low end ratio**, **High end ratio**). See Section 26.6.2 for definitions of the high and low sides.
- For tetrahedral meshes only:
 - Initial element height of the first element next to the vent (**Tetra size**).

2D Polygon Vents

The meshing parameters for a 2D polygon vent are as follows:

- For both hexahedral and tetrahedral meshes:
 - Number of elements along each side of the vent (**Side 1 count**, **Side 2 count**, etc.).
 - Initial element heights in the outward direction from the bottom and top surfaces of the vent (**Bottom height**, **Top height**).
 - Element height ratios in the outward direction from the bottom and top surfaces of the vent (**Bottom ratio**, **Top ratio**).

26.6.8 Meshing Parameters for Openings

The meshing parameters for a rectangular, circular, 2D polygon, or inclined opening are the same as for a rectangular, circular, 2D polygon, or inclined vent. See Section 26.6.7 for details.

26.6.9 Meshing Parameters for Person Objects

The meshing parameters for the head, body, arms, legs, or thighs of a person are the same as for a prism block. See Section 26.6.5 for details.

The meshing parameters for the face of a person are the same as for a rectangular vent. See Section 26.6.7 for details.

26.6.10 Meshing Parameters for Walls

The meshing parameters for a wall are described separately for each wall shape.

Rectangular Walls

The meshing parameters for a rectangular wall, shown in Figure 26.6.6 for a hexahedral mesh, are as follows:

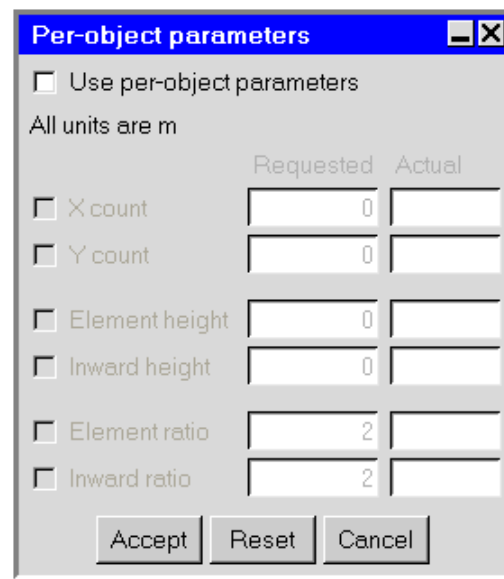


Figure 26.6.6: The Meshing Parameters Available for a Wall

- For hexahedral meshes only:
 - Number of elements along the edges of the wall (X count, Y count; Y count, Z count; or X count, Z count; depending on the plane in which the wall lies).
 - Initial element height in the direction normal to the wall (Element height).
 - Initial element height in the inward direction (Inward height) for both coordinate directions on the surface of the wall.

- Element height ratio in the direction normal to the wall (**Element ratio**).
- Element height ratio in the inward direction (**Inward ratio**) for both coordinate directions on the surface of the wall.



Only two of the three parameters (number of elements, inward height, and inward ratio) can be specified for any coordinate direction. The third parameter is determined from the other two.

- For tetrahedral meshes only:
 - Initial element height of the first element next to the wall (**Tetra size**).

Circular, 2D Polygon, and Inclined Walls

The meshing parameters for a circular, 2D polygon, or inclined wall are the same as for a circular, 2D polygon, or inclined vent, except no distinction is made between low and high end (side) for a wall. See Section 26.6.7 for details.

26.6.11 Meshing Parameters for Partitions

The meshing parameters for a rectangular, circular, 2D polygon, or inclined partition are the same as for a rectangular, circular, 2D polygon, or inclined vent. See Section 26.6.7 for details.

Note that you can specify the number of elements in the direction of the partition thickness using **Inward height**, or **Thickness count** for an inclined partition. If the partition has a zero thickness, this parameter is not applicable.

26.6.12 Meshing Parameters for Sources

The meshing parameters for a prism, cylinder, ellipsoid, or elliptical cylinder source are the same as for a prism, cylinder, ellipsoid, or elliptical cylinder block. See Section 26.6.5 for details.

The meshing parameters for a rectangular, circular, 2D polygon, or inclined source are the same as for a rectangular, circular, 2D polygon, or inclined vent. See Section 26.6.7 for details.

26.6.13 Meshing Parameters for Resistances

The meshing parameters for a prism, cylinder, or polygon resistance are the same as for a prism, cylinder, or polygon block. See Section 26.6.5 for details.

26.6.14 Meshing Parameters for Heat Exchangers

The meshing parameters for a rectangular, circular, inclined, or 2D polygon heat exchanger are the same as for a rectangular, circular, inclined, or 2D polygon vent. See Section 26.6.7 for details.

26.6.15 Meshing Parameters for Hoods

The meshing parameters for the neck of a hood are the same as for a prism block. See Section 26.6.5 for details.

The meshing parameters for the canopy of a hood are the same as for a 3D polygon block. See Section 26.6.5 for details.

The meshing parameters for the individual parts of the flange or the exhausts in the hood are the same as for a rectangular vent. See Section 26.6.7 for details.

26.6.16 Meshing Parameters for Assemblies

The meshing parameters for the bounding box of an assembly are essentially a combination of the per-object parameters for a hollow block and a room, and are available only for hexahedral meshes.

The external meshing parameters for an assembly begin with **Outside** and are the same as those for a hollow block (see Section 26.6.5), with the exception that the inward height and inward ratio are not applicable. The internal meshing parameters begin with **Inside**, and are the same as those for a room (see Section 26.6.4), with the addition of **Inside X max size**, **Inside Y max size**, and **Inside Z max size**. These three parameters are similar to the global maximum size parameters (i.e., **Max X size**, **Max Y size**, **Max Z size**) for the entire **Airpak** model (see Section 26.5.1), except that they apply to the interior of the assembly bounding box.

26.7 Controlling the Meshing Order for Objects

By default, objects are meshed in the order in which they were created. If two or more objects intersect, meshing the objects in the creation order may not give the result that you want. For such cases, you can modify the meshing order by changing the object priorities for the intersecting objects. The object with a higher priority (higher number) will retain the mesh where it intersects with an object with a lower priority.

To modify the meshing priorities, select **Edit priorities** in the **Model** menu.

Model→**Edit priorities**

This will open the **Object priority** panel (Figure 26.7.1), which allows you to change the meshing order of the objects that have been created. Simply edit the priorities as required, and then click **Accept** to save the new meshing order. You can also change the priorities by dragging an object up or down in the **Model manager** window.

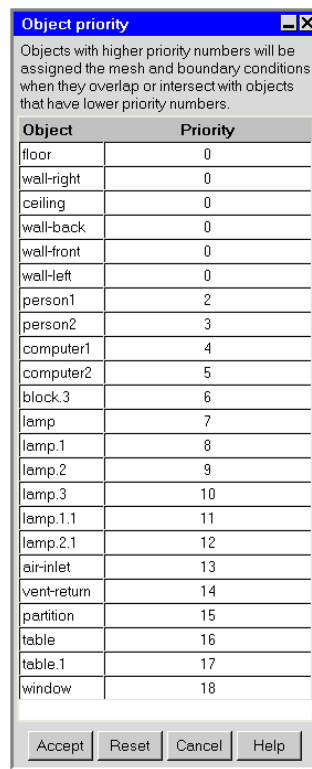


Figure 26.7.1: The Object priority Panel

26.8 Non-Conformal Meshing Procedures for Assemblies

In **Airpak**, assemblies of objects can be meshed separately. A region can be defined around a particular assembly and this region can be meshed independently of the mesh outside it. At the interface with the main model, there is a many-to-one transition between the mesh cells inside the region and those outside it, and a non-conformal interface is created at the boundary between the meshes. This allows for the embedding of a fine mesh within the sub-region around an assembly where it is needed, and for a coarse mesh in the rest of the model. To use non-conformal meshing for assemblies, you will use the following procedure:

1. Right-click on the assembly in the **Model manager**, and select **Edit object** from the context menu.
2. In the **Meshing** tab of the **Assemblies** panel, turn on the **Mesh separately** option.
3. If required, add a “slack” region around the assembly.
 - (a) Under **Slack**, enter the desired offset distances in the **Min X, Y, Z** and **Max X, Y, Z** directions. Positive values prescribe a slack region toward the exterior of the assembly.

By default the bounding box of the assembly has the exact extent necessary to enclose the objects that make the assembly. Defining a slack region allows you to make the bounding box slightly larger. This will make it possible to mesh more assemblies separately. Where a foreign object crosses the bounding box of an assembly, the non-conformal interface will cross through that object. See Section 7.8.2 for more details on specifying the slack distance around the bounding box of the assembly.



The bounding box must be large enough so that the cells on both sides of the box are either both fluid or both solid, with the same material.

Multiple assemblies can be meshed separately provided that:

- Assemblies are not partially embedded within one another (i.e., the bounding box of one assembly cannot intersect the bounding box of another assembly).
4. If required, specify the **Minimum gap** that separates objects in the assembly. See Section 7.8.2 for more details on specifying the **Minimum gap**.
 5. If required, specify the **Mesh type** to mesh the objects in the assembly. See Section 7.8.2 for more details on specifying the **Mesh type**.
 6. In the **Mesh control** panel, turn on the **Mesh assemblies separately** option.

Note that both the hexa-unstructured and hexa-cartesian meshers can be used to mesh assemblies separately, using non-conformal meshing. See Section 26.3.1 for

more information on hexahedral meshing in **Airpak**. The tetra mesher does not currently allow non-conformal meshing.

7. If required, turn on the **Allow different subgrid types** option in the **Mesh control** panel. This allows you to generate a mesh type for the assemblies different from the global mesh type, as specified under **Mesh type** in the **Meshing** tab of the **Assemblies** panel.


i When the meshing procedure has been completed, any assembly that has been meshed separately from the rest of the model will have its name displayed in a different color (default is pink) in the **Model manager** window.

26.9 Displaying the Mesh

Once you have generated a mesh, you can examine it to determine if it meets your needs. **Airpak** provides tools for viewing the mesh on individual objects, as well as on cross-sectional planes that extend throughout the room.

26.9.1 Displaying the Mesh on Individual Objects

The procedure for displaying the mesh on the surfaces or edges of objects or inside volumetric objects is as follows:

1. Open the **Mesh control** panel by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.

Model—→Generate mesh

2. Click on the **Display** tab to show the mesh display tools (see Figure 26.9.1).
3. Select one or more of the following display options:
 - Choose **Surface** to display the mesh on the exterior faces of objects. An example is shown in Figure 26.9.2.
 - Choose **Volume** to display the mesh inside volumetric objects (e.g., cylinder blocks, prism resistances). An example is shown in Figure 26.9.3.

i Note that the **Wire** option is selected by default. If the **Wire** option is not selected, the mesh lines will not be displayed.

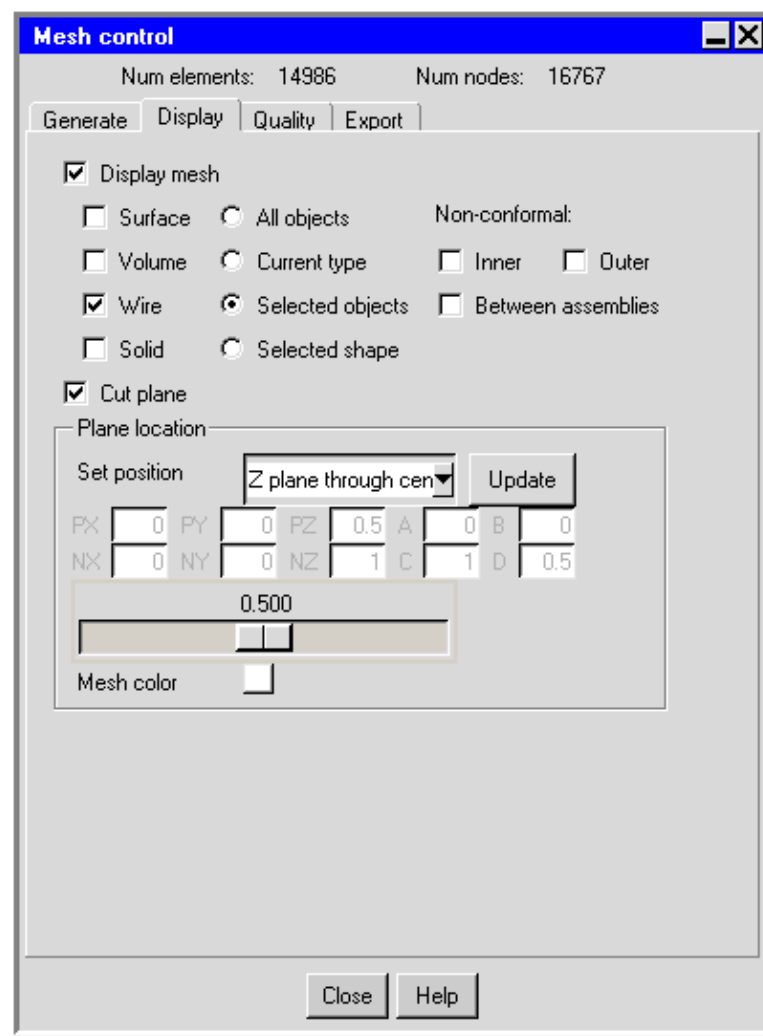


Figure 26.9.1: The Display Tab of the Mesh control Panel

4. Specify the objects for which you want to display the mesh:

- Choose **All objects** to display the mesh on all objects in the model.
- Choose **Current type** to display the mesh only on objects of the type currently selected in the **Model manager** window. For example, if a block object is selected, the mesh will be displayed only on block objects.
- Choose **Selected objects** to display the mesh only on the objects that are currently selected in the **Model manager** window. You can select the object either in the graphics window or in the **Model manager** window.
- Choose **Selected shape** to display the mesh only on objects of the shape currently selected in the **Model manager** window. For example, if a rectangular

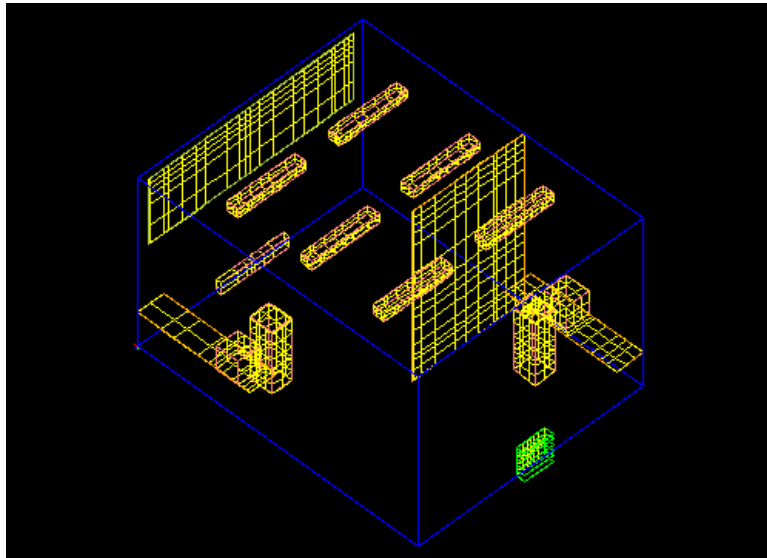


Figure 26.9.2: Display of Surface Elements

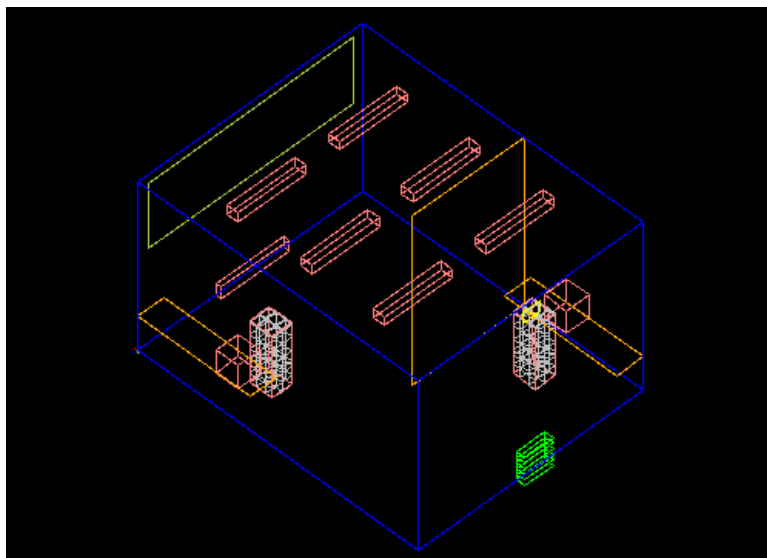


Figure 26.9.3: Display of Volume Elements

partition is selected, the mesh will be displayed only on 2D rectangular objects (e.g., rectangular vent, rectangular fan, etc.).

5. If you want to add filled shading to the mesh display, rather than displaying just the mesh lines, turn on the **Solid** option. When this option is off, the mesh will appear as shown in Figure 26.9.2. When it is on, the mesh will appear as shown in Figure 26.9.4. The mesh lines will not be displayed when the **Wire** option is off. When the **Solid** option is on and the **Wire** option is off, the mesh will appear as a solid without the mesh lines.

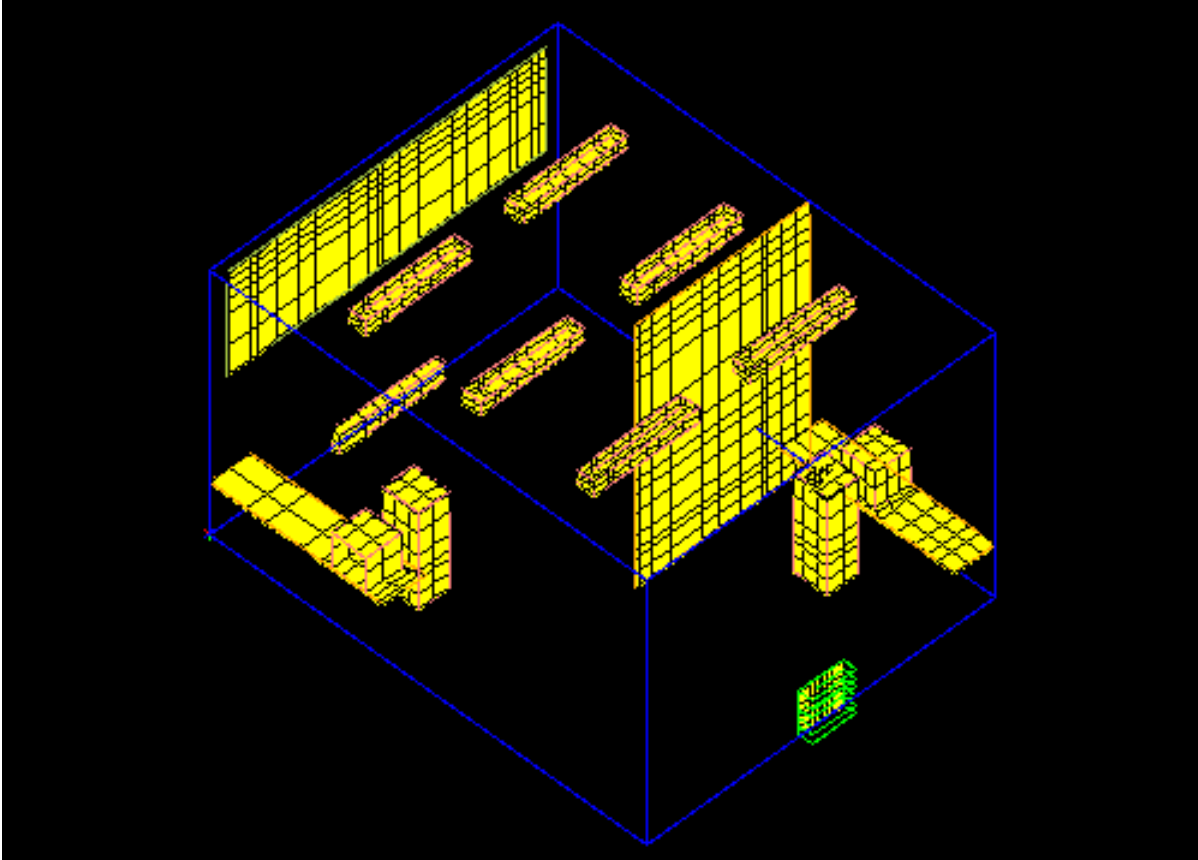


Figure 26.9.4: Filled Solid Display

6. Under **Non-conformal**, turn on **Inner** or **Outer**, or both, if you want **Airpak** to display the mesh on the inner side and/or the outer side of the non-conformal interface for assemblies that were meshed separately.
7. To display the common mesh faces between assemblies touching other assemblies, turn on **Between assemblies**.

8. Turn on the Display mesh option.

You can change the color of the mesh on objects using the Graphical styles panel. See Section 6.3.6 for details.

26.9.2 Displaying the Mesh on a Cross-Section of the Model

In addition to displaying the mesh on individual objects, as described in Section 26.9.1, you can also display the mesh on the intersection of a plane with the model (i.e., on a cross-section of the model). An example is shown in Figure 26.9.5.

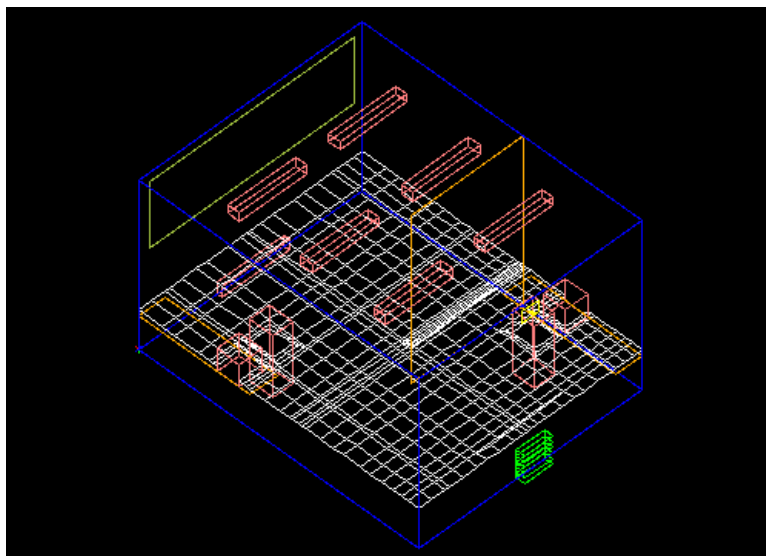



Figure 26.9.5: Mesh Display on a Cross-Sectional Plane

The procedure for displaying the mesh on a cross-sectional plane is as follows:

1. Open the Mesh control panel by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model** and solve toolbar.
Model—→Generate mesh
2. Click on the **Display** tab to show the mesh display tools (see Figure 26.9.1).
3. Turn on the **Cut plane** option.
4. Define the plane on which to display the mesh, using one of the following four methods:

- Specify a plane cut through the center of the model that is aligned with any Cartesian axis by selecting **X plane through center**, **Y plane through center**, or **Z plane through center** in the **Set position** drop-down list.
- Specify a point on the plane and the normal direction to the plane:
 - (a) Select **Point and normal** in the **Set position** drop-down list.
 - (b) Enter the coordinates of a point on the plane (PX, PY, PZ).

i If the **Display mesh** option is already turned on, you must press the **<Enter>** key on your keyboard after entering each value.

- (c) Enter a vector defining the direction normal to the plane (NX, NY, NZ). For example, entering (1, 0, 0) for the vector will define a normal pointing in the x direction.

i If the **Display mesh** option is already turned on, you must press the **<Enter>** key on your keyboard after entering each value.

- Specify an equation that defines the plane:
 - (a) Select the **Coeffs (Ax + By + Cz = D)** option.
 - (b) Enter the coefficients A, B, C, and D for the equation

$$Ax + By + Cz = D \quad (26.9-1)$$

i If the **Display mesh** option is already turned on, you must press the **<Enter>** key on your keyboard after entering each value.

- Specify the plane using your mouse:
 - (a) Use the **Orient** menu to specify the desired orientation. If you require a horizontal or vertical plane through your model, choose the orientation such that the plane of the display screen is perpendicular to the desired mesh-display plane. For example, if you want to display the mesh on a y - z or x - y plane, choose **Orient positive Y** as the orientation so that the display-screen plane is the x - z plane (see Figure 26.9.6). If you want to specify the plane by selecting three points on it, orient your model as desired.
 - (b) Select **Horizontal - screen select**, **Vertical - screen select**, or **3 points - screen select** in the **Set position** drop-down list.
 - (c) If you selected **Horizontal - screen select** or **Vertical - screen select**, click your left mouse button in the graphics window to indicate a point on the desired plane. **Airpak** will display the mesh on a horizontal or vertical plane perpendicular to the plane of the graphics screen and passing through the

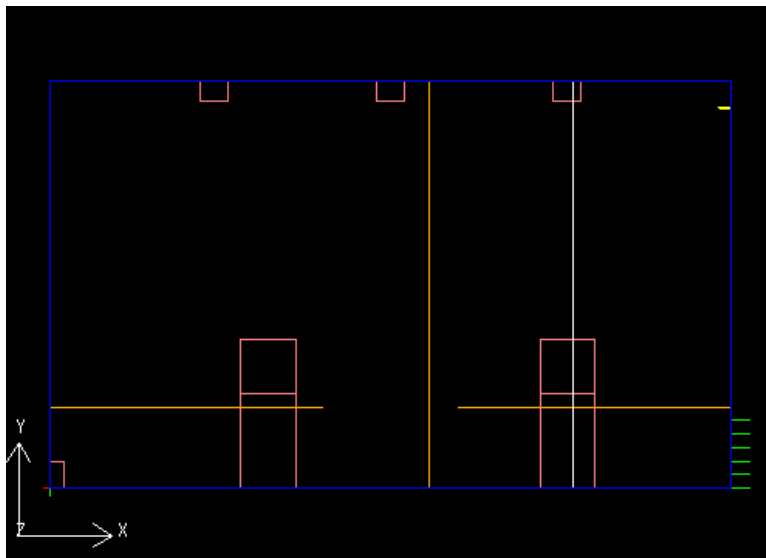


Figure 26.9.6: Specification of a Vertical Cross-Section

selected point. In Figure 26.9.6, the vertical mesh plane is indicated by a vertical line passing through the model. Figure 26.9.7 shows an isometric view of the same mesh plane.

If you selected 3 points - screen select, select the first, second, and third points on the plane in the graphics window using the left mouse button. Each point must be on the edge of an object or the room. If it is not, **Airpak** will move the point to the nearest location on the edge of an object or the room. **Airpak** will display the mesh on the plane defined by the three points.

i Note that if you display a hexahedral mesh on a plane that is not parallel or perpendicular to the sides of the room, the mesh elements may look like the elements of a tetrahedral mesh. This is an optical illusion caused by **Airpak** when it displays a hexahedral mesh on a plane that is not horizontal or vertical relative to the sides of the room.

5. Turn on the **Display mesh** option.
6. If you want to change the color of the mesh display, click on the colored square next to **Mesh color**, and select a new color. Acceptable entries include white (the default), red, blue, green, orange, yellow, etc.
7. To advance the currently-displayed mesh plane through the room so that you can easily view the mesh on different cross-sectional planes, use the slider bar next to

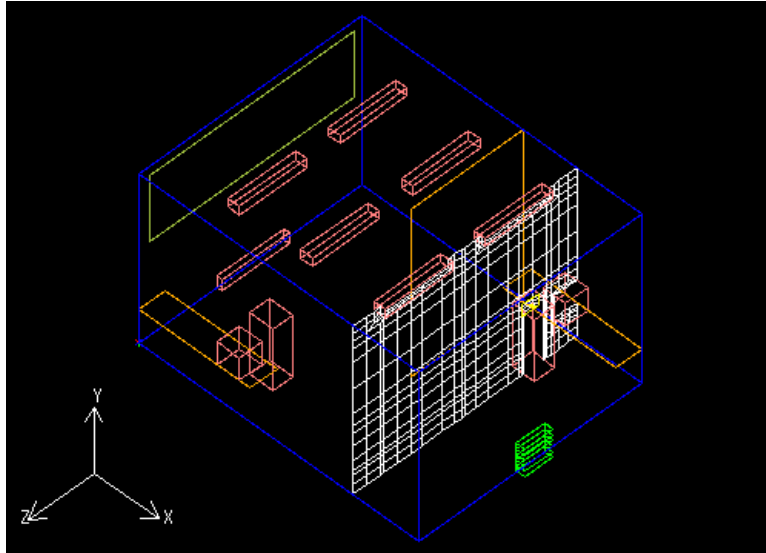


Figure 26.9.7: Isometric View of Cross-Sectional Mesh Display

Cut plane. Airpak will move the plane forward or backward along an axis normal to the mesh plane by a specified percentage of the total room length.


26.10 Checking the Mesh

Airpak provides three ways to check the quality of your mesh: distortion, face alignment, and element volume.

26.10.1 Checking the Element Quality

For a hexahedral element, the quality of an element is defined as the determinant of the Jacobian matrix, which is a measure of the element's distortion. For a tetrahedral element, the quality is defined as the ratio of its volume to that of the largest equilateral tetrahedron that will fit inside the circumsphere of the element. The best elements are generally those with a quality close to 1. A value less than 0.15 indicates a highly distorted element.

The procedure for checking the quality is as follows:

1. Open the Mesh control panel by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.

Model → Generate mesh

2. Click on the **Quality** tab to show the mesh diagnostic tools.

3. Select the **Quality** option. **Airpak** will show a histogram of the element quality, as shown in Figure 26.10.1.

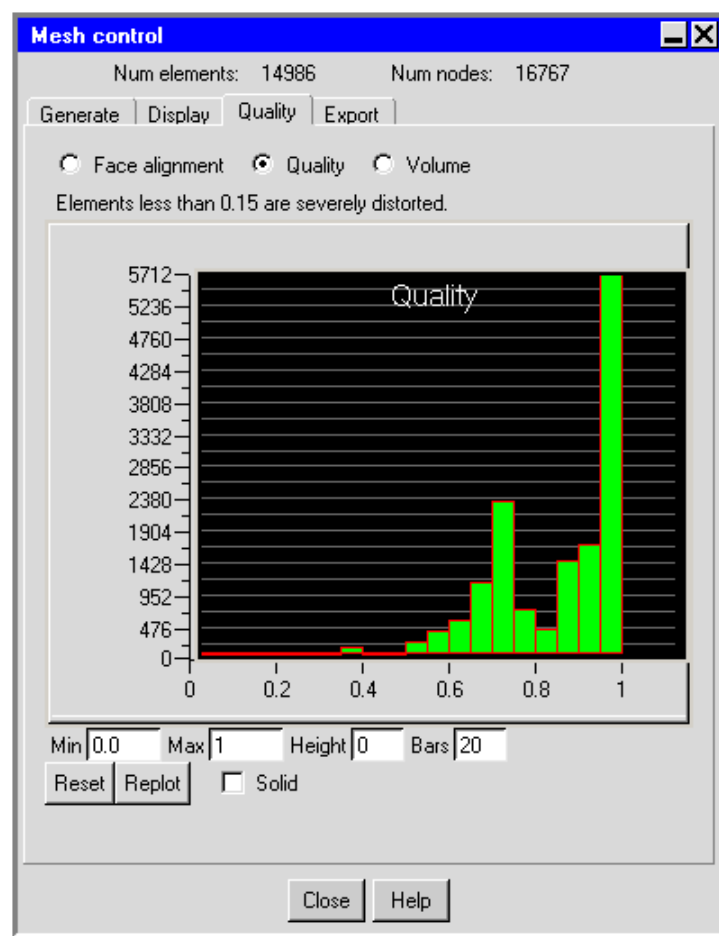


Figure 26.10.1: The Quality Histogram in the Mesh control Panel

4. If you want to modify the range of quality values viewed, enter a new value in the **Min** or **Max** field and then press the **<Enter>** key on your keyboard or click **Replot** to update the histogram. To modify the maximum height of the bars or the number of bars in the histogram, enter a new value in the **Height** and/or **Bars** fields and click **Replot**. (Note that a **Height** of 0 instructs **Airpak** to display the bars of the histogram at their full height.) To return to the default ranges, click the **Reset** button.
5. To view the elements of the mesh within a particular range of quality values, click on a bar in the histogram. **Airpak** will display the elements in the selected range in the graphics window. Select the **Solid** option if you want to view these elements with solid shading.

26.10.2 Checking the Face Alignment

Face alignment is a measure of mesh quality defined by

$$\text{face alignment index} = c_0 \vec{c}_1 \cdot \vec{f} \quad (26.10-1)$$

where c_0 and c_1 are the centroids of two adjacent elements, and \vec{f} is the normal vector to the face between the two elements, as shown in the 2D example in Figure 26.10.2.

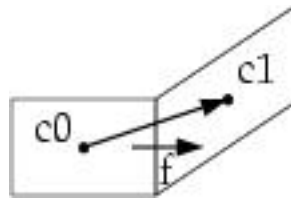



Figure 26.10.2: Definition of Face Alignment

Adjacent mesh faces that are not aligned can result in long, narrow elements. A value of 1 indicates perfect alignment. Values less than 0.15 indicate a severely distorted mesh.

The procedure for checking the face alignment is as follows:

1. Open the **Mesh control** panel by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.
Model → Generate mesh
2. Click on the **Quality** tab to show the mesh diagnostic tools.
3. Select the **Face alignment** option. **Airpak** will show a histogram of the face alignment, as shown in Figure 26.10.3.
4. If you want to modify the range of face alignments viewed, enter a new value in the **Min** or **Max** field and then press the **<Enter>** key on your keyboard or click **Replot** to update the histogram. To modify the maximum height of the bars or the number of bars in the histogram, enter a new value in the **Height** and/or **Bars** fields and click **Replot**. (Note that a **Height** of 0 instructs **Airpak** to display the bars of the histogram at their full height.) To return to the default ranges, click the **Reset** button.
5. To view the elements of the mesh within a particular range of face alignments, click on a bar in the histogram. **Airpak** will display the elements in the selected range in the graphics window. Select the **Solid** option if you want to view these elements with solid shading.

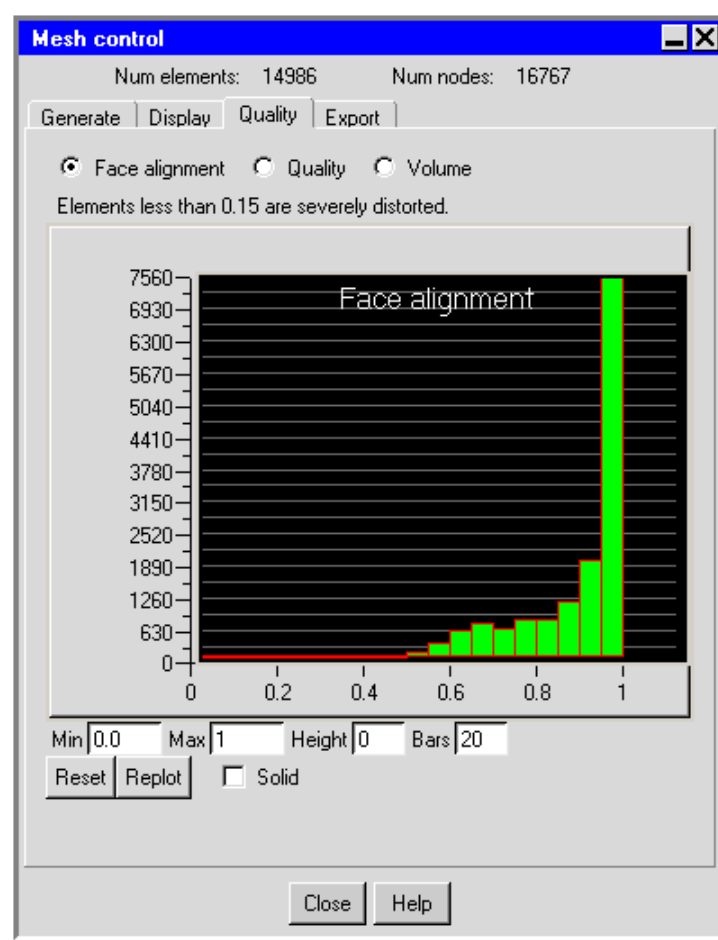



Figure 26.10.3: The Face alignment Histogram in the Mesh control Panel

26.10.3 Checking the Element Volume

Extremely small elements (on the order of 10^{-12} or lower) can lead to problems in the solver. It is therefore important to check the minimum element volumes in your mesh. Although this situation should and can be avoided in most models, if your model must have very small elements, you may want to use the double precision version of the solver (see Section 27.6).

The procedure for checking the element volume is as follows:

1. Open the Mesh control panel by selecting **Generate mesh** in the Model menu, or by clicking on the  button in the Model and solve toolbar.
Model → Generate mesh
2. Click on the **Quality** tab to show the mesh diagnostic tools.

3. Select the **Volume** option. **Airpak** will show a histogram of the element volume, as shown in Figure 26.10.4.

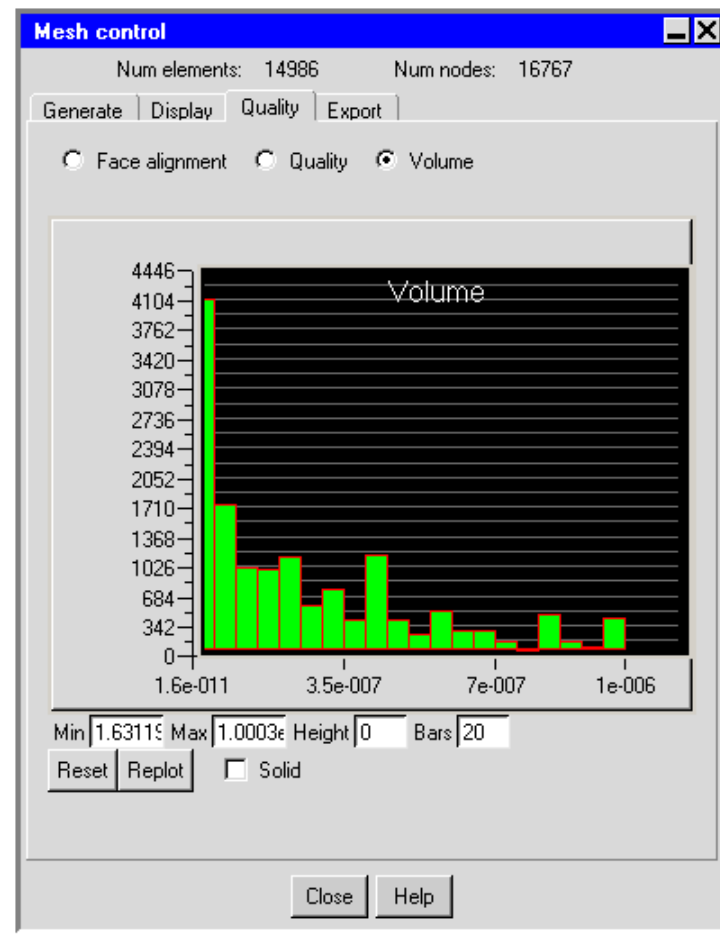



Figure 26.10.4: The Volume Histogram in the Mesh control Panel

4. If you want to modify the range of volumes viewed, enter a new value in the **Min** or **Max** field and then press the **<Enter>** key on your keyboard or click **Replot** to update the histogram. To modify the maximum height of the bars or the number of bars in the histogram, enter a new value in the **Height** and/or **Bars** fields and click **Replot**. (Note that a **Height** of 0 instructs **Airpak** to display the bars of the histogram at their full height.) To return to the default ranges, click the **Reset** button.
5. To view the elements of the mesh within a particular range of volumes, click on a bar in the histogram. **Airpak** will display the elements in the selected range in the graphics window. Select the **Solid** option if you want to view these elements with solid shading.

26.11 Exporting a Mesh

Airpak allows you to export mesh data to I-DEAS or ANSYS. The general procedure for exporting a mesh file from Airpak is as follows:

1. Open the Mesh control panel by selecting **Generate mesh** in the **Model** menu, or by clicking on the  button in the **Model and solve** toolbar.

Model—→Generate mesh

2. Click on the **Export** tab to show the export options, as shown in Figure 26.11.1.

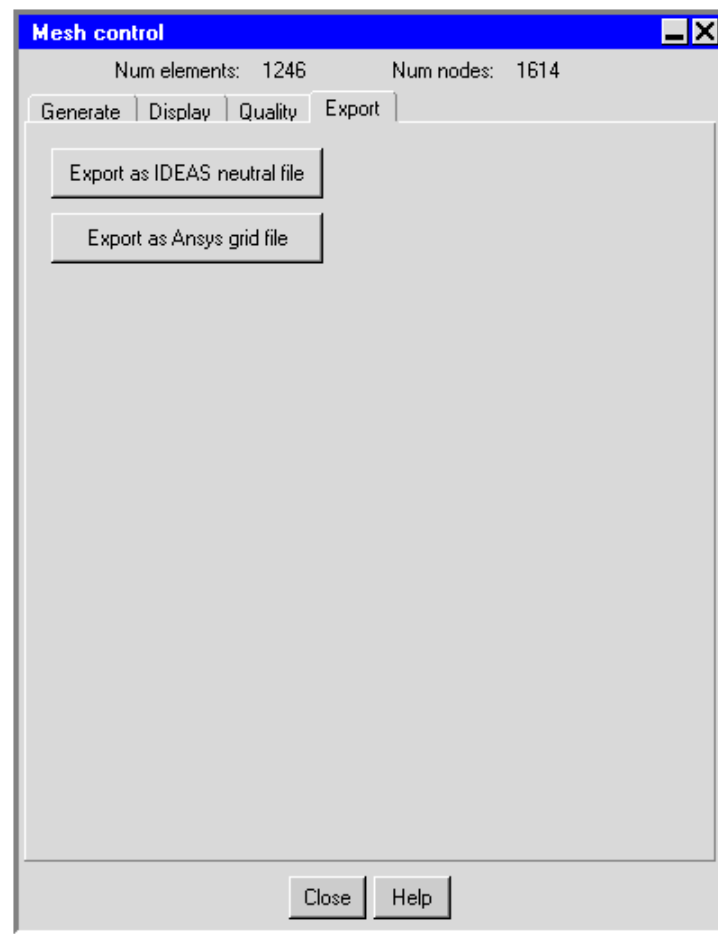


Figure 26.11.1: The Export Tab in the Mesh control Panel

3. Click the appropriate button for exporting either an I-DEAS or an ANSYS file and follow the procedure in Section 26.11.1 or 26.11.2.

26.11.1 Exporting an I-DEAS Neutral File

To export an I-DEAS neutral file from Airpak, use the following procedure:

1. In the Export tab of the Mesh control panel, click Export as IDEAS neutral file. This will open the I-DEAS neutral file panel (Figure 26.11.2).

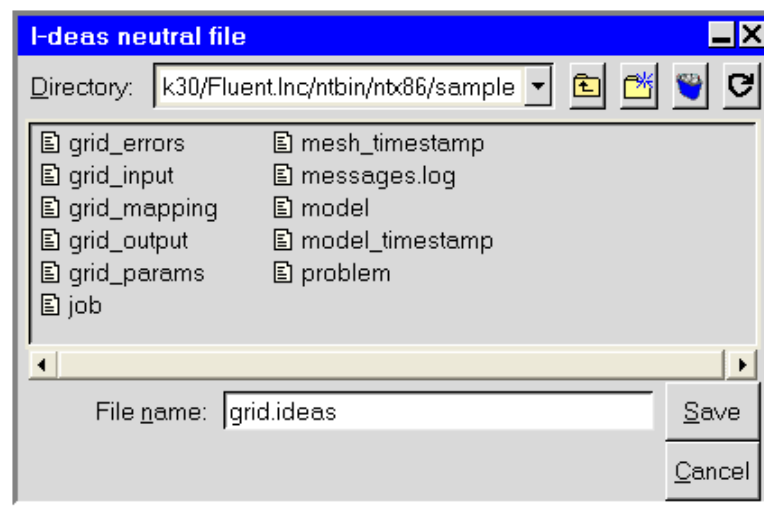


Figure 26.11.2: The I-DEAS neutral file Panel

2. In the I-DEAS neutral file panel, enter a name for the file to be saved (with a `.ideas` extension) next to File name and click Save. The Message window will report when the file has been written.

26.11.2 Exporting an ANSYS Grid File

To export an ANSYS grid file from Airpak, use the following procedure:

1. In the Export tab of the Mesh control panel, click Export as Ansys grid file. This will open the ANSYS file panel (Figure 26.11.3).
2. In the ANSYS file panel, enter a name for the file to be saved (with a `.ansys` extension) next to File name and click Save. The Message window will report when the file has been written.

26.12 Loading an Existing Mesh

When you read an existing job into Airpak, the mesh is not automatically loaded. If you have already created a mesh for the model, you can load it into Airpak by clicking the Load mesh button in the Mesh control panel (Figure 26.4.1 or 26.4.2).

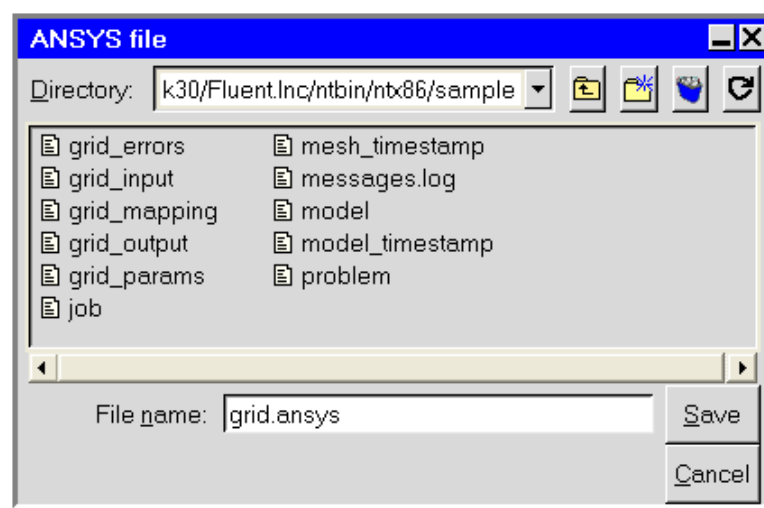


Figure 26.11.3: The ANSYS file Panel

Once you have built your model (see Chapters 8–19) and generated a mesh (see Chapter 26), you are ready to calculate a solution. **Airpak** allows you to specify the parameters that control the solution procedure and to monitor the solution. The functions that are needed to define the solution procedure are found in the **Model manager** window, under the **Problem setup** and **Solution settings** nodes, and in the **Solve** menu. This chapter begins with an overview of the **Solve** menu and the solution settings in the **Model manager** window, and then describes how to define the solution procedure for your model. Once you have calculated a solution for your simulation, you will go on to examine the results of the simulation, as described in Chapters 28 and 29.

The information in this chapter is divided into the following sections:

- Section 27.1: Overview
- Section 27.2: General Procedure for Setting Up and Calculating a Solution
- Section 27.3: Choosing the Discretization Scheme
- Section 27.4: Setting Under-Relaxation Factors
- Section 27.5: Selecting the Multigrid Scheme
- Section 27.6: Selecting the Version of the Solver
- Section 27.7: Initializing the Solution
- Section 27.8: Monitoring the Solution
- Section 27.9: Defining Postprocessing Objects
- Section 27.10: Defining Reports
- Section 27.11: Setting the Solver Controls
- Section 27.12: Performing Calculations
- Section 27.13: Diagnostic Tools for Technical Support

27.1 Overview

The Solve menu (Figure 27.1.1) is used in conjunction with the **Problem setup** and **Solution settings** nodes of the **Model manager** window to define the solution parameters for your Airpak model.

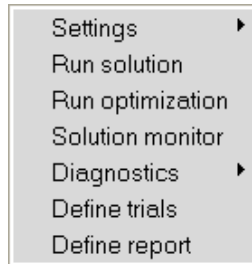








Figure 27.1.1: The Solve Menu

The functions you need to define the solution parameters for your Airpak simulation include the following:

- specifying the parameters that control the solver
 **Problem setup** →  **Basic parameters**
- changing the variables to be monitored during the solution calculation
Solve → **Solution monitor**
- defining the solution procedure
 **Solution settings** →  **Basic settings** or **Solve** → **Settings** → **Basic**
 **Solution settings** →  **Advanced settings** or **Solve** → **Settings** → **Advanced**
- accessing or creating files for use by your technical support engineer
Solve → **Diagnostics**
- defining the format for reports
Solve → **Define report**

In addition to solution control functions, the **Solve** menu contains a function related to managing multiple solution trials (**Solve** → **Define trials**). This function is described in Chapter 24.

27.2 General Procedure for Setting Up and Calculating a Solution

Once you have built your model in **Airpak**, you are ready to calculate a solution. The following steps outline a general procedure you can follow:



1. Choose the discretization scheme (see Section [27.3](#)).
2. Set the under-relaxation factors (see Section [27.4](#)).
3. Select the multigrid scheme (see Section [27.5](#)).
4. Select the version of the solver (single precision or double precision) (see Section [27.6](#)).
5. Initialize the solution (see Section [27.7](#)).
6. Enable the appropriate solution monitors (see Section [27.8](#)).
7. Define postprocessing objects prior to solving the model (see Section [27.9](#)).
8. Define the reports that you want **Airpak** to create when the solution is complete (see Section [27.10](#)).
9. Specify the parameters that control the solver (see Section [27.11](#)).
10. Start calculating (see Section [27.12](#)).

The default settings for the first three items listed above are suitable for most problems and need not be changed. The following sections outline how these and other solution parameters can be changed, and when you may wish to change them. The steps listed above apply for all steady-state calculations. See Chapter [20](#) for information about unsteady solution procedures.

27.3 Choosing the Discretization Scheme

Airpak allows you to choose the discretization scheme for the convection terms of each governing equation. See Section 31.6.2 for a complete description of the discretization schemes available in Airpak.

You can choose the discretization scheme to be used in the Advanced solver setup panel (Figure 27.3.1).

 Solution settings →  Advanced settings (or Solve → Settings → Advanced)

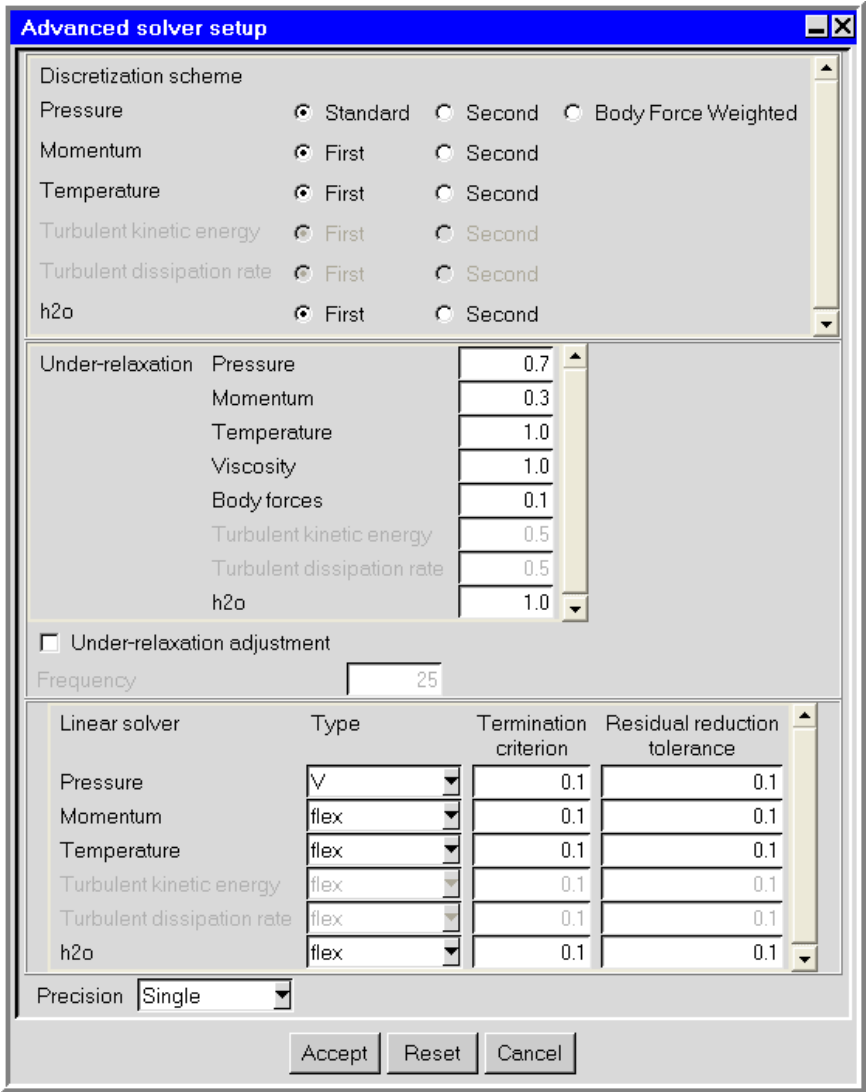


Figure 27.3.1: The Advanced solver setup Panel

The discretization scheme to be used for each of the equations that Airpak solves is

specified under **Discretization scheme** in the **Advanced solver setup** panel. By default, all equations (except the pressure equation) are solved using a first-order scheme. This scheme gives you a relatively quick and accurate solution. The second-order scheme is available for cases in which you require a more accurate solution. Be aware, however, that a second-order solution may take longer to converge. Also, for a second-order calculation it is recommended that you first compute a first-order solution and use it as a starting point.

By default, the pressure equation is solved using the **Standard** scheme. The **Body Force Weighted** scheme is also available, but it is recommended that you keep the default standard scheme for most cases. A second-order scheme is also available for cases in which you require a more accurate solution.

27.4 Setting Under-Relaxation Factors

As discussed in Section 31.6.2, **Airpak** uses under-relaxation to control the update of computed variables at each iteration. This means that all equations solved using **Airpak** will have under-relaxation factors associated with them.

You can set the under-relaxation factors in the **Advanced solver setup** panel (Figure 27.3.1).



Solution settings → **Advanced settings** (or **Solve** → **Settings** → **Advanced**)

The value of the “under-relaxation factor” for each of the equations that **Airpak** solves is specified under **Under-relaxation** in the **Advanced solver setup** panel. Because of the nonlinearity of the equation set, it is necessary to reduce the change of a variable from one iteration to the next. This is referred to as under-relaxation. For example, if the pressure under-relaxation factor is 0.3, the change in pressure value from one iteration to the next will be restricted to 30% of the difference between the initial value and the newly calculated value.

In **Airpak**, the default under-relaxation parameters for all variables are set to values that are near optimal for the largest possible number of cases. These values are suitable for many problems, but for some particularly non-linear problems (e.g., some turbulent flows or high-Rayleigh-number natural-convection problems) it is prudent to reduce the under-relaxation factors initially. If you click **Reset** in the **Basic settings** panel (Figure 27.7.1), **Airpak** adjusts the under-relaxation factors to values that are recommended for the type of problem you are trying to solve.

It is good practice to begin a calculation using the default settings in the **Advanced solver setup** panel. For most flows, the default under-relaxation factors do not usually require modification. If the solution exhibits unstable or divergent behavior, you may need to modify the under-relaxation factors; however, inappropriate under-relaxation factors may decrease the rate of convergence for your problem. Please contact your technical support engineer if your solution displays divergent behavior.

27.5 Selecting the Multigrid Scheme

As discussed in Section 31.6.4, **Airpak** uses a multigrid scheme to accelerate solution convergence. You can set the parameters related to the multigrid solver in the **Advanced solver setup** panel (Figure 27.3.1).



Solution settings → **Advanced settings** (or **Solve** → **Settings** → **Advanced**)

You can specify parameters related to the multigrid solver under **Linear solver** in the **Advanced solver setup** panel. The multigrid solver accelerates solution convergence by using a sequence of coarse grids based on the computational grid created by **Airpak**. Solutions can be computed more quickly on a coarse grid, but coarse-grid solutions are not as accurate as fine-grid solutions. For this reason, **Airpak** uses the coarser grid levels to obtain a starting point for the final solution, which is obtained for the actual **Airpak** grid. The following options are available for the multigrid solver.

Type specifies the multigrid cycle type for each equation that **Airpak** solves. (See Section 31.6.4 for a description of the different multigrid cycles.) By default, the **V** cycle is used for the pressure equation and the **flex** cycle is used for all other equations. You should generally not need to modify these settings.

Termination criterion controls the multigrid solver in different ways for different cycles. For the **flex** cycle, the **Termination criterion** governs when the solver should return to a finer grid level (i.e., when the residuals have improved sufficiently on the current level). For the **V** and **W** cycles, the **Termination criterion** determines whether or not another cycle should be performed on the finest (original) grid level. If the current residual on the finest level does not satisfy the **Termination criterion**, **Airpak** will perform another multigrid cycle.

For most cases, you should not need to modify the settings for the **Termination criterion**.

Residual reduction tolerance dictates when a coarser grid level must be visited (due to insufficient improvement in the solution on the current level). This parameter is used only by the **flex** cycle. With a larger value of the **Residual reduction tolerance**, coarse levels will be visited less often (and vice versa).

For most cases, you should not need to modify the settings for the **Residual reduction tolerance**.

27.6 Selecting the Version of the Solver

In the **Advanced solver setup** panel (Figure 27.3.1), select the version of the solver you want to use in the **Precision** drop-down list.

Single specifies that the single-precision version of the solver is to be used.

Double specifies that the double-precision version of the solver is to be used.

Both single-precision and double-precision versions of **Airpak** are available on all computer platforms. For most cases, the single-precision solver will be sufficiently accurate, but certain types of problems may benefit from the use of a double-precision solver. Several examples are listed below:

- For conjugate problems with high thermal-conductivity ratios and/or high-aspect-ratio grids, convergence and/or accuracy may be impaired with the single-precision solver, due to inefficient transfer of boundary information.
- If your geometry has features of very disparate length scales (e.g., a quasi 2D model), single-precision calculations may not be adequate to represent the node coordinates.

27.7 Initializing the Solution

Before starting your CFD simulation, you must supply **Airpak** with the number of iterations to be performed and the criteria **Airpak** should use to check for convergence. You can specify the number of iterations and the convergence criteria in the **Basic settings** panel (Figure 27.7.1).



Solution settings → **Basic settings** (or **Solve** → **Settings** → **Basic**)

The following values should be specified in the **Solver setup** panel.

1. Specify the **Number of iterations** to be performed by **Airpak** during the calculation. This specifies the number of solution iterations to be performed in a steady-state calculation. The calculation will stop when these iterations have been performed or the **Convergence criteria** are satisfied, whichever happens first. For relatively simple models, the default of 100 should be sufficient for the solution to converge, but for more complex models you may need to increase this value. See Chapter 20 for details on specifying the number of iterations for a transient simulation.
2. Specify the **Convergence criteria**. These are the solution-residual values used to determine convergence. Solution residuals measure the error or imbalance in the conservation equations that **Airpak** solves, and are defined in Section 31.6.5. When

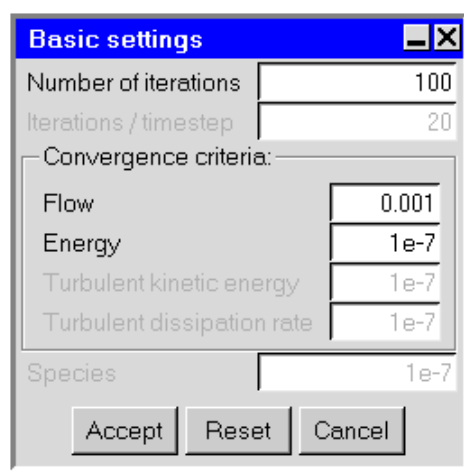


Figure 27.7.1: The Basic settings Panel

all solution residuals (e.g., flow, energy, turbulence, species) are less than or equal to their specified convergence criteria, the solution will be considered converged. If you click **Reset** in the **Basic settings** panel, **Airpak** will adjust the convergence criteria to values that are suitable for the type of problem you are trying to solve.




Note that when you click on **Reset** in the **Basic settings** panel, **Airpak** computes approximate values of the Reynolds, Peclet, Rayleigh, and Prandtl numbers, based on the physical characteristics of the model you have defined, and displays them in the **Message** window. See Section 6.4.4 for more details on the Reynolds, Peclet, Rayleigh, and Prandtl numbers.

27.8 Monitoring the Solution

The equations of flow and heat transfer are highly non-linear. To solve these equations numerically, an iterative procedure is required. Monitoring the convergence visually allows you to observe the ongoing results of the iterative procedure for each of the primary variables, and to determine the rate of convergence and how close the solution is to completion. See Section 31.6.5 for more details on solution convergence.

27.8.1 Defining Solution Monitors

The **Solution monitor parameters** panel (Figure 27.8.1) allows you to specify the variables to be monitored during the calculation. To open the **Solution monitor parameters** panel, select **Run solution** in the **Solve** menu or click on the  button in the **Model and solve** toolbar.

Solve—>Run solution

This opens the **Solve** panel. Next, select the **Start monitor** option and click on the **Edit parameters** button. This will open the **Solution monitor parameters** panel.

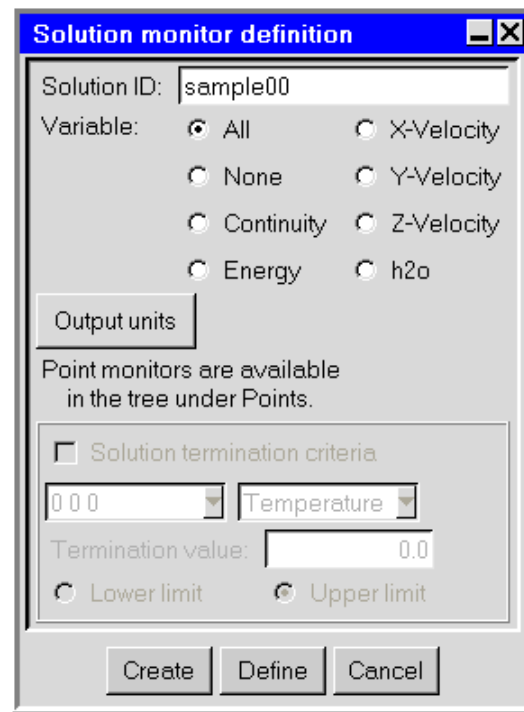


Figure 27.8.1: The Solution monitor definition Panel

To define the solution monitors for your simulation, follow the steps below.

1. Specify a **Solution ID** for the solver calculation for which you wish to view the solution residuals and point monitors. This defines a unique identifier for the solution results. **Airpak** automatically creates a default name that you can alter by typing a new name in the **Solution ID** text entry field. The default name consists of the project name plus a two-digit sequential number, starting with 00. When you click **Create** the solution residuals and point monitors for the selected calculation will be displayed in separate **Solution residuals** and **Monitor** graphics display and control windows.
2. Specify the **Variable** for which the convergence is to be displayed. The following options are available:
 - All** enables the display of residuals for pressure, temperature, and the x , y , and z velocity components.
 - None** disables the display of all residuals.

Continuity enables the display of the pressure residual.

Energy enables the display of the temperature residual.

X-Velocity, **Y-Velocity**, **Z-Velocity** enable, respectively, the residuals for the x -velocity, y -velocity, and z -velocity components.

Airpak will also display the names of the species that you have included in your problem in the **Variable** list (e.g., **h2o** in Figure 27.8.1). You can enable the display of the residual for an individual species by selecting the name of the species under **Variable**.

3. To specify the units for postprocessing the variables, click on **Output units**. See Section 5.5 for details on selecting units for postprocessing.
4. (transient cases only) You can specify a limit for a solution monitor variable at a specific point. If this limit is met while the solver is running, the solution will be terminated.

The procedure for defining such a limit is as follows:

- (a) Turn on the **Solution termination criteria** option.
- (b) Specify the location of the point in the model where you want to monitor the solution. See step 2 for details.
- (c) Select the variable that you want to monitor from the variable drop-down list.
- (d) Specify the **Termination value** for the selected variable and whether this value is a **Lower limit** or an **Upper limit**.

See Chapter 20 for more information about transient simulations.

To accept any changes you have made to the panel and then close the panel, click **Accept**. To undo all the changes you have made in the panel and restore all items in the panel to their original states when the panel was opened, click on the **Reset** button. To close the panel and ignore any changes made to the panel, click **Cancel**.

Note that solution residuals and point monitors will be displayed in different **Monitor** graphics display and control windows (see Section 27.12.2).

Object Monitor Points

As noted at the bottom of the **Solution monitor definition** and **Solution monitor parameters** panels, you can create per-object monitors in the **Model manager** window. To define a monitor point for an object, drag the object from under the **Model** node into the **Points** node. Alternatively, you can copy the object to the clipboard, right-click on the **Points** node, and select **Paste from clipboard** in the pull-down menu. Double-click on the object item to open the **Modify point** panel (Figure 27.8.2).

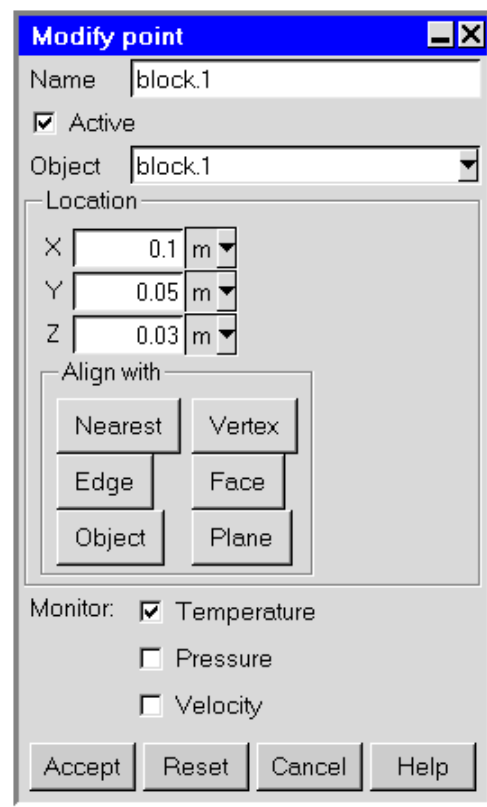


Figure 27.8.2: The Modify point Panel

The monitor point is defined, by default, as the centroid (2D or 3D) of the selected object. The values of the variables at this point are calculated as the average of the values at the vertices of the object. You can modify the location of the point by entering the appropriate values under **Location**, or by using the alignment tools under **Align with**. By using the alignment tools, you can align the point with a selected point (**Nearest**), **Edge**, **Object**, **Vertex**, **Face**, or **Plane**. To select any of these elements from the graphics window, you will use the left mouse button. To accept the modification, click the middle mouse button, and to cancel the modification, click the right mouse button. On a per-object basis, you can monitor the **Temperature**, **Pressure**, and **Velocity**. Like solution monitors, per-object monitors will be displayed in different **Monitor** graphics display windows.

Note that all objects are available for per-object monitoring, except networks.

Note that it is possible to change the display of the solution residuals and point monitors during the solution calculation, as described in Section 27.12.3.

27.8.2 Plotting Residuals

The solution residual history and point monitor history can be displayed using XY plots when the solution has finished. For a solution residual plot, the number of iterations is plotted on the x axis and the log-scaled residual values are plotted on the y axis. For a point monitor plot, the number of iterations is plotted on the x axis and the point monitor values are plotted on the y axis.

The **Solution monitor definition** panel allows you to display XY plots of solution residual history and point monitor history after the calculation has finished. To open the **Solution monitor definition** panel, select **Solution monitor** in the **Solve** menu. See Section 27.8.1 to set parameters in the **Solution monitor definition** panel.

Solve—>Solution monitor



Note that alternatively, you can open this panel by selecting **Convergence plot** in the **Post** menu.


The **Solution residuals** and **Monitor** graphics display and control windows used for displaying solution residuals and point monitors when the solution has finished is very similar to the **Solution residuals** window shown in Figure 27.12.1. The difference is that the **Lower pri** and **Terminate** buttons are not available when you are viewing the plots after the solution has finished. See Section 27.12.2 for details on using the **Solution residuals** window.

27.9 Defining Postprocessing Objects


You can define the following postprocessing objects prior to solving the model, by using the options in the **Post** menu or **Postprocessing** toolbar: **Object face**, **Plane cut**, **Isosurface**, and **Point**.

When you define a new postprocessing object, the object is added under the **Post-processing** node in the **Model manager** window. Similarly, as in the case of modeling objects, each postprocessing object has a unique object name, e.g., `iso.1`. (see Figure 28.2.1).

For details on defining an **Object face**, **Plane cut**, **Isosurface**, and **Point** see Sections 28.2.4, 28.2.5, 28.2.6, and 28.2.7, respectively.

To display the postprocessing objects defined prior to solving the model, you can load the solution data using the **Version selection** panel. To open this panel, select **Load solution ID** in the **Post** menu, or click the  button in the **Postprocessing** toolbar.

27.10 Defining Reports

You can define a summary report for a variable on any or all objects in your **Airpak** model. **Airpak** will generate the report after it has finished calculating the solution if you select **Write report when finished** in the **Solve** panel (see Section 27.11.2). You can use the **Define summary report** panel (Figure 27.10.1) to define a summary report for the results of your **Airpak** simulation. To open the **Define summary report** panel, select **Define report** in the **Solve** menu or click on the  button in the **Postprocessing** toolbar.

Solve—>Define report

For details on how to define a summary report see Section 29.5.

You can also create a summary report after **Airpak** has finished the calculation, as described in Section 29.5.

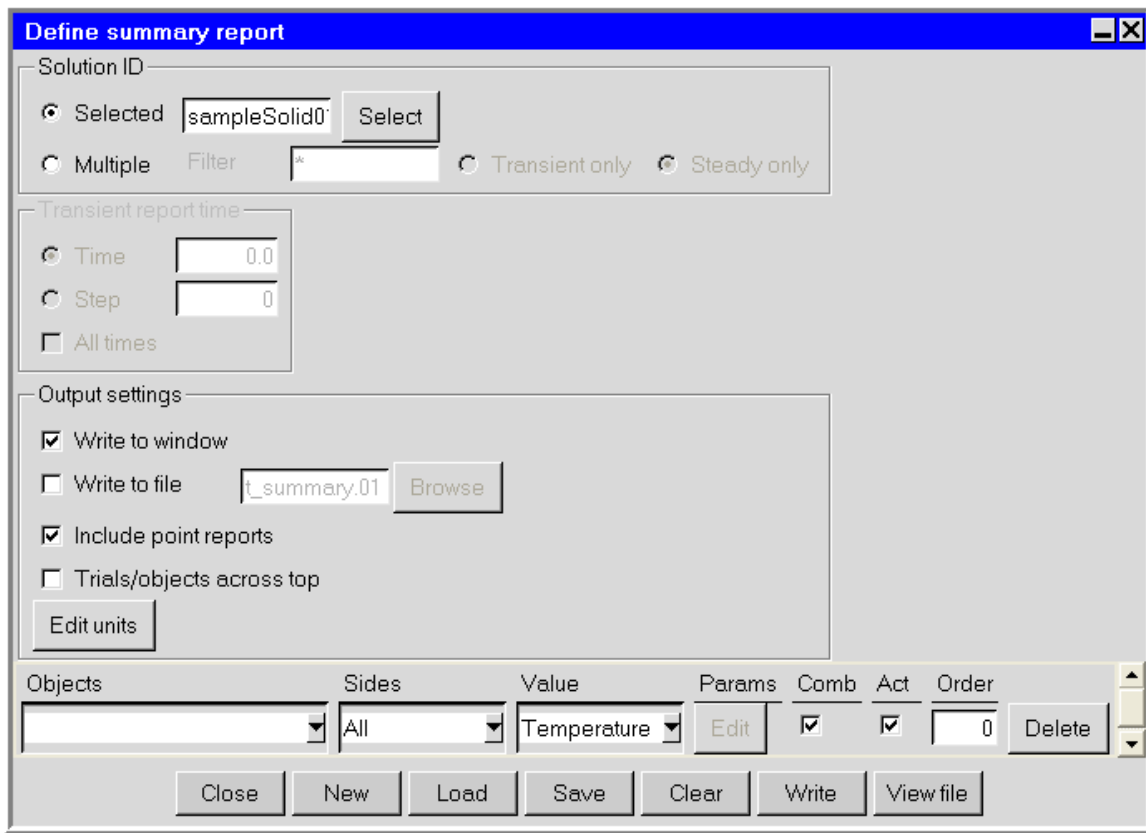



Figure 27.10.1: The Define summary report Panel

27.11 Setting the Solver Controls

Before Airpak executes the solution procedure, it allows you to set various controls for the solution process. The procedure for setting the solver controls is described in Section 27.11.1 and the solution control options are described in Section 27.11.2. Parallel processing options are described in Section 27.11.3.

27.11.1 Using the Solve Panel to Set the Solver Controls

The Solve panel (Figures 27.11.1 and 27.11.2) allows you to specify the controls for the solution. To open the Solve panel, select Run solution in the Solve menu or click on the  button in the Model and solve toolbar.

Solve—>Run solution

To set the controls for the solver, follow the steps below.

1. Specify the Solution ID. This defines a unique identifier for the solution results.

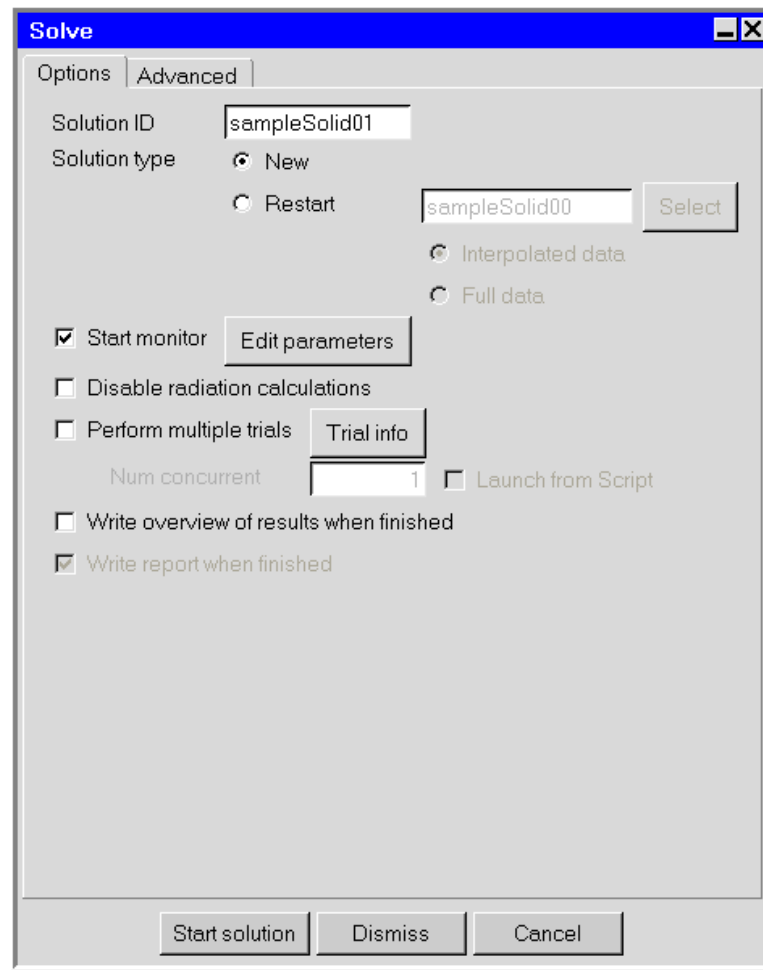


Figure 27.11.1: The Solve Panel (Options Tab)

Airpak automatically creates a default name that you can alter by typing a new name in the **Solution ID** text entry field. The default name consists of the project name plus a two-digit sequential number, starting with 00.

2. Specify the type of the solver calculation (the **Solution type**). There are two options:

New specifies that the calculation is a new one. Any previous data stored under the specified **Solution ID** will be overwritten.

Restart specifies that the calculation is the continuation of a solution that has converged or that was previously terminated before completion, either because you intentionally stopped the calculation to look at the results, or because the calculation reached the iteration limit you specified before convergence was achieved. When Airpak saves the results of a simulation, it saves two files: `projectname.dat` and `projectname.fdat`. The `projectname.fdat` file (bi-

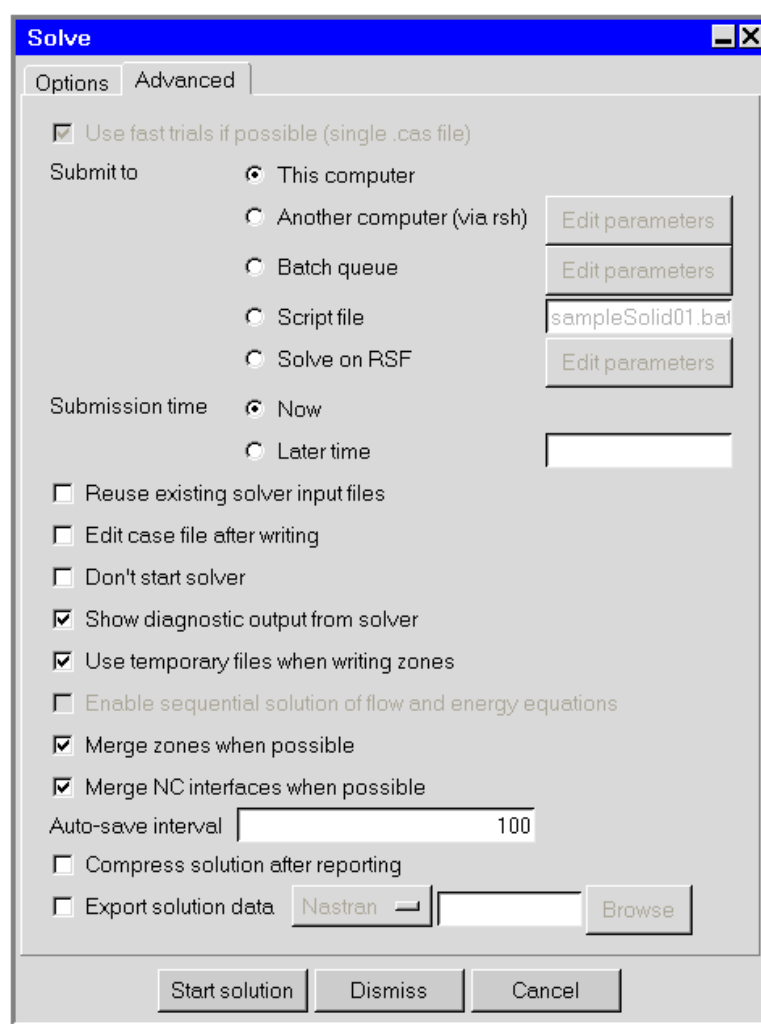


Figure 27.11.2: The Solve Panel (Advanced Tab)

nary format) contains all the information about the results of the simulation; the `projectname.dat` file (ASCII format) contains a subset of all the results of the simulation (velocity, pressure, and temperature data). You can choose which of these data files Airpak should use when it restarts the calculation by selecting **Interpolated data** or **Full data** in the **Solve** panel, as described below.

If you choose the **Restart** option, you must specify the solution ID of the previous calculation from which you want to continue. You can either enter the solution ID in the **Restart** text entry box, or select the solution ID using the **Version** selection panel. Click **Select** to open a **File** selection dialog box.



If the solution ID you specify in the **Solution ID** field is the same as the solution ID in the **Restart** box, the results of the previous calculation will

be overwritten. If you want to retain the previous results, be sure to enter a new name in the **Solution ID** field.

There are two options for restarting a calculation:

Interpolated data instructs **Airpak** to restart the calculation using the `projectname.dat` file. This is the default option and you should use it if you have calculated a solution on a coarse mesh for your model and then you have refined the mesh. You will decrease the time to calculate a solution on the fine mesh if you start your calculation using the results obtained for the coarse mesh.

Full data instructs **Airpak** to restart the calculation using the `projectname.fdat` file. You should use this option if you have previously calculated the flow solution for your problem and you want to use this solution data when you solve for temperature. Note that the geometry and mesh must be identical for the two problems.

3. Enable the appropriate basic solution control options in the **Options** tab of the **Solve** panel (Figure 27.11.1). The following options are available:

Start monitor instructs **Airpak** to display the convergence history for the calculation while the calculation is in progress. See Section 27.8.1 for details on enabling the display of solution residuals and point monitors.

Disable radiation calculations instructs **Airpak** not to calculate radiation parameters for the model. See Chapter 22 for details on radiation modeling.

Perform multiple trials instructs **Airpak** to perform multiple trials using the parameters you have specified for your model. See Chapter 24 for details on using parameters in your **Airpak** model.

Write overview of results when finished instructs **Airpak** to generate an overview of results when it has finished the calculation.

Write report when finished instructs **Airpak** to generate a report when it has finished the calculation. See Section 27.10 for details on defining a report to be generated when the calculation is complete.

4. Set the appropriate advanced solution controls in the **Advanced** tab of the **Solve** panel (Figure 27.11.2). See Section 27.11.2 for details.
5. Specify whether you want the solution to be run in serial or in parallel. To run the solution in serial, keep the default selection of **Serial** in the **Parallel settings** panel (Figure 27.11.6). See Section 27.11.3 for details about running a solution in parallel.
6. Click **Start solution** to start the calculation.

You can click **Cancel** to close the **Solve** panel without accepting the specifications and without starting the calculation, or you can click **Dismiss** to close the **Solve** panel and accept the specifications but without starting the calculation.

27.11.2 Advanced Solution Control Options

To set the advanced controls for the solver, follow the steps below.

1. Click the **Advanced** tab in the **Solve** panel (Figure 27.11.2)
2. To reduce the time taken per trial if the trials do not involve changes to the model geometry, enable **Use fast trials if possible**. If this option is enabled, memory storage space will be reduced because only a single case file will be written. If this option is disabled, a separate mesh will be created and a separate case file will be written for every trial regardless of whether the geometry has changed or not.
3. Specify the computer on which you want the solution to run. There are four options:

This computer runs the solution on the computer you are currently using.

Another computer (via rsh) runs the solution on another computer on your network. To specify parameters related to the remote execution, click **Edit parameters** next to **Another computer**. This will open the **Remote execution parameters** panel (Figures 27.11.3 and 27.11.4).

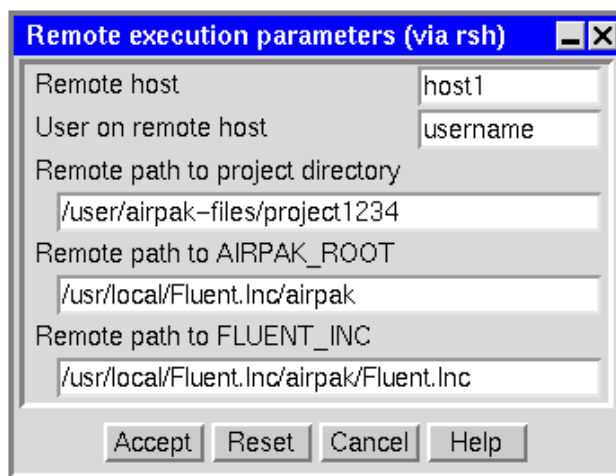


Figure 27.11.3: The Remote execution parameters Panel (UNIX)

- (a) Specify the name of the host computer to which the project is to be submitted next to **Remote host**.
- (b) Specify the user account name on the remote computer next to **User on remote host**. By default, this is the same as the user account you used to log onto the machine on which you are currently running **Airpak**.
- (c) (UNIX machines only) Specify the **Remote path to project directory**, which is the project directory on the remote computer. The default is the project directory path on the current machine.

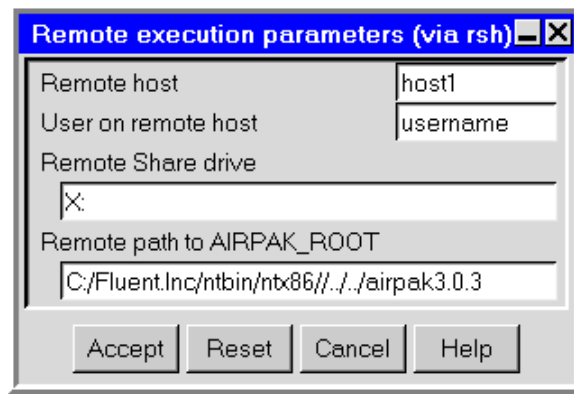


Figure 27.11.4: The Remote execution parameters Panel (Windows)

- (d) (Windows machines only) Specify the Remote Share drive by entering a drive letter followed by a colon (e.g., X:). This is the shared drive that will be used to mount the project directory on the remote computer. The drive letter that you enter must not be previously mapped on the remote computer.
- (e) Specify the Remote path to AIRPAK_ROOT, which tells the remote computer the location where Airpak is installed on the local computer. For example, if Airpak 3.0 has been installed under /usr/local on a local UNIX machine, there will be a Fluent.Inc directory present as /usr/local/Fluent.Inc. You would then need to set your PATH environment variable to include /usr/local/Fluent.Inc/bin.

On a Windows machine, you must first follow steps 1–3 in the section [Installing and Configuring Airpak Network Parallel](#) on the [Fluent User Services Center](#). In the Remote execution parameters panel, you must then specify the full network path to the local installation of Airpak unless the local path has been mapped as a network drive on the remote machine or Airpak has been installed on the remote machine. In either of these exceptions, you can specify an appropriate drive letter as a part of the path.

- (f) (UNIX machines only) Specify the Remote path to FLUENT_INC, which is the location where Airpak's version of the FLUENT solver was installed (i.e., AIRPAK_ROOT/Fluent.Inc).
- (g) Click Accept to accept any changes made to the panel and close the panel.

You can click Reset to undo all the changes made in the panel and restore all items to their original states when the panel was opened, or click Cancel to close the panel and ignore any changes made to the panel.

Batch queue allows you to run the solution on a remote server. To specify pa-

parameters related to the remote server execution, click **Edit parameters** next to **Batch queue**. This will open the **Remote execution parameters** panel, which is similar to the **Remote execution parameters** panel for remote simulation on a remote host. Follow the same steps previously defined for running the solution on via **rsh**. In lieu of the remote host, you will specify the server where the project will be submitted for batch processing. Note that you will first need to add the server to server list in the **Add Server** panel.

Script file specifies that the commands necessary to run the solution are to be output to a script file. The commands in the script file can be used to run the solution on the computer you are currently using or on a remote computer. The default name for the script file is **ident.SCRIPT**, where **ident** is the identifier specified in the **Solution ID** field. This option can also be used to batch process a number of **Airpak** projects. An example of batch processing on a Windows machine is described in Section 27.11.4.

Solve on RSF allows you to run your **Airpak** solution on the Remote Simulation Facility (RSF). See Section 27.11.5 for details.

4. Specify the **Submission time**. You can instruct **Airpak** to run the solution immediately (**Now**) or delay the calculation to a **Later time**. If you select **Later time**, you must enter the date and time in the **Later time** text field in the format **hh:mm month day** (e.g., 08:15 Jan 24).
5. Enable the appropriate advanced solution control options. The following options are available in the **Advanced** tab of the **Solve** panel (Figure 27.11.2):

Reuse existing solver input files enables the use of a pre-existing case file and script file. This is typically used if you want to continue from a previous solution, without changing any parameters in the problem.

Edit case file after writing allows you to view the case file before starting the solver to check for correctness. This is recommended only if you are an expert in the case file structure or if asked to do so by your technical support engineer.

Don't start solver allows you to set up the solver parameters without actually running the solver.

Show diagnostic output from solver prints the numeric values of the residuals to a separate window.

Use temporary files when writing zones instructs **Airpak** to write out temporary files when writing zones before creating the **FLUENT** case file. This option is on by default.

Enable sequential solution of flow and energy equations instructs **Airpak** to solve the flow equations first and then solve the energy equation. This is suitable for cases where there is no coupling between the flow and energy equations (e.g., forced convection problems with no gravity). This option may reduce the time required to obtain a solution.

Merge zones when possible instructs **Airpak** to optimize the solver by merging zones whenever possible before writing the model information to a **FLUENT** case file. This option is on by default.

Merge NC interfaces when possible instructs **Airpak** to optimize the solver by merging the non-conformal interfaces whenever possible before writing the model information to a **FLUENT** case file.

Auto-save interval specifies the frequency with which **Airpak** should save the results of a non-transient calculation. A value of 100 specifies that **Airpak** should save the results after every 100 iterations.

Compress solution after reporting instructs **Airpak** to compress some solver-related files when it has finished the calculation to reduce the usage of disk space. This will not affect postprocessing of the solution.

Export solution data instructs **Airpak** to export the results of the simulation to a **NASTRAN**, **PATRAN**, **I-deas**, or **Ansys** file when the simulation is complete. Note that this option is available only if you are using the serial solver. Select a file format from the drop-down list to the right of **Export solution data**. The following formats are available:

Nastran: A single file containing coordinates, connectivity, velocity, pressure, heat flux, and temperature is written.

Patran: Two files are written. One is a **PATRAN** neutral file with a **.ptr** extension, which contains coordinate, connectivity, and zonal information; the other is a **PATRAN** nodal file with a **.ptn** extension, which contains nodal values of velocity, pressure, heat flux, and temperature.

IDEAS: A single file containing coordinates, connectivity, velocity, pressure, heat flux, and temperature is written.

Ansys: A single file containing coordinates, connectivity, and various scalars (velocity components, pressure, temperature, etc.) is written.

Specify the name of the file for the solution data in the text entry box to the right of **Export solution data**. You can enter your own filename, which can be a full pathname to the file (beginning with a **/** character on UNIX or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started. Alternatively, you can choose a filename by clicking on the **Browse** button. This will open the **Export solution data** panel (Figure 27.11.5). Select the desired file and click **Open**.

27.11.3 Parallel Processing

The parallel version of the solver simultaneously computes the solution using multiple compute nodes (processes). The solver will partition the mesh into multiple subdomains

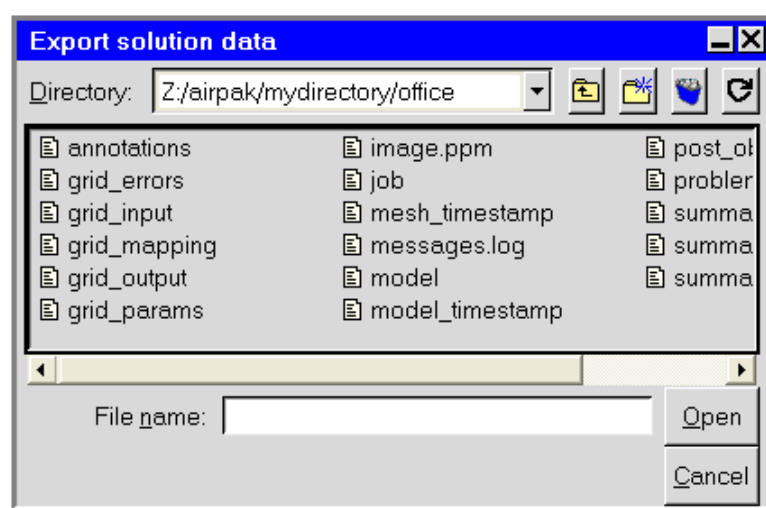


Figure 27.11.5: The Export solution data Panel

and each partition will reside on a different compute node. These processes can be compute nodes on a multiple-CPU workstation, or processes on a network of heterogeneous workstations or a cluster of Windows machines. In general, as the number of compute nodes increases, the turnaround time for the solution will decrease. However, parallel efficiency decreases as the ratio of communication to computation increases, so you should use parallel processing only for suitably large problems.

You will set up all the parameters for the parallel version of the solver in the **Parallel settings** panel (Figure 27.11.6).

 **Solution settings** →  **Parallel settings** (or **Solve** → **Settings** → **Parallel**)

The following options are available in the **Parallel settings** panel:

Serial specifies that the solution should be run as a serial process.

Parallel specifies that the solution should be run as a parallel process on a dedicated parallel computer. See below for further details on parallel processing.

Network parallel specifies that the solution should be run as a parallel process on a network of computers. See below for further details on parallel processing.

Dedicated Parallel Machine

To run Airpak on a dedicated parallel machine (i.e., a multiprocessor workstation), select **Parallel** in the **Parallel settings** panel (Figure 27.11.6) and specify the number of processors to be used in the **# processors** field.

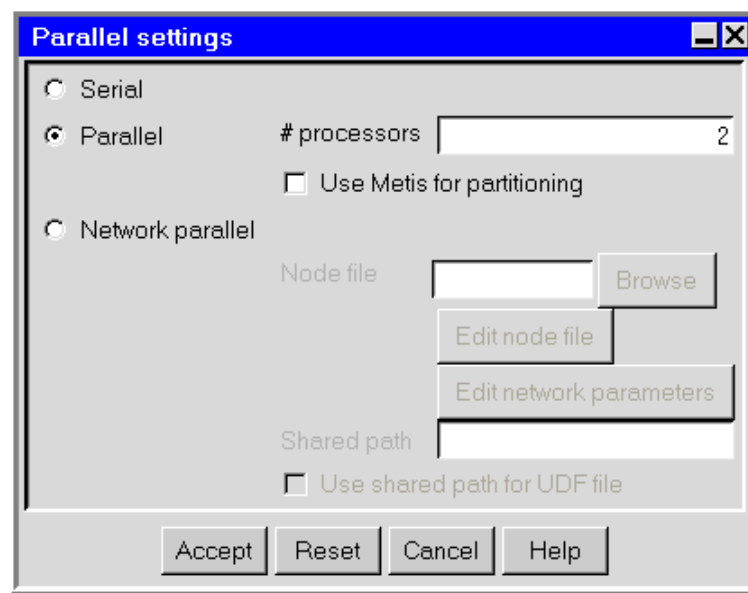


Figure 27.11.6: The Parallel settings Panel

Partitioning the Grid

The default option for partitioning the grid is the **Principal Axes** method. This method bisects the domain based on a coordinate frame aligned with the principal axes of the domain. There is another option, called **METIS**, that can be used for grid partitioning, by selecting the **Use Metis for partitioning** option in the **Parallel settings** panel.

METIS is a software package for partitioning irregular graphs developed by Karypis and Kumar at the University of Minnesota and the Army HPC Research Center. It uses a multilevel approach in which the vertices and edges on the fine graph are coalesced to form a coarse graph. The coarse graph is partitioned, and then uncoarsened back to the original graph. During coarsening and uncoarsening, algorithms are applied to permit high-quality partitions. Detailed information about METIS can be found in its manual [17].

Workstation Cluster

To run **Airpak** on a network of computers, you need to provide **Airpak** with a list of names of the computers that you want to use. This “node” file is a text file that you create that contains a list of machines, one per line, to be used by **Airpak** in the calculation. An example of a node file is shown below.

```
balin
bilbo
```

```
bofur  
dain  
dwalin
```



Note that the network must contain either all UNIX machines or all Windows machines.

To run **Airpak** on a network of computers, select **Network parallel** in the **Parallel settings** panel (Figure 27.11.6). You must specify the name of the node file in the **Node file** text entry field. You can enter your own filename, which can be a full pathname to the file (beginning with a / character on UNIX or a drive letter on Windows) or a pathname relative to the directory in which **Airpak** was started. Alternatively, you can choose a filename by clicking on the **Browse** button located next to the **Node file** text field and then selecting the file in the resulting **Node file for parallelism** dialog box.

To make changes to the node file you selected, click **Edit node file** under **Node file** in the **Parallel settings** panel and then create or select the file in the resulting **File** panel. **Airpak** will open the node file in a text editor, where you can edit the list of machines. Save your changes and exit the text editor to return to the **Parallel settings** panel in **Airpak**. Note that you do not need to specify the number of processors when using the **Network parallel** option. The number of processors is equal to the number of machines listed in the node file.

To make changes to the network parameters, click **Edit network parameters** to open the **Network parameter settings** panel (Figure 27.11.7). These advanced options allow you to specify fast interconnects and MPI communicators for running in network parallel mode.

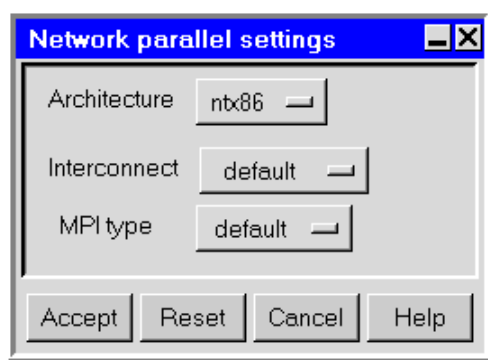


Figure 27.11.7: The Network parallel settings Panel

The following options are available in the **Network parallel settings** panel:

Architecture specifies the architecture that the interconnect and MPI communicator is dependent on.

Interconnect specifies the interconnect you choose to use for the network parallel run.

Note that **Airpak** doesn't check whether the hardware interconnects that are in the pull-down list are actually available on the network. You must choose the appropriate interconnect. You may also choose the **default** interconnect, which is typically an ethernet connection.

MPI type specifies the MPI communicator you wish to use for the network parallel run. Note that the list of MPI communicators will change depending on the architecture and interconnect. You can choose to use the **default** MPI communicator, which is chosen by **Airpak**.

For Windows networks, you must also specify the **Shared path** in the **Parallel settings** panel, which is the directory where **Airpak's** version of the **FLUENT** solver is installed. If the solver is installed on a Windows server, you should enter the following in the **Shared path** text entry box:

```
\\servername\fluent.inc
```

where **servername** is the name of the server where **Airpak's** version of the **FLUENT** solver is installed. If the solver is installed on a Windows machine, you should enter the following in the **Shared path** text entry box:

```
\\pcname\fluent.inc
```

where **pcname** is the name of the Windows machine where **Airpak's** version of the **FLUENT** solver is installed.

The **Use shared path for UDF file** option should be on by default for network parallel runs on Windows machines. **Airpak** generates a user-defined function (UDF) if model properties are varied, and the **FLUENT** solver will look in the **Shared path** directory for the UDF file if this option is enabled. This option should only be disabled if you are working in a separate networked drive that does not contain the **FLUENT** solver.

For more detailed information about configuring **Airpak** for Windows clusters, see the section entitled **Installing and Configuring Airpak Network Parallel** on the [Fluent User Services Center](#) .

27.11.4 Batch Processing of Airpak Projects on a Windows Machine

Airpak can be used in a batch mode in which inputs are obtained from and outputs are stored in files. Generally you will perform problem setup and postprocessing of results in an interactive mode. However, when you perform calculations for several Airpak projects outside of Airpak, you may want to run the projects in batch mode. This allows the computer resources to be prioritized, enables you to control the process from a file (eliminating the need for you to be present during the calculation), and also provides a record of the calculation history (residuals) in an output file.

To process multiple Airpak projects in batch mode, follow the steps below:

1. Set up the first project in Airpak.
2. Save the project to a batch file using the Solve panel (Figure 27.11.8).

Solve → Run solution

- (a) Select the Script file option under Submit to and enter the name of the batch file in the Script file text entry box using a .bat extension.
- (b) Turn on the Don't start solver option.
- (c) Click Start solution in the Solve panel to save the batch file. The contents of a typical batch file are shown below.

```
set AIRPAK_LICENSING=1
set FLUENT_INC=e:\fluent.inc\airpak3.0\ntx86
  \Fluent.Inc
set FLUENT_WAIT=yes
del project701.resd
del project701.*.out
e:\fluent.inc\airpak3.0\ntx86\Fluent.Inc\ntbin
  \ntx86\fluent.exe 3d -g -i project701.uns_in
echo. > project701.uns_done
del project701.uns_pid
```

3. Repeat the steps above to create batch files for all the projects you want to run in batch mode.
4. Using a text editor, combine all the individual batch files into a combined batch file with a .bat extension. Note that you must add a cd (change directory) command before each individual batch file to ensure that the file is executed in the correct directory. An example is shown below, where the batch files for project701 and project1000 are combined into a single batch file.

```
cd e:\airpak\project7
set AIRPAK_LICENSING=1
```

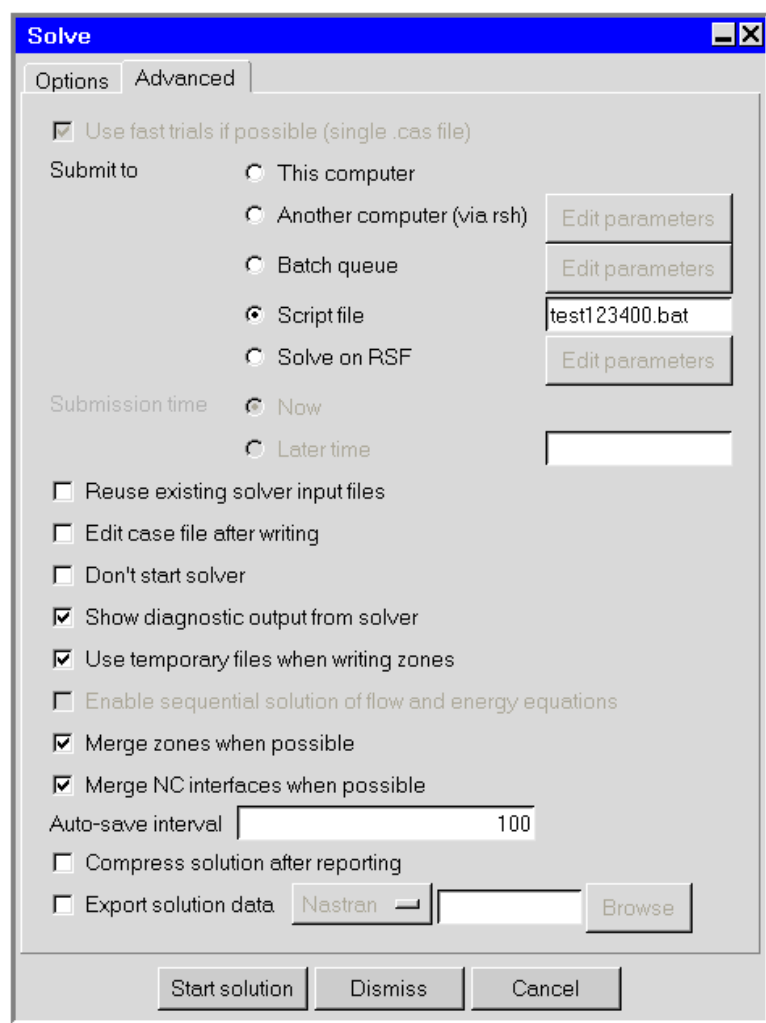


Figure 27.11.8: The Solve Panel for Batch Processing

```
set FLUENT_INC=e:\fluent.inc\airpak3.0\ntx86\Fluent.Inc
set FLUENT_WAIT=yes
del project701.resd
del project701.*.out
e:\fluent.inc\airpak3.0\ntx86\Fluent.Inc\ntbin\ntx86
    \fluent.exe 3d -g -i project701.uns_in
echo. > project701.uns_done
del project701.uns_pid
cd e:\airpak\project10
set AIRPAK_LICENSING=1
set FLUENT_INC=e:\fluent.inc\airpak3.0\ntx86\Fluent.Inc
set FLUENT_WAIT=yes
del project1000.resd
del project1000.*.out
e:\fluent.inc\airpak3.0\ntx86\Fluent.Inc\ntbin\ntx86
    \fluent.exe 3d -g -i project1000.uns_in
echo. > project1000.uns_done
del project1000.uns_pid
```

5. Open Windows Explorer and double-click on the combined batch file to start the execution of the batch process.
6. When the batch process is complete, you can use Airpak to examine the results for the individual projects, as described in [Chapter 28](#).

27.11.5 Running the Solution Using the Remote Simulation Facility (RSF)

Airpak allows you to solve a project using the Remote Simulation Facility (RSF) from within Airpak, or independently from outside of the Airpak application.

Solving an RSF Project From an Airpak Session

To solve an Airpak project using the Remote Simulation Facility (RSF) from an Airpak session, follow the steps below.

1. Specify the RSF host and proxy settings (if necessary) in the Preferences panel.

Edit—>Preferences



The same proxy settings are used for error reporting and RSF. If one is set, the other need not be set explicitly.

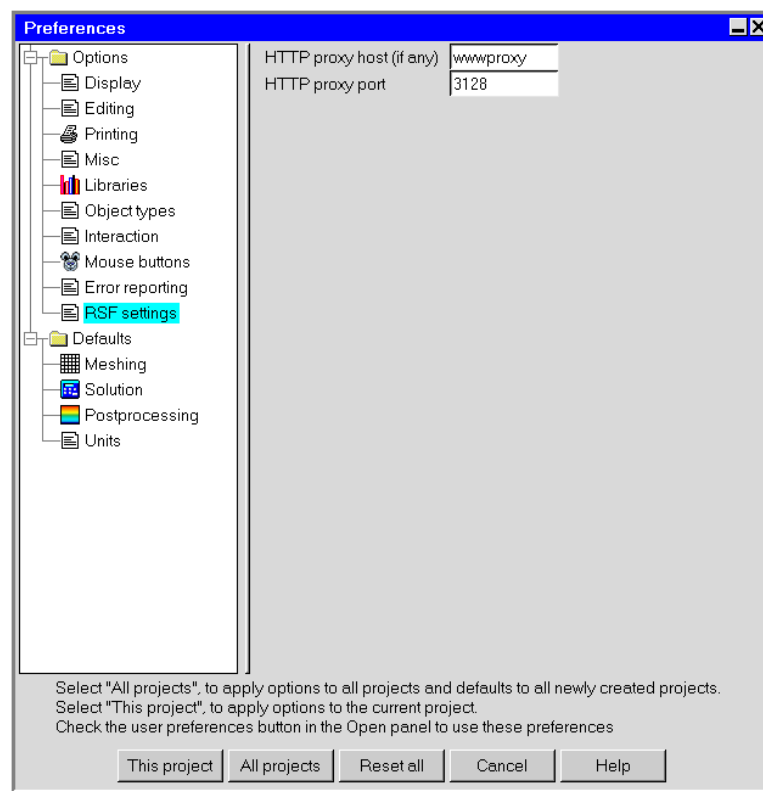


Figure 27.11.9: The Preferences panel (RSF settings)

- (a) Click the RSF settings icon in the Options folder in the Preferences panel to display the RSF options (Figure 27.11.9).

- (b) Enter the HTTP proxy host (e.g., `wwwproxy`).
- (c) Enter the HTTP proxy port (e.g., `3128`).
- (d) Click **This project** to apply the options to the current project, only. Alternatively, click **All projects** to apply the options and the defaults to all newly-created projects.

- Specify the project that is to be submitted to the RSF (Figure 27.11.10).

Solve → Run solution

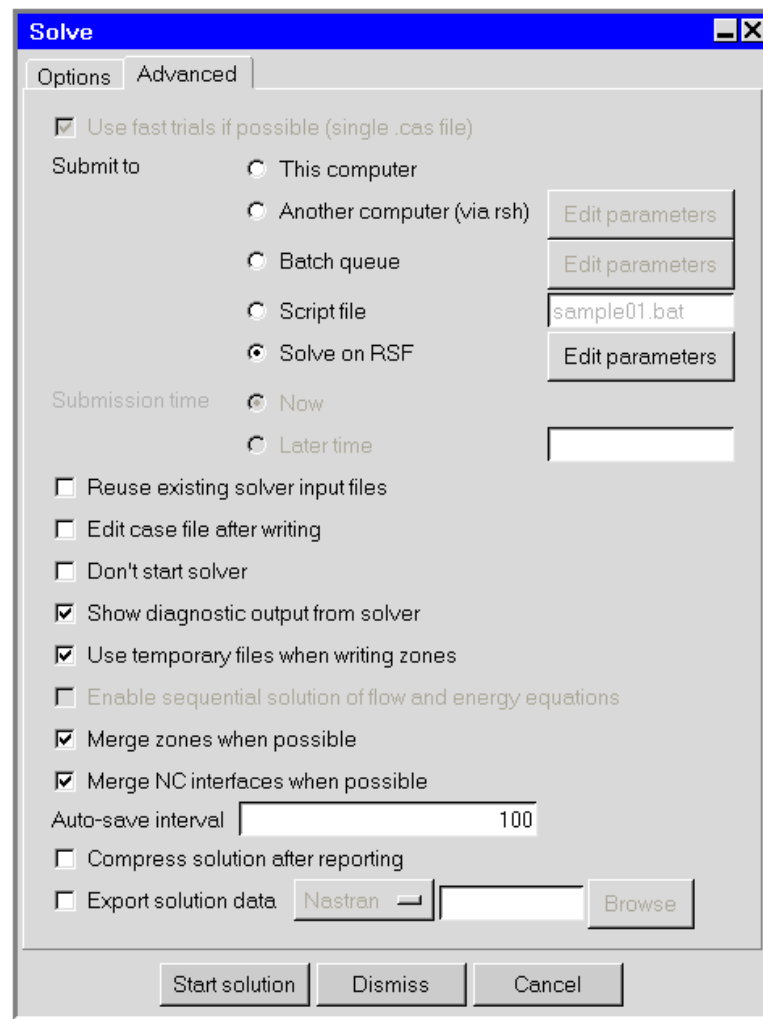


Figure 27.11.10: The Solve Panel for RSF Processing (Advanced Tab)

- (a) In the **Options** tab of the **Solve** panel, set the solution options (see Section 27.11 for details).
- (b) In the **Advanced** tab of the **Solve** panel, select **Solve on RSF** from the list of **Submit to** options and click the **Edit parameters** button to open the RSF parameters panel (Figure 27.11.11).

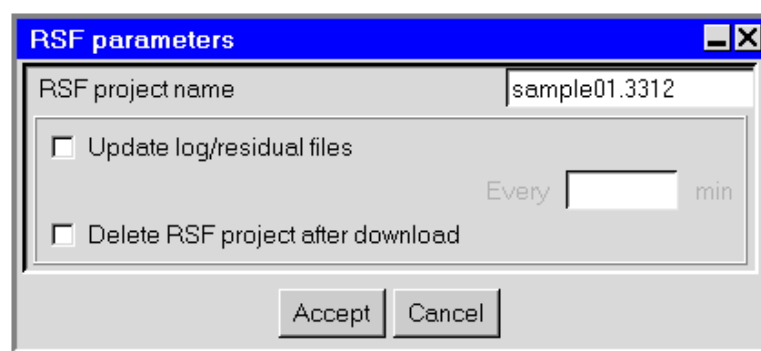


Figure 27.11.11: The RSF parameters Panel

- i. Enter the RSF project name in the RSF parameters panel. You can enter a different project name, but it has to be unique on the RSF site. If it is not unique, you will be prompted to enter another project name.
 - ii. Select **Update log/residual files** and enter a time interval if you want to download the residual and log files on a periodic basis. The default time is every 5 minutes.
 - iii. Select **Delete RSF project after download** if you want to delete the project and related files on the RSF after a successful download of result files.

i

Note that if you don't specify that the RSF project be automatically deleted after download, then you can subsequently log on to the RSF site (<https://rsolve.fluent.com/fluent/jsp/index.jsp>) using your UserID and password, and manually delete the project. If you want **Airpak** to always delete files after download, you can save this preference in the **RSF Job Settings** panel. See Figure 27.11.27 for details.
 - iv. Click **Accept** to close the RSF parameters panel.
- (c) To start the solution on the RSF, click the **Start solution** button in the **Solve** panel (**Advanced** tab). This will close the **Solve** panel and launch the **Connect to Remote Simulation Facility** login prompt (Figure 27.11.30). The project file will be saved and the RSF connection startup will be reported to the **Message** window.

3. Logon to the RSF by entering your RSF username and RSF password in the Connect to Remote Simulation Facility panel (Figure 27.11.30).

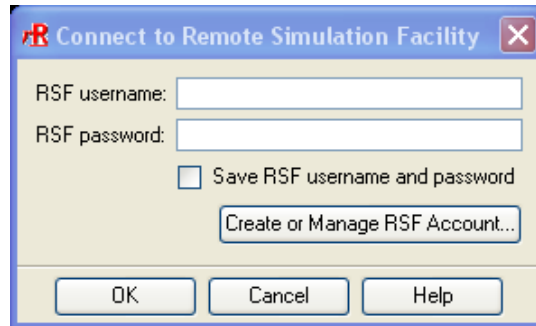


Figure 27.11.12: The Connect to Remote Simulation Facility Panel

- (a) Select **Save RSF username and password** if you don't want to have to retype it in future sessions. If you don't yet have a RSF username and password, click the **Create or Manage RSF Account...** button to open the RSF Home page on the User's Services web site (Figure 27.11.13).
- (b) Click **OK** to close the **Connect to Remote Simulation Facility** panel and begin the remote simulation. Once processing begins, a separate **runrsf** progress panel will open and will monitor the job. The panel will display the four stages of running your project on the RSF: **Upload files**, **Queue solution job**, **Calculate solution**, and **Download results**. As the RSF project runs, the stages will be updated in the panel. These stages are shown in Figures 27.11.14 – 27.11.17.

During the solution process, you can click the **View Log** button at any time in the **runrsf** progress panel to view the output log file. You can also choose to **Disconnect** from the RSF while the solution is being calculated. In this case, the solution will continue to run on the RSF but your real-time session will be disconnected. You can later connect to the RSF and monitor/download the results (see below for details). To kill the job on the RSF when it is calculating, you can click the **Interrupt** button. If you interrupt the job while it is downloading, the download process will be terminated.

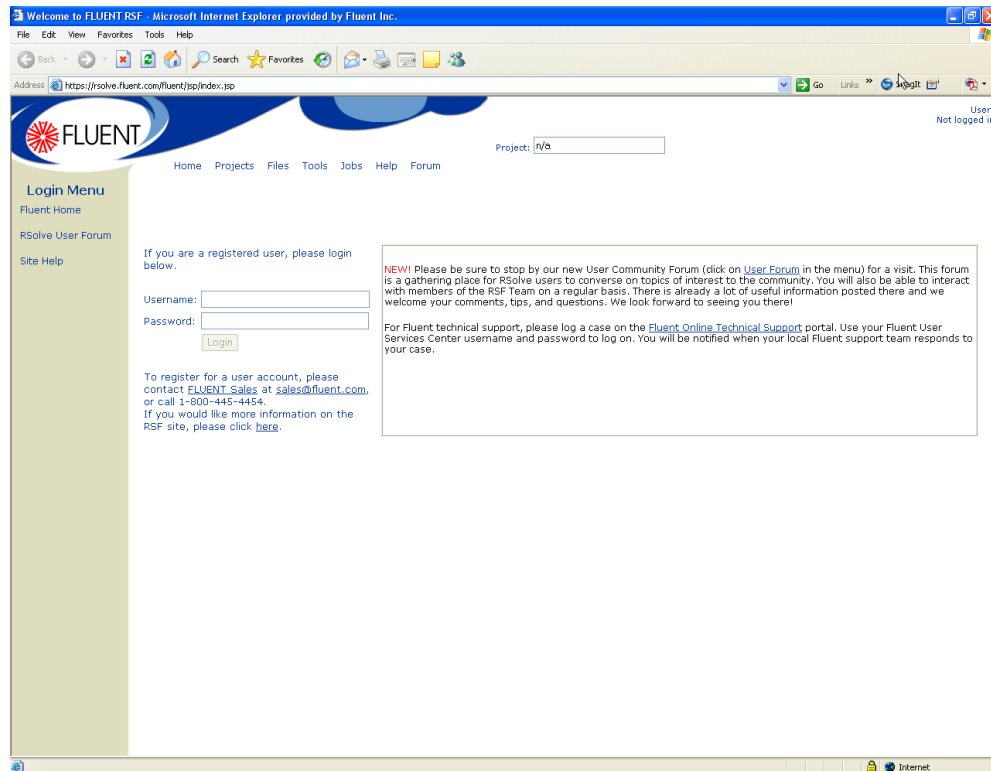


Figure 27.11.13: The RSolve Home Page on the User's Services Web Site

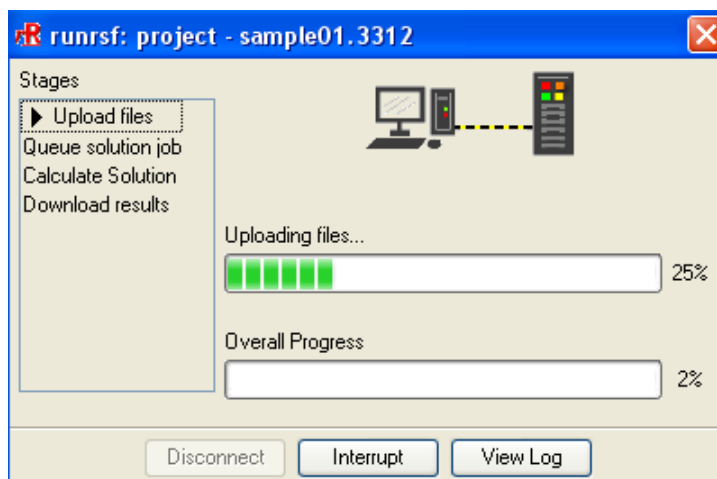


Figure 27.11.14: The runrsf Panel (Update files Stage)

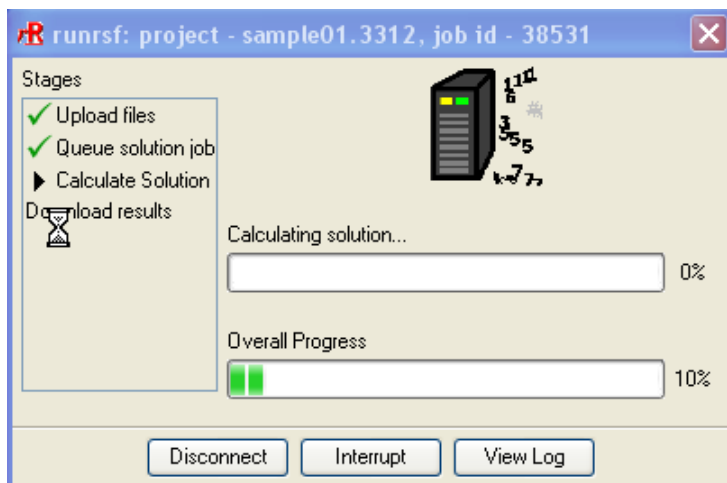


Figure 27.11.15: The runrsf Panel (Queue solution job Stage)

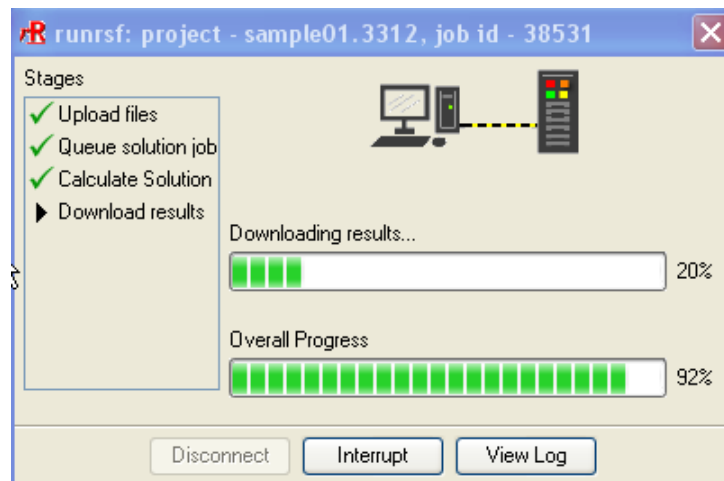


Figure 27.11.16: The runrsf Panel (Calculate Solution Stage)

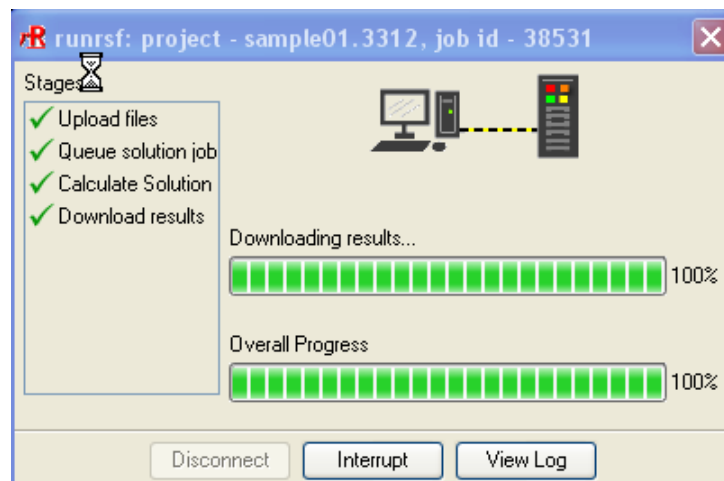


Figure 27.11.17: The runrsf Panel (Download results Stage)

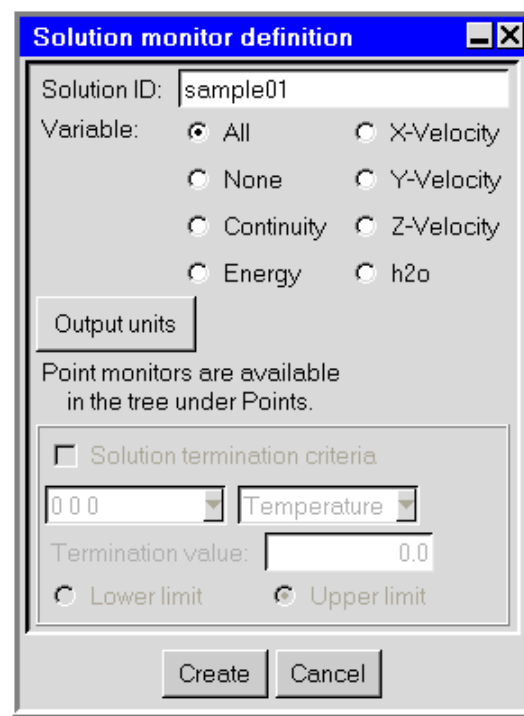



Figure 27.11.18: Solution monitor definition Panel

4. Once the iteration starts and the **Calculate Solution** progress is reported in the **runrsf** panel, you can create a convergence plot to monitor the solution. (Figure 27.11.18)

Post—>Convergence plot

- (a) Enter the **Solution ID**.
 - (b) Select a **Variable**.
 - (c) To change the postprocessing units, select the **Output units** button.
 - (d) Select **Create** to create a convergence plot or **Cancel** to close the panel.
5. Once the solution is successfully completed and downloaded, you can postprocess the results and continue with your Airpak modeling.
6. To download the results of any RSF project, open the **Download RSF project** panel by either selecting **Download RSF project** from the **Post** menu or clicking the  in the **Postprocessing** toolbar. (Figure 27.11.19)

Post—>Download RSF project

- (a) Airpak remembers the last submitted RSF project name in the same session. To download the last project, simply click the **Download** button in the panel and the project files will be downloaded to your current working directory. Alternatively, you can supply the name in the entry box.



Figure 27.11.19: Download RSF project Panel

- (b) To select a project for download (or in the absence of a project name entered in the name field), click **Find Project**. **Airpak** will gather information regarding all the available project/jobs on the RSF site.

i It may take some time to gather all the relevant **Airpak** projects available on the RSF to provide you the complete list of projects. An RSF dialog will open, displaying the **Gathering job information** progress. Once this task is completed the **Select job** panel will open (Figure 27.11.20).

- (c) Select a project from the list of jobs in the **Select job** panel that you wish to download, and either keep the default **Download area** or choose another. Click **OK** and **Airpak** will download the result files (or monitor the job if it is not yet completed). If the solution is not yet completed when you try to download it, **Airpak** will wait for the job to finish running on the RSF before downloading.

Running a RSF Job From Outside of Airpak

To run Remote Simulation Facility (RSF) job on an **Airpak** project independently from the **Airpak** application, follow the steps below.

1. Open the **RunRSF** panel which is the interface you will use to specify tasks to be executed on the RSF (Figure 27.11.21). You can do this several ways:

- (a) Run the `runrsf_w.exe` executable from your **Airpak** directory (e.g., Windows NT version)

`C:\Fluent.Inc\airpak3.0.6\Fluent.Inc\runrsf1.0\bin\ntx86\runrsf_w.exe`

- (b) (Windows only) Run the executable from the Start Menu.

Start→Programs→Fluent Inc Products→Airpak3.0.*→runRSF

2. To run a solution on the RSF for a new project, select **New** from the File menu in the **Runrsf** panel (Figure 27.11.22).

File→New

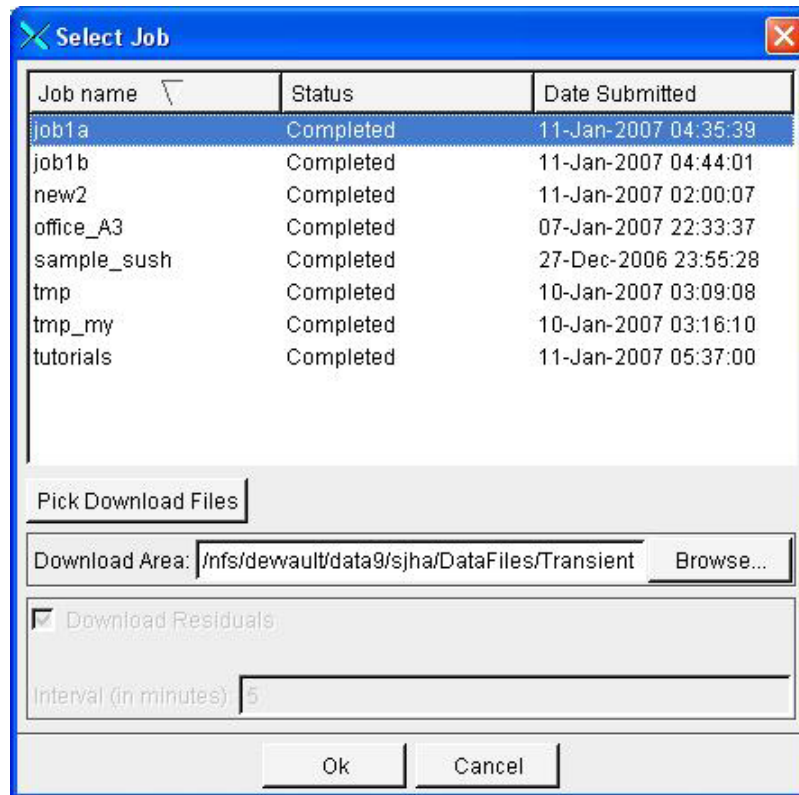


Figure 27.11.20: The Select job Panel

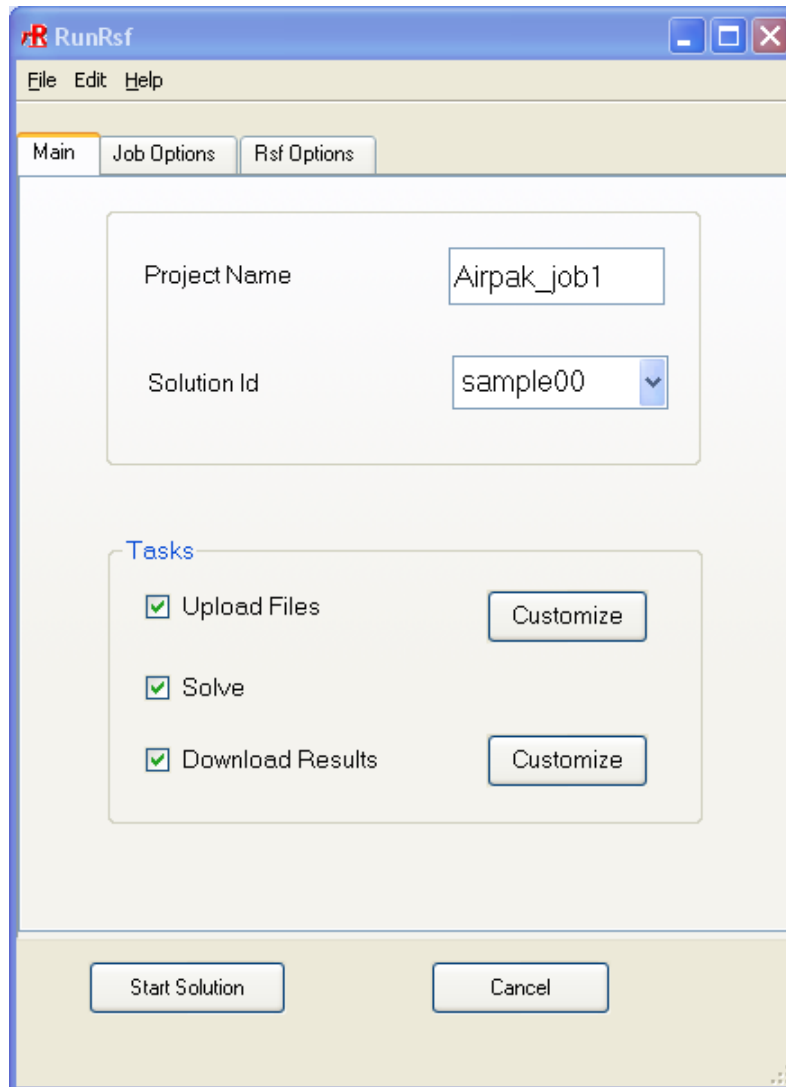


Figure 27.11.21: The Main Tab of the RunRsf Panel



Figure 27.11.22: The File menu in the RunRsf Panel

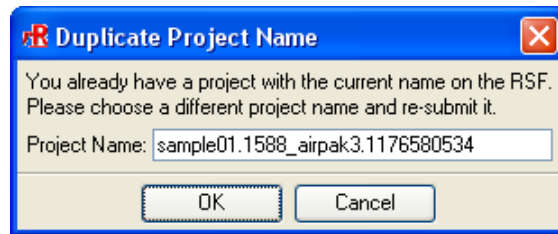


Figure 27.11.23: The Duplicate project name Panel

This will open a directory browser where you can designate the directory where RSF solution files will be stored. When you click **OK** in the directory browser panel, the name of the directory you chose will appear in the **Project Name** field in the **Main** tab of the **RunRSF** panel (Figure 27.11.21).

3. To connect to an existing project, select **Connect** from the **File** menu and input your login information.
4. In the **Main** tab of the **RunRsF** panel (Figure 27.11.21), set the following parameters.
 - (a) You can keep the existing **Project Name** (which is taken from the name of the current directory from where **RunRsF** is launched), or enter another. If you enter a name that already exists on the RSF, a **Duplicate project name** dialog pops up prompting you to change the project name (Figure 27.11.23).
 - (b) Select a **Solution Id** from the drop-down list. The list contains the name of the jobs you can choose to be uploaded to the RSF for processing. By default, the RSF looks for ***.cas** and ***.uns** files in the current directory (or the one specified) and uses the project name as the prefix. For example, if **sample00.cas** and **sample00.uns_in** files are present in the directory, then the **Solution Id** will be set to **sample00**.
 - (c) Under **Tasks**, you can set the following parameters.
 - i. Select **Upload Files** to specify whether you want to upload files on RSF. By default, **Airpak** takes the mesh file (***.uns_in**), case file (***.cas**), ***.pnom** and ***.udf.c** files. This selection of files can be customized. Click **Customize** to display the **Upload Files** panel. Highlight the file and choose **Add** to add the file to list of files to upload or click **Remove** to remove the file from the upload file list. The **Reset** button resets the list of files displayed to that of the default upload file list. When you are finish selecting files, click **Done** to save your changes and close the panel or click **Cancel** to close the panel.
 - ii. Select **Solve** if you want to start and solve the job on RSF.
 - iii. Select **Download Results** to download project files once the solution is completed on RSF. By default it selects ***.uns_out**, ***.res**, ***.resd**, ***.dat** and

*.fdat files. This selection of files can be altered by the **Customize** button. Click the **Customize** button to display the **Download Files** panel. The **Download Files** panel list files which are a combination of default upload and download files. The download files are selected by default. This selection can be changed by reselecting the desired files. Click **Reset** to reset the list of files selected for downloading. If the solution is transient, the the **Transient Data** section of the panel is activated. The transient data can be downloaded either for all time steps or specifying the interval of the time steps by providing the start and end time. Click **Select** to save changes in this dialog box. To close the panel without saving your changes, click **Cancel**.

- (d) In the **Job Options** tab, you can set solution parameters (Figure 27.11.24).
 - i. Select **New** or **Restart** as the **Solution type**. By selecting **New** a new set of calculations starts. If you select **Restart**, **Airpak** will restart the solution which has been terminated before from the given restart solution name. **Restart** calculations can be carried out with full data or interpolated data. Select **Interpolated data** or **Full data**.
 - ii. Select **Serial** or **Parallel** under **Parallel settings**. If you select **Parallel**, enter the number of processors in the space provided.
 - iii. Select **Transient** if the problem is transient.
 - iv. Select **Download Residuals** to save the residuals at a desired time interval. Enter the time interval in minutes.
- (e) In the **Rsf Options** tab, you can set the following parameters (Figure 27.11.25).

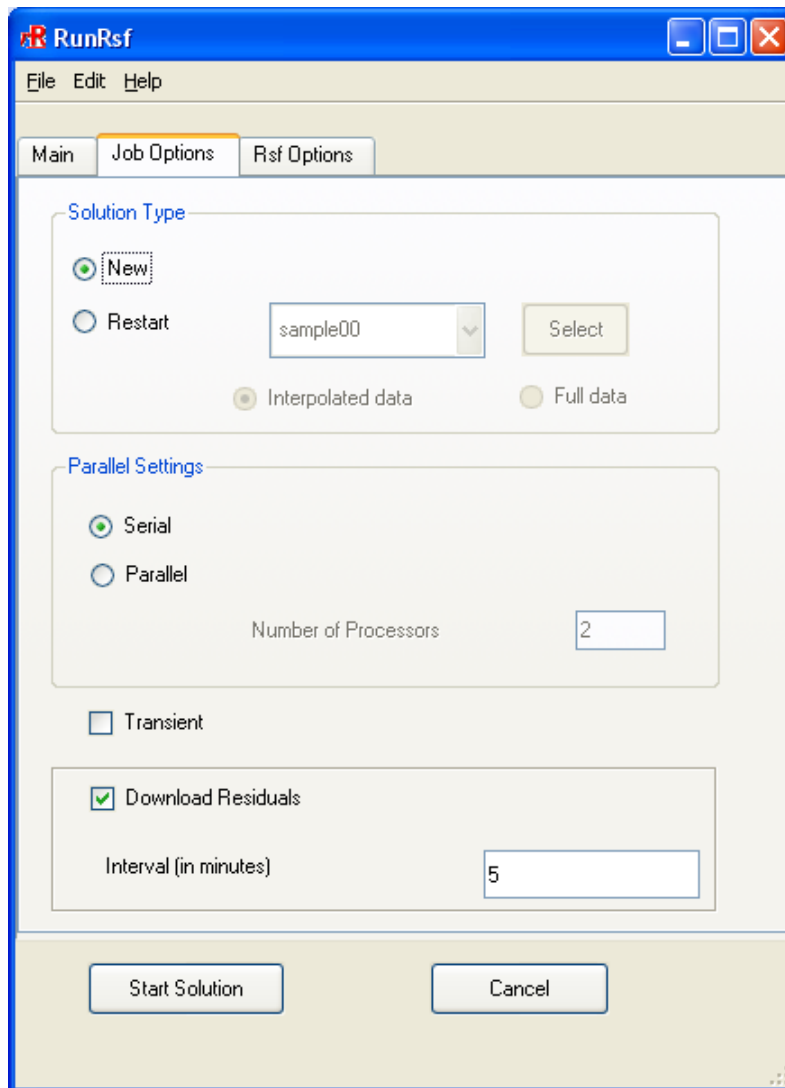


Figure 27.11.24: The Job Options Tab of the RunRsf Panel

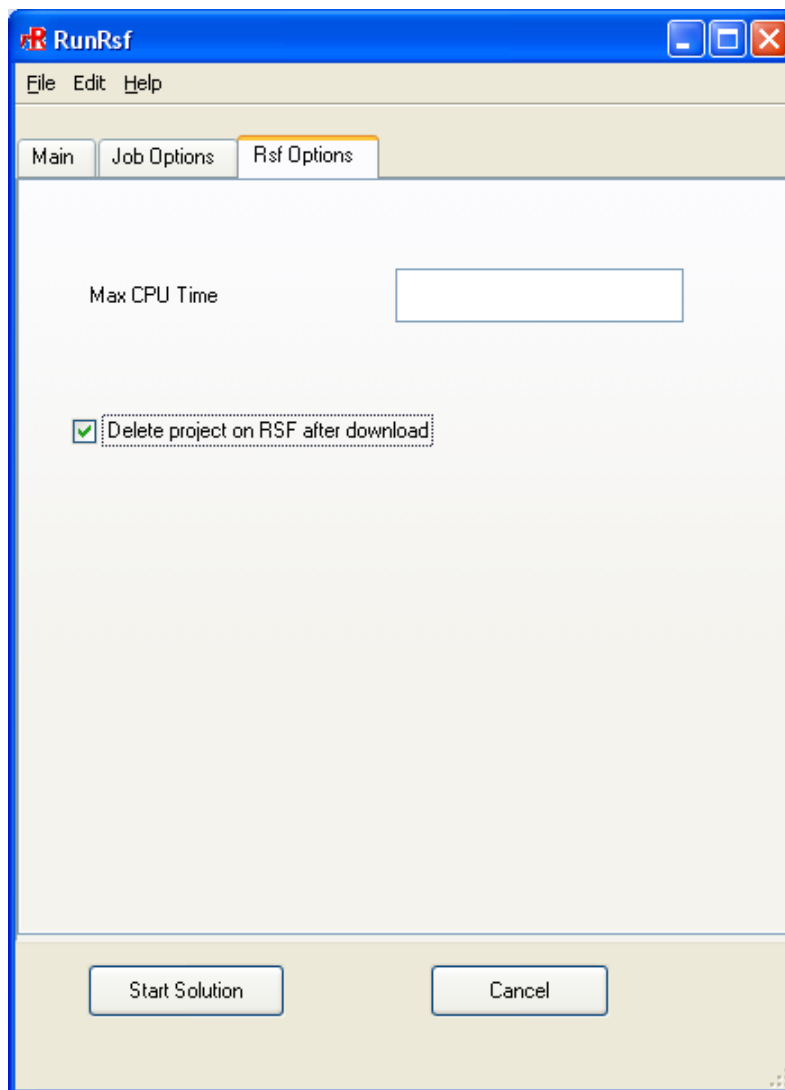


Figure 27.11.25: The Rsf Options Panel

- i. Enter the **Maximum CPU time**. This is the predicted CPU time that will be treated by **Airpak** as the maximum CPU time.
- ii. Select **Delete project on RSF after download** to delete the project and all its associated files on the RSF after the solution is completed and the results are successfully downloaded.

i Note that if you don't specify that the RSF project be automatically deleted after download, then you can subsequently delete the project by logging on to the RSF site (<https://rsolve.fluent.com/fluent/jsp/index.jsp>) using your UserID and password, and manually deleting the project files.

- (f) Set the RSF preferences by selecting **Preferences** from the **Edit** menu in the **RunRSF** panel. This will open the **RSF Options** panel Figure 27.11.26.

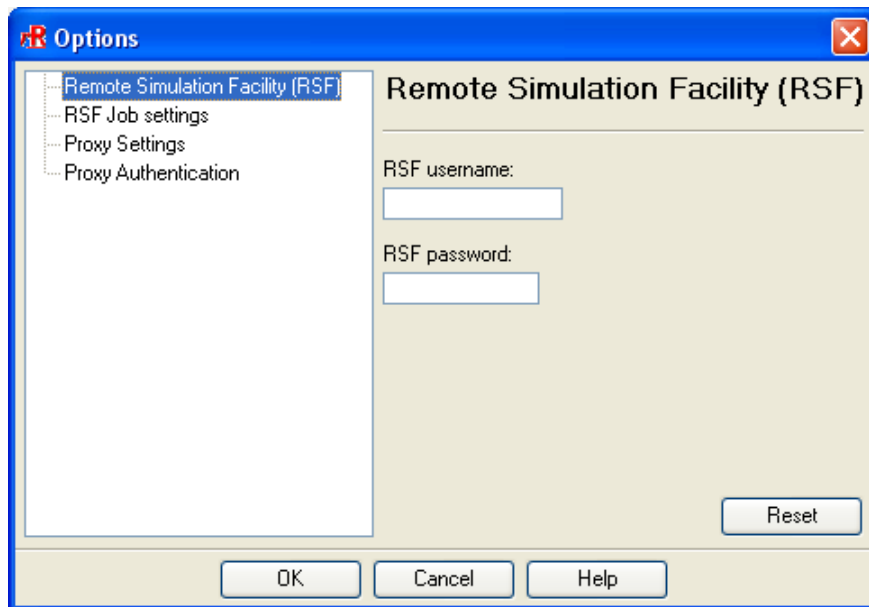


Figure 27.11.26: The Remote Simulation Facility (RSF) Options Panel

- i. Select the **Remote Simulation Facility (RSF)** node from the tree in the **Options** panel (Figure 27.11.26) to save login information for future runs. Enter your RSF username and RSF password.

- ii. Select the **RSF Job settings** node from the tree (Figure 27.11.27) to set the RSF job settings. Select **Delete project on RSF after download** to delete the project and all its associated files on the RSF after the solution is completed and the results are successfully downloaded.

Enter the **Airpak Application Id** to identify the version of Airpak that you want to run on your project (e.g., 2112 for Airpak 2.1.12). By default, the latest version of Airpak will be run.

i Note that if you don't specify that the RSF project be automatically deleted after download, then you can subsequently log on to the RSF site (<https://rsolve.fluent.com/fluent/jsp/index.jsp>) using your UserID and password, and manually delete the project.

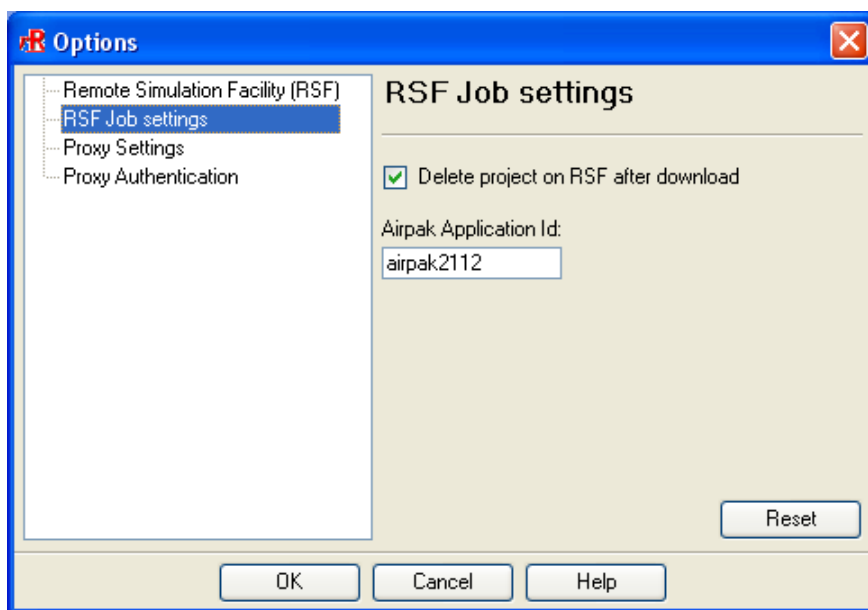


Figure 27.11.27: The RSF Job settings Panel

- iii. Select the **Proxy settings** node in the tree (Figure 27.11.28) to configure the proxy to access the RSF. Enter the server and server port information. Select **Use a proxy server to access the Internet** to access the Internet using a proxy server.

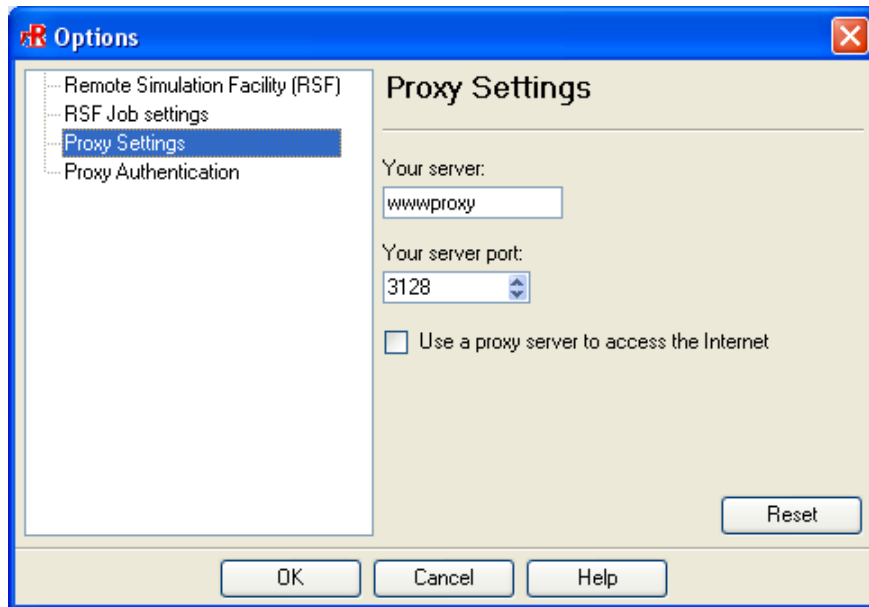


Figure 27.11.28: The Proxy Settings Panel

- iv. Select the **Proxy Authentication** node in the tree (Figure 27.11.29) to set the proxy parameters. Select the **Authentication type** from the drop-down list and enter the **User name**, **Password**, and **Domain**. The **Reset** button allows you to return to your original settings.

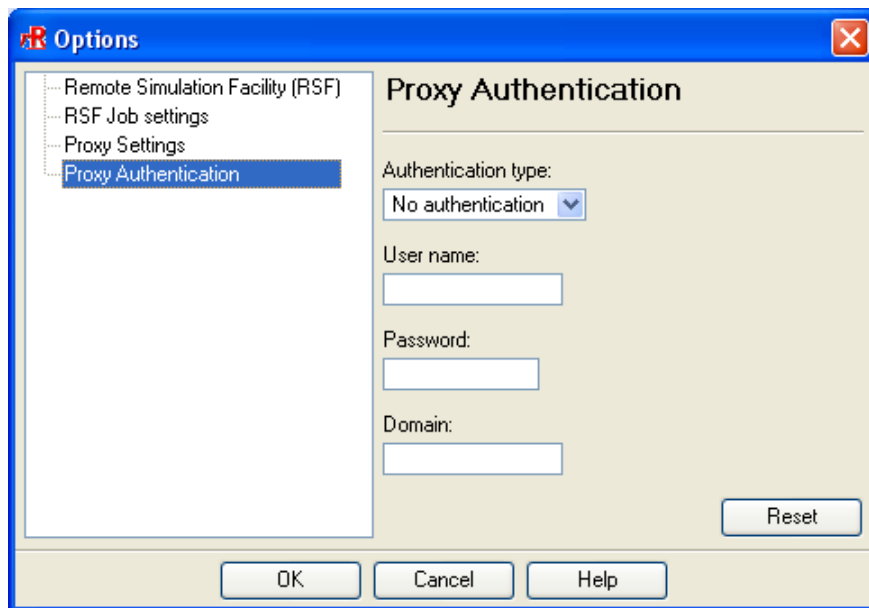


Figure 27.11.29: The Proxy Authentication Panel

- v. Click **OK** in the **Options** panel to save the preferences you have set for your RSF solution and close the panel.
- (g) To start the solution on the RSF, click the **Start Solution** button in the **Main** tab of the **RunRSF** panel (Figure 27.11.21). This will open the **Connect to Remote Simulation Facility** panel (Figure 27.11.30).

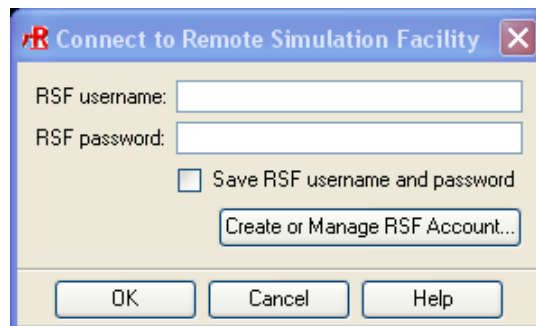


Figure 27.11.30: The Connect to Remote Simulation Facility panel

- (h) Follow Step 3 of the previous procedure for running RSF from within **Airpak** to log on to RSF, run your solution, and download your results for postprocessing.

27.12 Performing Calculations

The following sections provide information about what **Airpak** does when a calculation is started, during the calculation, and when the calculation ends. Information is also provided about judging the convergence of your solution.

27.12.1 Starting the Calculation

When you click on the **Start solution** button in the **Solve** panel (Figure 27.11.1), **Airpak** performs various preprocessing operations, including checking the model, saving a project file, and creating a case file for the solver. The solver will then begin to calculate. See Section 31.6 for theoretical information on the solution procedures.

If **Show diagnostic output from solver** is selected in the **Solve** panel (the default setting), a separate window will open and the solver will print the numeric values of the residuals to the window.

If **Start monitor** is selected in the **Solve** panel (the default setting), **Airpak** will display the convergence history for the calculation in the **Solution residuals** graphics display and control window. See Section 27.12.2 for details on the **Solution residuals** graphics display and control window.

If **Enable sequential solution of flow and energy equations** is selected in the **Solve** panel, **Airpak** will first solve the flow equations, and then solve the energy equation alone. See Section 27.11.2 for details.

The solver runs as a background task separate from **Airpak**, so you can work on another model once the solver starts to solve the current model.

Airpak will save the results of the calculation for non-transient problems at the frequency specified next to **Auto-save interval** in the **Solve** panel. See Section 27.11.2 for details. For a transient simulation, **Airpak** will save the results at the frequency specified next to **Solution save interval** in the **Transient parameters** panel, as described in Section 20.1.

27.12.2 The Solution residuals Graphics Display and Control Window

Airpak opens the **Solution residuals** graphics display and control window (Figure 27.12.1) by default when the solver starts to perform calculations, and displays the convergence history for the calculation.

The **Solution residuals** window is fully interactive. For example, at any time during the calculation, you can zoom in on a specific portion of the graph. To zoom into an area, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box to the desired size, and then release the mouse button. The selected area will then fill the **Solution residuals** window, with appropriate changes to the axes. After you have zoomed into an area, click on **Full range** to restore the graph to its original axes and scale.

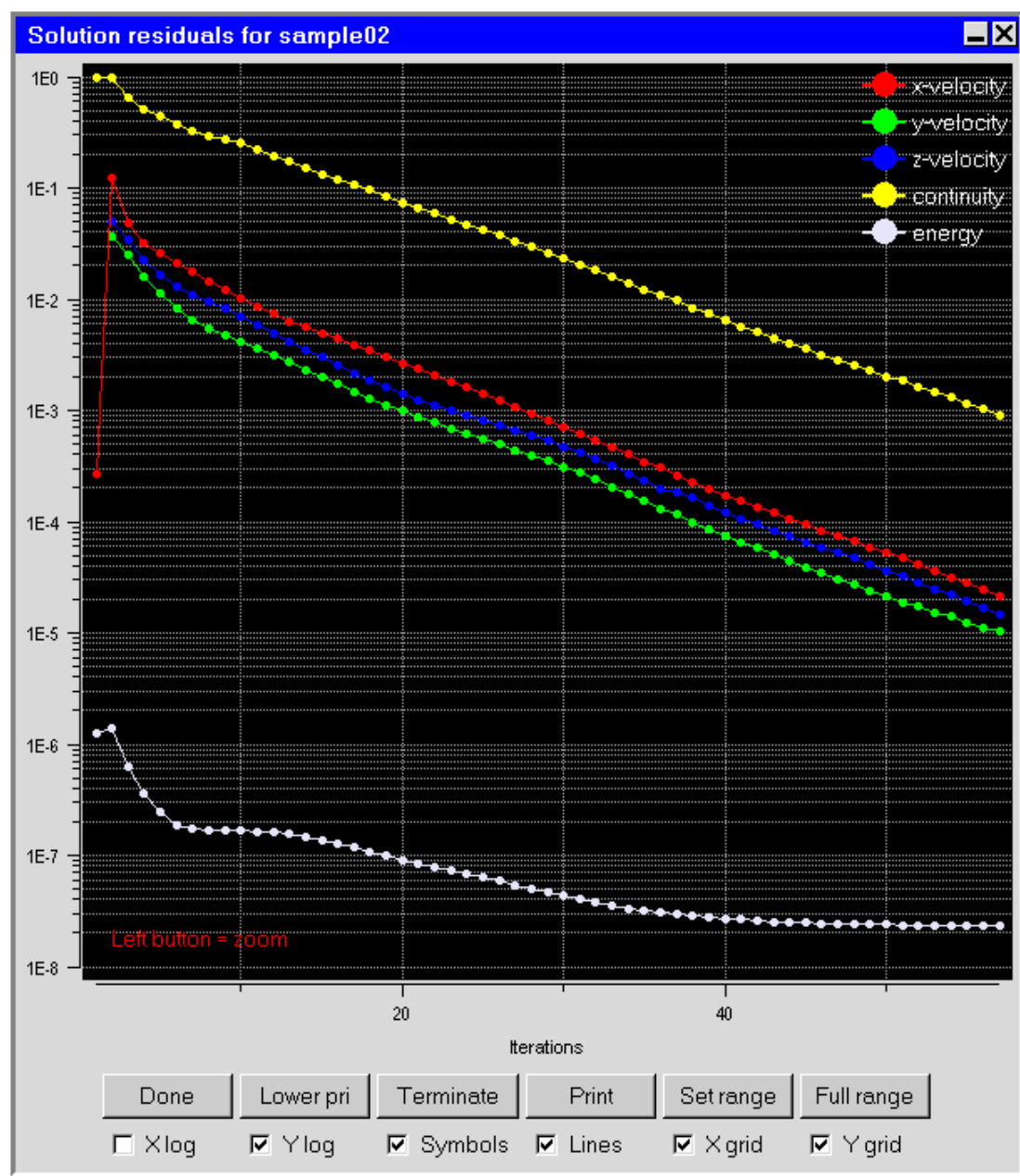


Figure 27.12.1: The Solution residuals Graphics Display and Control Window

Other options available in the **Solution residuals** window include the following:

X log converts the horizontal axis to a logarithmic scale.

Y log converts the vertical axis to a logarithmic scale.

Symbols displays data points on the plotted line.

Lines displays the plotted line.

X grid displays the vertical grid lines on the plot.

Y grid displays the horizontal grid lines on the plot.

Done cancels the monitor and closes the **Solution residuals** window, but does not terminate the solver.

Lower pri changes the solver to a lower priority on the machine permanently for the duration of the calculation.

Terminate stops the solver project *after* the current iteration is completed. **Airpak** acknowledges the termination request in the **Message** window, but the solver does not stop until the **Solution (projectname) is finished** message appears. Note that the **Terminate** button does not close the **Solution residuals** window.

Print generates a hardcopy of the residuals in the **Solution residuals** window. See Section 3.5 for details on saving an image file.

Set range allows you to specify a range for the values of a variable at a monitor point.

Full range allows you to restore the graph to its original axes and scale, if you have used the **Set range** option.



Note that you can change the solution-residual display in the **Solution residuals** window during the calculation, as described in Section 27.12.3.

27.12.3 Changing the Solution Monitors During the Calculation

During the calculation, **Airpak** will update the solution residuals and point monitors after each iteration. You can change the display of the convergence monitors while the solver is running using the **Solution monitor definition** panel.

Solve—→Solution monitor

This panel is very similar to the **Solution monitor parameters** panel (see Figure 27.8.1). You can also specify a **Solution ID** of the solver calculation you wish to monitor in the **Solution monitor definition** panel. The default **Solution ID** will correspond to the solution

ID for the current calculation when the solver is running. You can make changes to the Solution monitor definition panel while the solver is running. Click **Create** to make the changes appear in the Solution residuals graphics display and control windows.

27.12.4 Ending the Calculation

The solution will stop after the specified number of iterations or if the convergence criteria are met. You can also stop the solver during the calculation using the **Terminate** button in the Monitor window (see Section 27.12.2 for details). The message **Solution (projectname) is finished** appears in the Message window when the solver is done. Note that the Solution residuals windows will remain open until you close them by clicking **Done**.

If **Write overview of results when finished** is selected in the Solve panel, Airpak will generate an overview of results when it has finished the calculation.

If **Write report when finished** is selected in the Solve panel, Airpak will generate the report(s) that you defined in the Define summary report panel (see Section 27.10).

If **Compress solution after reporting** is selected in the Solve panel, Airpak will compress some solver-related files when it has finished the calculation to reduce the usage of disk space. This will not affect postprocessing of the solution.

If **Export solution data** is selected in the Solve panel, Airpak will export the solution data to the file specified in the Export solution data text entry box (see Section 27.11.2 for details).

27.12.5 Judging Convergence

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. For most Airpak models, the default convergence criterion is sufficient. This criterion requires that the scaled residuals defined by Equations 31.6-42 and 31.6-44 in Section 31.6.5 decrease to 10^{-3} for all equations except the energy equation, for which the criterion is 10^{-7} .

27.13 Diagnostic Tools for Technical Support

Airpak includes a number of tools designed to aid your technical support engineer in troubleshooting problems in the solver execution. These tools are intended for diagnostic purposes only. The Diagnostics sub-menu provides access to these tools. To open the Diagnostics sub-menu, select **Diagnostics** in the **Solve** menu.

Solve→Diagnostics

The following options are available for diagnostic purposes.

Edit .cas file opens a text-editor window in which you can edit the case file that contains the model information used by the solver.

Edit .diag file opens a text-editor window in which you can edit or view the diagnostic file created for input to the solver. Typically, this will be of use to your technical support engineer in diagnosing the problem.

Edit .uns_out file opens a text-editor window in which you can edit or view the residuals file created by the solver. This is useful to either check the amount of time taken by the solver, or to check for any errors printed out during the course of the solution.

Edit Optimization log opens a text-editor window in which you can edit or view the optimization log created by **Airpak**. This is useful to get detailed information regarding the optimization run. The quantities reported are the values of design variables for all trials performed, the corresponding objective and constraint functions, and the relative changes in each of these quantities.

To combine all relevant files for your model into a single file that you can send to your technical support engineer, select **Pack** from the **File** menu.

File→**Pack**

This opens the **File selection** dialog box, in which you can select a name for the packaged file. See Section 2.1.8 for details on selecting a file.

To email the packaged project, select **Email project** from the **File** menu.

File→**Email project**

This will open the **File selection** dialog box, in which you can select a name for the packaged file. See Section 2.1.8 for details on selecting a file. By clicking the **Save** button in the **File selection** dialog box, the **Email Project** panel will open up (Figure 27.13.1). In this panel enter **Recipient email address**, **Subject for email**, and **Message text**, then click **Send email**.

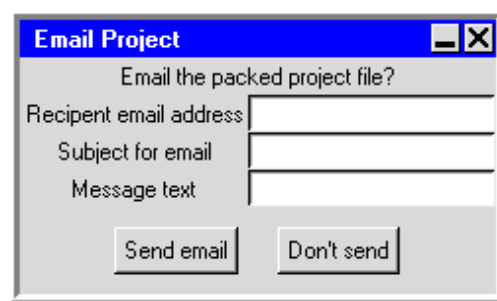


Figure 27.13.1: The Email Project Panel

Airpak provides a number of different methods for examining the results of your simulation (also referred to as postprocessing your results). You can create graphical displays of data on different portions of the model, or two-dimensional (XY) plots of solution data.

The information in this chapter is divided into the following sections:

- Section 28.1: Overview: The Post Menu and Postprocessing Toolbar
- Section 28.2: Graphical Displays
- Section 28.3: XY Plots
- Section 28.4: Selecting a Solution Set to be Examined
- Section 28.5: Zoom-in Modeling

28.1 Overview: The Post Menu and Postprocessing Toolbar

28.1.1 The Post Menu

The Post menu is accessible from the Main Menu bar. When you select Post in the Main Menu bar, a set of options is displayed as shown in Figure 28.1.1.

The Post menu contains all of the functions you need to examine your results in Airpak. These functions include the following:

- displaying results on one or more faces of an object (Post→Object face)
- displaying results on a cross-section of the model (Post→Plane cut)
- displaying results on an isosurface (Post→Isosurface)
- displaying results at a point (Post→Point)
- displaying results at a point on a surface (Post→Surface probe)
- displaying the locations of the minimum and maximum values of postprocessing variables (Post→Min/max locations)
- plotting a variable along a line through the model (Post→Variation plot)

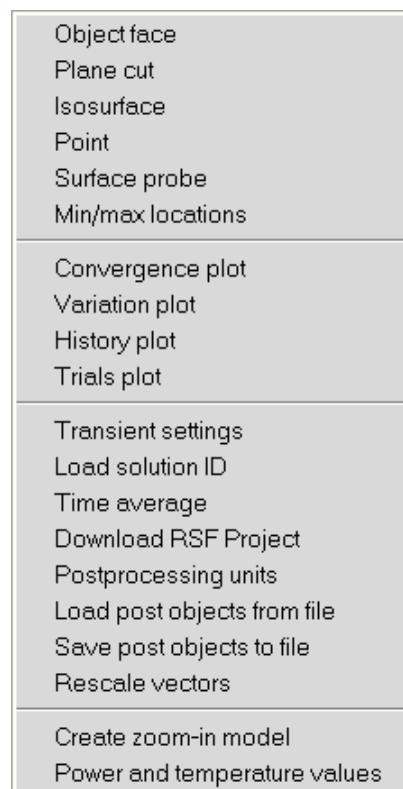


Figure 28.1.1: The Post Menu

- plotting a variable over different solutions (Post→Trials plot)
- selecting a specific solution set to be examined (Post→Load solution ID)
- loading postprocessing objects from a file (Post→Load post object from file)
- saving postprocessing objects to a file (Post→Save post objects to file)
- rescaling vectors to their original sizes (Post→Rescale vectors)
- creating zoom-in models (Post→Create zoom-in model)

In addition to these functions, the **Post** menu contains options for plotting solution residuals (Post→Convergence plot, described in Chapter 27), solution variable histories over time (Post→History plot, described in Chapter 20). Functions for units (Post→Post-processing units) and transient simulations (Post→Transient settings) are described in Chapters 5 and 20, respectively. The time-averaging function (Post→Time average) for transient results is described in Chapter 20. The function for downloading your RSF-run project for postprocessing (Post→Download RSF project) is described in Section 27.11.5.

If you have saved any postprocessing objects in your project file using the **Save postprocessing objects** option in the **Save project** panel (as described in Section 3.4), Airpak will ask you if you want to load them when you enter the **Post** menu for the first time.


28.1.2 The Postprocessing Toolbar


The Postprocessing toolbar (Figure 28.1.2) contains options that allow you to examine your results using Airpak's postprocessing objects. A description of the Postprocessing toolbar options is provided below.



Figure 28.1.2: The Postprocessing Toolbar


Object face () allows you to display results on object faces in the model.


Plane cut () allows you to display results on cross-sections of the model.


Isosurface () allows you to display results on defined isosurfaces in the model.


Point () allows you to display results at points in the model.

Surface probe () allows you to display results at a point on a surface in the model.


Variation plot () allows you to plot a variable along a line through the model.


History plot () allows you to plot solution variable histories over time.


Trials plot () allows you to plot trial solution variables.

Transient settings () opens the **Post-processing time** panel where you can set parameters for transient simulations.

Load solution ID () allows you to select a specific solution set to be examined.

Time average () allows you to view the results of a time averaged (transient) problem as if it were steady-state.

Download RSF project () opens the **Download RSF project** panel, where you can run an Airpak job using the Remote Simulation Facility (RSF) and download the results for postprocessing in Airpak.

Summary report () opens the **Define summary report** panel where you can specify a summary report for a variable on any or all objects in your Airpak model.

28.2 Graphical Displays

28.2.1 Overview of Generating Graphical Displays

Airpak allows you to view solution results for the variables listed in Chapter 30 in four types of locations in the domain:

- on one or more faces of a selected object in the model (e.g., the side of a PCB or the face of a block)
- on a cross-sectional plane that extends throughout the room or a truncated plane
- on a surface (called an isosurface) defined by the constant value of a specified variable, such as temperature
- at a particular point in the model, which can be easily moved to examine results at various locations

Sections 28.2.4–28.2.7 describe how to display results on these four types of locations in the model, which are referred to as postprocessing objects. Section 28.2.3 describes how to manage these postprocessing objects (e.g., how to delete, deactivate, and save them).

In the specified portion of the domain, you can display the data in four ways:

- contours of a specified variable, such as temperature or pressure
- vectors of a specified variable
- particles released from the object or surface
- mesh on the specified object or surface in a specified color

Parameters that control contour, vector, and particle displays are described in Sections 28.2.8–28.2.10.

28.2.2 The Significance of Color in Graphical Displays

Contour, vector, and particle displays are associated with a color spectrum representing the range of values for the variable being displayed (e.g., temperature). The spectrum ranges from violet (minimum value) to red (maximum value).

Airpak displays a legend (in the graphics window) showing the color spectrum and its associated values for the currently-selected postprocessing object. To reposition the legend, hold down the <Control> key, press and hold down the middle mouse button, and drag the legend to any location in the graphics window.

The default color spectrum is evenly distributed across the range of values for the specified variable (i.e., each band of the color spectrum is associated with an equal portion of the variable range). It is possible, however, to change the ranges in order to increase or decrease the size of the color band associated with a particular portion of the variable range. To do so, hold down the <Control> key, press and hold down the right mouse button, and drag the legend value lines up or down the spectrum. To return the legend to its original appearance, select **Rescale vectors** in the **Post** menu.

You can change the format of the labels that define the color divisions in the legend using the **Settings** panel. See Section 6.3 for details of the data formats available in **Airpak**, and how to specify the precision for each of the available formats.

28.2.3 Managing Postprocessing Objects

Once you have created postprocessing objects (according to the instructions in Sections 28.2.4–28.2.7), you can manipulate them using the **Postprocessing** node in the **Model manager** window, shown in Figure 28.2.1.

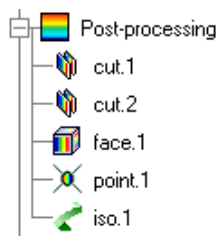


Figure 28.2.1: The Post-processing Node in the Model manager Window

When you define a new postprocessing object using one of the options in the **Post** menu or **Postprocessing** toolbar (i.e., **Object face**, **Plane cut**, **Isosurface**, **Surface probe**, or **Point**), the object is added under the **Postprocessing** node in the **Model manager** window. As is the case for modeling objects, each postprocessing object has a unique object name, such as **iso.1**. In the **Model manager** window shown in Figure 28.2.1, for example, one of each type of postprocessing object is listed.

Summary data associated with the postprocessing object selected in the **Model manager** window is displayed in the **Edit** window (Figure 28.2.2). The values displayed are the global and object-specific minima and maxima, location of the maximum of the variable that is being plotted, total, mean, and standard deviation of the variable, and area of the postprocessing object (or a value of the specified variable and the location for a point object). You can save the object's summary information listed in the **Edit** window, by clicking the **Save** button. This will open the **Save summary data** panel, in which you can select or specify the name of your summary file.

Object:	face.1	Variable:	Temperatur	
Global min:	19.9999	Total:	1.1	C-m2
Global max:	40.763	Mean:	36.3	C
Object min:	31.2907	Std dev:	2.59	C
Object max:	40.7229	Area:	0.0304	m2
Max loc:	0.14 0.0323077 0.1633			
Save				

Figure 28.2.2: The Edit Window for postprocessing objects

The following sections describe the ways in which you can manipulate postprocessing objects.


Activating and Deactivating Postprocessing Objects

By default, all postprocessing objects that you create will remain in the graphics window. If you have created several postprocessing objects, the display can become very cluttered, making it difficult to examine your results carefully. In such cases, you can remove an object from the display temporarily by selecting it in the **Model manager** window, right-clicking to open the context menu, and turning off the **Active** option. You can repeat this for each object that you do not want to include in the display. You can also deactivate an object by selecting it in the **Model manager** window and dragging it to the **Inactive** node in the **Model manager** window. When an object is deactivated, it is simply removed from the display, *not* deleted from **Airpak**. Since it still exists, you can add it to the display again by turning the **Active** option back on, or by dragging the object from the **Inactive** node into the **Postprocessing** node of the **Model manager** window.

Modifying Postprocessing Objects

If you want to modify an existing postprocessing object (e.g., select different faces of an object for display), double-click on the postprocessing object in the **Model manager** window, or right-click on the postprocessing object and select the **Edit** option in the context menu. **Airpak** will open the appropriate panel (e.g., the **Object face** or **Isosurface** panel), where you can make the desired changes to the object.

Deleting Postprocessing Objects

To delete a postprocessing object, select the object in the **Model manager** window and click the  button. The object will be removed from the display. Alternatively, you can right-click on the selected object in the **Model manager** window and select **Delete** from

the drop-down list. You can also drag the selected postprocessing object into the Trash node of the Model manager window.



Unlike other Airpak objects, postprocessing objects are permanently deleted when a delete operation is performed.

Rescaling Vectors in the Display

If one or more postprocessing objects include vectors, zooming in or out on the display will cause both the model and the vectors to increase or decrease in size. To redisplay the model with the vectors drawn at their original sizes, select **Rescale vectors** in the **Post** menu.

Saving and Reloading Postprocessing Objects

After creating a number of postprocessing objects, you may want to save them to a file so that you can reuse them the next time you view this model in Airpak. To save all existing postprocessing objects to a file, select **Save post objects to file** in the **Post** menu. Enter a name in the **File name** text field in the resulting **File selection** dialog box, and click **Save** to save the file.

To read the postprocessing objects back into Airpak, select **Load post objects from file** in the **Post** menu. Enter the appropriate file name in the **File name** text field in the resulting **File selection** dialog box, and click **Open** to load the objects from the file.

Saving, Reloading, and Clearing Airpak Views

After obtaining a certain Airpak view during postprocessing, you may want to save the view so that you can revisit it at a later time during your Airpak session. To save an Airpak view, select **Save user view** in the **Orient** menu. Airpak will open a **Query** panel, prompt you for a view name, and save the current view using your specified name. The new view name is attached to the list of user views at the bottom of the **Orient** menu. To reload the view back into Airpak at a later time during the same session, select the view name listed at the bottom of the **Orient** menu. To remove all saved views from the listing, select **Clear user views** in the **Orient** menu.



Note that views are only temporarily saved during a particular Airpak work session. The views are not saved when you quit Airpak.

28.2.4 Displaying Results on Object Faces

An Airpak object face is defined as a surface consisting of one or more faces of one or more modeling objects. Note that an object face can be specified not only for single objects or all objects of a single type, but for groups as well. (You will need to create the group, as described in Section 7.6, before displaying the results on a group of objects.) By default, all sides of an object are selected as the object face. Thus, all six sides of a block comprise its default object face. Figure 28.2.3 shows a block for which only two sides and the top are specified as parts of the object face.

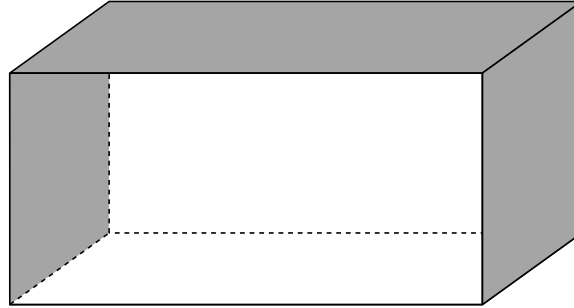



Figure 28.2.3: Block Object Face

An object face can be used to isolate a single object in the model, such as a block employed as a power supply, to determine the effects of the simulation on the surface of the object. In conjunction with particle traces, an object face can also be instrumental in observing flow. For example, if the surface of a fan is defined as an object face and particles are introduced, the flow from the fan's surface through the model can be observed.

Defining an Object Face

To define a postprocessing object face, you will use the **Object face** panel (Figure 28.2.4). To open the panel, click on the  button in the **Postprocessing** toolbar or select **Object face** in the **Post** menu.

Post—>Object face



Note that **Object** and **Object sides** in the panel refer to *modeling* objects (e.g., blocks, vents, fans), *not* postprocessing objects.

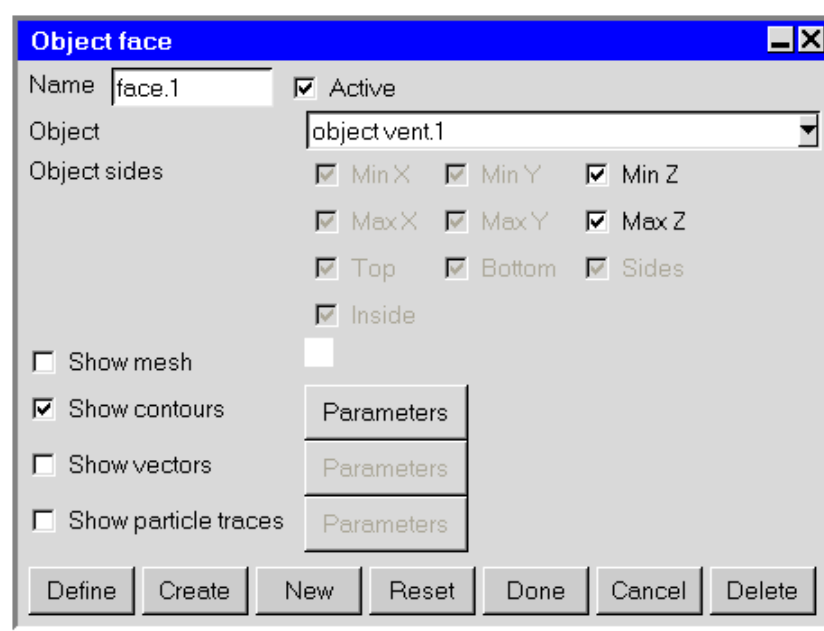


Figure 28.2.4: The Object face Panel

The procedure for defining an object face is as follows:

1. In the **Name** field, enter a new name for the object face. The default name is **face.n**, where **n** is a sequential number distinct for each object face created. You can keep the default name if you want, but a customized name is generally more useful.
2. To select the specific modeling object for which you want to create an object face, select the desired object (e.g., **vent.1**) in the **Object** drop-down list. The object name is displayed in the text field. (e.g., **object vent.1**).
3. To specify more than one object for which you want to create the object face, select the desired object (e.g., **vent.1**) from the **Object** drop-down list, hold down the **<Control>** key, and select another object from the list. When multiple objects are selected, the object names are listed in the **Object** text field separated by spaces.
To select multiple items that are listed consecutively in the drop-down list, you can select the first item, hold down the **<Shift>** key, and then select the last item in the list. All items between the first selected item and the last selected item will be selected.
4. Specify which sides of the modeling object should be included in the object face by enabling/disabling the corresponding options for **Object sides**. Only those sides appropriate for the object selected in the **Object** list will be available. Rectangular-block specifications, for example, include **Min X**, **Max X**, **Min Y**, **Max Y**, **Min Z**, and

Max Z. Cylindrical and polygonal blocks, on the other hand, include **Top**, **Bottom**, and **Sides**. (For a rectangular block, **Min X** and **Max X** represent the sides of the block on the x axis with the minimum and maximum x values, respectively.) By default, all object sides are active.

5. Specify which type of display you want to show on the object face by turning on the appropriate option: **Show mesh**, **Show contours**, **Show vectors**, or **Show particle traces**. (You can choose more than one, but the resulting display may become too busy.)
6. To modify the default attributes of a contour, vector, or particle display, click the corresponding **Parameters** button. See Sections 28.2.8–28.2.10 for details. If you choose to display the mesh, you can specify the mesh color by selecting a color in the drop-down color palette. To display the color palette, click on the small color rectangle next to the **Show mesh** option.
7. Click **Define** to define the object face according to the specifications in the panel, without loading the data. If the data had already been loaded, this button will be called the **Display** button. The **Define** (or **Display**) button will change to the **Update** button, which you can use to update the object face if you make further changes to the specifications in the panel. To create and display the object face by loading the data, click the **Create** button.

Modifying an Existing Object Face


If you want to modify an existing object face, make the desired changes to the entries in the **Object face** panel, and then click **Update**. If you start to make changes but then wish to return to the original settings in the panel, click **Reset**. **Airpak** will update the panel with the settings that were last saved (i.e., the settings that were present when you last clicked **Create**, **Update**, or **Done**).

Creating Another Object Face

If you are already in the **Object face** panel and you want to create a new object face, click **New** at the bottom of the panel. **Airpak** will increment the number on the default **Name** and clear the previous settings so that you can start defining a new object face.

Deactivating or Deleting an Object Face

If you want to temporarily remove the object face from the display, you can turn off the **Active** option at the top of the **Object face** panel. This has the same effect as turning off the **Active** option in the postprocessing object context menu, described in Section 28.2.3.

If you want to permanently delete the object face from **Airpak**, click the **Delete** button at the bottom of the **Object face** panel. This has the same effect as the  button, described in Section 28.2.3.

28.2.5 Displaying Results on Cross-Sections of the Model

To display results on a cross-section of the model, you will need to create a plane cut. An **Airpak** plane cut is defined as the region of a specified plane that intersects the model. Plane cuts allow you to examine areas between objects in the model (in which the fluid alone exists), as well as regions inside objects that contain a fluid or solid material.

There are three ways to specify the plane:

- Specify a point on the plane and the normal to the plane (see Figure 28.2.5).

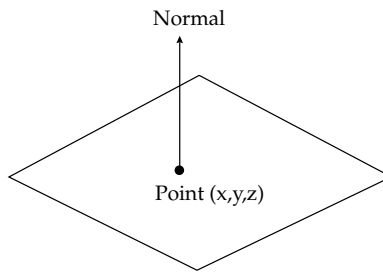



Figure 28.2.5: Plane Cut Defined by Point and Normal

- Specify the equation for the plane.
- Use the mouse to specify the plane in the graphics window.

Defining a Plane Cut

To define a plane cut, you will use the **Plane cut** panel (Figure 28.2.6). To open the panel, click on the  button in the **Postprocessing** toolbar or select **Plane cut** in the **Post** menu.

Post→Plane cut

The procedure for defining a plane cut is as follows:

1. In the **Name** field, enter a new name for the plane cut. The default name is **cut.n**, where **n** is a sequential number distinct for each plane cut created. You can keep the default name if you want, but a customized name is generally more useful.

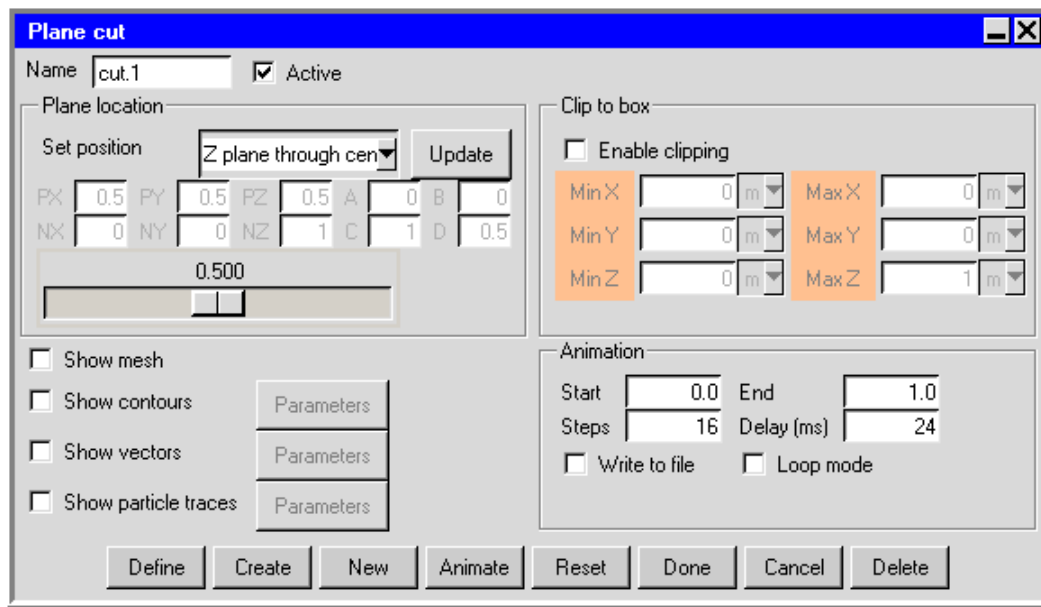


Figure 28.2.6: The Plane cut Panel

2. Define the plane using one of the following four methods:

- Specify a plane through the center of the model:
 - (a) Select X plane through center, Y plane through center, or Z plane through center in the Set position drop-down list under Plane location in the Plane cut panel. The plane cut created is perpendicular to the specified axis. For example, selecting Y plane through center creates a plane in the x - z plane located in the center of the model.
- Specify a point on the plane and the normal direction to the plane:
 - (a) Select Point and normal in the Set position drop-down list.
 - (b) Enter the coordinates of a point on the plane (PX, PY, PZ).
 - (c) Enter a vector defining the direction normal to the plane (NX, NY, NZ). For example, entering (1, 0, 0) for the vector will define a normal pointing in the x direction.
- Specify an equation that defines the plane:
 - (a) Select Coeffs ($Ax + By + Cz = D$) in the Set position drop-down list.
 - (b) Enter the coefficients A, B, C, and D for the equation

$$Ax + By + Cz = D \quad (28.2-1)$$

- Specify the plane using your mouse:
 - (a) Use the **Orient** menu or the **Orientation commands** toolbar to specify the desired orientation. If you require a horizontal or vertical plane through your model, choose the orientation such that the plane of the display screen is perpendicular to the desired plane cut. For example, if you want to display results on a y - z or x - y plane, choose **Orient positive Y** as the orientation so that the display-screen plane is the x - z plane. If you want to specify the plane by selecting three points on it, orient your model as desired.
 - (b) Select **Horizontal-screen select**, **Vertical-screen select**, or **3 Points-screen select** in the **Set position** drop-down list in the **Plane cut** panel.
 - (c) If you selected the **Horizontal-screen select** or **Vertical-screen select** option, click your left mouse button in the graphics window to indicate a point on the desired plane. **Airpak** will create a plane cut that is a horizontal or vertical plane perpendicular to the plane of the graphics window and passing through the selected point.

If you selected the **3 Points-screen select** option, select the first, second, and third points on the plane in the graphics window using the left mouse button. Each point must be on the edge of an object or the room. If it is not, **Airpak** will move the point to the nearest location on the edge of an object or the room.



The procedure for defining the plane cut with your mouse is the same as for defining a plane for displaying the mesh. See Section 26.9.2 for additional information (including diagrams).

3. Specify which type of display you want to show on the plane cut by turning on the appropriate option: **Show mesh**, **Show contours**, **Show vectors**, or **Show particle traces**. You can choose more than one, but the resulting display may become too busy.
4. To modify the default attributes of a contour, vector, or particle display, click the corresponding **Parameters** button. See Sections 28.2.8–28.2.10 for details. If you choose to display the mesh, you can specify the mesh color by selecting a color in the drop-down color palette. To display the color palette, click the small color rectangle next to the **Show mesh** option.
5. Click **Define** to define the plane cut according to the specifications in the panel, without loading the data. If the data had already been loaded, this button will be called the **Display** button. The **Define** (or **Display**) button will change to the **Update** button, which you can use to update the plane cut if you make further changes to the specifications in the panel. To create and display the plane cut by loading the data, click the **Create** button.

6. To advance the currently-displayed plane cut through the room so that you can easily view results on different cross-sectional planes, use the slider bar under **Plane location**. You can also generate an animation of the plane cut moving through the room, as described below.

Modifying an Existing Plane Cut

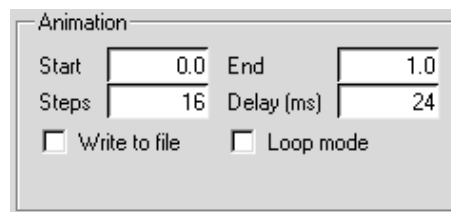
If you want to modify an existing plane cut, make the desired changes to the entries in the **Plane cut** panel, and then click **Update**. If you start to make changes but then wish to return to the original settings in the panel, click **Reset**. **Airpak** will update the panel with the settings that were last saved (i.e., the settings that were present when you last clicked **Create**, **Update**, or **Done**).

Creating Another Plane Cut

If you are already in the **Plane cut** panel and you want to create a new plane cut, click **New** at the bottom of the panel. **Airpak** will increment the number on the default **Name** and clear the previous settings so that you can start defining a new plane cut.

Animating a Plane Cut

If you want to create an animation that progresses from one static view of a plane cut to the next, go to the **Animation** section of the **Plane cut** panel (Figure 28.2.6).



To create an animation showing how relevant contours, vectors, particle displays, and/or the mesh change as you move through the room, follow the steps below.

1. Specify the starting plane cut (**Start**) and the ending plane cut (**End**) for the animation. A **Start** of 0 and an **End** of 1 indicates that the animation will begin at one end of the room and finish at the opposite end. The direction of the animation through the room is defined by the direction of the normal to the plane specified in the **Plane cut** panel.
2. Specify the number of steps (**Steps**), i.e., the number of plane cuts **Airpak** should display between the starting plane and the ending plane. **Airpak** will interpolate smoothly between the starting plane and the ending plane that you define, creating

the specified number of planes. Note that the number of steps includes the starting and ending planes.

3. Specify the **Delay (ms)** or the time between each frame in the animation in milliseconds. This will determine the duration of the animation. For example, if you specify 10 plane cuts next to **Steps**, and then specify 500 as the **Delay (ms)**, **Airpak** will create a five-second animation of the plane cuts. Note that the delay value is added to however long it takes **Airpak** to create a snapshot for the animation. The time it takes **Airpak** to create a snapshot is dependent on the speed of your system.
4. If you want **Airpak** to play the animation only in the graphics window, and you want the playback to repeat continuously, turn on the **Loop mode** option. To play the animation once through from start to finish, turn off the **Loop mode** option. You can also use the **Loop mode** option if you want to save an animated GIF or FLI file, as described below.
5. To save the animation to a file, select the **Write to file** option and click the **Write** button. This displays the **Save animation** file selection dialog box where you can save animations in MPEG, AVI, animated GIF, FLI, or Flash format. See below for details about saving animations.

Note that when the **Write to file** option is selected, the **Delay (ms)** field becomes the **Frames/s** field. The **Frames/s** field designates the number of animation frames displayed per second.

6. Click **Animate** to start the animation. To stop the animation during playback, click the **Interrupt** button in the upper right hand corner of the **Airpak** interface.

Saving an Animation

You can save an animation to an MPEG, AVI, animated GIF, FLI, or Flash file using the **Save animation** file selection dialog box.

In either format, there are two options for you to consider when saving an animation file: **Print region** and **Scale factor**. These options are available when you click the **Options...** button in the **Save animation** file selection dialog box.

Print region allows you to define the region of the graphics window that should be written to the file. See Section 3.5.2 for details about specifying the **Print region**.

The **Scale factor** is the factor by which **Airpak** will scale the image of the graphics window when it creates the file. If you specify a **Scale factor** of less than 1 for any of the files described below, you may want to thicken the outlines of the room and the objects in the graphics window; otherwise these outlines might not appear in the animation file. You can use the **Graphical styles** panel (described in Section 6.3.6) to thicken the lines.

Saving an MPEG File

To save the animation as an MPEG file, select **MPEG files (*.mpg)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. Note that you must specify the **Scale factor** for the MPEG file. If the MPEG file is to be viewed on a Windows machine, a **Scale factor** of 0.3 is recommended.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single MPEG file that you can view using an MPEG player.

For MPEG files to be played properly, you must use one of the following frame rates: 23.976, 24, 25, 29.97, 30, 50, 59.94, or 60.

Saving an AVI File

To save the animation as an AVI file, select **AVI files (*.avi)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single AVI file. You can then import the AVI file into Microsoft **PowerPoint** and use the animation in a presentation.

Saving an Animated GIF File

To save the animation as an animated GIF file, select **GIF files (*.gif)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animated GIF file, as described above. If you want the playback of the animation in the GIF file to repeat continuously, select the **Loop mode** option.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single animated GIF file that you can view using a web browser. You can also import the animated GIF file into Microsoft **PowerPoint** and use the animation in a presentation.

Saving an Autodesk FLI File

To save the animation as an FLI file (an Autodesk animation file format), select **FLI files (*.fli)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above. If you want the playback of the animation in the FLI file to repeat continuously, select the **Loop mode** option.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single FLI file (e.g., **movie.fli**) that

you can view using a web browser if you have an FLI viewer plug-in. It will also create an HTML file (e.g., `movie.html`) that can be viewed using a web browser. **Airpak** also creates Java applets that allow you to view the HTML file as an animation.

When you view the HTML file in a web browser, the following commands are available:

- To continue the animation in the chosen direction (forward or backward), use the `<Space>` bar on your keyboard.
- To play the animation forward, use the `.` key.
- To play the animation backward, use the `,` key.
- To reverse the direction of the animation during playback, use the `r` key.
- To speed up the animation, use the `+` key.
- To slow down the animation, use the `-` key.
- To return to the original animation speed, use the `0` (zero) key.

Note that **Airpak** also produces two additional files when it creates the FLI file: `fli-play.class` and `flickframe.class`. All four files (e.g., `movie.fli`, `movie.html`, `fli-play.class`, and `flickframe.class`) must be present for the animation in the HTML file to be played back.

Saving a Flash File

To save the animation as a Flash file, select **Flash files (*.swf)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single Flash file. You can then import the Flash file into Microsoft **PowerPoint** and use the animation in a presentation.

Clipping a Plane Cut

If you have created a plane cut, but you do not want to use the whole plane cut to display data, you can clip it to create a new plane cut that spans a specified subrange. The clipped plane cut consists of those points on the selected plane cut where the x , y , and z values are within the specified range.

To clip an existing plane cut, turn on the **Enable clipping** option in the **Clip to** box section in the **Plane cut** panel (Figure 28.2.6).

Clip to box	
<input checked="" type="checkbox"/> Enable clipping	
Min X	Max X
0 m	0.3454 m
Min Y	Max Y
0 m	0.0313 m
Min Z	Max Z
0 m	0.2355 m

Enter values for Min X, Min Y, Min Z, Max X, Max Y, and Max Z, and click Update.


These values can also be automatically determined by aligning the appropriate edges of the clipped plane cut to the existing edges of objects or bounding boxes of assemblies. The procedure is similar to that described in Section 7.3.8.

For example, to set the Min X value, you would first click Min X displayed in orange in the Plane cut panel, then click the edge in the graphics window that you want to align with the Min X edge of the clipped plane cut box.

Just as with other postprocessing objects, data associated with the clipped plane cut object (e.g., specified variable as well as global and object-specific minima and maxima) are displayed in the Edit window (Figure 28.2.2), and you can save them in a file by using the Save button.

Deactivating or Deleting a Plane Cut

If you want to temporarily remove the plane cut from the display, you can turn off the Active option at the top of the Plane cut panel. This has the same effect as turning off the Active option in the postprocessing object context menu, described in Section 28.2.3.

If you want to permanently delete the plane cut from Airpak, click the Delete button at the bottom of the Plane cut panel. This has the same effect as the  button, described in Section 28.2.3.

28.2.6 Displaying Results on Isosurfaces

An isosurface is defined as a surface representing a constant value of any primary or derived solution variable, such as temperature. Isosurfaces can take virtually any shape and are sometimes discontinuous. An example is shown in Figure 28.2.7.

An isosurface can be used to examine surfaces of constant value for a given solution variable. For example, an isosurface can be helpful if you are trying to determine how quantities such as temperature or pressure are convected through a model.

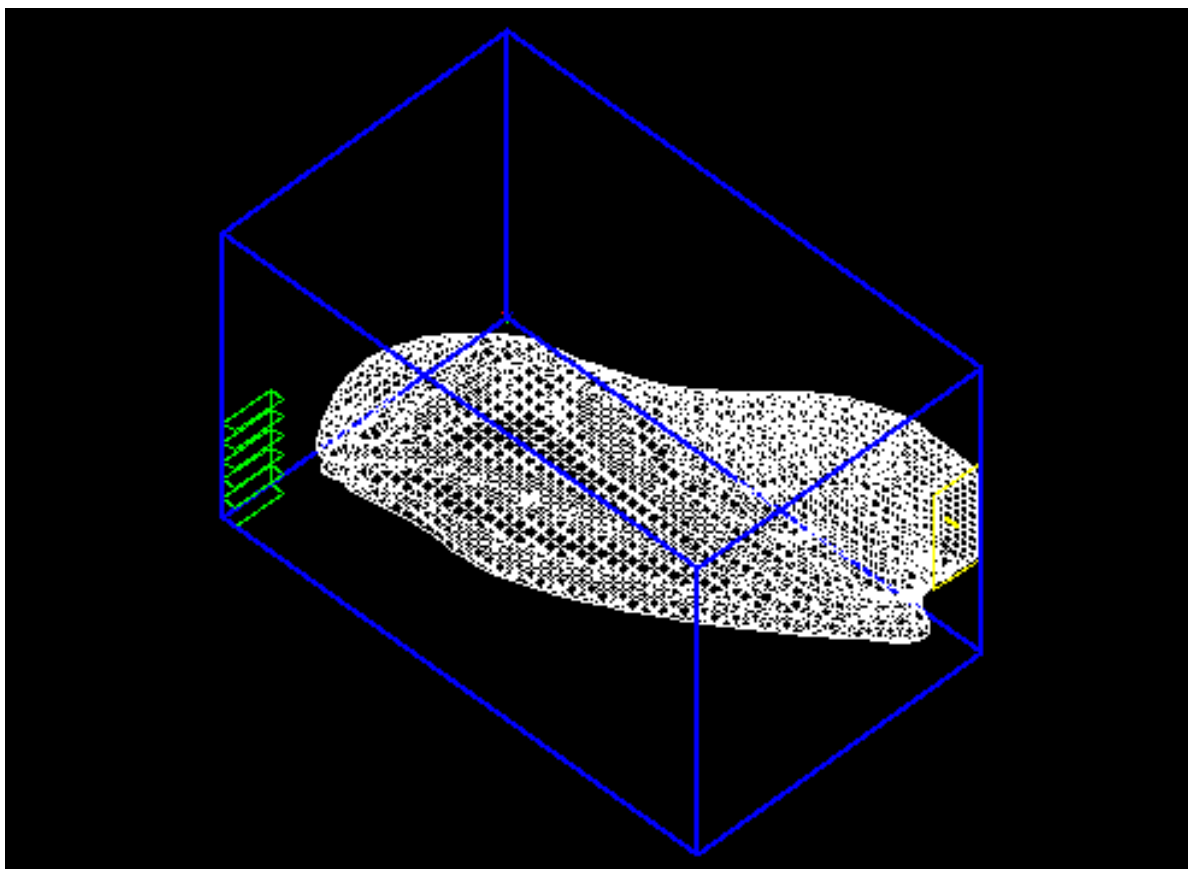



Figure 28.2.7: Isosurface Example

Defining an Isosurface

To define an isosurface, you will use the **Isosurface** panel (Figure 28.2.8). To open the panel, click on the  button in the Postprocessing toolbar or select **Isosurface** in the Post menu.

Post → Isosurface

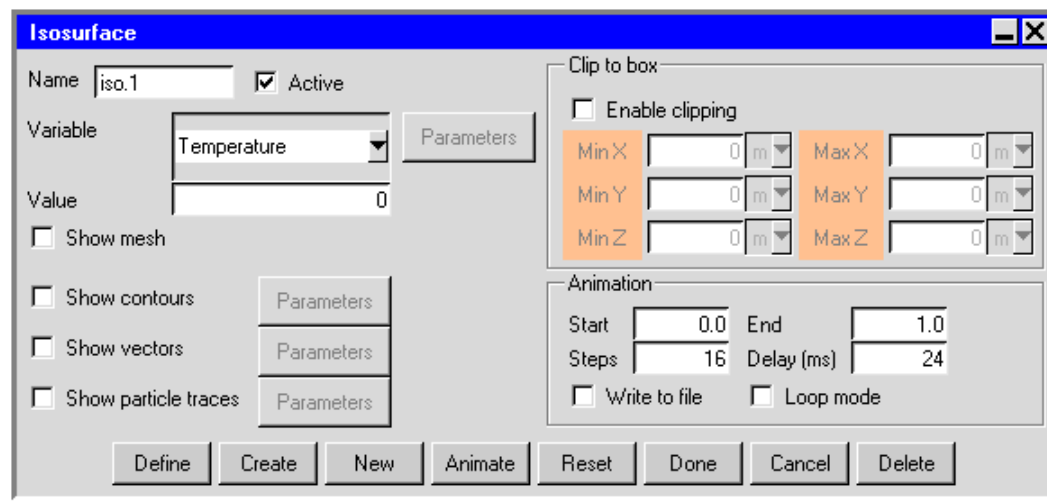


Figure 28.2.8: The Isosurface Panel

The procedure for defining an isosurface is as follows:

1. In the **Name** field, enter a new name for the isosurface. The default name is **iso.n**, where **n** is a sequential number distinct for each isosurface created. You can keep the default name if you want, but a customized name is generally more useful.
2. Specify the variable on which the isosurface is based. Select the desired variable (e.g., **Temperature**) from the **Variable** drop-down list. The available variables are described in Chapter 30.
3. Specify the constant **Value** of the specified **Variable** that defines the isosurface.
4. Specify which type of display you want to show on the isosurface by turning on the appropriate option: **Show mesh**, **Show contours**, **Show vectors**, or **Show particle traces**. (You can choose more than one, but the resulting display may become too busy.)
5. To modify the default attributes of a contour, vector, or particle display, click the corresponding **Parameters** button. See Sections 28.2.8–28.2.10 for details. If you choose to display the mesh, you can specify the mesh color by selecting a color

in the drop-down color palette. To display the color palette, click the small color rectangle next to the **Show mesh** option.

6. Click **Define** to define the isosurface according to the specifications in the panel, without loading the data. If the data had already been loaded, this button will be called the **Display** button. The **Define** (or **Display**) button will change to the **Update** button, which you can use to update the isosurface if you make further changes to the specifications in the panel. To create and display the object face by loading the data, click the **Create** button.

Modifying an Existing Isosurface

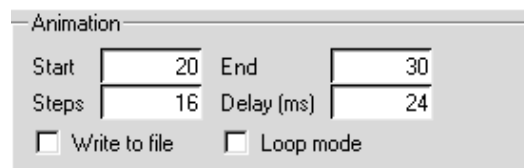
If you want to modify an existing isosurface, make the desired changes to the entries in the **Isosurface** panel, and then click **Update**. If you start to make changes but then wish to return to the original settings in the panel, click **Reset**. **Airpak** will update the panel with the settings that were last saved (i.e., the settings that were present when you last clicked **Create**, **Update**, or **Done**).

Creating Another Isosurface

If you are already in the **Isosurface** panel and you want to create a new isosurface, click **New** at the bottom of the panel. **Airpak** will increment the number on the default **Name** and clear the previous settings so that you can start defining a new isosurface.

Animating an Isosurface

If you want to create an animation that progresses from one static view of an isosurface to the next, go to the **Animation** section of the **Isosurface** panel (Figure 28.2.8).



To create an animation showing how relevant contours, vectors, particle displays, and/or the mesh change as you move through the room, follow the steps below.

1. Specify the starting isosurface (**Start**) and the ending isosurface (**End**) for the animation. A **Start** of 20 and an **End** of 30 indicates that the animation will begin with the display of an isosurface value of 20, and finish with the display of an isosurface value of 30.

2. Specify the number of steps (**Steps**), i.e., the number of isosurfaces **Airpak** should display between the starting value and the ending value. **Airpak** will interpolate smoothly between the starting value and the ending value that you define, creating the specified number of isosurfaces. Note that the number of steps includes the starting and ending isosurface values.
3. Specify the **Delay (ms)** or the time between each frame in the animation in milliseconds. This will determine the duration of the animation. For example, if you specify 10 isosurfaces next to **Steps**, and then specify 500 as the **Delay (ms)**, **Airpak** will create a five-second animation of the isosurfaces. Note that the delay value is added to however long it takes **Airpak** to create a snapshot for the animation. The time it takes **Airpak** to create a snapshot is dependent on the speed of your system.
4. If you want **Airpak** to play the animation only in the graphics window, and you want the playback to repeat continuously, turn on the **Loop mode** option. To play the animation once through from start to finish, turn off the **Loop mode** option. You can also use the **Loop mode** option if you want to save an animated GIF or FLI file, as described below.
5. To save the animation to a file, select the **Write to file** option and click the **Write** button. This displays the **Save animation** file selection dialog box where you can save animations in MPEG, AVI, animated GIF, FLI, or Flash format. See below for details about saving animations.

Note that when the **Write to file** option is selected, the **Delay (ms)** field becomes the **Frames/s** field. The **Frames/s** field designates the number of animation frames displayed per second.
6. Click **Animate** to start the animation. To stop the animation during playback, click the **Interrupt** button in the upper right hand corner of the **Airpak** interface.

Saving an Animation

You can save an animation to an MPEG, AVI, animated GIF, FLI, or Flash file using the **Save animation** file selection dialog box.

In either format, there are two options for you to consider when saving an animation file: **Print region** and **Scale factor**. These options are available when you click the **Options...** button in the **Save animation** file selection dialog box.

Print region allows you to define the region of the graphics window that should be written to the file. See Section 3.5.2 for details about specifying the **Print region**.

The **Scale factor** is the factor by which **Airpak** will scale the image of the graphics window when it creates the file. If you specify a **Scale factor** of less than 1 for any of the files described below, you may want to thicken the outlines of the room and the objects in the graphics window; otherwise these outlines might not appear in the animation file. You can use the **Graphical styles** panel (described in Section 6.3.6) to thicken the lines.

Saving an MPEG File

To save the animation as an MPEG file, select **MPEG files (*.mpg)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. Note that you must specify the **Scale factor** for the MPEG file. If the MPEG file is to be viewed on a Windows machine, a **Scale factor** of 0.3 is recommended.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single MPEG file that you can view using an MPEG player.

For MPEG files to be played properly, you must use one of the following frame rates: 23.976, 24, 25, 29.97, 30, 50, 59.94, or 60.

Saving an AVI File

To save the animation as an AVI file, select **AVI files (*.avi)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single AVI file. You can then import the AVI file into Microsoft PowerPoint and use the animation in a presentation.

Saving an Animated GIF File

To save the animation as an animated GIF file, select **GIF files (*.gif)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animated GIF file, as described above. If you want the playback of the animation in the GIF file to repeat continuously, select the **Loop mode** option.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single animated GIF file that you can view using a web browser. You can also import the animated GIF file into Microsoft PowerPoint and use the animation in a presentation.

Saving an Autodesk FLI File

To save the animation as an FLI file (an Autodesk animation file format), select **FLI files (*.fli)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above. If you want the playback of the animation in the FLI file to repeat continuously, select the **Loop mode** option.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single FLI file (e.g., **movie.fli**) that

you can view using a web browser if you have an FLI viewer plug-in. It will also create an HTML file (e.g., `movie.html`) that can be viewed using a web browser. **Airpak** also creates Java applets that allow you to view the HTML file as an animation.

When you view the HTML file in a web browser, the following commands are available:

- To continue the animation in the chosen direction (forward or backward), use the `<Space>` bar on your keyboard.
- To play the animation forward, use the `.` key.
- To play the animation backward, use the `,` key.
- To reverse the direction of the animation during playback, use the `r` key.
- To speed up the animation, use the `+` key.
- To slow down the animation, use the `-` key.
- To return to the original animation speed, use the `0` (zero) key.

Note that **Airpak** also produces two additional files when it creates the FLI file: `fli-play.class` and `flickframe.class`. All four files (e.g., `movie.fli`, `movie.html`, `fli-play.class`, and `flickframe.class`) must be present for the animation in the HTML file to be played back.

Saving a Flash File

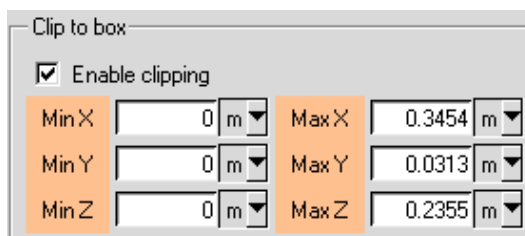
To save the animation as a Flash file, select **Flash files (*.swf)** in the **Files of type** drop-down list. Specify a file name in the **File name** text field and click **Save**. You can specify a **Scale factor** for the animation, as described above.

Airpak will play the animation in the graphics window and save each frame to a separate scratch file. It will then combine all the files into a single Flash file. You can then import the Flash file into Microsoft **PowerPoint** and use the animation in a presentation.

Clipping an Isosurface

If you have created an isosurface, but you do not want to use the whole isosurface to display data, you can clip it to create a new isosurface that spans a specified subrange. The clipped isosurface consists of those points on the selected isosurface where the x , y , and z values are within the specified range.

To clip an existing isosurface, turn on the **Enable clipping** option in the **Clip to** box section of the **Isosurface** panel (Figure 28.2.8).



Enter values for Min X, Min Y, Min Z, Max X, Max Y, and Max Z, and click **Update**.


These values can also be automatically determined by aligning the appropriate edges of the clipped isosurface to the existing edges of objects or bounding boxes of assemblies. The procedure is similar to that described in Section 7.3.8.

For example, to set the Min X value, you would first click Min X displayed in orange in the **Isosurface** panel, then click the edge in the graphics window that you want to align with the Min X edge of the clipped isosurface box.

Just as with other postprocessing objects, data associated with the clipped isosurface object (e.g., specified variable as well as global and object-specific minimum and maximum values) are displayed in the **Edit** window (Figure 28.2.2), and you can save them in a file by using the **Save** button.

Deactivating or Deleting an Isosurface

If you want to temporarily remove the isosurface from the display, you can turn off the **Active** option at the top of the **Isosurface** panel. This has the same effect as turning off the **Active** option in the postprocessing object context menu, described in Section 28.2.3.

If you want to permanently delete the isosurface from **Airpak**, click the **Delete** button at the bottom of the **Plane cut** panel. This has the same effect as the  button, described in Section 28.2.3.

28.2.7 Displaying Results at a Point

A point is defined as a geometric location in the model and is specified either as a three-dimensional coordinate or as a mesh node number. After a point has been created, you can move it through the model interactively (as described later in this section).

As the point is dragged through the model, its current location will be displayed and continuously updated on the postprocessing objects **Edit** panel (described in Section 28.2.3). The value of the velocity field and the variable for which the point is defined are also displayed, thereby allowing you to interactively acquire information about the solution. If you select the option to leave a trail behind the point, its path through the model will be clearly marked by a continuous curve, as shown in Figure 28.2.9. If the option to

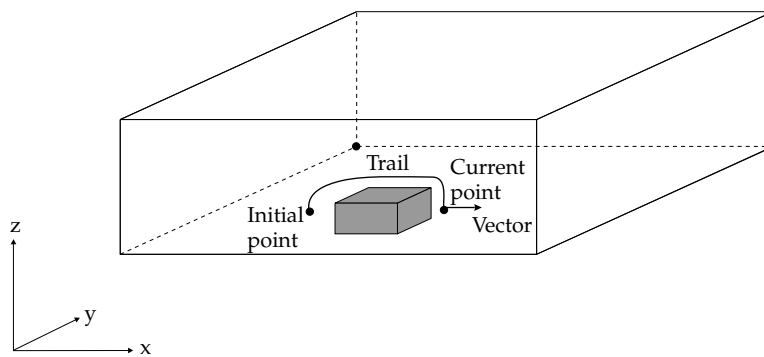


Figure 28.2.9: Point Object

display the vector is selected, the vector associated with the selected vector variable at the point's current location will also be shown.


If the option to display particles is selected, particle traces starting from the point will be displayed.

A point object can be used to probe the model, either to determine the value of a solution variable at a specific point (e.g., the location of a thermocouple in a physical model) or as a starting point for a broader analysis.



When using a point object, you may find it necessary to re-orient the model (e.g., along the coordinate axes) in order to avoid visual confusion regarding the exact location of the point relative to the rest of the model.

Defining a Point

To define a point, you will use the **Point** panel (Figure 28.2.10). To open the panel, click on the  button in the **Postprocessing** toolbar or select **Point** in the **Post** menu.

Post→Point

The procedure for defining a point is as follows:

1. In the **Name** field, enter a new name for the point. The default name is **point.n**, where **n** is a sequential number distinct for each point created. You can keep the default name if you want, but a customized name is generally more useful.
2. Specify the solution variable associated with the point. Select the desired variable (e.g., **Temperature**) from the **Variable** drop-down list. The available variables are described in Chapter 30.

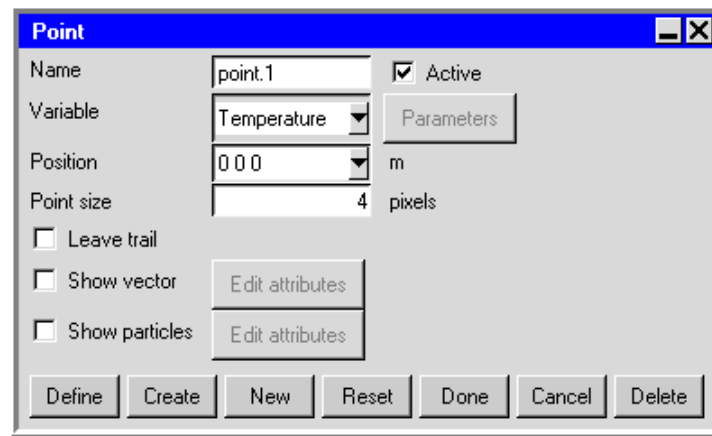


Figure 28.2.10: The Point Panel

3. Enter the coordinates of the point (separated by spaces) next to **Position**.
4. (optional) Specify the size of the point (in pixel units) in the **Point size** field. The default size is 4; if the point is too small for you to see easily, you may want to increase the **Point size**.
5. (optional) If you want to mark the point path by a visible trail as you move it from its initial location through the model (as shown in Figure 28.2.9), turn on the **Leave trail** option.
6. (optional) If you want to show the magnitude and direction of the velocity at the current location of the point (as shown in Figure 28.2.9), turn on the **Show vector** option. To modify the default vector definition, click **Edit attributes**. See Section 28.2.9 for details.
7. (optional) If you want to display particle traces starting from the current location of the point, turn on the **Show particles** option. To modify the default particle-trace definition, click **Edit attributes**. See Section 28.2.10 for details.
8. Click **Define** to define the point according to the specifications in the panel, without loading the data. If the data had already been loaded, this button will be called the **Display** button. The **Define** (or **Display**) button will change to the **Update** button, which you can use to update the point if you make further changes to the specifications in the panel. To create and display the object face by loading the data, click the **Create** button.

Moving a Point Interactively

As mentioned earlier, you can move a point through the model interactively. Hold down the <Shift> key, press and hold down any one of the mouse buttons, and drag the

point through the model in the graphics display. As the point is dragged through the model, its current location will be displayed and continuously updated in the **Edit** window (described in Section 28.2.3).

Modifying an Existing Point


If you want to modify an existing point, make the desired changes to the entries in the **Point** panel, and then click **Update**. If you start to make changes but then wish to return to the original settings in the panel, click **Reset**. **Airpak** will update the panel with the settings that were last saved (i.e., the settings that were present when you last clicked **Create**, **Update**, or **Done**).

Creating Another Point


If you are already in the **Point** panel and you want to create a new point, click **New** at the bottom of the panel. **Airpak** will increment the number on the default **Name** and clear the previous settings so that you can start defining a new point.

Deactivating or Deleting a Point

If you want to temporarily remove the point from the display, you can turn off the **Active** option at the top of the **Point** panel. This has the same effect as turning off the **Active** option in the postprocessing objects context menu, described in Section 28.2.3.

If you want to permanently delete the point from **Airpak**, click the **Delete** button at the bottom of the **Plane cut** panel. This has the same effect as the  button, described in Section 28.2.3.

Displaying the Value of a Variable at a Point

To display the value of a variable at a point on any postprocessing object that is displayed in the graphics window, select **Surface probe** in the **Post** menu or click on the  button.

Post→Surface probe

Click on the postprocessing object in the graphics window using the left mouse button. **Airpak** will display the value of the variable at the point you selected in the graphics window. Additionally, the value of the variable and the coordinate location of the point will be printed in the **Message** window. If you cannot see the value in the graphics window, press the **F9** key on the keyboard to switch the mode of the mouse to the graphics manipulation functions described in Section 2.2.4 and then rotate the model in the usual way. Use the **F9** key to return to the **Surface probe** function.

Displaying Minimum and Maximum Point Values

If you would like Airpak to report the location of the minimum and maximum values of an available postprocessing variable, select Min/max locations in the **Post** menu to open the Min/max locations panel (Figure 28.2.11).

Post → Min/max locations

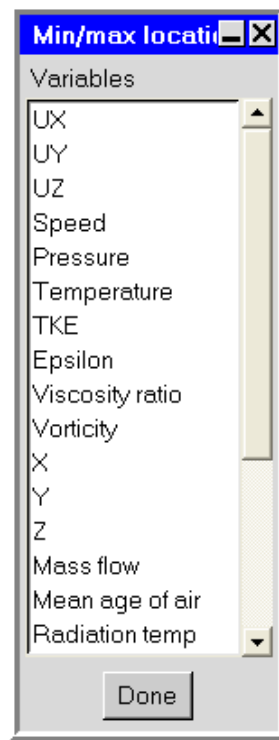


Figure 28.2.11: The Min/max locations Panel

In the Min/max locations panel, click on the name of the variable for which you want Airpak to display the locations of the minimum and maximum values. Airpak will display the points in the graphics window along with their coordinate locations and their values. The values and coordinate locations of the points will also be printed in the Message window. The variables available in the Min/max locations panel are described in Chapter 30.

28.2.8 Contour Attributes

Contours are representations of the variation of a specified variable drawn as lines or solid bands. An individual contour follows a single value of a variable and can curve around or through objects. Contour plots are used to examine how a variable changes locally or throughout the model, and are often useful for locating severe gradients and conditions (e.g., hot spots on the surfaces of objects).

Line contours (e.g., Figure 28.2.12) resemble topological maps. Solid contours can be specified as either smooth or bands. For smooth contours the colors change in continuous gradations from one boundary of the contour to the other. For banded contours, the colors change abruptly at the contour boundaries.

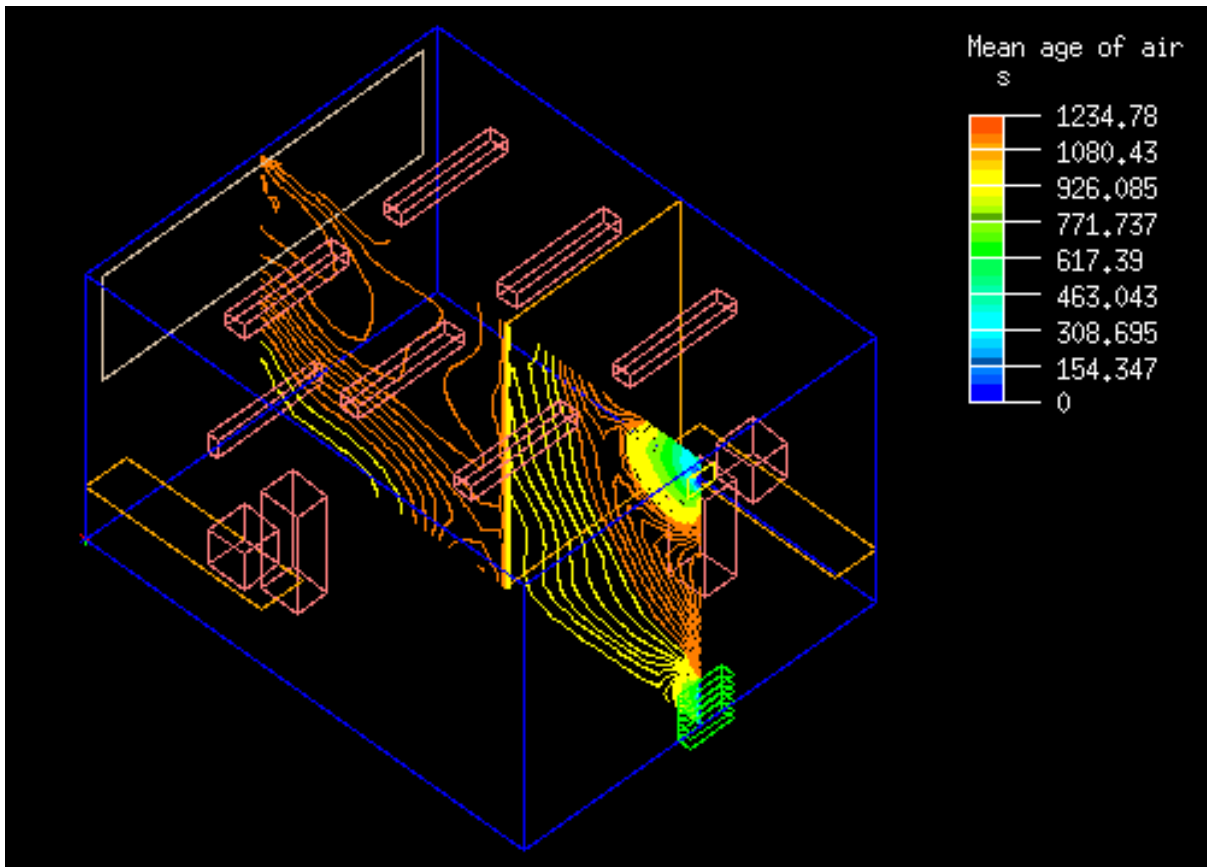


Figure 28.2.12: Contour Plot

Overview of Defining a Contour Plot

If you have selected **Show contours** as the type of display to be shown on an object face, plane cut, or isosurface (in the **Object face**, **Plane cut**, or **Isosurface** panel), you can click the associated **Parameters** button to modify the definition of the contour plot. Figure 28.2.13 shows an example of the **Object face contours** panel.

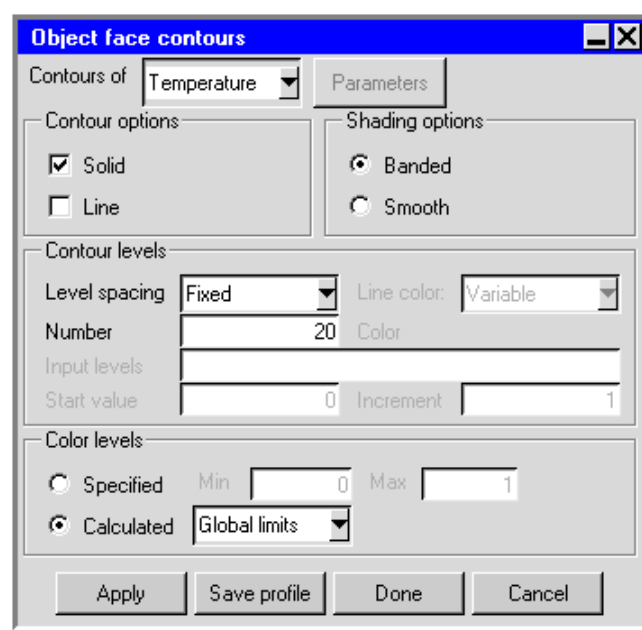


Figure 28.2.13: The Object face contours Panel

In the **Object face contours** panel, the **Plane cut contours** panel, or the **Isosurface contours** panel, you can specify the variable to be plotted, the type of contour plot (line or solid), the color levels for the plot, and several characteristics of the contour lines or solid contours. When you are satisfied with the new settings, click **Apply** to see the results in the graphics window.

Selecting the Variable to be Plotted

To choose the variable to be displayed, select the desired variable (e.g., **Temperature**) from the **Variable** drop-down list. The available variables are described in Chapter 30.

Specifying a Line or Solid Contour Plot

To specify whether you want a line contour plot or a solid contour plot (or both), turn on the **Solid** or **Line** option for **Contour options**. If you select both, the contours will be displayed as solid color bands with lines superimposed on them.

For line contour plots, you will specify how the line color is determined, and for solid contour plots you will specify the shading. For both, you will also specify how the spacing or band width between contours is determined.

Constant or Variable Color for a Line Contour Plot

There are two ways to color the lines on a line contour plot. To vary the color of the lines based on the value of the variable being plotted, select the **Variable** option in the **Line color** drop-down list. To have a single color for all the lines, select the **Fixed** option in the **Line color** drop-down list and then specify a color in the **Color** drop-down color palette.

Shading for a Solid Contour Plot

There are two ways to shade the contours on a solid contour plot. To vary the colors smoothly in continuous gradations from one boundary of the contour to the other, select the **Smooth** option under **Shading options**. To have the colors change abruptly at the contour boundaries, select the **Banded** option.

Specifying the Line Spacing or Band Width

For both line and solid contour plots, there are three ways to specify the line spacing or band width:

- Specify the number of contour levels or bands explicitly:
 1. Select the **Fixed** option in the **Level spacing** drop-down list.
 2. Enter the number of lines or bands in the **Number** field.
 3. Specify how the maximum and minimum levels of the contour range should be determined:
 - To set the color levels explicitly, select **Specified** under **Color levels** and enter the **Min** and **Max** values.
 - To have **Airpak** compute the levels for you, select **Calculated**, and then choose **Global limits**, **This object**, or **Visible on screen** from the drop-down list as the method used to compute the levels. **Global limits** uses the maximum and minimum values of the variable throughout the whole model. **This object** uses the maximum and minimum values of the variable for the current postprocessing object only. **Visible on screen** uses the maximum and minimum values of the variable for all postprocessing objects currently visible in the graphics window.
- Specify the width of the bands or the size of the interval between lines explicitly:
 1. Select the **Interval** option in the **Level spacing** drop-down list.

2. Specify the band width or the size of the interval between lines in the **Increment** field. The units for the **Increment** are the units of the variable being plotted.
3. Specify the minimum value for the contour range in the **Start value** field.

Airpak will generate subsequent intervals by repeatedly adding the **Increment** to the **Start value**.

- Specify the individual values for each line or band as discrete values of the variable:
 1. Select the **Input** option in the **Level spacing** drop-down list.
 2. In the field to the right of **Input levels**, enter the desired values for the contour lines or bands separated by spaces (e.g., 5 10 15).

You can specify as many values as you want with the **Input** option. If you enter more values than the text field can accommodate, the numbers entered first will disappear at the left side of the field, but will not be lost as data. You can drag the middle mouse button across the text field to view the hidden values at the beginning or end of the field, or use the left and right arrows on your keyboard.

Specifying the Color Levels

The color spectrum for a line or solid contour plot is distributed across a range of values for the variable being plotted. There are two ways to specify the minimum and maximum limits for this range:

- To set the color levels explicitly, select **Specified** under **Color levels** and enter the **Min** and **Max** values.
- To have **Airpak** compute the color levels for you, select **Calculated** under **Color levels**, and then select **Global limits**, **This object**, or **Visible on screen** from the drop-down list as the method used to compute the color levels. **Global limits** uses the maximum and minimum values of the variable throughout the whole model. **This object** uses the maximum and minimum values of the variable for the current postprocessing object only. **Visible on screen** uses the maximum and minimum values of the variable for all postprocessing objects currently visible in the graphics window.



Note that the selection of **Specified** or **Calculated** differs from the selection associated with the **Fixed** option for line spacing or band width. The range for the line spacing or band width indicates the minimum and maximum values that should be plotted, and the range for the color indicates the minimum and maximum values associated with the color spectrum. It is possible for the color range to be wider than the range of values being plotted, in which case not all colors in the spectrum will be used.

Saving a Contour Plot

After creating a contour plot, you can save the contour data to a file so that you can read it back into **Airpak** as a profile. This is useful if you have solved a large complicated problem and you want to zoom in to a specific region of your model and solve this region in more detail. You can use the contour data as the boundary conditions for the region you want to study in more detail, and then calculate a solution only for this region.

To save contour data, create a contour plot for one of the boundaries of the region to be studied in detail, and then click **Save profile** in the associated contour panel (e.g., Figure 28.2.13). Enter a name for the contour plot in the resulting **File selection** dialog box and click **Accept** to save the data. You can then create a wall or an opening at the same boundary of the region to be studied in detail. To use the contour data as the boundary condition for the wall or opening, define a spatial profile on the wall or opening using the contour data. (For information on specifying a spatial profile for an opening, see Chapter 11; for a wall, see Chapter 13.)

i Note that the shape and size of the plane containing the contour data you save does not have to be exactly the same shape and size as the opening or wall that you specify. For example, if you save contour data on a plane of size 10 cm × 10 cm, and you use this data to define a boundary profile for an opening, the size of the opening does not have to be 10 cm × 10 cm. If the size of the opening is 5 cm × 5 cm, **Airpak** will use the contour data in this area and ignore the contour data outside this area. If the size of the opening is 15 cm × 15 cm, **Airpak** will use the contour data at the edge of the 10 cm × 10 cm plane in the area of the opening that is larger than 10 cm × 10 cm.

28.2.9 Vector Attributes

An **Airpak** postprocessing vector is an arrow, of which the length and direction represent the magnitude and direction of the velocity at a specific location in the model. In addition, the color of the arrow can represent the value of a scalar solution variable at the vector location. Figure 28.2.14 shows an example vector plot.

Overview of Defining a Vector Plot

If you have selected **Show vectors** as the type of display to be shown on an object face, plane cut, or isosurface (in the **Object face**, **Plane cut**, or **Iso-surface** panel), you can click the associated **Parameters** button to modify the definition of the vector plot in the corresponding vector panel (e.g., Figure 28.2.15). If you are displaying the vector at a point, you can turn on the **Show vector** option and click the **Edit attributes** button in the **Point** panel.

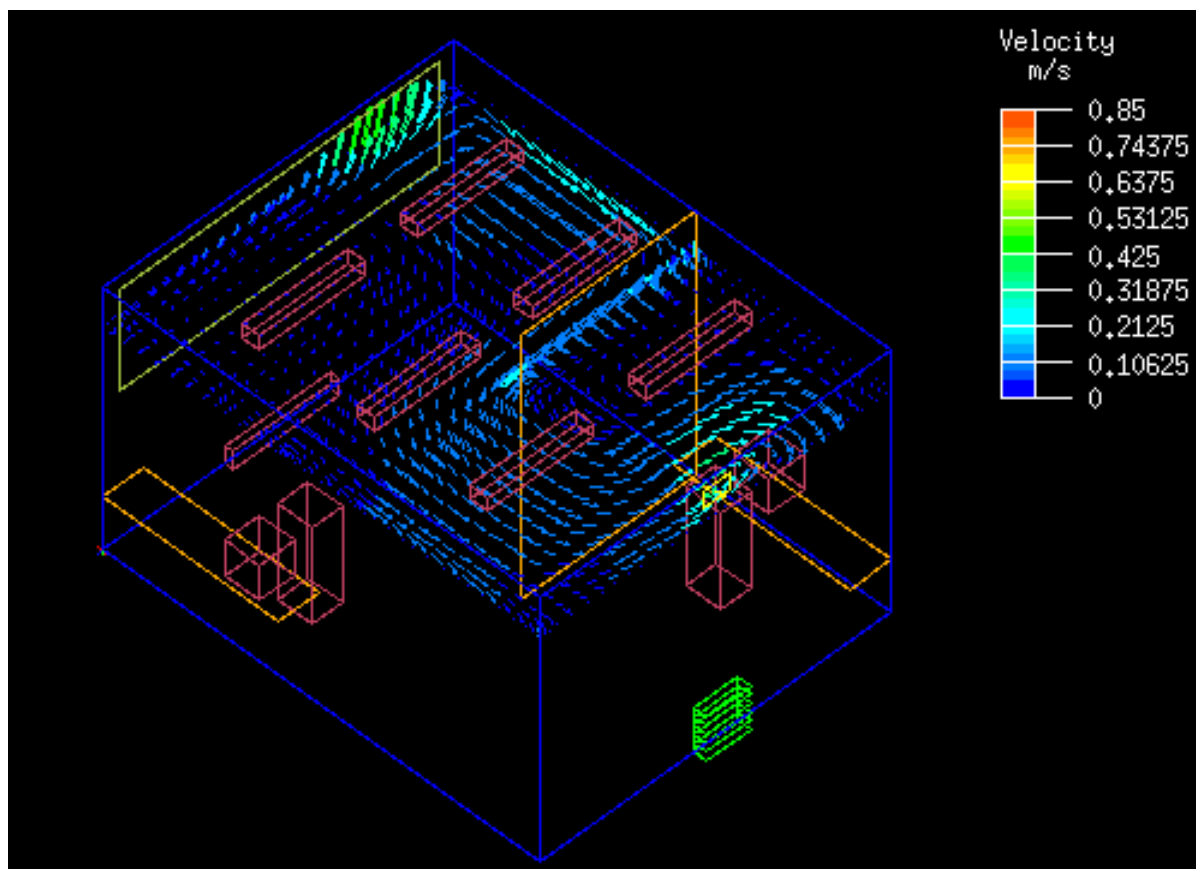


Figure 28.2.14: Vector Plot

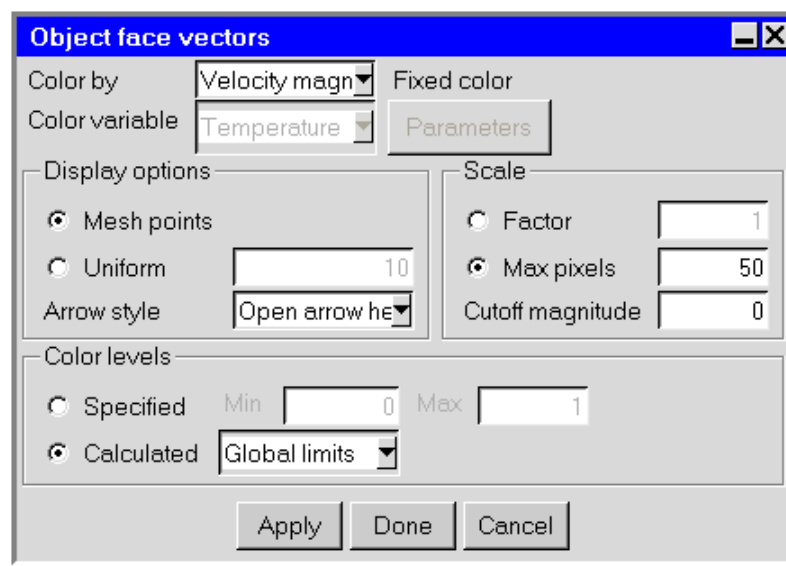


Figure 28.2.15: The Object face vectors Panel

In the Object face vectors panel, Plane cut vectors panel, or Isosurface vectors panel, you can specify the distribution, color, and other characteristics of the vectors, and the color limits for the plot. When you are satisfied with the new settings, click **Apply** to see the results in the graphics window.

Specifying the Distribution of the Vectors

By default, a vector will be displayed at each node point on the surface of the postprocessing object. (This is indicated by the selection of **Mesh points** for the Display options.) In regions where there are more mesh elements (e.g., near modeling objects), there will be more vectors than in regions with fewer mesh elements (e.g., in open spaces). If you prefer to display vectors that are uniformly distributed, select the **Uniform** option for Display options. You can then specify the total number of vectors in the field to the right of Uniform.

Specifying the Color of the Vectors

Vector color can be based on the magnitude of the velocity or on the value of a scalar variable. You can also assign a single color to all vectors.

- To base the vector color on the magnitude of the velocity (the default), select **Velocity magnitude** in the Color by drop-down list.
- To base the color on the value of a scalar variable, select **Scalar variable** in the Color by drop-down list and then select the desired scalar variable (e.g., **Temperature**)

from the **Color variable** drop-down list. The available variables are described in Chapter 30.

- To display single-color vectors, select **Fixed** in the **Color by** drop-down list and then specify a color in the **Fixed color** drop-down color palette.

If you choose a non-fixed vector color (i.e., if you choose **Velocity magnitude** or **Scalar variable**), the color spectrum for the vector plot will be distributed across a range of values for the velocity or specified scalar variable. There are two ways to specify the minimum and maximum limits for this range:

- To set the color levels explicitly, select **Specified** under **Color levels** and enter the **Min** and **Max** values.
- To have **Airpak** compute the color levels for you, select **Calculated** under **Color levels**, and then select **Global limits**, **This object**, or **Visible on screen** from the drop-down list as the method used to compute the color levels. **Global limits** uses the maximum and minimum values of the variable throughout the whole model. **This object** uses the maximum and minimum values of the variable for the current postprocessing object only. **Visible on screen** uses the maximum and minimum values of the variable for all postprocessing objects currently visible in the graphics window.

Scaling the Vectors

There are two ways to scale the size of the vectors:

- Specify a scaling factor for all vectors by selecting the **Factor** option for **Scale** and entering the factor in the text field. The default factor of 1 results in a maximum vector size of approximately 1/2 inch on the screen.
- Specify a scaling factor for the largest vector by selecting the **Max pixels** option for **Scale** and entering the maximum number of pixels in the text field. The size of the largest vector on the screen will be set to the specified number of pixels, and all other vectors will be scaled proportionally.

Limiting the Display of Vectors

If there are large differences in magnitude among displayed vectors, you may want to limit the display of the larger vectors so that you can see the smaller vectors more easily. To limit the display, enter the desired **Cutoff magnitude**. Any vector with a magnitude greater than the specified **Cutoff magnitude** value will not be displayed.

Turning Off Display of Arrowheads on the Vectors

By default, the vectors are displayed with arrowheads. To display the vectors as lines without arrowheads, select the **No arrow heads** option in the **Arrow style** drop-down list.

28.2.10 Particle Trace Attributes

A particle trace represents the path of a hypothetical “massless” particle through the model. The path of the particle is based on the computed flow field. Particle traces provide information similar to that obtained by introducing dye or smoke into the fluid of a real model. They are used primarily to observe flow in the model (e.g., to display the path of an air stream coming from a fan, as shown in Figure 28.2.16).

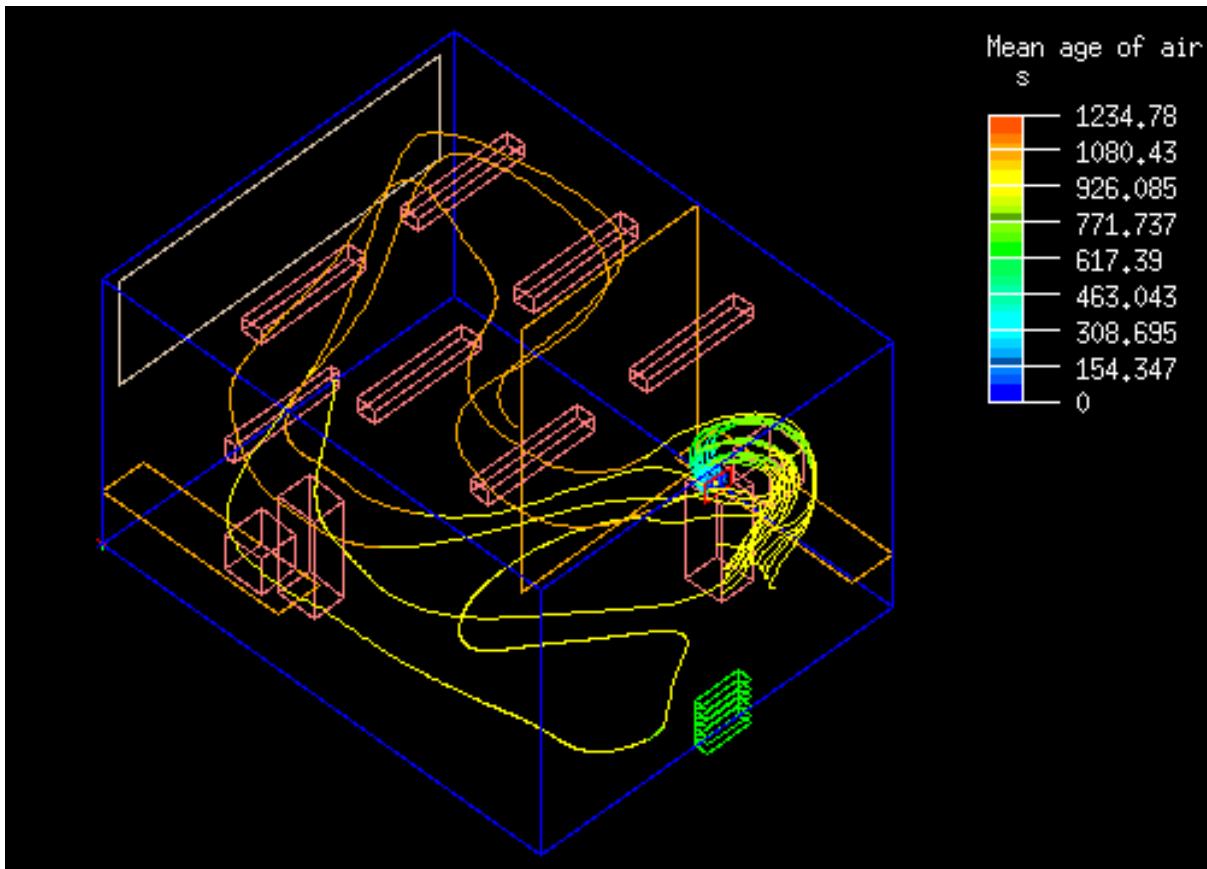


Figure 28.2.16: Particle Trace Plot

Overview of Defining a Particle Trace Plot

If you have selected **Show particle traces** as the type of display to be shown on an object face, plane cut, isosurface, or point (in the **Object face**, **Plane cut**, **Isosurface**, or **Point** panel), you can click the associated **Parameters** button to modify the definition of the particle trace plot in the respective particles panel (e.g., Figure 28.2.17).

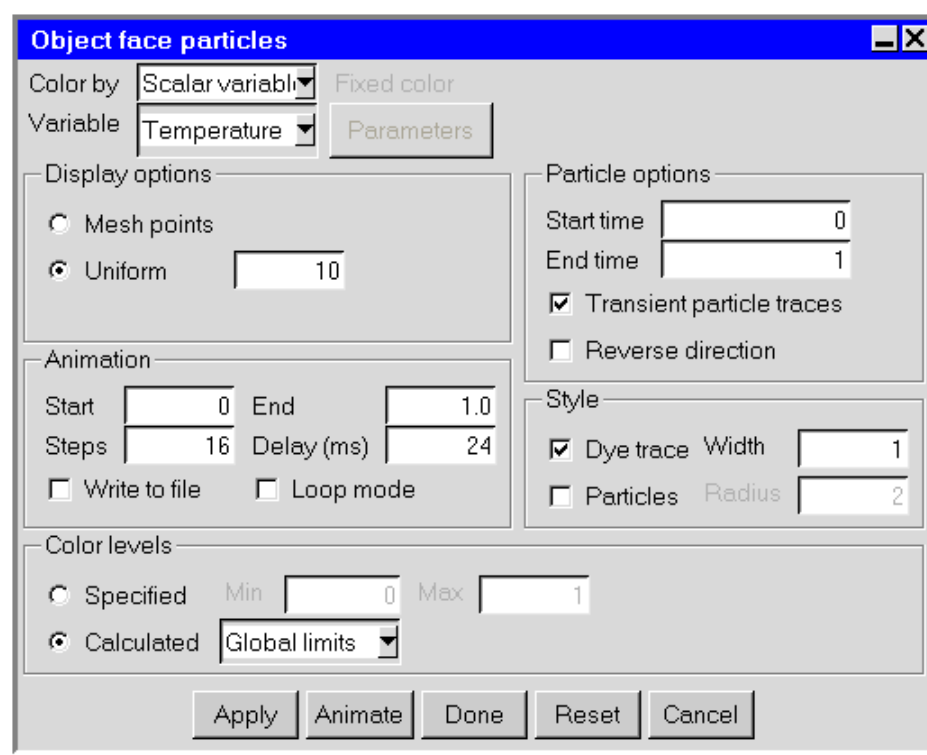


Figure 28.2.17: The Object face particles Panel

In the **Object face particles** panel, **Plane cut particles** panel, **Isosurface particles** panel, or **Point particles** panel, you can specify the color of the particle traces, the frequency of their position calculation, their distribution, and other characteristics. When you are satisfied with the new settings, click **Apply** to see the results in the graphics window.

Specifying Parameters for the Particle Position Calculation

The following parameters control the computation of the particle positions:

- The **Start time** specifies the time at which the particle traces begin.
- The **End time** specifies when the particle traces should end. **Airpak** estimates an end time from the time scale of the flow such that a particle introduced with the

highest velocity occurring in the solution will traverse the length of the enclosure. You can specify a smaller value if you want the particle traces to end before they reach the end of the domain.

- **Transient particle traces** (used for transient simulations only) creates particle traces for a transient solution at a particular time by utilizing the velocity vector field at that time.
- **Reverse direction** can be used to reverse the direction of the particle traces by tracing their path prior to their introduction at an object face, plane cut, isosurface, or point (e.g., to display the path of an air stream coming into and exiting the outlet vent shown in Figure 28.2.16).

Specifying the Distribution of the Particle Traces

By default, particles traces will be uniformly distributed. (This is indicated by the selection of **Uniform** for the **Display options**.) You can specify the total number of particles to be released in the field to the right of **Uniform**. If you prefer to display particles released from each node point on the surface of the postprocessing object, select the **Mesh points** option for **Display options**. In regions where there are more mesh elements, there will be more particles than in regions with fewer mesh elements.

Selecting the Display Style for the Particle Traces

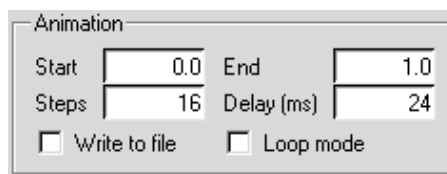
Particles can be represented as discrete points or they can leave dye traces in their wake (or both).

For discrete points, select the **Particles** option under **Style**. You can then specify the **Radius** of the particles in pixel units. Particle traces displayed with this option will consist of individual particles.

For continuous particle traces, select the **Dye trace** option under **Style**. Particle traces displayed with this option will consist of a line. The width of the lines can be specified as integers next to **Width**.

Animating the Particle Traces

By default, the complete particle trace will be displayed all at once. If you want to see the particle trace appear gradually, as the particles move through the domain, you can animate them using the **Animation** section of the **Object face particles** panel, **Plane cut particles** panel, **Isosurface particles** panel, or **Point particles** panel.



To create an animation of a particle trace, follow the steps below.

1. Specify the starting animation “frame” (**Start**) and the ending frame (**End**) for the animation. The default value for **End** is computed from the values found in the **Particle options** section of the corresponding particles panel. Changing this default value will change the number of particle steps.
2. Specify the number of **Steps**, i.e., the number of frames **Airpak** should display between the start and end of the animation. **Airpak** will interpolate smoothly between the start and end of the animation, creating the specified number of frames. Note that the number of steps includes the starting and ending frames.
3. Specify the **Delay (ms)** or the time between each frame in the animation in milliseconds. This will determine the duration of the animation. For example, if you specify 10 frames next to **Steps**, and then specify 500 as the **Delay (ms)**, **Airpak** will create a five-second animation of the particle trace. Note that the delay value is added to however long it takes **Airpak** to create a snapshot for the animation. The time it takes **Airpak** to create a snapshot is dependent on the speed of your system.
4. If you want **Airpak** to play the animation only in the graphics window, and you want the playback to repeat continuously, turn on the **Loop mode** option. To play the animation once through from start to finish, turn off the **Loop mode** option. You can also use the **Loop mode** option if you want to save an animated GIF or FLI file, as described in Section 28.2.5.
5. If you want **Airpak** to save the animation to a file, select the **Write to file** option and click the **Write** button. This displays the **Save animation** file selection dialog box where you can save animations in MPEG, AVI, animated GIF, or FLI format. See Section 28.2.5 for details on saving an animation.

Note that when the **Write to file** option is selected, the **Delay (ms)** field becomes the **Frames/s** field. The **Frames/s** field designates the number of animation frames displayed per second.

6. Click **Animate** to start the animation. To stop the animation during playback, click on the **Interrupt** button in the upper right hand corner of the **Airpak** interface.

Specifying the Color of the Particle Traces

Particle trace color can be based on the value of a scalar variable, or all particle traces can be the same color.

- To base the color on the value of a scalar variable, select **Scalar variable** in the **Color by** drop-down list and then select the desired scalar variable (e.g., **Temperature**) from the **Variable** drop-down list. The available variables are described in Chapter 30.
- To display single-color particle traces, select **Fixed** in the **Color by** drop-down list and specify a color in the **Fixed color** drop-down color palette.

If you choose a non-fixed particle color (i.e., if you choose **Scalar variable** or **Time**), the color spectrum for the particle trace plot will be distributed across a range of values for the specified variable. There are two ways to specify the minimum and maximum limits for this range:

- To set the color levels explicitly, select **Specified** under **Color levels** and enter the **Min** and **Max** values.
- To have **Airpak** compute the color levels for you, select **Calculated** under **Color levels**, and then select **Global limits**, **This object**, or **Visible on screen** from the drop-down list as the method used to compute the color levels. **Global limits** uses the maximum and minimum values of the variable throughout the whole model. **This object** uses the maximum and minimum values of the variable for the current postprocessing object only. **Visible on screen** uses the maximum and minimum values of the variable for all postprocessing objects currently visible in the graphics window.

28.3 XY Plots

Airpak allows you to create four types of XY plots: convergence, history, variation, and trials. The first two are described in Sections 27.8.1 and 20.3.5, respectively, while the variation plot and the trials plot are described here.

28.3.1 Variation Plots

A variation plot allows you to examine a variable along any line in 3D space through the model. Figure 28.3.1 shows an example. The distance along the line is plotted on the x axis and the corresponding value of the variable is plotted along the y axis.

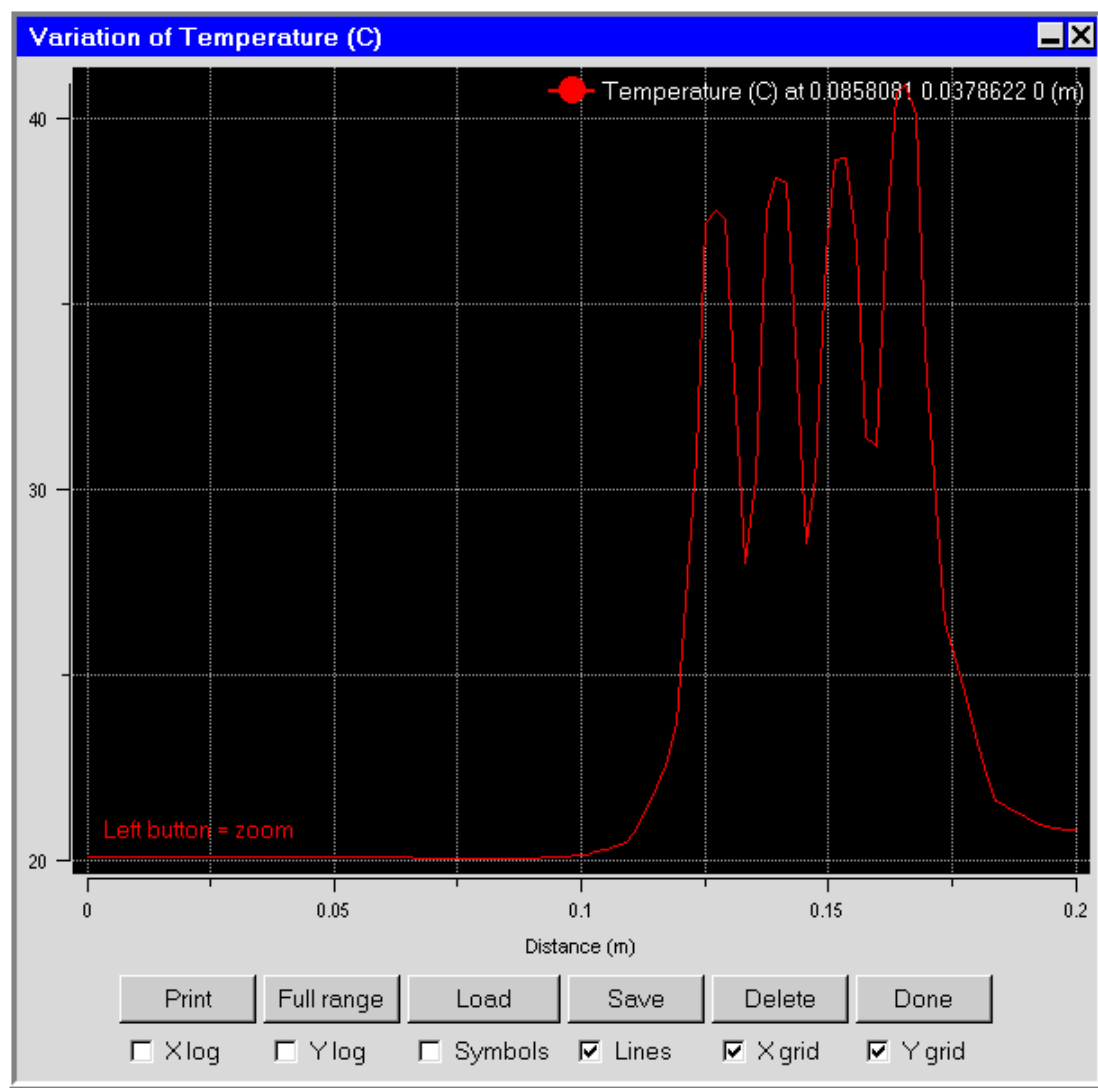



Figure 28.3.1: Variation Plot

Generating a Variation Plot

To generate a variation plot, you will use the **Variation plot** panel (Figure 28.3.2). To open the panel, click on the  button in the Postprocessing toolbar or select **Variation plot** in the Post menu.

Post → Variation plot

The procedure for defining a variation plot is as follows:

1. Select the variable to be plotted (e.g., **Temperature**) from the **Variable** drop-down list. The available variables are described in Chapter 30.

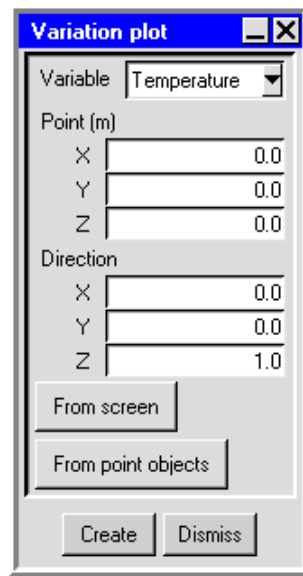


Figure 28.3.2: The Variation plot Panel

2. Specify the initial point of the line (X , Y , Z) and the vector (X , Y , Z) that defines the direction of the line. There are three methods available for defining the line:

- Enter the **Point** and **Direction** explicitly in the panel.
- Use your mouse to define the line in the graphics window:
 - (a) Use the **Orient** menu or the **Orientation commands** toolbar to specify the desired orientation. Choose the orientation such that the plane of the display screen is perpendicular to the desired line. For example, if you want to display results on a line in the z direction, choose **Home** as the orientation so that the display-screen plane is the x - y plane.
 - (b) Click **From screen** in the **Variation plot** panel.
 - (c) Click your left mouse button in the graphics window to indicate a point on the desired line. The resulting line will be normal to the plane of the graphics window and pass through the selected point.

Airpak will update the **Point** and **Direction** to reflect the line definition.

- Define the line from two existing point objects:
 - (a) Click **From point objects** in the **Variation plot** panel.
 - (b) In the **Model manager** window, select two point objects from the **Postprocessing** node. If there are only two postprocessing point objects to choose from, Airpak will automatically use them. If there are more than two point objects, Airpak will open a **Selection** panel that will prompt you to choose

two postprocessing point objects. The first point you select defines the initial point, and the direction is defined as the vector from the first point to the second point.

Airpak will update the **Point** and **Direction** to reflect the line definition.

3. Click **Create** to display the plot.

Modifying the Range of the Variation Plot

Once you have generated the variation plot, you can easily zoom in on a specific portion of the plot. To zoom on an area, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box window to the desired size, and then release the mouse button. The selected area will then fill the plot window, with appropriate changes to the axes. After you have zoomed into an area, click the **Full range** button to restore the graph to its original axes and scale.

Modifying the Appearance of the Plot

Several options for controlling the appearance of the plot are available at the bottom of the plot window:

X log converts the horizontal axis to a logarithmic scale.

Y log converts the vertical axis to a logarithmic scale.

Symbols displays data points on the plotted line.

Lines displays the plotted line.

X grid displays the vertical grid lines on the plot.

Y grid displays the horizontal grid lines on the plot.

Printing the Variation Plot

If you want to print an image of the variation plot, click the **Print** button at the bottom of the variation plot window. See Section 3.5 for details on saving an image.

Saving and Reloading Variation Plot Data

After generating a variation plot, you may want to save the data curve to a file so that you can reuse it the next time you view this model in **Airpak**. To save variation plot data to a file, click the **Save** button at the bottom of the variation plot window. Enter a file name in the resulting **Save curve** dialog box, and click **Accept** to save the file.

To read the data curve back into **Airpak**, click the **Load** button at the bottom of the variation plot window. Enter the appropriate file name in the resulting **Load curve** dialog box, and click **Accept** to load the data.

28.3.2 Trials Plots

A trials plot allows you to compare variables at a particular point in 3D space between different solutions for the current model. In the example shown in Figure 28.3.3, the trial solution is plotted on the x axis and the corresponding value of the variable (pressure) is plotted along the y axis. You may, however, select any variable to be plotted on the x axis against the y axis variable, in which case the plot points will represent one variable as a function of the other at the designated point in the model domain for the specified trials.

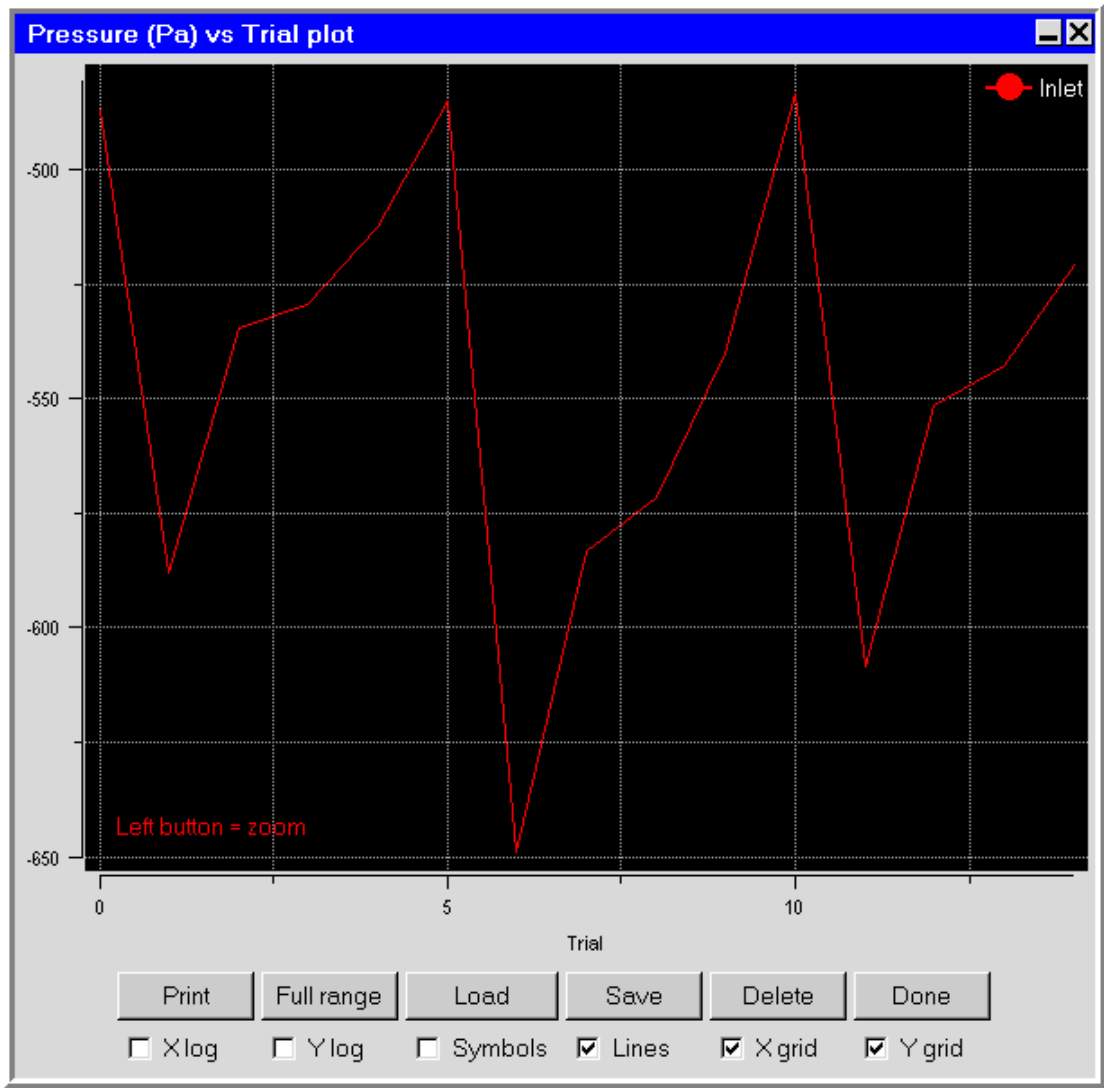



Figure 28.3.3: Trials Plot

Generating a Trials Plot

To generate a trials plot, you will use the Trials plot panel (Figure 28.3.4). To open the panel, click on the  button in the Postprocessing toolbar or select Trials plot in the Post menu.

Post → Trials plot

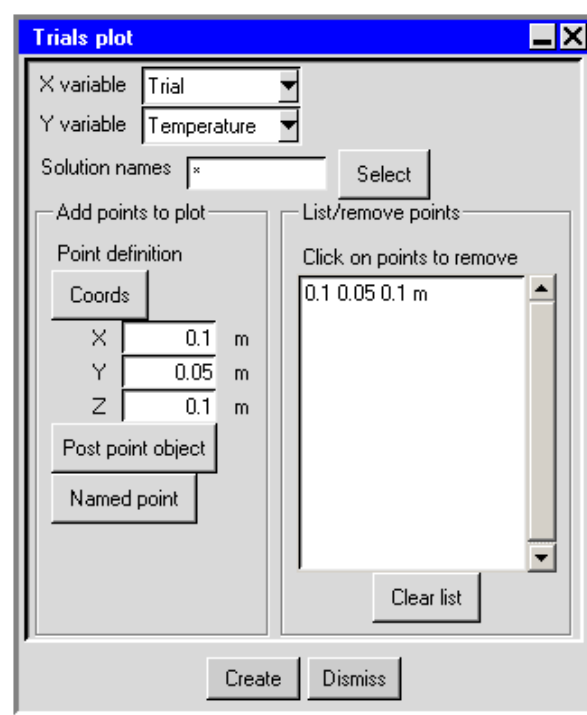


Figure 28.3.4: The Trials plot Panel

The procedure for defining a trials plot is as follows:

1. Select the variables to be plotted (e.g., **Temperature**) from the **X Variable** and **Y Variable** drop-down lists. If you select **Trial** for either the x or y axis, the name of the solutions specified below will appear as the axis label. The other available variables are described in Chapter 30.
2. Specify a set of solutions to compare using the **Solution names** field. You can type a solution name that contains an asterisk or a question mark in place of characters or a character, respectively. For example, typing **rad*** compares solutions **radiation00**, **radiation01**, and **radiation02**, but not **no-rad00**. Typing **rad?** compares solutions **rad1**, **rad2**, and **rad3**, but not **rad10**, or **rad100**. Use the default ***** to specify all solutions for the current model.

3. Under **Add points to plot**, specify the location of the point of interest for the history plot. There are three options:

Coords allows you to specify the coordinates of a point. Enter the **X**, **Y**, and **Z** coordinates and click on the **Coords** button. The point will appear in the **List/remove points** list.

Post point object allows you to select an existing point object from the **Selection** panel (Figure 28.3.5).

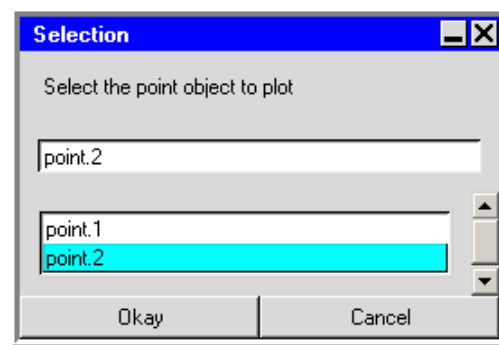


Figure 28.3.5: The Selection Panel

In the **Selection** panel, select the point in the list and click **Okay**. The point will appear in the **List/remove points** list in the **History plot** panel. See Section 28.2.7 for details on creating a point object.

Named point allows you to select an existing named point from the point drop-down list or create a new named point. See Section 20.3.5 for details about creating named points.

4. Click **Create** to display the XY trials plot of the selected variable(s) at the point(s) specified. Click **Cancel** to close the **Trials plot** panel without creating a trials plot.

Modifying the Range of the Trials Plot

Once you have generated the trials plot, you can easily zoom in on a specific portion of the plot. To zoom on an area, position the mouse pointer at a corner of the area to be zoomed, hold down the left mouse button and drag open a selection box window to the desired size, and then release the mouse button. The selected area will then fill the plot window, with appropriate changes to the axes. After you have zoomed into an area, click the **Full range** button to restore the graph to its original axes and scale.

Modifying the Appearance of the Plot

Several options for controlling the appearance of the plot are available at the bottom of the plot window:

X log converts the horizontal axis to a logarithmic scale.

Y log converts the vertical axis to a logarithmic scale.

Symbols displays data points on the plotted line.

Lines displays the plotted line.

X grid displays the vertical grid lines on the plot.

Y grid displays the horizontal grid lines on the plot.

Printing the Trials Plot


If you want to print an image of the trials plot, click the **Print** button at the bottom of the trials plot window. See Section 3.5 for details on saving an image.

Saving and Reloading Trials Plot Data

After generating a trials plot, you may want to save the data curve to a file so that you can reuse it the next time you view this model in **Airpak**. To save trials plot data to a file, click the **Save** button at the bottom of the trials plot window. Enter a file name in the resulting **Save curve** dialog box, and click **Accept** to save the file.

To read the data curve back into **Airpak**, click the **Load** button at the bottom of the trials plot window. Enter the appropriate file name in the resulting **Load curve** dialog box, and click **Accept** to load the data.

28.4 Selecting a Solution Set to be Examined

If there is more than one solution set available for your model, you can specify which solution version you want to examine. To open the **Version selection** panel (Figure 28.4.1), click on the  button in the **Postprocessing** toolbar or select **Load solution ID** in the **Post** menu.

Post→**Load solution ID**

Select the desired solution set from the list at the bottom of the panel, and click **Okay** to load it. The new solution will be applied automatically to the model in the graphics window.

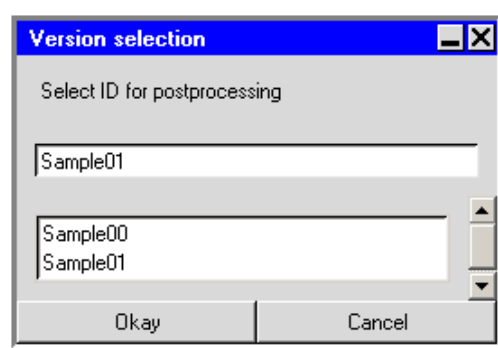


Figure 28.4.1: The Version selection Panel

28.5 Zoom-In Modeling

Airpak allows you to model sub-systems with given results from a system model using a process known as zoom-in or cascade modeling. This process creates a zoom-in model with the appropriate profiles applied automatically to the boundaries of the sub-system.

To create a zoom-in model, you will use the **Zoom-in modeling** panel (Figure 28.5.1). To open the panel, select **Create zoom-in model** in the **Post** menu.

Post → Create zoom-in model

The procedure for creating a zoom-in model is as follows:

1. Specify a name for the zoom-in model project in the **Zoom-in project** text entry box. You can also click **Browse** next to the text field and select the desired project name in the resulting **File selection** dialog box. See Section 2.1.8 for more information on the **File selection** dialog box.
2. Specify the solution ID of the results from the previous system model. You can either enter the solution ID in the **Solution ID** text entry box, or select the solution ID using the **Version selection** panel. Click **Select** to open the **Version selection** panel. Note that you do not need to provide a file extension for the solution ID.
3. Specify the coordinates for each of the zoom-in model boundaries (**Min X**, **Max X**, **Min Y**, **Max Y**, **Min Z**, and **Max Z**). There are two methods available for choosing the coordinates:
 - Enter the coordinate values in the appropriate text entry boxes and press the **<Enter>** key on the keyboard.
 - Click **Select** to the right of the appropriate text entry box and then click the left mouse button on the desired point in the graphics window. You may want to orient your view depending upon the coordinate being selected to ensure a more accurate selection.

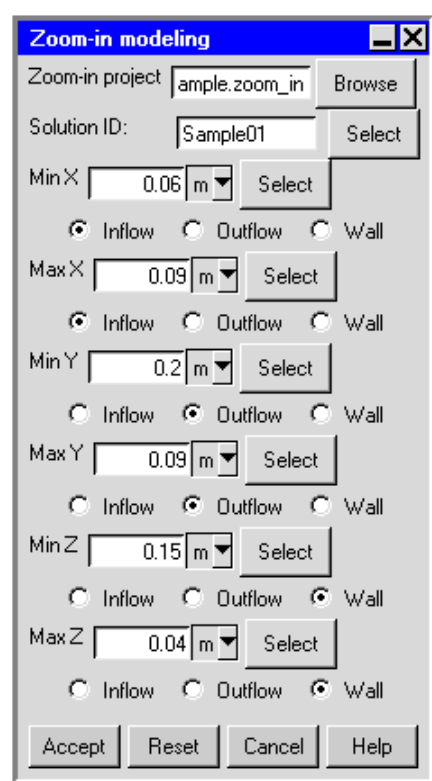


Figure 28.5.1: The Zoom-in modeling Panel

The boundaries of the zoom-in model will be displayed in the graphics window as you update them. Once you have specified the boundaries, you can also change them in the graphics window by first holding down the <Shift> key on the keyboard and then dragging the desired boundary with the right mouse button.

4. Select the type of boundary for each side of the zoom-in model. There are three options:

Inflow boundaries are all open boundaries that have not been chosen as the outflow boundary. For inflow boundaries, a velocity profile is used as a boundary condition.

Outflow boundaries should only be defined as the plane or boundary through which the maximum mass outflow occurs. For an outflow boundary, a static pressure profile is used as a boundary condition.

Wall boundaries are all closed or solid boundaries. For wall boundaries, a fixed temperature profile is used as a boundary condition.

By default, all walls in the zoom-in model will have a fixed temperature profile. While this is suitable for most applications, it is not applicable for the case

where the wall in question is an adiabatic wall in the system-level model. In such a case, you will need to change the boundary condition for the zoom-in model from the automatically-generated temperature profile to an adiabatic boundary condition.

5. Click **Accept** to close the panel and create the zoom-in model. The **Message** window will print the details of the process and alert you when the new project has been written.

Note that it is best to create plane cuts and display the velocity vectors in the system-level model at the locations where you would like the zoom-in boundary to be before writing out the sub-system model to a project file.

i Do not use the zoom-in modeling feature for natural convection problems if any of the boundary conditions in the zoom-in model need walls with specified profiles of heat transfer coefficient.

i In cases where the zoom-in model boundary truncates any solid objects, it is necessary to create additional walls in the zoom-in model to represent the boundaries of the solid objects. The temperature profile at the boundary will have already been written out and can be loaded for the new wall using the **Load** button in the **Curve specification** panel.

Airpak provides tools to create reports of solution results and print them either to a file or to the report window. These reports represent tables of any primary or derived solution variable for all nodal mesh points that lie within a specified region of the model. The reports also include minimum and maximum values of the variable within the region. The region can be the entire model, a zoomed-in portion of the model, a region in the shape of a rectangular prism, the faces of one or more modeling objects, or a point in the model. You can also create a report of the fan operating point for characteristic curve fans, indoor air quality and thermal comfort level quantities, and of the Air Diffusion Performance Index (ADPI) for a particular region in the model.

These features are described in the following sections.

- Section 29.1: Overview: The Report Menu
- Section 29.2: Variables Available for Reporting
- Section 29.3: HTML Reports
- Section 29.4: Reviewing a Solution
- Section 29.5: Summary Reports
- Section 29.6: Point Reports
- Section 29.7: Full Reports
- Section 29.8: Fan Operating Point Report
- Section 29.9: ADPI Report

29.1 Overview: The Report Menu

The Report menu is used to generate reports of results for your Airpak model. When you select **Report** in the Main Menu bar, a set of sub-menus is displayed, as shown in Figure 29.1.1.

The Report menu contains all the functions you need to create reports of results for your simulation. These functions include the following:

- creating an HTML report (Report→HTML report)

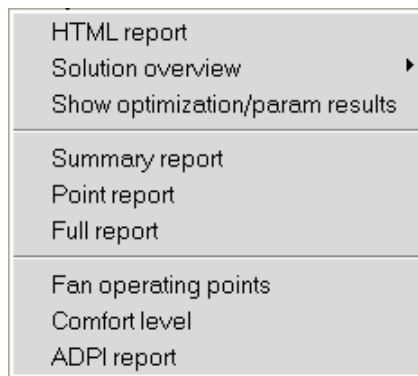


Figure 29.1.1: The Report Menu

- creating a solution overview (Report→Solution overview)
- creating a summary report (Report→Summary report)
- creating a point report (Report→Point report)
- creating a full report (Report→Full report)
- creating a report of the fan operating point for characteristic curve fans (Report→Fan operating points)
- creating an Air Diffusion Performance Index (ADPI) report (Report→ADPI report)

The **Report** menu also contains a function that allows you to view the optimization/parametrization results (Report→Show optimization/param results), described in Chapter 23. You can also create a report of indoor air quality and thermal comfort level quantities (Report→Comfort level). See Section 30.2.8 for details.

29.2 Variables Available for Reporting

Listed below are the variables that are available for reporting and the formats in which each is reported. Note that, for a summary report, the average value of a variable is calculated as the area-based average. For a full report, **Airpak** calculates the average value as the arithmetic mean of the nodal values of the variable.

UX, UY, UZ, Speed, Pressure, Temperature, TKE, Epsilon, Viscosity ratio, X, Y, Z, Mean age of air, Radiation temp, Relative humidity, Angular deviation, PPD, PMV, species (mole), species (mass) are reported in the following format:

1. Number of nodes
2. Minimum, maximum, and average values of the variable
3. Node #, x , y , z , value (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Heat flux is reported in the following format:

1. Total heat flux
2. Surface area
3. Number of nodes
4. Minimum, maximum, and average pointwise values of heat flux
5. Node #, x , y , z , \dot{q} (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Note that the reported pointwise value of heat flux (\dot{q}) represents heat flux normal to the surface of the specified object.

Heat flow, Radiative heat flow are reported in the following format:

1. Total heat flow
2. Surface area
3. Number of nodes
4. Minimum, maximum, and average pointwise values of the variable
5. Node #, x , y , z , value (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Mass flow is reported in the following format:

1. Total mass flow
2. Surface area
3. Number of nodes
4. Minimum, maximum, and average pointwise values of mass flow

5. Node #, x , y , z , value (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Note that the reported pointwise value represents mass flow normal to the surface of the specified object.

Volume flow is reported in the following format:

1. Total volume flow
2. Surface area
3. Number of nodes
4. Minimum, maximum, and average pointwise values of volume flow
5. Node #, x , y , z , value (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Note that the reported pointwise value represents volume flow normal to the surface of the specified object.

Heat tr. coeff is reported in the following format:

1. Average heat transfer coefficient
2. Surface area
3. Number of nodes
4. Minimum, maximum, and average pointwise values of heat transfer coefficient
5. Node #, x , y , z , h (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Note that the heat transfer coefficient (h) is determined relative to a user-defined reference temperature (**Ref temp**).

Velocity is reported in the following format:

1. Number of nodes
2. Minimum, maximum, and average velocity in the x , y , and z directions
3. Node #, x , y , z , u_x , u_y , u_z (displayed if you deselect **Only summary information** in the **Full report** panel described in Section 29.7)

Note that the variables u_x , u_y , and u_z represent velocity in the x , y , and z directions, respectively.

29.3 HTML Reports

You can create a web-based report displaying the results of your **Airpak** simulation using the HTML report panel. To open the HTML report panel (e.g, Figure 29.3.1), select HTML report in the Report menu.

Report—>HTML report

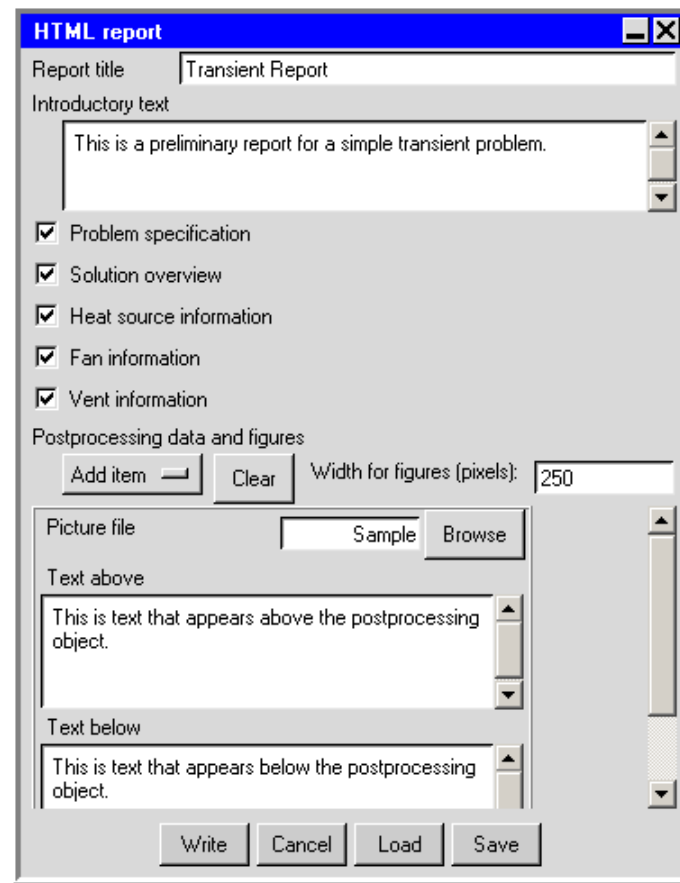


Figure 29.3.1: The HTML report Panel

To create and HTML report, follow the steps below:

1. Specify a title for the report in the **Report title** field.
2. Enter introductory notes in the **Introductory text** field.
3. Specify the level of detail you need to include in the report. You can choose from **Problem specification**, **Solution overview**, **Heat source information**, **Fan information**, and **Vent information**. The HTML report includes all of these options by default.

4. Specify postprocessing objects and data under **Postprocessing data and figures**. Use the **Add item** drop-down list to add postprocessing objects and data to the HTML report. Note that when you use the **Add item** drop-down list to add multiple items to the report, the **Postprocessing data and figures** section of the **HTML report** panel expands and you must use the scroll bars to access the report data fields. There are three categories of data you can include in the HTML report:

Picture (.gif etc.) allows you to include graphic images in your HTML report. To add a picture to the report, use the following procedure:

- (a) Specify a width in pixels for the images using the **Width for figures (pixels)** field.
- (b) Enter a file name for the image file you want to include in the HTML report in the **Picture file** text field, or browse the directory structure for a specific file by clicking **Browse**.
- (c) (optional) To include text above and below the figure, use the **Text above** and **Text below** fields.
- (d) (optional) To keep the image from being included in the generated HTML, turn off the **Active** option.

To remove the image from the **HTML report** panel, click **Delete**.

Postprocessing object allows you to include postprocessing object data in your HTML report. To add a postprocessing object to a report, use the following procedure:

- (a) Select a postprocessing object from your **Airpak** model using the **Post-proc object** drop-down list.
- (b) **Airpak** preloads the **Viewing position** text field with position, rotation, scale, and centering data for the current view. You can re-enter data into the field, or you can adjust the viewing position in the **Airpak** graphics window and click **Current** in the **HTML report** panel.
- (c) If there is more than one solution set available for your model, you will need to specify which solution you want to include in the HTML report. Enter a solution ID in the **Solution ID** text field or click **Select** to open the **Version selection** panel. See [Section 28.4](#) for more information about selecting a solution set.
- (d) (optional) To include text above and below the figure, use the **Text above** and **Text below** fields.
- (e) (optional) To keep the postprocessing object from being included in the generated HTML, turn off the **Active** option.

To remove the postprocessing object from the **HTML report** panel, click **Delete**.

Summary report allows you to include summary report data in your HTML report. To add a summary report to an HTML report, use the following procedure:

- (a) Select the type of summary report (**Fan operating points**, **Vent flow rates**, **Heat source values**, or **User-defined**) to include in the HTML report using the **Summary type** drop-down list.

When the **Summary type** is **User-defined**, the **Summary definition** field becomes active, allowing you to enter the file name of a summary. You can also click **Browse** to navigate through your directory structure to locate the summary file.

- (b) If there is more than one solution set available for your model, you will need to specify which solution you want to include in the HTML report. Enter a solution ID in the **Solution ID** text field or click **Select** to open the **Version selection** panel. See Section 28.4 for more information about selecting a solution set.
- (c) (optional) To include text above and below the figure, use the **Text above** and **Text below** fields.
- (d) (optional) To keep the summary report from being included in the generated HTML, turn off the **Active** option.

To remove the summary report from the **HTML report** panel, click **Delete**. Click **Clear** to remove all postprocessing data and objects from the **HTML report** panel.

To load a report format, click on **Load**. This opens the **File selection** dialog box, in which you can specify the filename and the directory from which the chosen format is to be loaded. See Section 2.1.8 for details on selecting a file.

To save the report format, click on **Save**. This opens the **File selection** dialog box, in which you can specify the filename and the directory to which the chosen format is to be saved. See Section 2.1.8 for details on selecting a file.

5. Click **Write** in the **HTML report** panel to create the report (or click **Cancel** to close the panel without creating the report).

29.4 Reviewing a Solution

You can create a summary for each solution that you generate in Airpak. To create a summary data file for a particular solution, select **Solution overview** and then **Create** in the **Report** menu. This opens a **Version selection** panel (see Figure 29.5.2) where you can select the solution ID to use to create the summary report.

Report→Solution overview→Create

To view summary data stored for a particular solution, select **Solution overview** and then **View** in the **Report** menu. This opens a **File selection** dialog box where you can open a solution overview file (*.overview). Once an overview file is selected, the summary data is displayed in the **Solution panel** (Figure 29.4.1).

Report→Solution overview→View

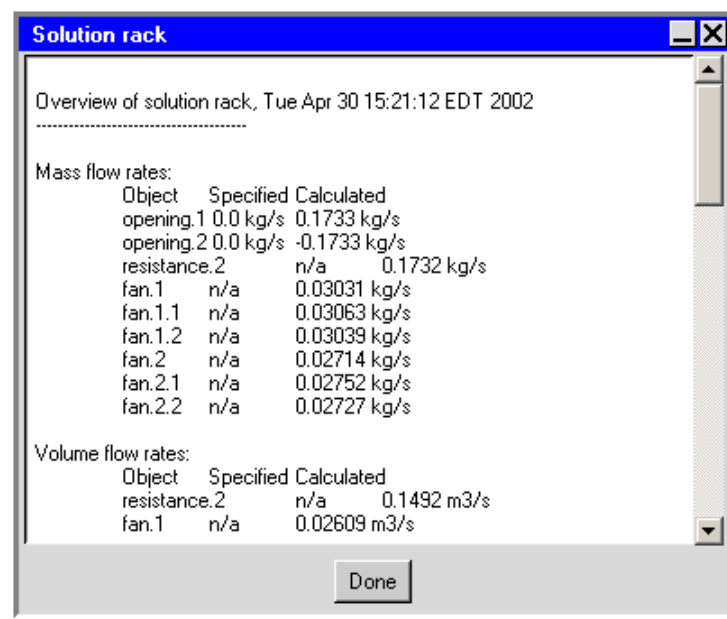



Figure 29.4.1: The Solution Panel

The **Solution** panel contains heat and mass transfer information for the objects in your Airpak model.

29.5 Summary Reports

You can create a summary report for a variable on any or all objects in your Airpak model. You can use the **Define summary report** panel to create a summary report for the results of your Airpak simulation. To open the **Define summary report** panel (Figure 29.5.1), select the **Summary report** option in the **Report** menu or click on the  button in the **Postprocessing** toolbar.

Report—>Summary report

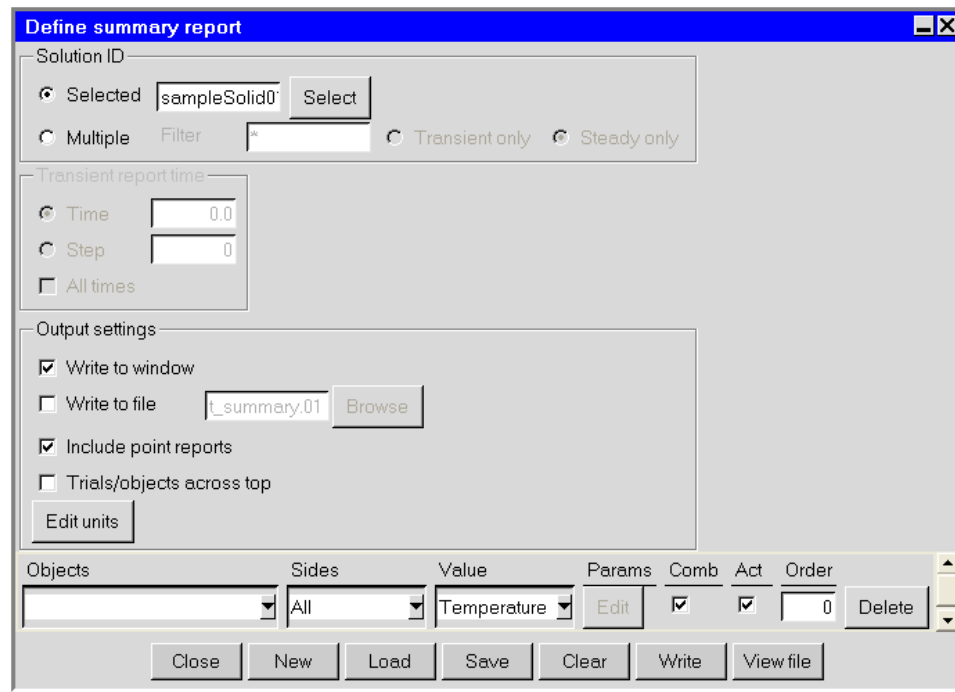


Figure 29.5.1: The Define summary report Panel

To create a summary report, follow the steps below.

1. Specify the **Solution ID** for which you wish to create a report. There are two options:

Selected allows you to either enter the solution ID in the **Solution ID** text entry box or select the solution ID using the **Version selection** panel. Click **Select** to open the **Version selection** panel (Figure 29.5.2).

Select the version you want to use to create the report in the list at the bottom of the **Version selection** panel and click **Okay** to accept the selection.

Multiple allows you to generate a report on multiple solutions by entering a wild-card expression or a text string in the **Filter** text field, and selecting either

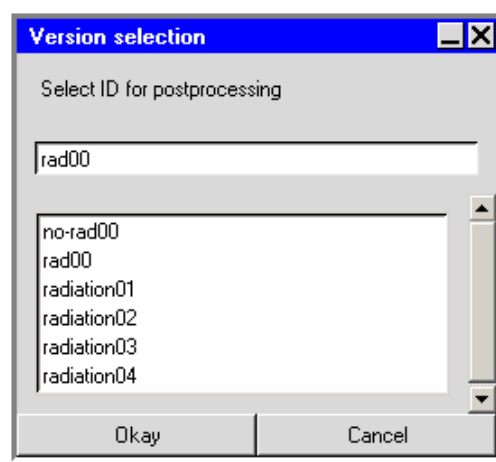


Figure 29.5.2: The Version selection Panel

Transient only or Steady only solutions. By default, **Airpak** generates a report for all steady solutions, denoted using the * in the **Filter** text field.

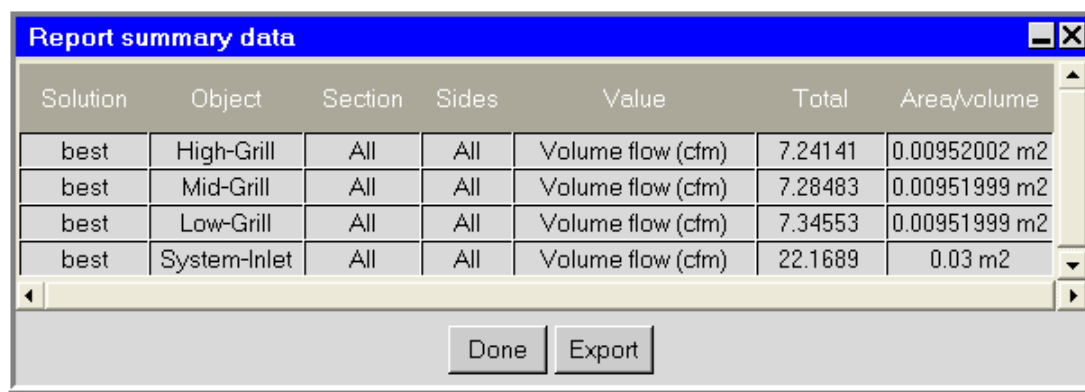
You can type a solution name that contains an asterisk or a question mark in place of characters or a character, respectively. For example, typing **rad*** will allow a report to be written for all solutions whose names start with **rad**; typing **rad?** will allow a report to be written for all solutions whose names consist of the word **rad** plus one character. Any solution whose name matches this text pattern will be analyzed and data written to the summary report when you click **Write** in the **Define summary report** panel.

2. (transient simulations only) Specify the **Transient report time**, which is the time in the transient analysis for which the report is to be written. The time can be specified as a particular **Time**, time **Step**, or **All times**. See Chapter 20 for details on transient simulations.
3. Under **Output settings**, specify where **Airpak** should write the report. There are four options:

Write to window directs the report to the **Report summary data** panel (e.g., Figure 29.5.3).

Write to file directs the report to a file. **Airpak** assigns a default name to the file, but it can be overwritten with a user-specified name. The user-specified filename can be a full pathname to the file or a pathname relative to the directory in which execution of **Airpak** was initiated. You can also click **Browse** to navigate through your directories to locate a file in which to write.

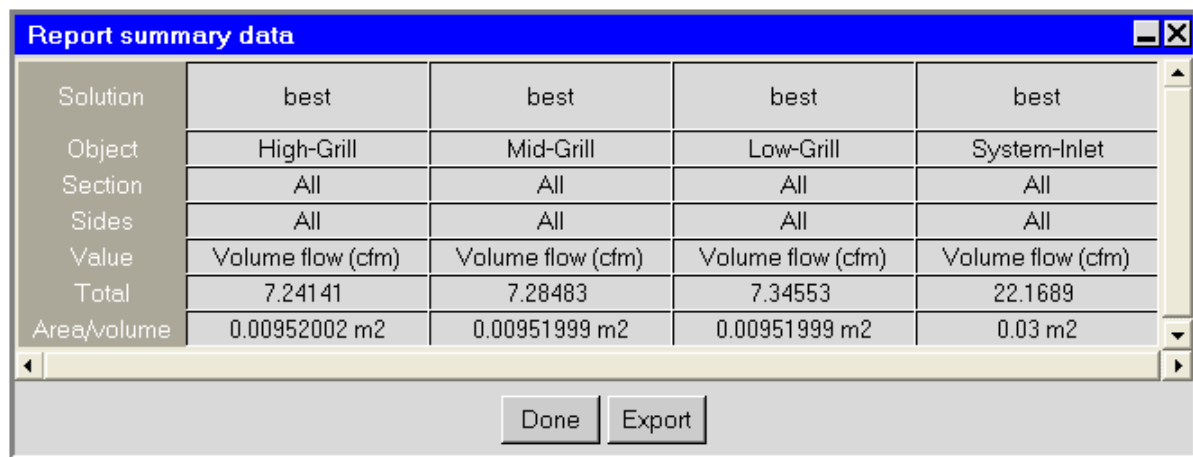
Include point reports includes the point report (if previously defined) in the summary report.



Solution	Object	Section	Sides	Value	Total	Area/volume
best	High-Grill	All	All	Volume flow (cfm)	7.24141	0.00952002 m2
best	Mid-Grill	All	All	Volume flow (cfm)	7.28483	0.00951999 m2
best	Low-Grill	All	All	Volume flow (cfm)	7.34553	0.00951999 m2
best	System-Inlet	All	All	Volume flow (cfm)	22.1689	0.03 m2

Figure 29.5.3: The Report summary data Panel

Trials/objects across top transposes the list of trials and objects (in the case of a single trial) and displays them by column, as shown in Figure 29.5.4.



Solution	best	best	best	best
Object	High-Grill	Mid-Grill	Low-Grill	System-Inlet
Section	All	All	All	All
Sides	All	All	All	All
Value	Volume flow (cfm)	Volume flow (cfm)	Volume flow (cfm)	Volume flow (cfm)
Total	7.24141	7.28483	7.34553	22.1689
Area/volume	0.00952002 m2	0.00951999 m2	0.00951999 m2	0.03 m2

Figure 29.5.4: The Report summary data Panel Displayed by Column

4. Click **Edit units** to specify the units for the report. See Section 5.5 for details on selecting units for reports.
5. Load a saved report or create a new report.
 - To load a previously saved format (e.g., a `.summ` file), click **Load**. Any one of many previously saved formats can be loaded.
 - To define the format for a new report, click **New** and select the object, section, sides, and variable of interest on the object.

- **Objects** specifies the type of modeling object that represents the report region. In the **Objects** drop-down list, you can select different types of objects from those available in the model. you can select an individual object from the drop-down list, or you can select all objects of any one given type. To do this, simply right-click over any object in the object tree of the drop-down list and a pop-up menu will be displayed allowing you to select all objects of that type. Note that groups are also valid object types.
- (recirculation openings only) Specify whether you want to create a report for the supply or extract section of the recirculation opening by selecting **supply** or **extract** in the **Section** drop-down list. Select **All** to create the report for both the supply and the extract sections of the opening.
- **Sides** specifies the side or sides of the object to be included in the report region. The **Sides** options include **All**, **Min X**, **Max X**, **Min Y**, **Max Y**, **Min Z**, **Max Z**, **Top**, **Bottom**, **Sides**, and **Interior**; however, only those options appropriate for the selected object are available for selection. All object sides are active by default.
- To choose the variable to be reported, select the desired variable (e.g., **Temperature**) from the **Value** drop-down list. The available variables are described in Section 29.2 and Chapter 30.



Note that, for a summary report, the average value of a variable is calculated as the area-based average.

If you select **Heat tr. coeff** under **Value**, you will need to specify the reference temperature for the heat transfer coefficient. To specify the reference temperature, click the **Edit** button under **Params** to the right of the **Value** text field and enter the **Ref temp** in the **Heat tr. coeff** panel.

6. If you want to report a combined (single) value for all the sides of an object, turn on the **Comb** option. To report them separately, turn off **Comb**.
7. (optional) If you want to temporarily disable reporting on the specified object, turn off the **Act** option to deactivate the report. If you want to permanently remove the object from the report, click **Delete**.
8. To change the order of items in a report, enter the appropriate values under **Order**.
9. Save the report format by clicking **Save**. This opens the **Report summary data** dialog box, in which you can specify the filename and the directory to which the chosen format is to be saved. See Section 2.1.8 for details on selecting a file.
10. Click **Write** in the **Define summary report** panel to create the report (or click **Close** to close the panel without creating the report).

i To clear the summary report and start over, click **Clear**.

You can also use the **Define summary report** panel to view the results of a previously created report. To view the information in a previously generated report file, click on **View file** in the **Define summary report** panel. Airpak will open the **File selection** dialog box, in which you can select the report you want to view. (See Section 2.1.8 for details on selecting a file.) When you click **Open** in the **File selection** dialog box, Airpak will open a window containing the information in the selected report.

You can also define a summary report before Airpak starts a calculation. Airpak will create the report when the calculation is complete. See Section 27.10 for details.

29.6 Point Reports

You can create a report for a variable at any point in your Airpak model. You can use the **Define point report** panel to define a report for the results of your Airpak simulation at a point in your model. To open the **Define point report** panel (Figure 29.6.1), select **Point report** in the **Report** menu.

Report→Point report

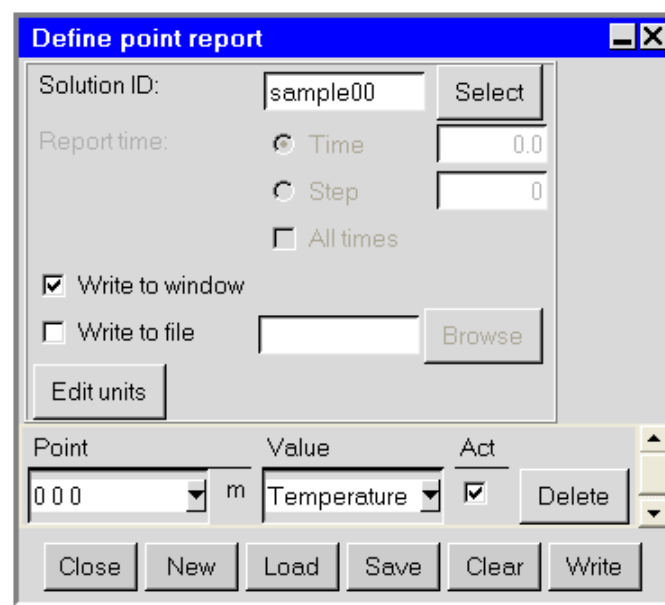


Figure 29.6.1: The Define point report Panel

To create a report at a point, follow the steps below.

1. Specify the **Solution ID** for which you wish to create a report. You can either enter the solution ID in the **Solution ID** text entry box, or select the solution ID using the **Version selection** panel. Click **Select** to open the **Version selection** panel (Figure 29.5.2).

Select the version you want to use to create the report in the list at the bottom of the **Version selection** panel and click **Okay** to accept the selection.

2. (transient simulations only) Specify the **Report time**, which is the time in the transient analysis for which the report is to be written. The time can be specified as a particular **Time**, time **Step**, or **All times**. See Chapter 20 for details on transient simulations.

3. Specify where **Airpak** should write the report. There are two options:

Write to window directs the report to the **Report point data** panel (e.g., Figure 29.6.2).

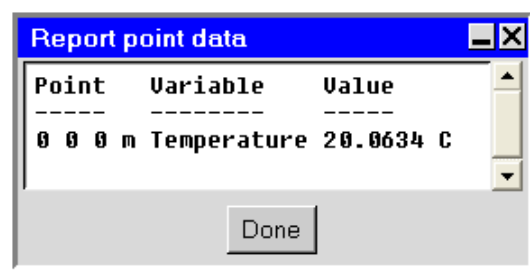


Figure 29.6.2: The Report point data Panel

Write to file directs the report to a file. The filename can be a full pathname to the file or a pathname relative to the directory in which execution of **Airpak** was initiated. You can also click **Browse** to navigate through your directories to locate a file in which to write.

4. Click **Edit units** to specify the units for the report. See Section 5.5 for details on selecting units for reports.
5. Load a saved report or create a new report.
 - To load a previously saved format (e.g., a **.rpt** file), click **Load**. Any one of many previously saved formats can be loaded.
 - To define the format for a new report, click **New**, specify the point of interest, and select the variable of interest.
 - (a) Enter values for the coordinates of the point (separated by spaces) in the **Point** text entry field.

- (b) Choose the variable to be reported by selecting the desired variable (e.g., **Temperature**) from the **Value** drop-down list. The available variables are described in Section 29.2 and Chapter 30.
- 6. (optional) If you want to temporarily disable reporting on the specified point, turn off the **Act** option to deactivate the report. If you want to permanently remove the point from the report, click **Delete**.
- 7. Save the report format by clicking **Save**. This opens the **Report summary data** dialog box, in which you can specify the filename and the directory to which the chosen format is to be saved. See Section 2.1.8 for details on selecting a file.
- 8. Click **Write** in the **Define point report** panel to create the report (or click **Close** to close the panel without creating the report).



To clear the point report and start over, click **Clear**.

29.7 Full Reports

You can use the **Full report** panel to customize the report of your results. To open the **Full report** panel (Figure 29.7.1), select **Full report** in the **Report** menu.

Report → **Full report**

To create a full report, follow the steps below.

1. Specify the name of the solution for which to generate the report next to **Solution ID**.
2. Specify the **Variable** to be reported. **Temperature** is selected by default in the **Variable** list. To change the variable to be reported, select a new variable from the **Variable** drop-down list. The available variables are described in Section 29.2 and Chapter 30.



Note that, for a full report, the average value of a variable is calculated as the arithmetic mean of the nodal values of the variable.

3. Define the region for which the report is to be generated under **Report region**. Three options are available:

Entire room specifies the complete model. The value of the variable is reported at each node point of the mesh.

Full report

Solution ID:

Variable:

Report region:

☒ Entire room

☐ Sub-region

Start xS m End xE m

yS m yE m

zS m zE m

☐ Specific objects

☒ All sides

☐ Sides

☒ Min X ☒ Min Y ☒ Min Z

☒ Max X ☒ Max Y ☒ Max Z

☒ Top ☒ Bottom ☒ Sides

Report time:

☒ Time

☐ Step

☒ Only summary information

☒ Write to window

☒ Write to file

Figure 29.7.1: The Full report Panel

Sub-region specifies a region of the model. If you want only a sub-domain of the model, you must specify the boundary coordinates that describe the sub-domain. The parameters **xS**, **yS**, **zS**, **xE**, **yE**, and **zE** specify the **Start** and **End** points of the rectangular region, with **xS**, **yS**, **zS**, **xE**, **yE**, and **zE** being the boundary coordinates of the subdomain (i.e., $xS \leq xE$, $yS \leq yE$, and $zS \leq zE$).

Specific objects specifies that the report is to be generated for a particular object or collection of objects.

- (a) Specify the modeling object using the **Specific object** drop-down list. Select an object name from the list of objects in the **Airpak** model. The object name is displayed in the text field. (e.g., **object fan.1**). You can specify more than one object by holding down the <Control> key and select another object from the list. Multiple names are listed in the **Object** text field separated by spaces.
- (b) Specify the part of the object where you want the report to be generated. There are two options:

All sides specifies the entire object.

Sides specifies the side or sides of the object to be included in the report region. The **Sides** options include **Min X**, **Max X**, **Min Y**, **Max Y**, **Min Z**, **Max Z**, **Top**, **Bottom**, and **Sides**; however, only those options appropriate for the selected object are available for specification. All object sides are active by default.

4. (transient simulations only) Specify the **Report time**, which is the time in the transient analysis for which the report is to be written. The time can be specified as a particular **Time** or time **Step**. See Chapter 20 for details on transient simulations.
5. Select from the following options available at the bottom of the **Full report** panel.

Only summary information includes only summary information in the report.

Write to window directs the report to the **Report** window (e.g., Figure 29.7.2).

Write to file directs the report to a file. **Airpak** assigns a default name to the file, but it can be overwritten with a user-specified name. The user-specified filename can be a full pathname to the file or a pathname relative to the directory in which execution of **Airpak** was initiated.

6. Click **Write** to create the report, click **Cancel** to close the panel without creating the report, or click **Done** to store the specifications in the panel and close the panel. If you modify the model (without closing the current **Airpak** session) and reopen the **Full report** panel, its default parameters and specifications will be those currently specified.

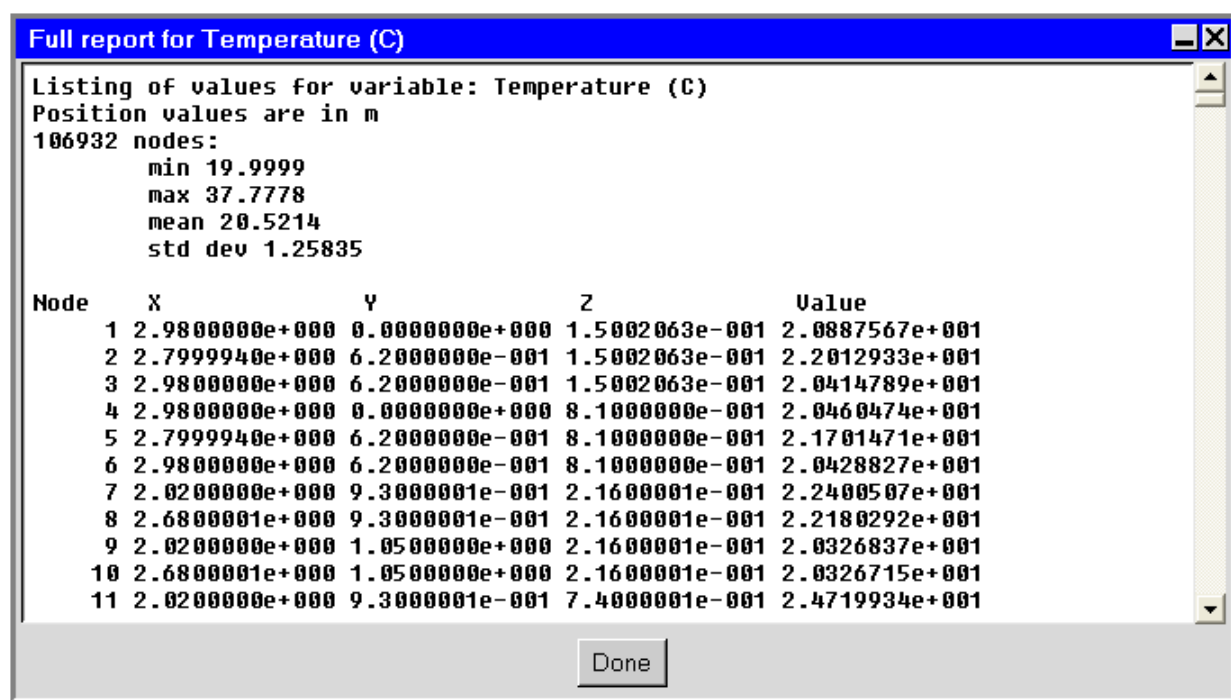


Figure 29.7.2: The Full report for Temperature window

29.8 Fan Operating Points Report

You can create a report of the fan operating points for the characteristic curve of fans in your model. To create a report, select **Fan operating points** in the **Report** menu.

Report → **Fan operating points**

The **Message** window will report the volume flow rate and pressure rise for the characteristic curve of fans in your model. An example is shown below.

Fan "fan.1" vol flow = 3 m3/s, press rise = 300 N/m2

Airpak automatically reports fan operating point data as well as other information at the end of a solution. The report is accessible by selecting **Solution overview** from the **Post** menu. See Section 29.4 for more information about the **Solution overview** option.

29.9 ADPI Report

You can create a report of the Air Diffusion Performance Index (ADPI) for a specified region in your model. The ADPI is an index for cooling ventilation applications that measures the ventilation performance in the occupied zone.

The ADPI of a region is defined as the percentage of points in the space for which the local air velocity is less than 70 ft/min and the effective draft temperature, θ_{ED} , is in the range $-3 < \theta_{ED} < 2$.

The effective draft temperature is defined as

$$\theta_{ED} = (T_{air} + T_{room,avg}) - 0.07(v_{air} - 30) \quad (29.9-1)$$

where

$$\begin{aligned} T_{air} &= \text{local air temperature (}^{\circ}\text{F)} \\ T_{room,avg} &= \text{average room temperature (}^{\circ}\text{F)} \\ v_{air} &= \text{local air velocity (ft/min)} \end{aligned}$$

You can use the **ADPI specification** panel to specify the region of the model on which to report your results. To open the **ADPI specification** panel, click on **ADPI** in the **Report** menu.

Report → **ADPI**

To create an ADPI report, use the following procedure:

1. Specify the region on which you want to report. There are two options:

Entire volume indicates that the report will be generated on the entire volume of your **Airpak** model.

Region indicates that the report will be generated on a specific region within your **Airpak** model. You will need to specify the bounding coordinates of the desired region (Min X, Min Y, Min Z, Max X, Max Y, and Max Z).

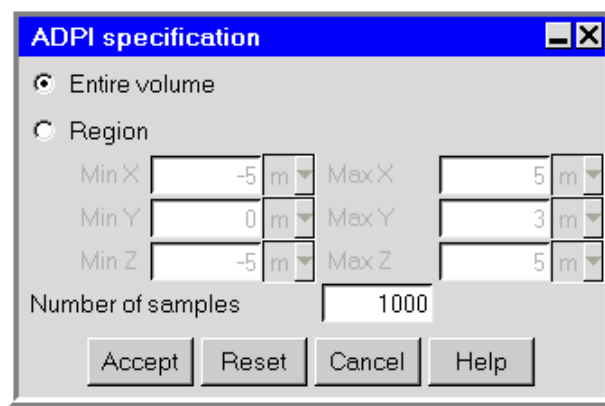


Figure 29.9.1: The ADPI specification Panel

2. Specify the **Number of samples** (or points) on which you want to base the ADPI report. The default value is 1000.
3. Click **Accept**.

The **Message** window will report the ADPI value. An example is shown below.

ADPI = 80%



Note that the ADPI is relevant only for cooling applications.

Chapter 30. Variables for Postprocessing and Reporting

In **Airpak**, variables are used within the various postprocessing and reporting operations for a variety of purposes, such as determining the color spectrum to be applied to a contour plot. This chapter provides definitions of the variables that are contained in the **Variable** and **Value** drop-down lists that appear in the postprocessing and reporting panels. See Chapter 28 for details on displaying and plotting variables and Chapter 29 for details on generating reports.

Information in this chapter is presented in the following sections:

- Section 30.1: General Information about Variables
- Section 30.2: Definitions of Variables

30.1 General Information about Variables

Airpak has two kinds of variables: scalar and vector. Scalar variables consist of a single value at each point in the solution domain, and vector variables consist of three values (one for each coordinate direction) at each point in the domain.

Scalar and vector variables are further divided into two types: primary and derived. A primary solution variable is one for which **Airpak** solves directly when performing the simulation. The primary variables are pressure, temperature, and the three components of velocity. If you are modeling turbulence with the two-equation (standard k - ϵ) model, the enhanced two-equation model, or the RNG k - ϵ model, the turbulent kinetic energy and dissipation rate will also be primary solution variables. Derived solution variables are variables that **Airpak** computes from the primary variables during postprocessing. For example, heat flux is a derived variable computed from the temperature field.

Not all variables, either primary or derived, are available for all simulations. In an isothermal simulation, for example, neither the temperature nor any derived variable based on temperature (e.g., heat flux) will be available for postprocessing analysis. For laminar solutions, or for turbulent solutions that use the zero-equation turbulence model, the turbulent kinetic energy and dissipation will not be available.

30.2 Definitions of Variables

30.2.1 Velocity-Related Quantities

Velocity-related quantities that can be reported are as follows:

- **UX** is the x component of the velocity vector and is available when flow is computed.
- **UY** is the y component of the velocity vector and is available when flow is computed.
- **UZ** is the z component of the velocity vector and is available when flow is computed.
- **Speed** is the magnitude of the velocity vector and is available when flow is computed.

$$\omega = \nabla \times \vec{u} \quad (30.2-1)$$

- **Mass flow** is a derived scalar quantity computed from the velocity field. The flow rate can be computed only on the surface of model objects (typically, a fan, vent, or opening); it cannot be computed for plane cuts or isosurfaces.
- **Volume flow** is a derived scalar quantity computed from the velocity field. The flow rate can be computed only on the surface of model objects (typically, a fan, vent, or opening); it cannot be computed for plane cuts or isosurfaces.
- **Angular deviation** is a derived scalar quantity computed from the velocity field. The angular deviation is computed as

$$\cos \theta = \frac{\mathbf{u} \cdot \mathbf{d}}{|\mathbf{u}||\mathbf{d}|} \quad (30.2-2)$$

where \mathbf{u} is the local velocity vector, \mathbf{d} is a reference direction vector, and θ is the angular deviation. To specify the reference direction vector, click on the **Parameters** button for the **Variable** list or the **Edit** button under **Params**, and enter the **X direction**, **Y direction**, and **Z direction** components of the reference direction vector in the **Angular deviation** panel (Figure 30.2.1). Click **Compute** to compute the angular deviation of the local velocity vector in degrees.

30.2.2 Pressure-Related Quantities

Pressure-related quantities that can be reported are as follows:

- **Pressure** is the relative static pressure of the fluid and is available when flow is computed.

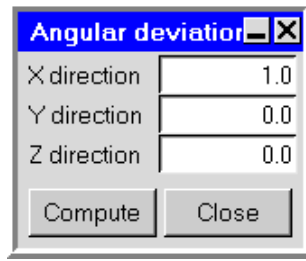


Figure 30.2.1: The Angular deviation Panel

30.2.3 Temperature-Related Quantities

Temperature-related quantities that can be reported are as follows:

- **Temperature** is available when temperature is computed.
- **Heat flux** is a derived scalar quantity computed from the temperature field. The heat flux can be computed only on the surface of model objects; it cannot be computed for plane cuts or isosurfaces. Two types of heat flux can be computed on an object face:

- the heat flux, \dot{q}_d , at a point on a surface (wall) is computed from

$$\dot{q}_d = -k\nabla T \cdot \vec{n} \quad (30.2-3)$$

where k is the conductivity of the material, T is the temperature, and \vec{n} is the normal to the surface.

- the heat flux, \dot{q}_c , at flow boundaries (e.g., openings, vents, fans) is computed from

$$\dot{q}_c = \rho c_p (T - T_{\text{ref}}) \vec{u} \cdot \vec{n} \quad (30.2-4)$$

where ρ is the density of the fluid, c_p is the specific heat, T_{ref} is 298 K, and \vec{u} is the velocity.

- **Heat flow** is a derived scalar quantity computed from the temperature field. The heat flow can be computed only on the surface of model objects; it cannot be computed for plane cuts or isosurfaces. Two types of heat flow can be computed on an object face:

- the heat flow, q_d , at a point on a surface (wall) is computed from

$$q_d = -Ak\nabla T \cdot \vec{n} \quad (30.2-5)$$

where A is the surface area of the face, k is the conductivity of the material, T is the temperature, and \vec{n} is the normal to the surface.

- the heat flow, q_c , at flow boundaries (e.g., openings, vents, fans) is computed from

$$q_c = A\rho c_p(T - T_{\text{ref}})\vec{u} \cdot \vec{n} \quad (30.2-6)$$

where A is the surface area of the face, ρ is the density of the fluid, c_p is the specific heat, T_{ref} is 298 K, and \vec{u} is the velocity.

- **Heat tr. coeff** is a derived scalar quantity computed from the temperature field. The heat transfer coefficient can be computed only on the surface of model objects; it cannot be computed for plane cuts or isosurfaces.

The heat transfer coefficient is computed as

$$h = \frac{q}{T - T_{\text{ref}}} \quad (30.2-7)$$

where q is the heat flux for the surface and T_{ref} is a reference temperature. To specify the reference temperature, click on the **Parameters** button for the **Contours** of drop-down list or click the **Edit** button under **Params** in the **Define summary report** panel. Specify the temperature next to **Ref temp** in the **Heat tr. coeff parameters** panel (Figure 30.2.2).

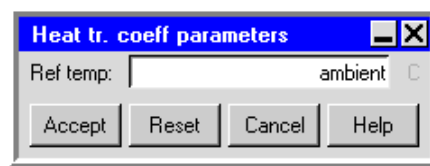


Figure 30.2.2: The Heat tr. coeff parameters Panel

30.2.4 Radiation-Related Quantities

Radiation-related quantities that can be reported are as follows:

- **Radiative heat flow** is a derived scalar quantity. It is available if you are solving your problem with radiation effects. It can be computed only on surfaces of objects.

30.2.5 Species-Transport-Related Quantities

Species-transport-related quantities that can be reported are as follows:

- **species (mole)** is the mole fraction of a particular species in the mixture. For perfect-gas mixtures, the mole fraction is equal to both the pressure fraction and the volume fraction, so the ppmv (parts per million by volume) for a species is 10^6 times the mole fraction. **Airpak** displays the mole fraction for **species (mole)** by default. To display the ppmv for the species, you can change the postprocessing units for concentration to ppmv using the **Postprocessing units** panel (see Section 5.5 for details on changing units for postprocessing). Typical concentration values in terms of mass per unit volume can be displayed by creating a user-defined unit for concentration based on the local mole fraction and reference temperatures and pressures. See Section 5.4.3 for details on creating a user-defined unit.
- **species (mass)** is the mass fraction of a particular species in the mixture.

30.2.6 Position-Related Quantities

Position-related quantities that can be reported are as follows:

- **X, Y, Z** are the Cartesian coordinates in the x -axis, y -axis, and z -axis directions respectively. They are primary variables in the sense that they are always available. These variables are useful for superimposing lines of constant x , y , or z value on the model.

30.2.7 Turbulence-Related Quantities

Turbulence-related quantities that can be reported are as follows:

- **TKE** is the turbulent kinetic energy. It can be reported if the two-equation (standard k - ϵ) turbulence model, the enhanced two-equation model, or the RNG k - ϵ turbulence model is used.
- **Epsilon** is the turbulent dissipation rate. It can be reported if the two-equation (standard k - ϵ) turbulence model, the enhanced two-equation model, or the RNG k - ϵ turbulence model is used.
- **Viscosity ratio** is the ratio of the turbulent viscosity to the laminar viscosity. It can be reported if any of the available turbulence models are used.

30.2.8 IAQ and Thermal Comfort Quantities

Indoor air quality (IAQ) and thermal comfort quantities that can be reported are as follows:

- **Mean age of air** is the average lifetime of air at a particular location in the room relative to the time when it first entered the room. It gives an indication of the air “freshness”.
- **Radiation temp** (the mean radiant temperature) is defined as the uniform temperature of an imaginary enclosure in which the radiant heat transfer from the human body equals the radiant heat transfer in the actual non-uniform enclosure. The mean radiant temperature is calculated using the results of the radiant heat transfer calculations in **Airpak**.
- **Relative humidity** is the percentage fraction of water vapor present in the air/water-vapor mixture in the room relative to a saturated air/water-vapor mixture at the same temperature.
- **PMV** (predicted mean vote) is an index indicating the mean vote concerning the thermal sensation of a large group of people expressed on a seven-point thermal sensation scale:

+3	+2	+1	0	−1	−2	−3
hot	warm	slightly warm	neutral	slightly cool	cool	cold

Airpak uses the ISO 7730 standard [?] to calculate PMV in moderate thermal environments based on air temperature, air velocity, mean radiant temperature, relative humidity or partial water-vapor pressure, activity level, and the amount and type of clothing worn. You can specify the values used by **Airpak** to compute PMV in the **Comfort level** panel as described below.

- **PPD** (predicted percentage dissatisfied) is an index that indicates the number of thermally dissatisfied people among a large group of people. **Airpak** uses the ISO 7730 standard [?] to calculate PPD in moderate thermal environments. You can specify the values used by **Airpak** to calculate PPD in the **Comfort level** panel as described below.

You can specify the values used by **Airpak** to compute PMV and PPD in the **Comfort level** panel (Figure 30.2.3). You can open this panel in two ways:

- Click on the **Parameters** button for the **Variable** list or click on the **Edit** button under **Params**.

Comfort level

External work (met) met

Clothing value (clo) ☒ Compute 1.0 ☐ Specify

Metabolic rate (met) ☒ Compute 1.0 ☐ Specify

HIGH-INSULATIVE, FIBRE-PELT
boiler suit [0.90]
trousers [0.35]
jacket [0.40]
vest [0.20]
None

reclining [0.8]
seated, relaxed [1.0]
sedentary activity (office, dv)
standing, light activity (shop)
standing, medium activity (s)
Walking on the level: 2 km/h
Walking on the level: 3 km/h

Relative humidity (%) ☒ Compute ☐ Specify

Rad temperature ☒ Compute ☐ Specify C

Air temperature ☒ Compute ☐ Specify C

Air velocity ☒ Compute ☐ Specify m/s

Compute Close

Figure 30.2.3: The Comfort level Panel

- Click on **Comfort level** in the **Report** menu.
Report → Comfort level

To compute the comfort level, follow the steps below.

1. Specify the **External work** being done by the person (usually input as 0).
2. Characterize the type and amount of clothing worn (in clo units, which represent the total thermal resistance of the garments worn). There are two options:

Compute allows you to select any number of clothing items to be worn by the person from the list of available garments. **Airpak** recomputes the **Clothing value** each time you select a new item of clothing and displays the total thermal resistance of the clothes selected next to **Compute**. To deselect all the items in the list, click **None**.

Specify allows you to specify a value for the total thermal resistance of the garments worn.

3. Specify the **Metabolic rate**. There are two options:

Compute allows you to select an activity from the list of available activities. You can select only one activity to set the metabolic rate. **Airpak** will display the metabolic rate for the selected activity next to **Compute**.

Specify allows you to specify a value for the metabolic rate.

4. Specify the **Relative humidity** in the room. There are two options:

Compute instructs **Airpak** to use the relative humidity values computed during the calculation of the solution. These values will be available only if **Airpak** performed a species calculation for water vapor.

Specify allows you to specify a global value for the relative humidity.

5. Specify the mean radiant temperature (**Rad temperature**) in the room. There are two options:

Compute instructs **Airpak** to use the radiation temperature computed during the calculation of the solution to determine the mean radiant temperature.

Specify allows you to specify a global value for the mean radiant temperature.

6. Specify the **Air temperature** in the room. There are two options:

Compute instructs **Airpak** to use the values for the air temperature it computed when calculating the solution.

Specify allows you to specify a global value for the air temperature.

7. Specify the **Air velocity** in the room. There are two options:

Compute instructs **Airpak** to use the values for the air velocity it computed when calculating the solution.

Specify allows you to specify a global value for the air velocity.

8. Click the **Compute** button to compute either PMV or PPD values, depending on your selection in the **Variable** or **Value** list. If you opened the **Comfort level** panel by clicking **Comfort level** in the **Report** menu, **Airpak** will compute PMV values. **Airpak** will display the results in the **Message** window.

This chapter provides some theoretical background for the models, equations, and solution procedures used by **Airpak**.

Information is presented in the following sections:

- Section 31.1: Governing Equations
- Section 31.2: Turbulence
- Section 31.3: Buoyancy-Driven Flows and Natural Convection
- Section 31.4: Radiation
- Section 31.5: Optimization
- Section 31.6: Solution Procedures

31.1 Governing Equations

Airpak solves the Navier-Stokes equations for transport of mass, momentum, species, and energy when it calculates laminar flow with heat transfer. Additional transport equations are solved when the flow is turbulent (see Section 31.2) or when radiative heat transfer is included (see Section 31.4).

31.1.1 The Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (31.1-1)$$

For an incompressible fluid, this reduces to

$$\nabla \cdot \vec{v} = 0 \quad (31.1-2)$$

31.1.2 Momentum Equations

Transport of momentum in an inertial (non-accelerating) reference frame is described by [3]

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (31.1-3)$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor (described below), and $\rho \vec{g}$ is the gravitational body force. \vec{F} contains other source terms that may arise from resistances, sources, etc.

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (31.1-4)$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right-hand side is the effect of volume dilation.

31.1.3 Energy Conservation Equation

The energy equation for a fluid region can be written in terms of sensible enthalpy h ($= \int_{T_{\text{ref}}}^T c_p dT$, where T_{ref} is 298.15 K) as

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho h \vec{v}) = \nabla \cdot [(k + k_t) \nabla T] + S_h \quad (31.1-5)$$

where k is the molecular conductivity, k_t is the conductivity due to turbulent transport ($k_t = c_p \mu_t / \text{Pr}_t$), and the source term S_h includes any volumetric heat sources you have defined.

In conducting solid regions, **Airpak** solves a simple conduction equation that includes the heat flux due to conduction and volumetric heat sources within the solid:

$$\frac{\partial}{\partial t}(\rho h) = \nabla \cdot (k \nabla T) + S_h \quad (31.1-6)$$

where ρ is density, k is conductivity, T is temperature, and S_h is the volumetric heat source.

Equation 31.1-6 is solved simultaneously with the energy transport equation, Equation 31.1-5, in the flow regions to yield a fully coupled conduction/convection heat transfer prediction.

31.1.4 Species Transport Equations

When you choose to solve conservation equations for species, **Airpak** predicts the local mass fraction of each species, Y_i , through the solution of a convection-diffusion equation for the i th species. This conservation equation takes the following general form:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + S_i \quad (31.1-7)$$

where S_i is the rate of creation by addition from user-defined sources. An equation of this form will be solved for $N - 1$ species where N is the total number of fluid phase species present in the system.

Mass Diffusion in Laminar Flows

\vec{J}_i is the diffusion flux of species i , which arises due to concentration gradients. **Airpak** uses the dilute approximation, under which the diffusion flux can be written as

$$\vec{J}_i = -\rho D_{i,m} \nabla Y_i \quad (31.1-8)$$

Here $D_{i,m}$ is the diffusion coefficient for species i in the mixture.

Mass Diffusion in Turbulent Flows

In turbulent flows, **Airpak** computes the mass diffusion in the following form:

$$\vec{J}_i = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right) \nabla Y_i \quad (31.1-9)$$

where Sc_t is the turbulent Schmidt number, $\frac{\mu_t}{\rho D_t}$ (with a default setting of 0.7).

Treatment of Species Transport in the Energy Equation

For many multicomponent mixing flows, the transport of enthalpy due to species diffusion

$$\nabla \cdot \left[\sum_{i=1}^n (h_i) \vec{J}_i \right]$$

can have a significant effect on the enthalpy field and should not be neglected. In particular, when the Lewis number

$$Le_i = \frac{k}{\rho c_p D_{i,m}}$$

is far from unity, this term cannot be neglected. **Airpak** will include this term by default.

31.2 Turbulence

Six turbulence models are available in **Airpak**: the zero-equation (mixing-length) model, the indoor zero-equation model, the two-equation (standard k - ϵ) model, the RNG k - ϵ model, the enhanced two-equation (standard k - ϵ with enhanced wall treatment) model, and the Spalart-Allmaras model.

31.2.1 Zero-Equation Turbulence Models

Airpak provides two zero-equation turbulence models: the mixing-length model and the indoor model. These models are described below.

Mixing-Length Zero-Equation Turbulence Model

The mixing-length zero-equation turbulence model (also known as the algebraic model) uses the following relation to calculate turbulent viscosity, μ_t :

$$\mu_t = \rho \ell^2 S \quad (31.2-1)$$

The mixing length, ℓ , is defined as

$$\ell = \min(\kappa d, 0.09 d_{\max}) \quad (31.2-2)$$

where d is the distance from the wall and the von Kármán constant $\kappa = 0.419$.

S is the modulus of the mean rate-of-strain tensor, defined as

$$S \equiv \sqrt{2S_{ij}S_{ij}} \quad (31.2-3)$$

with the mean strain rate S_{ij} given by

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \quad (31.2-4)$$

Indoor Zero-Equation Turbulence Model

The indoor zero-equation turbulence model was developed specifically for indoor airflow simulations [?]. It addresses the need of HVAC engineers for a simple but reliable turbulence model that can be used with modest desktop computing resources. It uses the following relationship to calculate the turbulent viscosity μ_t :

$$\mu_t = 0.03874 \rho v L \quad (31.2-5)$$

where v is the local velocity magnitude, ρ is the fluid density, L is defined as the distance from the nearest wall, and 0.03874 is an empirical constant.

Airpak determines the heat transfer at the boundary surfaces by computing a convective heat transfer coefficient:

$$h = \frac{\mu_{\text{eff}}}{\text{Pr}_{\text{eff}}} \frac{c_p}{\Delta x_j} \quad (31.2-6)$$

where c_p is the fluid specific heat, Pr_{eff} is the effective Prandtl number, and Δx_j is the grid spacing adjacent to the wall. μ_{eff} is the effective viscosity, given by

$$\mu_{\text{eff}} = \mu + \mu_t \quad (31.2-7)$$

where μ is the viscosity of the fluid.

This model is ideally suited for predicting indoor air flows that consider natural convection, forced convection, mixed convection, and displacement ventilation.

31.2.2 Advanced Turbulence Models

In turbulence models that employ the Boussinesq approach, the central issue is how the eddy viscosity is computed. The model proposed by Spalart and Allmaras [29] solves a transport equation for a quantity that is a modified form of the turbulent kinematic viscosity.

Both the standard and RNG k - ϵ models have similar forms, with transport equations for k and ϵ . The major differences in the models are as follows:

- the method of calculating turbulent viscosity
- the turbulent Prandtl numbers governing the turbulent diffusion of k and ϵ
- the generation and destruction terms in the ϵ equation

This section describes the Reynolds-averaging method for calculating turbulent effects and provides an overview of the issues related to choosing an advanced turbulence model in **Airpak**. The transport equations, methods of calculating turbulent viscosity, and model constants are presented separately for each model. The features that are essentially common to both models follow, including turbulent production, generation due to buoyancy, and modeling heat transfer.

Reynolds (Ensemble) Averaging

The advanced turbulence models in **Airpak** are based on Reynolds averages of the governing equations. In Reynolds averaging, the solution variables in the instantaneous (exact) Navier-Stokes equations are decomposed into the mean (ensemble-averaged or time-averaged) and fluctuating components. For the velocity components:

$$u_i = \bar{u}_i + u'_i \quad (31.2-8)$$

where \bar{u}_i and u'_i are the mean and instantaneous velocity components ($i = 1, 2, 3$).

Likewise, for pressure and other scalar quantities:

$$\phi = \bar{\phi} + \phi' \quad (31.2-9)$$

where ϕ denotes a scalar such as pressure or energy.

Substituting expressions of this form for the flow variables into the instantaneous continuity and momentum equations and taking a time (or ensemble) average (and dropping the overbar on the mean velocity, \bar{u}) yields the ensemble-averaged momentum equations. They can be written in Cartesian tensor form as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (31.2-10)$$

$$\begin{aligned} & \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \\ & - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{u'_i u'_j}) \end{aligned} \quad (31.2-11)$$

Equations 31.2-10 and 31.2-11 are called “Reynolds-averaged” Navier-Stokes (RANS) equations. They have the same general form as the instantaneous Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values. Additional terms now appear that represent the effects of turbulence. These “Reynolds stresses”, $-\rho \overline{u'_i u'_j}$, must be modeled in order to close Equation 31.2-11.

For variable-density flows, Equations 31.2-10 and 31.2-11 can be interpreted as Favre-averaged Navier-Stokes equations [11], with the velocities representing mass-averaged values. As such, Equations 31.2-10 and 31.2-11 can be applied to density-varying flows.

Boussinesq Approach

The Reynolds-averaged approach to turbulence modeling requires that the Reynolds stresses in Equation 31.2-11 be appropriately modeled. A common method employs the Boussinesq hypothesis [11] to relate the Reynolds stresses to the mean velocity gradients:

$$-\overline{\rho u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \quad (31.2-12)$$

The Boussinesq hypothesis is used in the Spalart-Allmaras model and the k - ϵ models. The advantage of this approach is the relatively low computational cost associated with the computation of the turbulent viscosity, μ_t . In the case of the Spalart-Allmaras model, only one additional transport equation (representing turbulent viscosity) is solved. In the case of the k - ϵ models, two additional transport equations (for the turbulence kinetic energy, k , and the turbulence dissipation rate, ϵ) are solved, and μ_t is computed as a function of k and ϵ . The disadvantage of the Boussinesq hypothesis as presented is that it assumes μ_t is an isotropic scalar quantity, which is not strictly true.

Choosing an Advanced Turbulence Model

This section provides an overview of the issues related to the advanced turbulence models provided in Airpak.

The Spalart-Allmaras Model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity for turbomachinery applications.

On a cautionary note, however, the Spalart-Allmaras model is still relatively new, and no claim is made regarding its suitability to all types of complex engineering flows. For instance, it cannot be relied on to predict the decay of homogeneous, isotropic turbulence. Furthermore, one-equation models are often criticized for their inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from a wall-bounded to a free shear flow.

The Standard k - ϵ Model

The simplest “complete models” of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k - ϵ model in Airpak falls within

this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [19]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

As the strengths and weaknesses of the standard k - ϵ model have become known, improvements have been made to the model to improve its performance. One of these variants is available in Airpak: the RNG k - ϵ model [32].

The RNG k - ϵ Model

The RNG k - ϵ model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard k - ϵ model, but includes the following refinements:

- The RNG model has an additional term in its ϵ equation that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard k - ϵ model uses user-specified, constant values.
- While the standard k - ϵ model is a high-Reynolds-number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds-number effects.

These features make the RNG k - ϵ model more accurate and reliable for a wider class of flows than the standard k - ϵ model.

The Enhanced Two-Equation Model

The k - ϵ models are primarily valid for turbulent core flows (i.e., the flow in the regions somewhat far from walls). Consideration therefore needs to be given as to how to make these models suitable for wall-bounded flows.

Turbulent flows are significantly affected by the presence of walls. Obviously, the mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. However, the turbulence is also changed by the presence of the wall in non-trivial ways. Very close to the wall, viscous damping reduces the tangential velocity fluctuations, while kinematic blocking reduces the normal fluctuations. Toward the outer part of the near-wall region, however, the turbulence is rapidly augmented by the production of turbulence kinetic energy due to the large gradients in mean velocity.

The near-wall modeling significantly impacts the fidelity of numerical solutions, inasmuch as walls are the main source of mean vorticity and turbulence. It is in the near-wall region where the solution variables have large gradients and where the momentum and other scalar transports are the greatest. Therefore, accurate representation of the flow in the near-wall region determines successful predictions of wall-bounded turbulent flows.

Numerous experiments have shown that the near-wall region can be largely subdivided into three layers. In the innermost layer, called the “viscous sublayer”, the flow is almost laminar, and the (molecular) viscosity plays a dominant role in momentum and heat or mass transfer. In the outer layer, called the fully-turbulent layer, turbulence plays a major role. Finally, there is an interim region between the viscous sublayer and the fully turbulent layer where the effects of molecular viscosity and turbulence are equally important.

To more accurately resolve the flow near the wall, the *enhanced* two-equation model combines the standard k - ϵ model with enhanced wall treatment.

Enhanced Wall Treatment

Enhanced wall treatment is a near-wall modeling method that combines a two-layer model with enhanced wall functions [4, 12, 15, 16, 30, 31].

In the two-layer model, the viscosity-affected near-wall region is completely resolved all the way to the viscous sublayer. The two-layer approach is an integral part of the enhanced wall treatment and is used to specify both ϵ and the turbulent viscosity in the near-wall cells. In this approach, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region. The demarcation of the two regions is determined by a wall-distance-based, turbulent Reynolds number.

If the near-wall mesh is fine enough to be able to resolve the laminar sublayer (typically $y^+ \approx 1$), then the enhanced wall treatment will be identical to the traditional two-layer zonal model. However, the restriction that the near-wall mesh must be sufficiently fine everywhere might impose too large a computational requirement. Ideally, then, one would like to have a near-wall formulation that can be used with coarse meshes (usually referred to as wall-function meshes) as well as fine meshes (low-Reynolds-number meshes). In addition, excessive error should not be incurred for intermediate meshes that are too fine for the near-wall cell centroid to lie in the fully turbulent region, but also too coarse to properly resolve the sublayer.

To achieve the goal of having a near-wall modeling approach that will possess the accuracy of the standard two-layer approach for fine near-wall meshes and will not significantly reduce accuracy for wall-function meshes, **Airpak** combines the two-layer model with enhanced wall functions to result in the enhanced wall treatment.

Computational Effort: CPU Time and Solution Behavior

The standard k - ϵ model clearly requires more computational effort than the Spalart-Allmaras model since an additional transport equation is solved. However, due to the extra terms and functions in the governing equations and a greater degree of nonlinearity, computations with the RNG k - ϵ model tend to take 10–15% more CPU time than with the standard k - ϵ model.

Aside from the time per iteration, the choice of turbulence model can affect the ability of **Airpak** to obtain a converged solution. For example, the standard k - ϵ model is known to be slightly over-diffusive in certain situations, while the RNG k - ϵ model is designed such that the turbulent viscosity is reduced in response to high rates of strain. Since diffusion has a stabilizing effect on the numerics, the RNG model is more likely to be susceptible to instability in steady-state solutions. However, this should not necessarily be seen as a disadvantage of the RNG model, since these characteristics make it more responsive to important physical instabilities such as time-dependent turbulent vortex shedding.

The Spalart-Allmaras Model

In its original form, the Spalart-Allmaras model is effectively a low-Reynolds-number model, requiring the viscous-affected region of the boundary layer to be properly resolved. In **Airpak**, however, the Spalart-Allmaras model has been implemented to use wall functions when the mesh resolution is not sufficiently fine. This might make it the best choice for relatively crude simulations on coarse meshes where accurate turbulent flow computations are not critical. Furthermore, the near-wall gradients of the transported variable in the model are much smaller than the gradients of the transported variables in the k - ϵ models. This might make the model less sensitive to numerical error when non-layered meshes are used near walls.

Transport Equation for the Spalart-Allmaras Model

The transported variable in the Spalart-Allmaras model, $\tilde{\nu}$, is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region. The transport equation for $\tilde{\nu}$ is

$$\frac{\partial}{\partial t}(\rho\tilde{\nu}) + \frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i) = G_\nu + \frac{1}{\sigma_\nu} \left[\frac{\partial}{\partial x_j} \left\{ (\mu + \rho\tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right\} + C_{b2}\rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] - Y_\nu + S_\nu \quad (31.2-13)$$

where G_ν is the production of turbulent viscosity and Y_ν is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. σ_ν and C_{b2} are constants and ν is the molecular kinematic viscosity. S_ν is a user-defined source term. Note that since the turbulence kinetic energy k is not calculated in the Spalart-Allmaras model, the last term in Equation 31.2-12 is ignored when estimating the Reynolds stresses.

Modeling the Turbulent Viscosity

The turbulent viscosity, μ_t , is computed from

$$\mu_t = \rho \tilde{\nu} f_{v1} \quad (31.2-14)$$

where the viscous damping function, f_{v1} , is given by

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3} \quad (31.2-15)$$

and

$$\chi \equiv \frac{\tilde{\nu}}{\nu} \quad (31.2-16)$$

Modeling the Turbulent Production

The production term, G_ν , is modeled as

$$G_\nu = C_{b1} \rho \tilde{S} \tilde{\nu} \quad (31.2-17)$$

where

$$\tilde{S} \equiv S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2} \quad (31.2-18)$$

and

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (31.2-19)$$

C_{b1} and κ are constants, d is the distance from the wall, and S is a scalar measure of the deformation tensor. By default in **Airpak**, as in the original model proposed by Spalart and Allmaras, S is based on the magnitude of the vorticity:

$$S \equiv \sqrt{2\Omega_{ij}\Omega_{ij}} \quad (31.2-20)$$

where Ω_{ij} is the mean rate-of-rotation tensor and is defined by

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (31.2-21)$$

The justification for the default expression for S is that, for the wall-bounded flows that were of most interest when the model was formulated, turbulence is found only where vorticity is generated near walls. However, it has since been acknowledged that one should also take into account the effect of mean strain on the turbulence production, and a modification to the model has been proposed [9] and incorporated into **Airpak**.

This modification combines measures of both rotation and strain tensors in the definition of S :

$$S \equiv |\Omega_{ij}| + C_{\text{prod}} \min(0, |S_{ij}| - |\Omega_{ij}|) \quad (31.2-22)$$

where

$$C_{\text{prod}} = 2.0, \quad |\Omega_{ij}| \equiv \sqrt{2\Omega_{ij}\Omega_{ij}}, \quad |S_{ij}| \equiv \sqrt{2S_{ij}S_{ij}}$$

with the mean strain rate, S_{ij} , defined as

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \quad (31.2-23)$$

Including both the rotation and strain tensors reduces the production of eddy viscosity and consequently reduces the eddy viscosity itself in regions where the measure of vorticity exceeds that of strain rate. One such example can be found in vortical flows, i.e., flow near the core of a vortex subjected to a pure rotation where turbulence is known to be suppressed. Including both the rotation and strain tensors more correctly accounts for the effects of rotation on turbulence. The default option (including the rotation tensor only) tends to overpredict the production of eddy viscosity and hence overpredicts the eddy viscosity itself in certain circumstances.

Modeling the Turbulent Destruction

The destruction term is modeled as

$$Y_\nu = C_{w1} \rho f_w \left(\frac{\tilde{\nu}}{d} \right)^2 \quad (31.2-24)$$

where

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{1/6} \quad (31.2-25)$$

$$g = r + C_{w2} (r^6 - r) \quad (31.2-26)$$

$$r \equiv \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2} \quad (31.2-27)$$

C_{w1} , C_{w2} , and C_{w3} are constants, and \tilde{S} is given by Equation 31.2-18. Note that the modification described above to include the effects of mean strain on S will also affect the value of \tilde{S} used to compute r .

Model Constants

The model constants C_{b1} , C_{b2} , $\sigma_{\tilde{\nu}}$, C_{v1} , C_{w1} , C_{w2} , C_{w3} , and κ have the following default values [29]:

$$C_{b1} = 0.1355, \quad C_{b2} = 0.622, \quad \sigma_{\tilde{\nu}} = \frac{2}{3}, \quad C_{v1} = 7.1$$

$$C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{(1 + C_{b2})}{\sigma_{\tilde{\nu}}}, \quad C_{w2} = 0.3, \quad C_{w3} = 2.0, \quad \kappa = 0.4187$$

Wall Boundary Conditions

At walls, the modified turbulent kinematic viscosity, $\tilde{\nu}$, is set to zero.

When the mesh is fine enough to resolve the laminar sublayer, the wall shear stress is obtained from the laminar stress-strain relationship:

$$\frac{u}{u_\tau} = \frac{\rho u_\tau y}{\mu} \quad (31.2-28)$$

If the mesh is too coarse to resolve the laminar sublayer, it is assumed that the centroid of the wall-adjacent cell falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho u_\tau y}{\mu} \right) \quad (31.2-29)$$

where u is the velocity parallel to the wall, u_τ is the shear velocity, y is the distance from the wall, κ is the von Kármán constant (0.4187), and $E = 9.793$.

Convective Heat and Mass Transfer Modeling

In *Airpak*, turbulent heat transport is modeled using the concept of Reynolds' analogy to turbulent momentum transfer. The “modeled” energy equation is thus given by the following:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + p)] = \frac{\partial}{\partial x_j} \left[\left(k + \frac{c_p \mu_t}{\text{Pr}_t} \right) \frac{\partial T}{\partial x_j} + u_i(\tau_{ij})_{\text{eff}} \right] + S_h \quad (31.2-30)$$

where k , in this case, is the thermal conductivity, E is the total energy, and $(\tau_{ij})_{\text{eff}}$ is the deviatoric stress tensor, defined as

$$(\tau_{ij})_{\text{eff}} = \mu_{\text{eff}} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_i}{\partial x_i} \delta_{ij}$$

The term involving $(\tau_{ij})_{\text{eff}}$ represents the viscous heating. The default value of the turbulent Prandtl number is 0.85. Turbulent mass transfer is treated similarly, with a default turbulent Schmidt number of 0.7.

Wall boundary conditions for scalar transport are handled analogously to momentum, using the appropriate “law-of-the-wall”.

Two-Equation (Standard k - ϵ) Turbulence Model

The two-equation turbulence model (also known as the standard k - ϵ model) is more complex than the zero-equation model. The standard k - ϵ model [19] is a semi-empirical model based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ϵ). The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the standard k - ϵ model, it is assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k - ϵ model is therefore valid only for fully turbulent flows.

Transport Equations for the Standard k - ϵ Model

The turbulent kinetic energy, k , and its rate of dissipation, ϵ , are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \epsilon \quad (31.2-31)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \quad (31.2-32)$$

In these equations, G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients, calculated as described later in this section. G_b is the generation of turbulent kinetic energy due to buoyancy, calculated as described later in this section. $C_{1\epsilon}$, $C_{2\epsilon}$, and $C_{3\epsilon}$ are constants. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ , respectively.

Modeling the Turbulent Viscosity

The “eddy” or turbulent viscosity, μ_t , is computed by combining k and ϵ as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (31.2-33)$$

where C_μ is a constant.

Model Constants

The model constants $C_{1\epsilon}$, $C_{2\epsilon}$, C_μ , σ_k , and σ_ϵ have the following default values [19]:

$$C_{1\epsilon} = 1.44, \quad C_{2\epsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3$$

These default values have been determined from experiments with air and water for fundamental turbulent shear flows including homogeneous shear flows and decaying isotropic grid turbulence. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

The RNG k - ϵ Model

The RNG-based k - ϵ turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called “renormalization group” (RNG) methods. The analytical derivation results in a model with constants different from those in the standard k - ϵ model, and additional terms and functions in the transport equations for k and ϵ . A more comprehensive description of RNG theory and its application to turbulence can be found in [6].

Transport Equations for the RNG k - ϵ Model

The RNG k - ϵ model has a similar form to the standard k - ϵ model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i} \left(\alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_i} \right) + G_k + G_b - \rho \epsilon \quad (31.2-34)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_i} \left(\alpha_\epsilon \mu_{\text{eff}} \frac{\partial \epsilon}{\partial x_i} \right) + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} - R_\epsilon \quad (31.2-35)$$

In these equations, G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients, calculated as described later in this section. G_b is the generation of turbulent kinetic energy due to buoyancy, calculated as described later in this section. The quantities α_k and α_ϵ are the inverse effective Prandtl numbers for k and ϵ , respectively.

Modeling the Effective Viscosity

The scale elimination procedure in RNG theory results in a differential equation for turbulent viscosity:

$$d \left(\frac{\rho^2 k}{\sqrt{\epsilon \mu}} \right) = 1.72 \frac{\hat{\nu}}{\sqrt{\hat{\nu}^3 - 1 + C_\nu}} d\hat{\nu} \quad (31.2-36)$$

where

$$\begin{aligned} \hat{\nu} &= \mu_{\text{eff}} / \mu \\ C_\nu &\approx 100 \end{aligned}$$

Equation 31.2-36 is integrated to obtain an accurate description of how the effective turbulent transport varies with the effective Reynolds number (or eddy scale), allowing the model to better handle low-Reynolds-number and near-wall flows.

In the high-Reynolds-number limit, Equation 31.2-36 gives

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (31.2-37)$$

with $C_\mu = 0.0845$, derived using RNG theory. It is interesting to note that this value of C_μ is very close to the empirically-determined value of 0.09 used in the standard k - ϵ model.

In **Airpak**, the effective viscosity is computed using the high-Reynolds-number form in Equation 31.2-37.

Calculating the Inverse Effective Prandtl Numbers

The inverse effective Prandtl numbers α_k and α_ϵ are computed using the following formula derived analytically by the RNG theory:

$$\left| \frac{\alpha - 1.3929}{\alpha_0 - 1.3929} \right|^{0.6321} \left| \frac{\alpha + 2.3929}{\alpha_0 + 2.3929} \right|^{0.3679} = \frac{\mu_{\text{mol}}}{\mu_{\text{eff}}} \quad (31.2-38)$$

where $\alpha_0 = 1.0$. In the high-Reynolds-number limit ($\mu_{\text{mol}}/\mu_{\text{eff}} \ll 1$), $\alpha_k = \alpha_\epsilon \approx 1.393$.

The R_ϵ Term in the ϵ Equation

The main difference between the RNG and standard k - ϵ models lies in the additional term in the ϵ equation given by

$$R_\epsilon = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \frac{\epsilon^2}{k} \quad (31.2-39)$$

where $\eta \equiv Sk/\epsilon$, $\eta_0 = 4.38$, $\beta = 0.012$.

The effects of this term in the RNG ϵ equation can be seen more clearly by rearranging Equation 31.2-35. Using Equation 31.2-39, the last two terms in Equation 31.2-35 can be merged, and the resulting ϵ equation can be rewritten as

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j} \left(\alpha_\epsilon \mu_{\text{eff}} \frac{\partial \epsilon}{\partial x_j} \right) + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon}^* \rho \frac{\epsilon^2}{k} \quad (31.2-40)$$

where $C_{2\epsilon}^*$ is given by

$$C_{2\epsilon}^* \equiv C_{2\epsilon} + \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \quad (31.2-41)$$

In regions where $\eta < \eta_0$, the R term makes a positive contribution, and $C_{2\epsilon}^*$ becomes larger than $C_{2\epsilon}$. In the logarithmic layer, for instance, it can be shown that $\eta \approx 3.0$, giving $C_{2\epsilon}^* \approx 2.0$, which is close in magnitude to the value of $C_{2\epsilon}$ in the standard k - ϵ model (1.92). As a result, for weakly to moderately strained flows, the RNG model tends to give results largely comparable to the standard k - ϵ model.

In regions of large strain rate ($\eta > \eta_0$), however, the R term makes a negative contribution, making the value of $C_{2\epsilon}^*$ less than $C_{2\epsilon}$. In comparison with the standard k - ϵ model, the smaller destruction of ϵ augments ϵ , reducing k and eventually the effective viscosity. As a result, in rapidly strained flows, the RNG model yields a lower turbulent viscosity than the standard k - ϵ model.

Thus, the RNG model is more responsive to the effects of rapid strain and streamline curvature than the standard k - ϵ model, which explains the superior performance of the RNG model for certain classes of flows.

Model Constants

The model constants $C_{1\epsilon}$ and $C_{2\epsilon}$ in Equation 31.2-35 have values derived analytically by the RNG theory. These values, used by default in **Airpak**, are

$$C_{1\epsilon} = 1.42, \quad C_{2\epsilon} = 1.68$$

Modeling Turbulent Production in the k - ϵ Models

From the exact equation for the transport of k , the term G_k , representing the production of turbulent kinetic energy, can be defined as

$$G_k = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \quad (31.2-42)$$

To evaluate G_k in a manner consistent with the Boussinesq hypothesis,

$$G_k = \mu_t S^2 \quad (31.2-43)$$

where S is the modulus of the mean rate-of-strain tensor, defined as

$$S \equiv \sqrt{2S_{ij}S_{ij}} \quad (31.2-44)$$

with the mean strain rate S_{ij} given by

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \quad (31.2-45)$$

Effects of Buoyancy on Turbulence in the k - ϵ Models

When a non-zero gravity field and temperature gradient are present simultaneously, the k - ϵ models in **Airpak** account for the generation of k due to buoyancy (G_b in Equations 31.2-31 and 31.2-34), and the corresponding contribution to the production of ϵ in Equations 31.2-32 and 31.2-35.

The generation of turbulence due to buoyancy is given by

$$G_b = \beta g_i \frac{\mu_t}{\text{Pr}_t} \frac{\partial T}{\partial x_i} \quad (31.2-46)$$

where Pr_t is the turbulent Prandtl number for energy. For the standard k - ϵ model, the default value of Pr_t is 0.85. In the case of the RNG k - ϵ model, $\text{Pr}_t = 1/\alpha$, where α is given by Equation 31.2-38, but with $\alpha_0 = 1/\text{Pr} = k/\mu c_p$. The coefficient of thermal expansion, β , is defined as

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \quad (31.2-47)$$

It can be seen from the transport equation for k (Equation 31.2-31 or 31.2-34) that turbulent kinetic energy tends to be augmented ($G_b > 0$) in unstable stratification. For stable stratification, buoyancy tends to suppress the turbulence ($G_b < 0$). In **Airpak**, the effects of buoyancy on the generation of k are always included when you have both a non-zero gravity field and a non-zero temperature (or density) gradient.

While the buoyancy effects on the generation of k are relatively well understood, the effect on ϵ is less clear. In **Airpak**, by default, the buoyancy effects on ϵ are neglected simply by setting G_b to zero in the transport equation for ϵ (Equation 31.2-32 or 31.2-35).

The degree to which ϵ is affected by the buoyancy is determined by the constant $C_{3\epsilon}$. In **Airpak**, $C_{3\epsilon}$ is not specified, but is instead calculated according to the following relation [10]:

$$C_{3\epsilon} = \tanh \left| \frac{v}{u} \right| \quad (31.2-48)$$

where v is the component of the flow velocity parallel to the gravitational vector and u is the component of the flow velocity perpendicular to the gravitational vector. In this way, $C_{3\epsilon}$ will become 1 for buoyant shear layers for which the main flow direction is aligned with the direction of gravity. For buoyant shear layers that are perpendicular to the gravitational vector, $C_{3\epsilon}$ will become zero.

Convective Heat Transfer Modeling in the k - ϵ Models

In **Airpak**, turbulent heat transport is modeled using the concept of Reynolds' analogy to turbulent momentum transfer. The “modeled” energy equation is thus given by the following:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + p)] = \frac{\partial}{\partial x_i} \left(k_{\text{eff}} \frac{\partial T}{\partial x_i} \right) + S_h \quad (31.2-49)$$

where E is the total energy and k_{eff} is the effective conductivity.

For the standard k - ϵ model, k_{eff} is given by

$$k_{\text{eff}} = k + \frac{c_p \mu_t}{\text{Pr}_t}$$

with the default value of the turbulent Prandtl number set to 0.85.

For the RNG k - ϵ model, the effective thermal conductivity is

$$k_{\text{eff}} = \alpha c_p \mu_{\text{eff}}$$

where α is calculated from Equation 31.2-38, but with $\alpha_0 = 1/\text{Pr} = k/\mu c_p$.

The fact that α varies with $\mu_{\text{mol}}/\mu_{\text{eff}}$, as in Equation 31.2-38, is an advantage of the RNG k - ϵ model. It is consistent with experimental evidence indicating that the turbulent Prandtl number varies with the molecular Prandtl number and turbulence [18]. Equation 31.2-38 works well across a very broad range of molecular Prandtl numbers, from liquid metals ($\text{Pr} \approx 10^{-2}$) to paraffin oils ($\text{Pr} \approx 10^3$), which allows heat transfer to be calculated in low-Reynolds-number regions. Equation 31.2-38 smoothly predicts the variation of effective Prandtl number from the molecular value ($\alpha = 1/\text{Pr}$) in the viscosity-dominated region to the fully turbulent value ($\alpha = 1.393$) in the fully turbulent regions of the flow.

31.3 Buoyancy-Driven Flows and Natural Convection

The importance of buoyancy forces in a mixed convection flow can be measured by the ratio of the Grashof and Reynolds numbers:

$$\frac{\text{Gr}}{\text{Re}^2} = \frac{g\beta\Delta TL}{\nu^2} \quad (31.3-1)$$

When this number approaches or exceeds unity, you should expect strong buoyancy contributions to the flow. Conversely, if it is very small, buoyancy forces may be ignored in your simulation. In pure natural convection, the strength of the buoyancy-induced flow is measured by the Rayleigh number:

$$\text{Ra} = \frac{g\beta\Delta TL^3\rho}{\mu\alpha} \quad (31.3-2)$$

where β is the thermal expansion coefficient:

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \quad (31.3-3)$$

and α is the thermal diffusivity:

$$\alpha = \frac{k}{\rho c_p} \quad (31.3-4)$$

Rayleigh numbers less than 10^8 indicate a buoyancy-induced laminar flow, with transition to turbulence occurring over the range of $10^8 < \text{Ra} < 10^{10}$.

Airpak uses either the Boussinesq model or the ideal gas law in the calculation of natural-convection flows, as described below.

31.3.1 The Boussinesq Model

By default, **Airpak** uses the Boussinesq model for natural-convection flows. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$(\rho - \rho_0)g \approx -\rho_0\beta(T - T_0)g \quad (31.3-5)$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. Equation 31.3-5 is obtained by using the Boussinesq

approximation $\rho = \rho_0(1 - \beta\Delta T)$ to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when $\beta(T - T_0) \ll 1$.

31.3.2 Incompressible Ideal Gas Law

In **Airpak**, if you choose to define the density using the ideal gas law, **Airpak** will compute the density as

$$\rho = \frac{p_{\text{op}}}{\frac{R}{M}T} \quad (31.3-6)$$

where R is the universal gas constant and p_{op} is defined by you as the **Oper. pressure** in the **Advanced problem setup** panel (see Section 6.4.4). In this form, the density depends only on the operating pressure and not on the local relative pressure field, local temperature field, or molecular weight.

Definition of the Operating Density

When the Boussinesq approximation is not used, the operating density, ρ_0 , appears in the body-force term in the momentum equations as $(\rho - \rho_0)g$.

This form of the body-force term follows from the redefinition of pressure in **Airpak** as

$$p'_s = p_s - \rho_0 g x \quad (31.3-7)$$

The hydrostatic balance in a fluid at rest is then

$$p'_s = 0 \quad (31.3-8)$$

The definition of the operating density is thus important in all buoyancy-driven flows.

31.4 Radiation

31.4.1 Overview

The terms radiative heat transfer and thermal radiation are commonly used to describe heat transfer caused by electromagnetic (EM) waves. All materials continually emit and absorb EM waves, or photons. The strength and wavelength of emission depends on the temperature of the emitting material. At absolute zero K, no radiation is emitted from a surface. For heat transfer applications, wavelengths in the infrared spectrum are generally of greatest importance and are, therefore, the only ones considered in Airpak.

While both conduction and convection (the other basic modes of heat transfer) require a medium for transmission, radiation does not. Therefore, thermal radiation can traverse a long distance without interacting with a medium. Also, for most applications, conductive and convective heat transfer rates are linearly proportional to temperature differences. Radiative heat transfer rates, on the other hand, are (for the most part) proportional to differences in temperature raised to the fourth power.

31.4.2 Gray-Diffuse Radiation

Airpak's radiation models assume the surfaces to be gray and diffuse. Emissivity and absorptivity of a gray surface are independent of the wavelength. Also, by Kirchhoff's law [20], the emissivity equals the absorptivity ($\epsilon = \alpha$). For a diffuse surface, the reflectivity is independent of the outgoing (or incoming) directions.

As stated earlier, for applications of interest, the exchange of radiative energy between surfaces is virtually unaffected by the medium that separates them. Thus, according to the gray-body model, if a certain amount of radiation (E) is incident on a surface, a fraction (ρE) is reflected, a fraction (αE) is absorbed, and a fraction (τE) is transmitted. Since for most indoor applications the surfaces in question are opaque to thermal radiation (in the infrared spectrum), the surfaces can be considered opaque. The transmissivity, therefore, can be neglected. It follows, from conservation of energy, that $\alpha + \rho = 1$, since $\alpha = \epsilon$ (emissivity), and $\rho = 1 - \epsilon$.

31.4.3 Radiative Transfer Equation

The radiative transfer equation (RTE) for an absorbing, emitting, and scattering medium at position \vec{r} in the direction \vec{s} is

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \Phi(\vec{s} \cdot \vec{s}') d\Omega' \quad (31.4-1)$$

where	\vec{r}	=	position vector
	\vec{s}	=	direction vector
	\vec{s}'	=	scattering direction vector
	s	=	path length
	a	=	absorption coefficient
	n	=	refractive index
	σ_s	=	scattering coefficient
	σ	=	Stefan-Boltzmann constant ($5.672 \times 10^{-8} \text{ W/m}^2\text{-K}^4$)
	I	=	radiation intensity, which depends on position (\vec{r}) and direction (\vec{s})
	T	=	local temperature
	Φ	=	phase function
	Ω'	=	solid angle

$(a + \sigma_s)s$ is the optical thickness or opacity of the medium. The refractive index n is important when considering radiation in semi-transparent media. Figure 31.4.1 illustrates the process of radiation heat transfer.

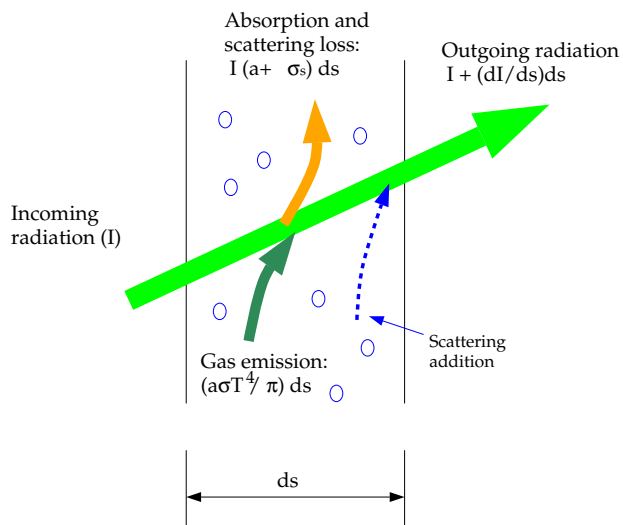


Figure 31.4.1: Radiation Heat Transfer

31.4.4 The Surface-to-Surface Radiation Model

The default radiation model used in **Airpak** is the surface-to-surface radiation model. The energy flux leaving a given surface is composed of directly emitted and reflected energy. The reflected energy flux is dependent on the incident energy flux from the surroundings, which then can be expressed in terms of the energy flux leaving all other surfaces. The energy reflected from surface k is

$$q_{\text{out},k} = \epsilon_k \sigma T_k^4 + \rho_k q_{\text{in},k} \quad (31.4-2)$$

where $q_{\text{out},k}$ is the energy flux leaving the surface, ϵ_k is the emissivity, σ is the Boltzmann constant, and $q_{\text{in},k}$ is the energy flux incident on the surface from the surroundings.

The amount of incident energy upon a surface from another surface is a direct function of the surface-to-surface “view factor,” F_{jk} . The view factor F_{jk} is the fraction of energy leaving surface k that is incident on surface j . The incident energy flux $q_{\text{in},k}$ can be expressed in terms of the energy flux leaving all other surfaces as

$$A_k q_{\text{in},k} = \sum_{j=1}^N A_j q_{\text{out},j} F_{jk} \quad (31.4-3)$$

where A_k is the area of surface k and F_{jk} is the view factor between surface k and surface j . For N surfaces, using the view factor reciprocity relationship gives

$$A_j F_{jk} = A_k F_{kj} \text{ for } j = 1, 2, 3, \dots, N \quad (31.4-4)$$

so that

$$q_{\text{in},k} = \sum_{j=1}^N F_{kj} q_{\text{out},j} \quad (31.4-5)$$

Therefore,

$$q_{\text{out},k} = \epsilon_k \sigma T_k^4 + \rho_k \sum_{j=1}^N F_{kj} q_{\text{out},j} \quad (31.4-6)$$

which can be written as

$$J_k = E_k + \rho_k \sum_{j=1}^N F_{kj} J_j \quad (31.4-7)$$

where J_k represents the energy that is given off (or radiosity) of surface k and E_k represents the emissive power of surface k . This represents N equations, which can be recast into matrix form as

$$\mathbf{KJ} = \mathbf{E} \quad (31.4-8)$$

where \mathbf{K} is an $N \times N$ matrix, \mathbf{J} is the radiosity vector, and \mathbf{E} is the emissive power vector.

Equation 31.4-8 is referred to as the radiosity matrix equation. The view factor between two finite surfaces i and j is given by

$$F_{ij} = \frac{1}{A_i} \int_{A_i} \int_{A_j} \frac{\cos \theta_i \cos \theta_j}{\pi r^2} \delta_{ij} dA_i dA_j \quad (31.4-9)$$

where δ_{ij} is determined by the visibility of dA_j to dA_i . $\delta_{ij} = 1$ if dA_j is visible to dA_i and 0 otherwise.

31.4.5 The Discrete Ordinates (DO) Radiation Model

The discrete ordinates (DO) radiation model solves the radiative transfer equation (RTE) for a finite number of discrete solid angles, each associated with a vector direction \vec{s} fixed in the global Cartesian system (x, y, z) . The fineness of the angular discretization is set by default parameters that depend on the type of problem you are solving. The DO model transforms Equation 31.4-1 into a transport equation for radiation intensity in the spatial coordinates (x, y, z) . The DO model solves for as many transport equations as there are directions \vec{s} . The solution method is identical to that used for the fluid flow and energy equations.

The implementation in **Airpak** uses a conservative variant of the discrete ordinates model called the finite-volume scheme [7, 23], and its extension to unstructured meshes [21].

The DO Equations

The DO model considers the radiative transfer equation (RTE) in the direction \vec{s} as a field equation. Thus, Equation 31.4-1 is written as

$$\nabla \cdot (I(\vec{r}, \vec{s}) \vec{s}) + (a + \sigma_s) I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \Phi(\vec{s} \cdot \vec{s}') d\Omega' \quad (31.4-10)$$

Angular Discretization and Pixelation

Each octant of the angular space 4π at any spatial location is discretized into $N_\theta \times N_\phi$ solid angles of extent ω_i , called control angles. The angles θ and ϕ are the polar and azimuthal angles respectively, and are measured with respect to the global Cartesian system (x, y, z) as shown in Figure 31.4.2. The θ and ϕ extents of the control angle, $\Delta\theta$ and $\Delta\phi$, are constant.

When Cartesian meshes are used, it is possible to align the global angular discretization with the control volume face, as shown in Figure 31.4.3. For generalized unstructured meshes, however, control volume faces do not in general align with the global angular discretization, as shown in Figure 31.4.4, leading to the problem of control angle overhang [21].

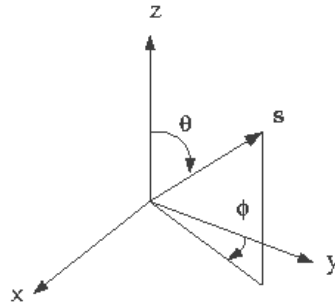


Figure 31.4.2: Angular Coordinate System

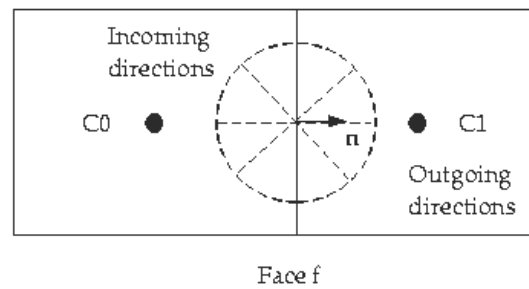


Figure 31.4.3: Face with No Control Angle Overhang

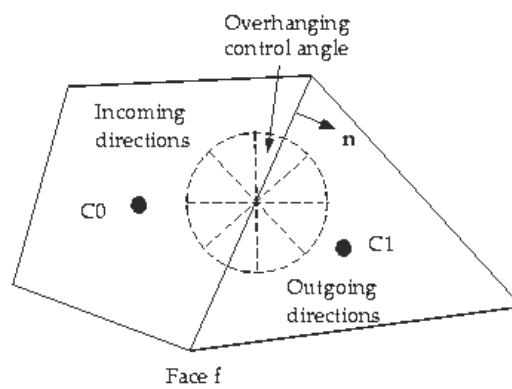


Figure 31.4.4: Face with Control Angle Overhang

Essentially, control angles can straddle the control volume faces, so that they are partially incoming and partially outgoing to the face. Figure 31.4.5 shows a 3D example of a face with control angle overhang.

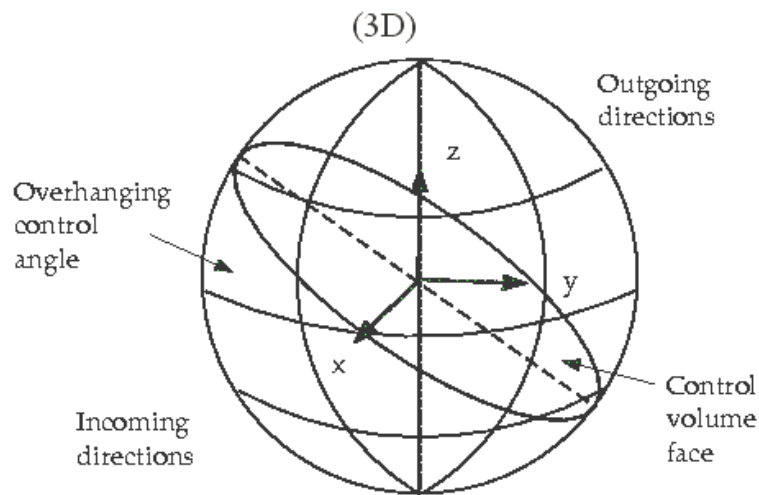


Figure 31.4.5: Face with Control Angle Overhang (3D)

The control volume face cuts the sphere representing the angular space at an arbitrary angle. The line of intersection is a great circle. Control angle overhang may also occur as a result of reflection and refraction. It is important in these cases to correctly account for the overhanging fraction. This is done through the use of pixelation [21].

Each overhanging control angle is divided into $N_{\theta_p} \times N_{\phi_p}$ pixels, as shown in Figure 31.4.6. The energy contained in each pixel is then treated as incoming or outgoing to the face. The influence of overhang can thus be accounted for within the pixel resolution. For problems involving gray-diffuse radiation, the default pixelation of 1×1 is usually sufficient. For problems involving symmetry boundaries, a pixelation of 3×3 is used.

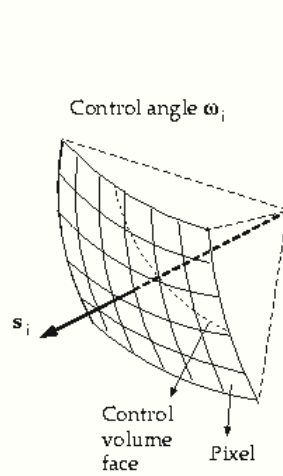


Figure 31.4.6: Pixelation of Control Angle

Boundary Condition Treatment at Gray-Diffuse Walls

For gray radiation, the incident radiation heat flux, q_{in} , at the wall is

$$q_{\text{in}} = \int_{\vec{s} \cdot \vec{n} > 0} I_{\text{in}} \vec{s} \cdot \vec{n} d\Omega \quad (31.4-11)$$

The net radiative flux leaving the surface is given by

$$q_{\text{out}} = (1 - \epsilon_w) q_{\text{in}} + n^2 \epsilon_w \sigma T_w^4 \quad (31.4-12)$$

where n is the refractive index of the medium next to the wall. The boundary intensity for all outgoing directions \vec{s} at the wall is given by

$$I_0 = q_{\text{out}} / \pi \quad (31.4-13)$$

Boundary Condition Treatment at Symmetry Boundaries

At symmetry boundaries, the direction of the reflected ray \vec{s}_r corresponding to the incoming direction \vec{s} is given by

$$\vec{s}_r = \vec{s} - 2(\vec{s} \cdot \vec{n})\vec{n} \quad (31.4-14)$$

Furthermore,

$$I_w(\vec{s}_r) = I_w(\vec{s}) \quad (31.4-15)$$

Boundary Condition Treatment at Flow Inlets and Exits

The net radiation heat flux at flow inlets and outlets is computed in the same manner as at walls, as described above. **Airpak** assumes that the emissivity of all flow inlets and outlets is 1.0 (black body absorption).

31.5 Optimization

Airpak uses the Dynamic-Q optimization method to solve design optimization problems. The Dynamic-Q method is an efficient constrained-optimization method consisting of applying a dynamic-trajectory-optimization algorithm to successive quadratic approximations of the actual optimization problem.

Due to its efficiency with respect to the number of function evaluations required for convergence, the Dynamic-Q method is primarily intended for optimization problems where function evaluations are computationally expensive. Such problems occur frequently in engineering applications where time consuming numerical simulations may be used for function evaluations. Among others, these numerical analysis may take the form of a computational fluid dynamics (CFD) simulation, a structural analysis by means of the finite element method (FEM), or a dynamic simulation of a multibody system. Because these simulations are usually computationally expensive to perform, and because the relevant functions may not be known analytically, standard classical optimization methods are normally not suited to these types of problems. Also the storage requirements of the Dynamic-Q method are minimal. The method is therefore particularly suitable for problems where the number of variables n is large.

In the following sections, the Dynamic-Q methodology is presented as well as the dynamic trajectory "leap-frog" algorithm, which is used for solving the quadratic subproblems.

31.5.1 The Dynamic-Q Optimization method

Consider the general nonlinear optimization problem:

$$\begin{aligned}
& \min_{\mathbf{x}} f(\mathbf{x}); \quad \mathbf{x} = (x_1, x_2, \dots, x_n) \in \mathcal{R}^n \\
& \text{subject to } g_j(\mathbf{x}) \leq 0; \quad j = 1, \dots, p \\
& h_k(\mathbf{x}) = 0; \quad k = 1, \dots, q
\end{aligned} \tag{31.5-1}$$

where $f(\mathbf{x})$, $g_j(\mathbf{x})$, and $h_k(\mathbf{x})$ are scalar functions of \mathbf{x} .

In the Dynamic-Q approach, successive subproblems $P[i]$, $i = 0, 1, 2, \dots$ are generated, at successive approximations \mathbf{x}^i to the solution \mathbf{x}^* , by constructing *spherically quadratic* approximations $\tilde{f}(\mathbf{x})$, $\tilde{g}_j(\mathbf{x})$, and $\tilde{h}_k(\mathbf{x})$ to functions $f(\mathbf{x})$, $g_j(\mathbf{x})$, and $h_k(\mathbf{x})$, respectively. These approximation functions, evaluated at a point \mathbf{x}^i , are given by

$$\begin{aligned}
\tilde{f}(\mathbf{x}) &= f(\mathbf{x}^i) + \nabla^T f(\mathbf{x}^i)(\mathbf{x} - \mathbf{x}^i) + \frac{1}{2}(\mathbf{x} - \mathbf{x}^i)^T \mathbf{A}(\mathbf{x} - \mathbf{x}^i); \\
\tilde{g}_j(\mathbf{x}) &= g_j(\mathbf{x}^i) + \nabla^T g_j(\mathbf{x}^i)(\mathbf{x} - \mathbf{x}^i) + \frac{1}{2}(\mathbf{x} - \mathbf{x}^i)^T \mathbf{B}_j(\mathbf{x} - \mathbf{x}^i); \quad j = 1, \dots, p \\
\tilde{h}_k(\mathbf{x}) &= h_k(\mathbf{x}^i) + \nabla^T h_k(\mathbf{x}^i)(\mathbf{x} - \mathbf{x}^i) + \frac{1}{2}(\mathbf{x} - \mathbf{x}^i)^T \mathbf{C}_k(\mathbf{x} - \mathbf{x}^i); \quad k = 1, \dots, q
\end{aligned} \tag{31.5-2}$$

where \mathbf{A} , \mathbf{B}_j , and \mathbf{C}_k are the Hessian matrices of the simple forms:

$$\mathbf{A} = \text{diag}(a, a, \dots, a) = a\mathbf{I}; \quad \mathbf{B}_j = b_j\mathbf{I}; \quad \mathbf{C}_k = c_k\mathbf{I} \tag{31.5-3}$$

The identical entries along the diagonal of the Hessian matrices indicate that the approximate subproblems $P[i]$ are indeed spherically quadratic.

For the first subproblem ($i = 0$), a linear approximation is formed by setting the curvatures a , b_j , and c_k to zero. Thereafter, a , b_j , and c_k are chosen so that the approximating functions (Equations 31.5-2) interpolate their corresponding actual functions at both \mathbf{x}^i and \mathbf{x}^{i-1} , which implies that for $i = 1, 2, 3, \dots$

$$\begin{aligned}
a &= \frac{2[f(\mathbf{x}^{i-1}) - f(\mathbf{x}^i) - \nabla^T f(\mathbf{x}^i)(\mathbf{x}^{i-1} - \mathbf{x}^i)]}{\|\mathbf{x}^{i-1} - \mathbf{x}^i\|^2} \\
b_j &= \frac{2[g_j(\mathbf{x}^{i-1}) - g_j(\mathbf{x}^i) - \nabla^T g_j(\mathbf{x}^i)(\mathbf{x}^{i-1} - \mathbf{x}^i)]}{\|\mathbf{x}^{i-1} - \mathbf{x}^i\|^2} \\
c_k &= \frac{2[h_k(\mathbf{x}^{i-1}) - h_k(\mathbf{x}^i) - \nabla^T h_k(\mathbf{x}^i)(\mathbf{x}^{i-1} - \mathbf{x}^i)]}{\|\mathbf{x}^{i-1} - \mathbf{x}^i\|^2}
\end{aligned} \tag{31.5-4}$$

If the gradient vectors ∇f , ∇g_j , and ∇h_k are not known analytically, they may be approximated from functional data by means of first-order forward finite differences.

The particular choice of spherically quadratic approximations in the Dynamic-Q algorithm has implications on the computational and storage requirements of the method. Since the second derivatives of the objective function and constraints are approximated using function and gradient data, the $\mathcal{O}(n^2)$ calculations and storage locations, which would usually be required for these second derivatives, are not needed. The computational and storage resources for the Dynamic-Q method are thus reduced to $\mathcal{O}(n)$. At most, $4 + p + q + r + s$ n -vectors need to be stored (where p , q , r , and s are, respectively, the number of inequality constraints, equality constraints, lower limit of variables, and upper limit of variables). These savings become significant when the number of variables becomes large. For this reason it is expected, and has also been shown by Snyman et al. [28], that the Dynamic-Q method is well suited to engineering problems, such as structural optimization problems, where a large number of variables are present.

In many optimization problems, additional simple side constraints of the form $\hat{k}_i \leq x_i \leq \check{k}_i$ occur, where constants \hat{k}_i and \check{k}_i represent, respectively, lower and upper bounds for variable x_i . Since these constraints are of a simple form (with zero curvature), they do not need to be approximated in the Dynamic-Q method. Instead, they are explicitly treated as special linear inequality constraints. Constraints corresponding to lower and upper limits are of the forms, respectively,

$$\begin{aligned} \hat{g}_l(\mathbf{x}) &= \hat{k}_{vl} - x_{vl} \leq 0; \quad l = 1, \dots, r \leq n, \\ \check{g}_m(\mathbf{x}) &= x_{wm} - \check{k}_{wm} \leq 0; \quad m = 1, \dots, s \leq n. \end{aligned} \quad (31.5-5)$$

Here, $vl \in \hat{I} = (v1, v2, \dots, vr)$, where \hat{I} is the set of r subscripts corresponding to the set of variables for which respective lower bounds \hat{k}_{vl} are prescribed, and $wm \in \check{I} = (w1, w2, \dots, ws)$, where \check{I} is the set of s subscripts corresponding to the set of variables for which respective upper bounds \check{k}_{wm} are prescribed. The subscripts vl and wm are used since, in general, there will not be exactly n lower and upper limits, i.e. usually $r \neq n$ and $s \neq n$.

In order to obtain convergence to the solution in a controlled and stable manner, move limits are placed on the variables. For each approximate subproblem $P[i]$ this move limit takes the form of an additional single inequality constraint

$$g_\delta(\mathbf{x}) = \|\mathbf{x} - \mathbf{x}^{i-1}\|^2 - \delta^2 \leq 0, \quad (31.5-6)$$

where δ is an appropriately chosen step limit and \mathbf{x}^{i-1} is the solution to the previous subproblem.

The approximate subproblem $P[i]$, constructed at x^i , to the optimization problem (Equation 31.5-1), including simple side constraints (Equation 31.5-5) and move limit (Equation 31.5-6), becomes

$$\begin{aligned}
& \min_{\mathbf{x}} \tilde{f}(\mathbf{x}); \quad \mathbf{x} = (x_1, x_2, \dots, x_n) \in \mathcal{R}^n \\
& \text{subject to } \tilde{g}_j(\mathbf{x}) \leq 0; \quad j = 1, \dots, p \\
& \quad \tilde{h}_k(\mathbf{x}) = 0; \quad k = 1, \dots, q \\
& \quad \hat{g}_l(\mathbf{x}) \leq 0; \quad l = 1, \dots, r \\
& \quad \check{g}_m(\mathbf{x}) \leq 0; \quad m = 1, \dots, s \\
& \quad g_\delta(\mathbf{x}) = \|\mathbf{x} - \mathbf{x}^{i-1}\|^2 - \delta^2 \leq 0
\end{aligned} \tag{31.5-7}$$

with a solution \mathbf{x}^{*i} .

The Dynamic-Q algorithm can now be stated as follows:

1. Choose a starting point \mathbf{x}^0 and step limit δ . Set $i := 0$.
2. Evaluate $f(\mathbf{x}^i)$, $g_j(\mathbf{x}^i)$, and $h_k(\mathbf{x}^i)$, as well as $\nabla f(\mathbf{x}^i)$, $\nabla g_j(\mathbf{x}^i)$, and $\nabla h_k(\mathbf{x}^i)$. If termination criteria are satisfied, then stop.
3. Construct a local approximation $P[i]$ to the optimization problem at \mathbf{x}^i using Equations 31.5-2, 31.5-3, and 31.5-4.
4. Solve the approximated subproblem $P[i]$ (Equations 31.5-7) using the constrained optimizer LFOPC (see Section 31.5.2) with $\mathbf{x}^0 := \mathbf{x}^i$ and get \mathbf{x}^{*i} .
5. Set $i := i + 1$, $\mathbf{x}^i := \mathbf{x}^{*(i-1)}$ and return to step 2.

31.5.2 The Dynamic-Trajectory (Leap-Frog) Optimization Method for Solving the Subproblems

In the Dynamic-Q method, generated subproblems are solved using the dynamic-trajectory ("leap-frog") method for unconstrained optimization ([25], [26]) applied to penalty function formulations for constrained optimization ([27], [28]).

In its unconstrained form, the Leap-Frog Optimizer (LFOP) determines the minimum of a function $f(\mathbf{x})$ by considering the associated dynamic problem of the motion of a particle of unit mass in an n -dimensional conservative force field. $f(\mathbf{x})$ is the potential energy of a particle at a point $\mathbf{x}(t)$ at time t . Therefore, the method requires the solution of the following equations of the motion:

$$\ddot{\mathbf{x}}(t) = -\nabla f(\mathbf{x}(t)) \tag{31.5-8}$$

with initial conditions

$$\mathbf{x}(0) = \mathbf{x}_0, \quad \dot{\mathbf{x}}(0) = \nu_0. \quad (31.5-9)$$

To explain how the dynamic-trajectory method works, consider the solution of the above problem over the time interval $[0, t]$. It follows that

$$\begin{aligned} \frac{1}{2}\|\dot{\mathbf{x}}(t)\|^2 - \frac{1}{2}\|\nu_0\|^2 &= f(\mathbf{x}_0) - f(\mathbf{x}(t)) \\ T(t) - T(0) &= f(0) - f(t), \quad \text{or} \quad f(t) + T(t) = f(0) + T(0) = K \end{aligned} \quad (31.5-10)$$

where $T(t)$ is kinetic energy of the particle at time t , and K is a constant determined by the initial values. The last expression in Equation 31.5-10 indicates that energy is conserved. It can also be seen that $\nabla f = -\nabla T$, therefore, as long as T increases, f decreases. This forms the basis of the dynamic-trajectory method.

The LFOP algorithm computes an approximation to the trajectory followed by the particle in the force field. Whenever T is increasing along the trajectory, f is decreasing and the algorithm is minimizing the function. However, whenever T is decreasing along the trajectory, the objective function (potential energy) is increasing. An interfering strategy is then applied to extract kinetic energy from the particle. The consequence of this strategy, based on an energy-conservation argument, is that a systematic reduction in the potential energy f of the particle is obtained. The particle is thus forced to follow a path to a local minimum at \mathbf{x}^* . The numerical integration of the initial value problem (Equations 31.5-8 and 31.5-9) is achieved using the "leap-frog" (Euler forward - Euler backward) method. The method contains some heuristic elements relating to time step selection and control.

The LFOP algorithm outlined above can be modified to handle constrained problems (LFOPC) by means of the penalty-function approach [27]. In particular, the penalty-function formulation for constrained-quadratic-optimization problem $P[i]$ is (by Snyman [27])

$$Q(\mathbf{x}) = f(\mathbf{x}) + \sum_{j=1}^{p+r+s+1} \alpha_j g_j^2(\mathbf{x}) + \sum_{k=1}^q \beta_k h_k^2(\mathbf{x}) \quad (31.5-11)$$

where the vector of inequality constraints functions $\mathbf{g}(\mathbf{x}) = [\tilde{\mathbf{g}}, \hat{\mathbf{g}}, \check{\mathbf{g}}, g_\delta]^T$ and $h_k(\mathbf{x}) = \tilde{h}_k(\mathbf{x})$, and

$$\alpha_j = \begin{cases} 0 & \text{if } g_j(\mathbf{x}) \leq 0 \\ \rho_j & \text{if } g_j(\mathbf{x}) > 0 \end{cases}. \quad (31.5-12)$$

For simplicity, the penalty parameters α_j and β_k usually take on the same positive value $\alpha_j = \beta_k = \mu$. It can be shown that as μ tends to infinity, the unconstrained minimum of $Q(\mathbf{x})$ yields the solution to the constrained problem 31.5-7.

The dynamic-trajectory method is applied to the penalty-function formulation of the constrained problem in the following three phases:

Phase 0: Given some starting point \mathbf{x}^0 , apply LFOP with some overall penalty parameter $\mu = \mu_0 (= 10^2)$ to $Q(\mathbf{x}, \mu_0)$ to get $\mathbf{x}^*(\mu_0)$.

Phase 1: With $\mathbf{x}^0 := \mathbf{x}^*(\mu_0)$, apply LFOP with increased overall penalty parameter $\mu = \mu_1 (= 10^4) \gg \mu_0$ to $Q(\mathbf{x}, \mu_1)$ to get $\mathbf{x}^*(\mu_1)$. Identify the set of n_a active constraints corresponding to the set of subscripts $I_a = (u1, u2, \dots, un_a)$ for which $g_{uj}(\mathbf{x}^*(\mu_1)) > 0$, $j = 1, 2, \dots, n_a$.

Phase 2: With $\mathbf{x}^0 := \mathbf{x}^*(\mu_1)$, apply LFOP to

$$\underset{\mathbf{x}}{\text{minimize}} \quad Q_a(\mathbf{x}, \mu_1) = \sum_{j=1}^{n_a} \mu_1 g_{uj}^2(\mathbf{x}) + \sum_{k=1}^q \mu_1 h_k^2(\mathbf{x})$$

to get \mathbf{x}^* .

The LFOPC algorithm possesses a number of outstanding characteristics, which makes it highly suitable for implementation in the Dynamic-Q methodology. The algorithm requires only gradient information and no explicit line searches or function evaluations are performed. These properties, together with the influence of the fundamental physical principles underlying the method, ensure that the algorithm is extremely robust. This has been proven over many years of testing by Snyman [27]. A further desirable characteristic related to its robustness, and the main reason for its application in the step 4 of the Dynamic-Q algorithm, is that if there is no feasible solution to the problem, the LFOPC algorithm will still find the best possible compromised solution without breaking down. The Dynamic-Q algorithm thus usually converges to a solution from an infeasible remote point without the need to use line searches between subproblems. The LFOPC algorithm used by Dynamic-Q is identical to that used in [27] except for a minor change to LFOP which is advisable if the subproblems become effectively unconstrained.

Given specified positive tolerances ε_x , ε_f , and ε_c , then at step i , termination of the algorithm occurs if the normalized step size $\nabla \mathbf{x}_{norm} = \frac{\|\mathbf{x}^i - \mathbf{x}^{i-1}\|}{1 + \|\mathbf{x}^i\|} < \varepsilon_x$, or if the normalized change in the function value $\nabla f_{norm} = \frac{|f^i - f_{best}|}{1 + |f_{best}|} < \varepsilon_f$, where f_{best} is the lowest previous feasible function value, and the current \mathbf{x}^i is feasible. The point \mathbf{x}^i is considered feasible if the absolute value of the violation of each constraint is less than ε_c .

31.6 Solution Procedures

31.6.1 Overview of Numerical Scheme

Airpak will solve the governing integral equations for mass and momentum, and (when appropriate) for energy and other scalars such as turbulence. A control-volume-based technique is used that consists of:

- Division of the domain into discrete control volumes using a computational grid.
- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables (“unknowns”) such as velocities, pressure, temperature, and conserved scalars.
- Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

The governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated in Figure 31.6.1 and outlined below:

1. Fluid properties are updated, based on the current solution. (If the calculation has just begun, the fluid properties will be updated based on the initialized solution.)
2. The u , v , and w momentum equations are each solved in turn using current values for pressure and face mass fluxes, in order to update the velocity field.
3. Since the velocities obtained in Step 2 may not satisfy the continuity equation locally, a “Poisson-type” equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied.
4. Where appropriate, equations for scalars such as turbulence, energy, and radiation are solved using the previously updated values of the other variables.
5. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met.

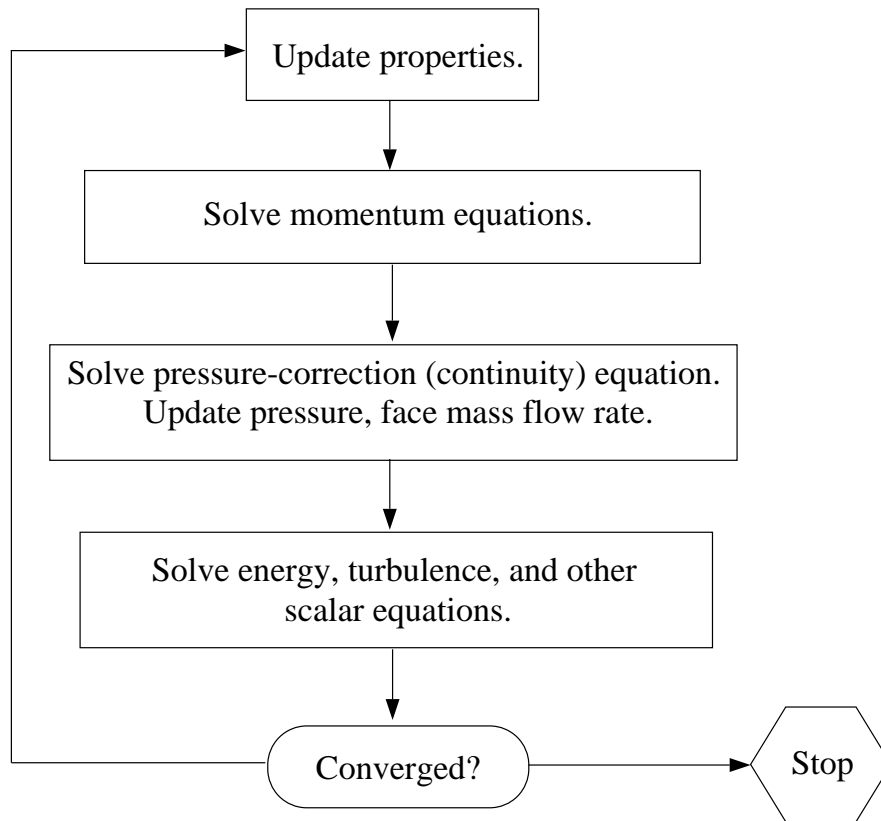


Figure 31.6.1: Overview of the Solution Method

Linearization

The discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized takes an “implicit” form with respect to the dependent variable (or set of variables) of interest. For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

This will result in a system of linear equations with one equation for each cell in the domain. Because there is only one equation per cell, this is sometimes called a “scalar” system of equations. A point implicit (Gauss-Seidel) linear equation solver is used in conjunction with an algebraic multigrid (AMG) method to solve the resultant scalar system of equations for the dependent variable in each cell. For example, the x -momentum equation is linearized to produce a system of equations in which u velocity is the un-

known. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated u -velocity field.

In summary, **Airpak** solves for a single variable field (e.g., p) by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time, and so on.

31.6.2 Spatial Discretization

Airpak uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis.

Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity ϕ . This is demonstrated by the following equation written in integral form for an arbitrary control volume V as follows:

$$\oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_{\phi} \nabla \phi \cdot d\vec{A} + \int_V S_{\phi} dV \quad (31.6-1)$$

where

ρ	=	density
\vec{v}	=	velocity vector ($= u \hat{i} + v \hat{j}$ in 2D)
\vec{A}	=	surface area vector
Γ_{ϕ}	=	diffusion coefficient for ϕ
$\nabla \phi$	=	gradient of ϕ ($= (\partial \phi / \partial x) \hat{i} + (\partial \phi / \partial y) \hat{j}$ in 2D)
S_{ϕ}	=	source of ϕ per unit volume

Equation 31.6-1 is applied to each control volume, or cell, in the computational domain. The two-dimensional, triangular cell shown in Figure 31.6.2 is an example of such a control volume. Discretization of Equation 31.6-1 on a given cell yields

$$\sum_f^{N_{\text{faces}}} \rho \vec{v}_f \phi_f \vec{A}_f = \sum_f^{N_{\text{faces}}} \Gamma_{\phi} (\nabla \phi)_n \vec{A}_f + S_{\phi} V \quad (31.6-2)$$

where

N_{faces}	=	number of faces enclosing cell
ϕ_f	=	value of ϕ convected through face f
$\rho \vec{v}_f \cdot \vec{A}_f$	=	mass flux through the face
\vec{A}_f	=	area of face f , $ A $ ($= A_x \hat{i} + A_y \hat{j} $ in 2D)
$(\nabla \phi)_n$	=	magnitude of $\nabla \phi$ normal to face f
V	=	cell volume

The equations solved by **Airpak** take the same general form as the one given above and apply readily to multi-dimensional, unstructured meshes composed of arbitrary polyhedra.

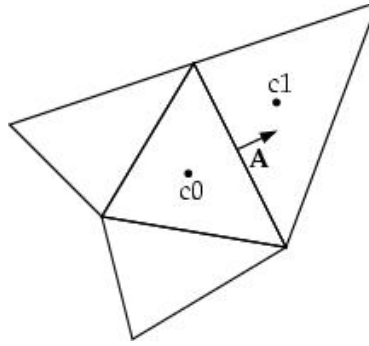


Figure 31.6.2: Control Volume Used to Illustrate Discretization of a Scalar Transport Equation

Airpak stores discrete values of the scalar ϕ at the cell centers ($c0$ and $c1$ in Figure 31.6.2). However, face values ϕ_f are required for the convection terms in Equation 31.6-2 and must be interpolated from the cell center values. This is accomplished using an upwind scheme.

Upwinding means that the face value ϕ_f is derived from quantities in the cell upstream, or “upwind,” relative to the direction of the normal velocity v_n in Equation 31.6-2. **Airpak** allows you to choose from two upwind schemes: first-order upwind, and second-order upwind. These schemes are described below.

The diffusion terms in Equation 31.6-2 are central-differenced and are always second-order accurate.

First-Order Upwind Scheme

When first-order accuracy is desired, quantities at cell faces are determined by assuming that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell; the face quantities are identical to the cell quantities. Thus when first-order upwinding is selected, the face value ϕ_f is set equal to the cell-center value of ϕ in the upstream cell.

Second-Order Upwind Scheme

When second-order accuracy is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach [2]. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. Thus when second-order upwinding is selected, the face value ϕ_f is computed using the following expression:

$$\phi_f = \phi + \nabla\phi \cdot \Delta\vec{s} \quad (31.6-3)$$

where ϕ and $\nabla\phi$ are the cell-centered value and its gradient in the upstream cell, and $\Delta\vec{s}$ is the displacement vector from the upstream cell centroid to the face centroid. This formulation requires the determination of the gradient $\nabla\phi$ in each cell. This gradient is computed using the divergence theorem, which in discrete form is written as

$$\nabla\phi = \frac{1}{V} \sum_f^{N_{\text{faces}}} \tilde{\phi}_f \vec{A} \quad (31.6-4)$$

Here the face values $\tilde{\phi}_f$ are computed by averaging ϕ from the two cells adjacent to the face. Finally, the gradient $\nabla\phi$ is limited so that no new maxima or minima are introduced.

Linearized Form of the Discrete Equation

The discretized scalar transport equation (Equation 31.6-2) contains the unknown scalar variable ϕ at the cell center as well as the unknown values in surrounding neighbor cells. This equation will, in general, be non-linear with respect to these variables. A linearized form of Equation 31.6-2 can be written as

$$a_P \phi = \sum_{nb} a_{nb} \phi_{nb} + b \quad (31.6-5)$$

where the subscript nb refers to neighbor cells, and a_P and a_{nb} are the linearized coefficients for ϕ and ϕ_{nb} .

The number of neighbors for each cell depends on the grid topology, but will typically equal the number of faces enclosing the cell (boundary cells being the exception).

Similar equations can be written for each cell in the grid. This results in a set of algebraic equations with a sparse coefficient matrix. For scalar equations, **Airpak** solves this linear system using a point implicit (Gauss-Seidel) linear equation solver in conjunction with an algebraic multigrid (AMG) method which is described in Section 31.6.4.

Under-Relaxation

Because of the nonlinearity of the equation set being solved by **Airpak**, it is necessary to control the change of ϕ . This is typically achieved by under-relaxation, which reduces the change of ϕ produced during each iteration. In a simple form, the new value of the variable ϕ within a cell depends upon the old value, ϕ_{old} , the computed change in ϕ , $\Delta\phi$, and the under-relaxation factor, α , as follows:

$$\phi = \phi_{\text{old}} + \alpha \Delta \phi \quad (31.6-6)$$

Discretization of the Momentum and Continuity Equations

In this section, special practices related to the discretization of the momentum and continuity equations and their solution are addressed. These practices are most easily described by considering the steady-state continuity and momentum equations in integral form:

$$\oint \rho \vec{v} \cdot d\vec{A} = 0 \quad (31.6-7)$$

$$\oint \rho \vec{v} \vec{v} \cdot d\vec{A} = - \oint p \mathbf{I} \cdot d\vec{A} + \oint \vec{\tau} \cdot d\vec{A} + \int_V \vec{F} dV \quad (31.6-8)$$

where \mathbf{I} is the identity matrix, $\vec{\tau}$ is the stress tensor, and \vec{F} is the force vector.

Discretization of the Momentum Equation

The discretization scheme described earlier in this section for a scalar transport equation is also used to discretize the momentum equations. For example, the x -momentum equation can be obtained by setting $\phi = u$:

$$a_P u = \sum_{nb} a_{nb} u_{nb} + \sum p_f A \cdot \hat{i} + S \quad (31.6-9)$$

If the pressure field and face mass fluxes were known, Equation 31.6-9 could be solved in the manner outlined earlier in this section, and a velocity field obtained. However, the pressure field and face mass fluxes are not known a priori and must be obtained as a part of the solution. There are important issues with respect to the storage of pressure and the discretization of the pressure gradient term; these are addressed later in this section.

Airpak uses a co-located scheme, whereby pressure and velocity are both stored at cell centers. However, Equation 31.6-9 requires the value of the pressure at the face between cells $c0$ and $c1$, shown in Figure 31.6.2. Therefore, an interpolation scheme is required to compute the face values of pressure from the cell values.

Pressure Interpolation Schemes

The default pressure interpolation scheme in **Airpak** is the standard scheme. This scheme interpolates the pressure values at the faces using momentum equation coefficients [24]. This procedure works well as long as the pressure variation between cell centers is smooth. When there are jumps or large gradients in the momentum source terms between control volumes, the pressure profile has a high gradient at the cell face, and cannot be

interpolated using this scheme. If this scheme is used, the discrepancy shows up in overshoots/undershoots of cell velocity.

Flows for which the standard pressure interpolation scheme will have trouble include flows with large body forces, such as in strongly swirling flows and in high-Rayleigh-number natural convection. In such cases, it is necessary to pack the mesh in regions of high gradient to resolve the pressure variation adequately.

Another source of error is that **Airpak** assumes that the normal pressure gradient at the wall is zero. This is valid for boundary layers, but not in the presence of body forces or curvature. Again, the failure to correctly account for the wall pressure gradient is manifested in velocity vectors pointing in/out of walls.

The other scheme available in **Airpak** is the body-force-weighted scheme. This scheme computes the face pressure by assuming that the normal acceleration of the fluid resulting from the pressure gradient and body forces is continuous across each face. This works well if the body forces are known a priori in the momentum equations (e.g., buoyancy and axisymmetric swirl calculations). This scheme is good for high-Rayleigh-number natural convection flows.

Discretization of the Continuity Equation

Equation 31.6-7 may be integrated over the control volume in Figure 31.6.2 to yield the following discrete equation

$$\sum_f^{N_{\text{faces}}} J_f A_f = 0 \quad (31.6-10)$$

where J_f is the mass flux through face f , ρv_n .

As described in Section 31.6.1, the momentum and continuity equations are solved sequentially. In this sequential procedure, the continuity equation is used as an equation for pressure. However, pressure does not appear explicitly in Equation 31.6-10 for incompressible flows, since density is not directly related to pressure. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm [22] is used for introducing pressure into the continuity equation. This procedure is outlined below.

To proceed further, it is necessary to relate the face values of velocity v_n to the stored values of velocity at the cell centers. Linear interpolation of cell-centered velocities to the face results in unphysical checker-boarding of pressure. **Airpak** uses a procedure similar to that outlined by Rhie and Chow [24] to prevent checkerboarding. The face value of velocity v_n is not averaged linearly; instead, momentum-weighted averaging, using weighting factors based on the a_P coefficient from equation 31.6-9, is performed. Using this procedure, the face flow rate J_f may be written as

$$J_f = \hat{J}_f + d_f (p_{c0} - p_{c1}) \quad (31.6-11)$$

where p_{c0} and p_{c1} are the pressures within the two cells on either side of the face, and \hat{J}_f contains the influence of velocities in these cells (see Figure 31.6.2). The term d_f is a function of \bar{a}_P , the average of the momentum equation a_P coefficients for the cells on either side of face f .

Pressure-Velocity Coupling with SIMPLE

Pressure-velocity coupling is achieved by using Equation 31.6-11 to derive an equation for pressure from the discrete continuity equation (Equation 31.6-10). Airpak uses the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) pressure-velocity coupling algorithm. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

If the momentum equation is solved with a guessed pressure field p^* , the resulting face flux J_f^* computed from Equation 31.6-11

$$J_f^* = \hat{J}_f^* + d_f (p_{c0}^* - p_{c1}^*) \quad (31.6-12)$$

does not satisfy the continuity equation. Consequently, a correction J_f' is added to the face flow rate J_f^* so that the corrected face flow rate J_f

$$J_f = J_f^* + J_f' \quad (31.6-13)$$

satisfies the continuity equation. The SIMPLE algorithm postulates that J_f' be written as

$$J_f' = d_f (p'_{c0} - p'_{c1}) \quad (31.6-14)$$

where p' is the cell pressure correction.

The SIMPLE algorithm substitutes the flux correction equations (Equations 31.6-13 and 31.6-14) into the discrete continuity equation (Equation 31.6-10) to obtain a discrete equation for the pressure correction p' in the cell:

$$a_P p' = \sum_{nb} a_{nb} p'_{nb} + b \quad (31.6-15)$$

where the source term b is the net flow rate into the cell:

$$b = \sum_f^{N_{\text{faces}}} J_f^* A_f \quad (31.6-16)$$

The pressure-correction equation (Equation 31.6-15) may be solved using the algebraic multigrid (AMG) method described in Section 31.6.4. Once a solution is obtained, the cell pressure and the face flow rate are corrected using

$$p = p^* + \alpha_p p' \quad (31.6-17)$$

$$J_f = J_f^* + d_f (p'_{c0} - p'_{c1}) \quad (31.6-18)$$

Here α_p is the under-relaxation factor for pressure (see Equation 31.6-6 and related description for information about under-relaxation). The corrected face flow rate J_f satisfies the discrete continuity equation identically during each iteration.

31.6.3 Time Discretization

In **Airpak** the time-dependent equations must be discretized in both space and time. The spatial discretization for the time-dependent equations is identical to the steady-state case (see Section 31.6.2). Temporal discretization involves the integration of every term in the differential equations over a time step Δt . The integration of the transient terms is straightforward, as shown below.

A generic expression for the time evolution of a variable ϕ is given by

$$\frac{\partial \phi}{\partial t} = F(\phi) \quad (31.6-19)$$

where the function F incorporates any spatial discretization. If the time derivative is discretized using backward differences, the first-order accurate temporal discretization is given by

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi) \quad (31.6-20)$$

where

- ϕ = a scalar quantity
- $n + 1$ = value at the next time level, $t + \Delta t$
- n = value at the current time level, t

Once the time derivative has been discretized, a choice remains for evaluating $F(\phi)$: in particular, which time level values of ϕ should be used in evaluating F ?

One method (the method used in **Airpak**) is to evaluate $F(\phi)$ at the future time level:

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi^{n+1}) \quad (31.6-21)$$

This is referred to as “implicit” integration since ϕ^{n+1} in a given cell is related to ϕ^{n+1} in neighboring cells through $F(\phi^{n+1})$:

$$\phi^{n+1} = \phi^n + \Delta t F(\phi^{n+1}) \quad (31.6-22)$$

This implicit equation can be solved iteratively by initializing ϕ^i to ϕ^n and iterating the equation

$$\phi^i = \phi^n + \Delta t F(\phi^i) \quad (31.6-23)$$

until ϕ^i stops changing (i.e., converges). At that point, ϕ^{n+1} is set to ϕ^i .

The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size.

31.6.4 Multigrid Method

This section describes the mathematical basis of the multigrid approach used in **Airpak**.

Approach

Airpak uses a multigrid scheme to accelerate the convergence of the solver by computing corrections on a series of coarse grid levels. The use of this multigrid scheme can greatly reduce the number of iterations and the CPU time required to obtain a converged solution, particularly when your model contains a large number of control volumes.

The Need for Multigrid

Implicit solution of the linearized equations on unstructured meshes is complicated by the fact that there is no equivalent of the line-iterative methods that are commonly used on structured grids. Since direct matrix inversion is out of the question for realistic problems and “whole-field” solvers that rely on conjugate-gradient (CG) methods have robustness problems associated with them, the methods of choice are point implicit solvers like Gauss-Seidel. Although the Gauss-Seidel scheme rapidly removes local (high-frequency) errors in the solution, global (low-frequency) errors are reduced at a rate inversely related to the grid size. Thus, for a large number of nodes, the solver “stalls” and the residual reduction rate becomes prohibitively low.

Multigrid techniques allow global error to be addressed by using a sequence of successively coarser meshes. This method is based upon the principle that global (low-frequency) error existing on a fine mesh can be represented on a coarse mesh where it again becomes accessible as local (high-frequency) error: because there are fewer coarse cells overall, the global corrections can be communicated more quickly between adjacent cells. Since computations can be performed at exponentially decaying expense in both CPU time and memory storage on coarser meshes, there is the potential for very efficient elimination of global error. The fine-grid relaxation scheme or “smoother”, in this case either the point-implicit Gauss-Seidel or the explicit multi-stage scheme, is not required to be particularly effective at reducing global error and can be tuned for efficient reduction of local error.

The Basic Concept in Multigrid

Consider the set of discretized linear (or linearized) equations given by

$$A \phi_e + b = 0 \quad (31.6-24)$$

where ϕ_e is the exact solution. Before the solution has converged there will be a defect d associated with the approximate solution ϕ :

$$A \phi + b = d \quad (31.6-25)$$

We seek a correction ψ to ϕ such that the exact solution is given by

$$\phi_e = \phi + \psi \quad (31.6-26)$$

Substituting Equation 31.6-26 into Equation 31.6-24 gives

$$A (\phi + \psi) + b = 0 \quad (31.6-27)$$

$$A \psi + (A \phi + b) = 0 \quad (31.6-28)$$

Now using Equations 31.6-25 and 31.6-28 we obtain

$$A \psi + d = 0 \quad (31.6-29)$$

which is an equation for the correction in terms of the original fine level operator A and the defect d . Assuming the local (high-frequency) errors have been sufficiently damped by the relaxation scheme on the fine level, the correction ψ will be smooth and therefore more effectively solved on the next coarser level.

Restriction and Prolongation

Solving for corrections on the coarse level requires transferring the defect down from the fine level (restriction), computing corrections, and then transferring the corrections back up from the coarse level (prolongation). We can write the equations for coarse level corrections ψ^H as

$$A^H \psi^H + R d = 0 \quad (31.6-30)$$

where A^H is the coarse level operator and R the restriction operator responsible for transferring the fine level defect down to the coarse level. Solution of Equation 31.6-30 is followed by an update of the fine level solution given by

$$\phi^{\text{new}} = \phi + P \psi^H \quad (31.6-31)$$

where P is the prolongation operator used to transfer the coarse level corrections up to the fine level.

Unstructured Multigrid

The primary difficulty with using multigrid on unstructured grids is the creation and use of the coarse grid hierarchy. On a structured grid, the coarse grids can be formed simply by removing every other grid line from the fine grids and the prolongation and restriction operators are simple to formulate (e.g., injection and bilinear interpolation).

Multigrid Cycles

A multigrid cycle can be defined as a recursive procedure that is applied at each grid level as it moves through the grid hierarchy. Three types of multigrid cycles are available in Airpak: the V, W, and flexible (“flex”) cycles.

The V and W Cycles

Figures 31.6.3 and 31.6.4 show the V and W multigrid cycles (defined below). In each figure, the multigrid cycle is represented by a square, and then expanded to show the individual steps that are performed within the cycle. You may want to follow along in the figures as you read the steps below.

For the V and W cycles, the traversal of the hierarchy is governed by three parameters, β_1 , β_2 , and β_3 , as follows:

1. β_1 “smoothings”, (sometimes called pre-relaxation sweeps), are performed at the current grid level to reduce the high-frequency components of the error (local error).

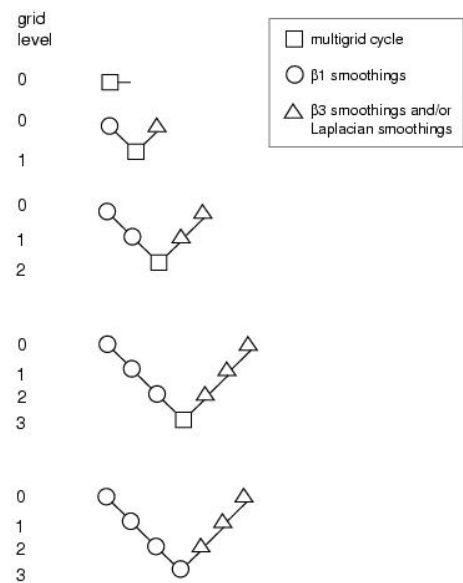


Figure 31.6.3: V-Cycle Multigrid

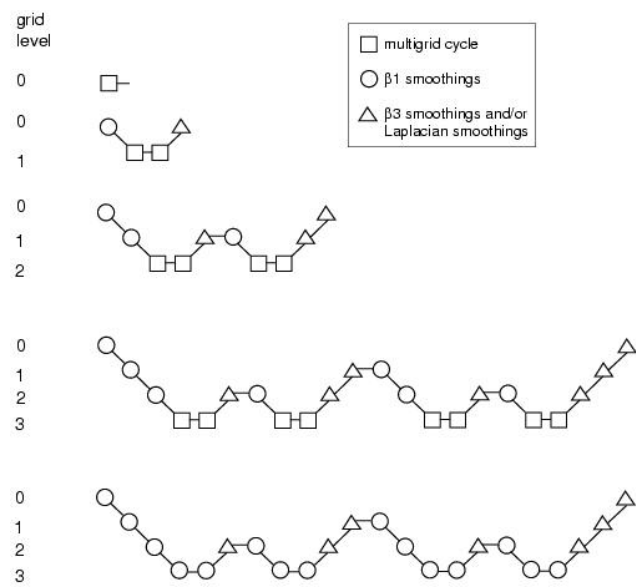


Figure 31.6.4: W-Cycle Multigrid

In Figures 31.6.3 and 31.6.4 this step is represented by a circle and marks the start of a multigrid cycle. The high-wave-number components of error should be reduced until the remaining error is expressible on the next coarser mesh without significant aliasing.

If this is the coarsest grid level, then the multigrid cycle on this level is complete. (In Figures 31.6.3 and 31.6.4 there are 3 coarse grid levels, so the square representing the multigrid cycle on level 3 is equivalent to a circle, as shown in the final diagram in each figure.)



In **Airpak**, β_1 is zero (i.e., pre-relaxation is not performed).

2. Next, the problem is “restricted” to the next coarser grid level using the appropriate restriction operator.

In Figures 31.6.3 and 31.6.4, the restriction from a finer grid level to a coarser grid level is designated by a downward-sloping line.

3. The error on the coarse grid is reduced by performing β_2 multigrid cycles (represented in Figures 31.6.3 and 31.6.4 as squares). Commonly, for fixed multigrid strategies β_2 is either 1 or 2, corresponding to V-cycle and W-cycle multigrid, respectively.
4. Next, the cumulative correction computed on the coarse grid is “interpolated” back to the fine grid using the appropriate prolongation operator and added to the fine grid solution.

In Figures 31.6.3 and 31.6.4 the prolongation is represented by an upward-sloping line.

The high-frequency error now present at the fine grid level is due to the prolongation procedure used to transfer the correction.

5. In the final step, β_3 “smoothings” (post-relaxations) are performed to remove the high-frequency error introduced on the coarse grid by the β_2 multigrid cycles.

In Figures 31.6.3 and 31.6.4, this relaxation procedure is represented by a single triangle.

The Flexible Cycle

For the flexible cycle, the calculation and use of coarse grid corrections is controlled in the multigrid procedure by the logic illustrated in Figure 31.6.5. This logic ensures that coarser grid calculations are invoked when the rate of residual reduction on the current grid level is too slow. In addition, the multigrid controls dictate when the iterative solution of the correction on the current coarse grid level is sufficiently converged and should thus be applied to the solution on the next finer grid. These two decisions are controlled by the parameters α and β shown in Figure 31.6.5, as described in detail

below. Note that the logic of the multigrid procedure is such that grid levels may be visited repeatedly during a single global iteration on an equation. For a set of 4 multigrid levels, referred to as 0, 1, 2, and 3, the flex-cycle multigrid procedure for solving a given transport equation might consist of visiting grid levels as 0-1-2-3-2-3-2-1-0-1-2-1-0, for example.

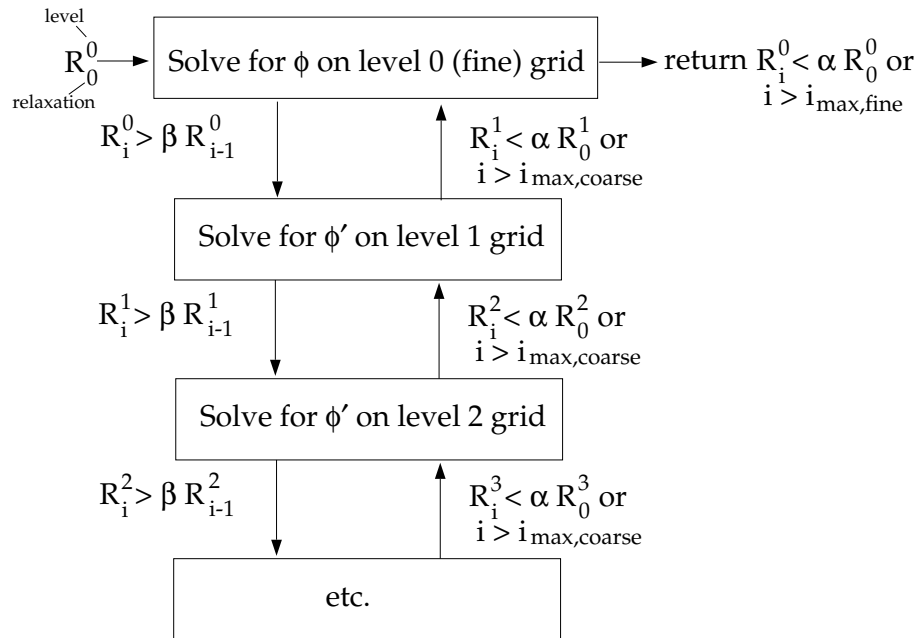


Figure 31.6.5: Logic Controlling the Flex Multigrid Cycle

The main difference between the flexible cycle and the V and W cycles is that the satisfaction of the residual reduction tolerance and termination criterion determine when and how often each level is visited in the flexible cycle, whereas in the V and W cycles the traversal pattern is explicitly defined.

The Residual Reduction Rate Criteria

The multigrid procedure invokes calculations on the next coarser grid level when the error reduction rate on the current level is insufficient, as defined by

$$R_i > \beta R_{i-1} \quad (31.6-32)$$

Here R_i is the absolute sum of residuals (defect) computed on the current grid level after the i th relaxation on this level. The above equation states that if the residual present in the iterative solution after i relaxations is greater than some fraction, β (between 0 and 1), of the residual present after the $(i - 1)$ th relaxation, the next coarser grid level should be visited. Thus β is referred to as the residual reduction tolerance, and determines when

to “give up” on the iterative solution at the current grid level and move to solving the correction equations on the next coarser grid. The value of β controls the frequency with which coarser grid levels are visited. The default value is 0.1. A larger value will result in less frequent visits, and a smaller value will result in more frequent visits.

The Termination Criteria

Provided that the residual reduction rate is sufficiently rapid, the correction equations will be converged on the current grid level and the result applied to the solution field on the next finer grid level.

The correction equations on the current grid level are considered sufficiently converged when the error in the correction solution is reduced to some fraction, α (between 0 and 1), of the original error on this grid level:

$$R_i < \alpha R_0 \quad (31.6-33)$$

Here, R_i is the residual on the current grid level after the i th iteration on this level, and R_0 is the residual that was initially obtained on this grid level at the current global iteration. The parameter α , referred to as the termination criterion, has a default value of 0.1. Note that the above equation is also used to terminate calculations on the lowest (finest) grid level during the multigrid procedure. Thus, relaxations are continued on each grid level (including the finest grid level) until the criterion of this equation is obeyed (or until a maximum number of relaxations has been completed, in the case that the specified criterion is never achieved).

Restriction, Prolongation, and Coarse-Level Operators

The multigrid algorithm in *Airpak* is referred to as an “algebraic” multigrid (AMG) scheme because, as we shall see, the coarse level equations are generated without the use of any geometry or re-discretization on the coarse levels; a feature that makes AMG particularly attractive for use on unstructured meshes. The advantage is that no coarse grids have to be constructed or stored, and no fluxes or source terms need be evaluated on the coarse levels. This approach is in contrast with FAS (sometimes called “geometric”) multigrid in which a hierarchy of meshes is required and the discretized equations are evaluated on every level. In theory, the advantage of FAS over AMG is that the former should perform better for non-linear problems since non-linearities in the system are carried down to the coarse levels through the re-discretization; when using AMG, once the system is linearized, non-linearities are not “felt” by the solver until the fine level operator is next updated.

AMG Restriction and Prolongation Operators

The restriction and prolongation operators used here are based on the additive correction (AC) strategy described for structured grids by Hutchinson and Raithby [13]. Inter-level transfer is accomplished by piecewise constant interpolation and prolongation. The defect in any coarse level cell is given by the sum of those from the fine level cells it contains, while fine level corrections are obtained by injection of coarse level values. In this manner the prolongation operator is given by the transpose of the restriction operator

$$P = R^T \quad (31.6-34)$$

The restriction operator is defined by a coarsening or “grouping” of fine level cells into coarse level ones. In this process each fine level cell is grouped with one or more of its “strongest” neighbors, with a preference given to currently ungrouped neighbors. The algorithm attempts to collect cells into groups of fixed size, typically two or four, but any number can be specified. In the context of grouping, strongest refers to the neighbor j of the current cell i for which the coefficient A_{ij} is largest. For sets of coupled equations A_{ij} is a block matrix and the measure of its magnitude is simply taken to be the magnitude of its first element. In addition, the set of coupled equations for a given cell are treated together and not divided amongst different coarse cells. This results in the same coarsening for each equation in the system.

AMG Coarse Level Operator

The coarse level operator A^H is constructed using a Galerkin approach. Here we require that the defect associated with the corrected fine level solution must vanish when transferred back to the coarse level. Therefore we may write

$$R d^{\text{new}} = 0 \quad (31.6-35)$$

Upon substituting Equations 31.6-25 and 31.6-31 for d^{new} and ϕ^{new} we have

$$\begin{aligned} R [A \phi^{\text{new}} + b] &= 0 \\ R [A (\phi + P \psi^H) + b] &= 0 \end{aligned} \quad (31.6-36)$$

Now rearranging and using Equation 31.6-25 once again gives

$$\begin{aligned} R A P \psi^H + R (A \phi + b) &= 0 \\ R A P \psi^H + R d &= 0 \end{aligned} \quad (31.6-37)$$

Comparison of Equation 31.6-37 with Equation 31.6-30 leads to the following expression for the coarse level operator:

$$A^H = R A P \quad (31.6-38)$$

The construction of coarse level operators thus reduces to a summation of diagonal and corresponding off-diagonal blocks for all fine level cells within a group to form the diagonal block of that group's coarse cell.

31.6.5 Solution Residuals

During the solution process you can monitor the convergence dynamically by checking residuals. At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. This history is also saved in the data file. The residual sum is defined below.

On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value (“round-off”) and then stop changing (“level out”). For “single precision” computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round-off. Double precision residuals can drop up to twelve orders of magnitude. Guidelines for judging convergence can be found in Section 27.12.5.

After discretization, the conservation equation for a general variable ϕ at a cell P can be written as

$$a_P \phi_P = \sum_{nb} a_{nb} \phi_{nb} + b \quad (31.6-39)$$

Here a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, and b is the contribution of the constant part of the source term S_c in $S = S_c + S_P \phi$ and of the boundary conditions. In Equation 31.6-39,

$$a_P = \sum_{nb} a_{nb} - S_P \quad (31.6-40)$$

The residual R^ϕ computed by **Airpak** is the imbalance in Equation 31.6-39 summed over all the computational cells P . This is referred to as the “unscaled” residual. It may be written as

$$R^\phi = \sum_{\text{cells } P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - a_P \phi_P \right| \quad (31.6-41)$$

In general, it is difficult to judge convergence by examining the residuals defined by Equation 31.6-41 since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual. **Airpak** scales the residual using a scaling factor representative of the flow rate of ϕ through the domain. This “scaled” residual is defined as

$$R^\phi = \frac{\sum_{\text{cells } P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - a_P \phi_P \right|}{\sum_{\text{cells } P} |a_P \phi_P|} \quad (31.6-42)$$

For the momentum equations the denominator term $a_P \phi_P$ is replaced by $a_P v_P$, where v_P is the magnitude of the velocity at cell P .

The scaled residual is a more appropriate indicator of convergence, and is the residual displayed by **Airpak**.

For the continuity equation, the unscaled residual is defined as

$$R^c = \sum_{\text{cells } P} |\text{rate of mass creation in cell } P| \quad (31.6-43)$$

The scaled residual for the continuity equation is defined as

$$\frac{R_{\text{iteration } N}^c}{R_{\text{iteration } 5}^c} \quad (31.6-44)$$

The denominator is the largest absolute value of the continuity residual in the first five iterations.

Bibliography

- [1] ASHRAE. *ASHRAE Handbook – 1997 Fundamentals*. American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Atlanta, Georgia, 1997.
- [2] T. J. Barth and D. Jespersen. The design and application of upwind schemes on unstructured meshes. Technical Report AIAA-89-0366, AIAA 27th Aerospace Sciences Meeting, Reno, Nevada, 1989.
- [3] G. K. Batchelor. *An Introduction to Fluid Dynamics*. Cambridge Univ. Press, Cambridge, England, 1967.
- [4] H. C. Chen and V. C. Patel. Near-Wall Turbulence Models for Complex Flows Including Separation. *AIAA Journal*, 26(6):641–648, 1988.
- [5] Q. Chen and J. Srebric. Simplified Diffuser Boundary Conditions for Numerical Room Airflow Models. ASHRAE RP-1009, 2000.
- [6] D. Choudhury. Introduction to the Renormalization Group Method and Turbulence Modeling. Fluent Inc. Technical Memorandum TM-107, 1993.
- [7] E. H. Chui and G. D. Raithby. Computation of Radiant Heat Transfer on a Non-Orthogonal Mesh Using the Finite-Volume Method. *Numerical Heat Transfer, Part B*, 23:269–288, 1993.
- [8] M. F. Cohen and D. P. Greenberg. The hemi-cube: a radiosity solution for complex environments. *Computer Graphics*, 19(3):31–40, 1985.
- [9] J. Dacles-Mariani, G. G. Zilliac, J. S. Chow, and P. Bradshaw. Numerical/Experimental Study of a Wingtip Vortex in the Near Field. *AIAA Journal*, 33(9):1561–1568, 1995.
- [10] R. A. W. M. Henkes, F. F. van der Flugt, and C. J. Hoogendoorn. Natural Convection Flow in a Square Cavity Calculated with Low-Reynolds-Number Turbulence Models. *Int. J. Heat Mass Transfer*, 34:1543–1557, 1991.
- [11] J.O. Hinze. *Turbulence*. McGraw-Hill Publishing Co., New York, 1975.
- [12] P. Huang, P. Bradshaw, and T. Coakley. Skin Friction and Velocity Profile Family for Compressible Turbulent Boundary Layers. *AIAA Journal*, 31(9):1600–1604, September 1993.

- [13] B. R. Hutchinson and G. D. Raithby. A Multigrid Method Based on the Additive Correction Strategy. *Numerical Heat Transfer*, 9:511–537, 1986.
- [14] I. E. Idelchick. *Handbook of Hydraulic Resistances*. Hemisphere Publishing Corp., 2nd edition, 1986.
- [15] T. Jongen. *Simulation and Modeling of Turbulent Incompressible Flows*. PhD thesis, EPF Lausanne, Lausanne, Switzerland, 1992.
- [16] B. Kader. Temperature and Concentration Profiles in Fully Turbulent Boundary Layers. *Int. J. Heat Mass Transfer*, 24(9):1541–1544, 1993.
- [17] G. Karypis and V. Kumar. METIS - A Software Package for Partitioning Unstructured Graphs, Partitioning Meshes, and Computing Fill-Reducing Orderings of Sparse Matrices, Version 3.0. Manual, University of Minnesota and Army HPC Research Center, 1997.
- [18] W. M. Kays. Turbulent Prandtl Number - Where Are We? *J. Heat Transfer*, 116:284–295, 1994.
- [19] B. E. Launder and D. B. Spalding. *Lectures in Mathematical Models of Turbulence*. Academic Press, London, England, 1972.
- [20] M. F. Modest. *Radiative Heat Transfer*. Series in Mechanical Engineering. McGraw Hill, 1993.
- [21] J. Y. Murthy and S. R. Mathur. A Finite Volume Method For Radiative Heat Transfer Using Unstructured Meshes. AIAA-98-0860, January 1998.
- [22] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Hemisphere, Washington, D.C., 1980.
- [23] G. D. Raithby and E. H. Chui. A Finite-Volume Method for Predicting a Radiant Heat Transfer in Enclosures with Participating Media. *J. Heat Transfer*, 112:415–423, 1990.
- [24] C. M. Rhie and W. L. Chow. Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation. *AIAA Journal*, 21(11):1525–1532, November 1983.
- [25] J. A. Snyman. A New and Dynamic Method for Unconstrained Minimization. *Applied Mathematical Modeling*, 6:449–462, 1982.
- [26] J. A. Snyman. An improved Version of the Original Leap-Frog Method for Unconstrained Minimization. *Applied Mathematical Modeling*, 7:216–218, 1983.
- [27] J. A. Snyman. The LFOPC Leap-Frog Algorithm for Constrained Optimization. *Computers and Mathematics with Applications*, 40:1085–1096, 2000.

- [28] J. A. Snyman, W. J. Roux, and N. Stander. A Dynamic Penalty Function Method for the Solution of Structural Optimization Problems. *Applied Mathematical Modeling*, 18:453–460, 1994.
- [29] P. Spalart and S. Allmaras. A one-equation turbulence model for aerodynamic flows. Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics, 1992.
- [30] F. White and G. Christoph. A Simple New Analysis of Compressible Turbulent Skin Friction Under Arbitrary Conditions. Technical Report AFFDL-TR-70-133, February 1971.
- [31] M. Wolfstein. The Velocity and Temperature Distribution of One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient. *Int. J. Heat Mass Transfer*, 12:301–318, 1969.
- [32] V. Yakhot and S. A. Orszag. Renormalization Group Analysis of Turbulence: I. Basic Theory. *Journal of Scientific Computing*, 1(1):1–51, 1986.

Index

- 1/4 Polygonal cylinder panel, 25-51
- 1/4 polygonal cylinder macro, 25-51
- Accept button, 2-42
- accuracy
 - diffusion terms, 31-39
 - first-order, 27-5, 31-39
 - second-order, 27-5, 31-39
- Active, 7-43
- Active parameter panel, 24-5
- activity level, 30-8
- Add marker panel, 2-8
- adding objects to a group, 7-69
- Adobe Acrobat Reader, 1-23
- ADPI button, 29-20
- ADPI reports, 29-20
- ADPI specification panel, 29-20
- Advanced parameters button, 26-15
- Advanced solver setup panel, 27-4
- air inlet diffuser, 25-10
- Airpak environment variables, 1-16
- airpak, 1-11
- airpak -unpack, 1-15
- airpak -x, 1-15
- airpak -xfast, 1-15
- airpak projectname, 1-15
- AIRPAK_JOB_DIRECTORY, 1-16
- AIRPAK_LIB_PATH, 1-16
- AIRPAK_LICENSE_FILE, 1-16
- algebraic multigrid (AMG), 31-40, 31-45
- Align and morph edges button, 2-31
- Align and morph faces button, 2-31
- Align and morph vertices button, 2-31
- Align centers button, 7-36
- Align edges button, 7-33
- Align face centers button, 2-31, 7-36
- Align faces button, 7-32
- Align object centers button, 2-31
- Align vertices button, 7-35
- All projects button, 6-12, 6-19
- ambient values, 6-39
- angular deviation, 29-3, 30-2
- Angular deviation panel, 30-2
- angular discretization, 31-26
- animated GIF files, 28-17, 28-24
- animation
 - isosurface, 28-22
 - particle traces, 28-41
 - plane cut, 28-15
 - saving, 28-16, 28-23
 - transient simulations, 20-21
- Anisotropic tensor panel, 7-78
- annotations, 2-36, 6-15
- Annotations panel, 2-36
- applications, 1-8
- Assemblies, 7-90
- Assemblies panel, 7-91
- assembly, *see also* object, 7-90
 - adding objects to, 7-91
 - copying, 7-96
 - creating, 7-90
 - using a group, 7-73
 - deleting, 7-98
 - editing, 7-91, 7-96
 - expanding into its components, 7-98
 - external, 7-92
 - internal, 7-93
 - loading, 7-97
 - merging, 7-97
 - meshing, 26-29
 - moving, 7-97
 - saving, 7-97
 - selecting, 7-96

- summary, 7-98
- total area, 7-99
- total volume, 7-99
- translating, 7-93
- viewing, 7-96
- Assembly contents panel, 7-98
- Assembly node, 2-57
- atmospheric boundary layer macro, 25-4, 25-6
 - northward direction, 6-39, 25-6
 - wind direction, 25-6
 - wind speed, 25-8
- Atmospheric boundary layer panel, 25-6
- augmentation efficiency, 21-9
- automatic mesh generation, 26-1
- Autoscale button, 7-9
- Available toolbars panel, 2-47
- background color, 2-9
- baffles, 10-2
- band width for contour plots, 28-33
- Basic parameters panel, 6-27
- Basic settings panel, 27-7
 - Reset button, 6-39, 27-5, 27-8
 - transient simulations, 20-11
- batch execution, 27-26
- batch file, 27-26
- block, *see also* object, 8-1
 - additional resistance, 8-13
 - combination, 8-3, 8-8
 - coincident surfaces, 8-3
 - intersecting volumes, 8-4
 - creating, 8-9
 - fixed heat, 8-3
 - fixed temperature, 8-3
 - fluid, 8-2, 8-14
 - hollow, 8-2, 8-16
 - individual sides, 8-3, 8-11
 - meshing, 26-21
 - 3D polygon, 26-22
 - cylinder, 26-22
 - ellipsoid, 26-23
 - elliptical cylinder, 26-23
 - prism, 26-21
 - on external wall, 8-7
 - partition
 - intersecting volume, 8-7
 - surface coincident, 8-6
 - radiation, 8-11, 22-2
 - emissivity, 22-7
 - specification, 22-5
 - solid, 8-2, 8-14
 - species, 21-6
 - surface roughness, 8-2
 - thermal properties, 8-2, 8-3, 8-13, 8-14
 - transient simulations, 8-16, 20-9
 - type, 8-2
- Block thermal conditions panel, 8-16, 8-17
- Blocks panel, 8-9
- blower, *see also* object
 - operating point, 29-19
- body forces, 31-42
- boundary conditions
 - parameterization, 24-1
 - radiation
 - discrete ordinates (DO) model, 31-29, 31-30
 - radiative heat transfer, 13-7
 - solar flux, 25-2
 - species, 21-6, 21-8
 - turbulence
 - input methods, 6-33
- boundary layer, 25-4
- boundary profiles, 11-11, 13-19
 - species mass fraction, 21-11
- Boussinesq approximation, 6-36, 31-22
- Boussinesq hypothesis, 31-7
- box method, 25-11
- Box method parameters panel, 25-44
- browser, 1-18
- bubble help, 2-49, 6-17
- building a model, 7-1
- buoyancy forces, 31-21
- CAD data operation options panel, 4-3, 4-16
- CAD data panel, 4-3
- CAD geometry, 4-1
- calculating a solution, 27-1, 27-49

- in batch mode, 27-26
 - using RSF, 27-29
- Cancel button, 2-43
- capabilities, 1-4
 - boundary conditions, 1-7
 - general, 1-4
 - materials, 1-6
 - meshing, 1-5
 - model building, 1-4
 - physical models, 1-6
 - reporting, 1-8
 - solver, 1-7
 - visualization, 1-7
- case files, 3-3, 27-20
- ceiling diffuser, 25-12
- cell values, 31-39
- check box, 2-43
- Check model button, 2-27
- checking the design, 7-99
- checking the mesh, 26-39
 - element quality, 26-39
 - element volume, 26-42
 - face alignment, 26-41
- circular object, 7-47
 - meshing, 26-24
- Clean up project data panel, 3-17
- cleaning up project data, 3-16
- closed box macro, 25-48
- Closed box panel, 25-48
- clothing, 30-7
- co-located scheme, 31-41
- coarse mesh
 - hexahedral, 26-5
 - tetrahedral, 26-7
- coarsening the mesh, 26-14
- cocooning, 26-2, 26-12
- color, 6-20
 - contour plot, 28-33
 - graphical displays, 28-5
 - group, 7-68
 - object, 7-43
 - particle trace, 28-43
 - room, 7-13
 - vector plot, 28-37
- color legend
 - data format, 6-12
 - moving, 2-61
- color levels
 - contour plot, 28-34
- color spectrum, 28-5
 - changing, 2-61
 - contour plot, 28-34
 - data format, 6-12
- comfort level, 6-30
 - variables, 30-6
 - postprocessing, 6-32
- Comfort level button, 30-7
- Comfort level panel, 30-7
- compass orientation of model, 6-39
- compressing project data, 3-16
- compute nodes, 27-21
- computing view factors, 22-10
 - adaptive method, 22-11
 - hemicube method, 22-11
- condenser, 17-1
- conduction, 31-2
 - transient simulations, 20-1
- configuring a project, 6-11
- conjugate-gradient methods, 31-45
- conservation equations
 - discretization, 31-38
 - integral form, 31-41
- Contents button, 1-20
- context-specific help, 2-49
- continuity equation, 31-1
- contours, 28-31
- control panels, 2-42
- control volume technique, 31-36
- convection, 31-21
 - forced, 6-36
 - natural, 6-36
 - transient simulations, 20-1
- conventions used in this manual, UTM-5
- convergence, 31-53
 - criteria, 27-7, 31-36
 - history, 27-49

- judging, 27-52, 31-53
- monitoring, 27-51
- conversion factors
 - units, 5-7
- coordinate axes
 - moving, 2-61
- coordinate systems, 7-28
- Copy assembly panel, 7-96
- Copy from button, 7-41
- Copy group panel, 7-71
- Copy object button, 2-31, 7-38, 7-82, 7-96
- Copy object panel, 7-38
- copying
 - assembly, 7-96
 - group, 7-71
 - materials, 7-81
 - mesh data, 3-10
 - meshing parameters, 26-19
 - object, 7-38, 7-41
 - solution data, 3-10
- Create assemblies button, 2-30, 7-91
- Create blocks button, 2-29, 8-9
- Create fans button, 2-29, 9-11
- Create heat exchangers button, 2-30, 17-3
- Create hoods button, 2-30, 18-3
- Create image file button, 2-25, 6-4
- Create materials button, 2-30, 7-82
- Create openings button, 2-29, 11-6
- Create partitions button, 2-29, 14-4
- Create persons button, 2-29, 12-3
- Create resistances button, 2-30, 16-4
- Create sources button, 2-29, 15-3
- Create vents button, 2-29, 10-6
- Create walls button, 2-29, 13-10
- Create wires button, 2-30, 19-1
- creating
 - assembly, 7-90
 - block, 8-9
 - fan, 9-11
 - group, 7-67
 - heat exchanger, 17-3
 - hood, 18-3
 - isosurface, 28-22
 - local coordinate system, 7-28
 - materials, 7-82
 - named point, 20-25
 - object, 7-19
 - object face, 28-11
 - opening, 11-6
 - partition, 14-4
 - person, 12-3
 - plane cut, 28-15
 - point, 28-29
 - project, 6-7
 - resistance, 16-4
 - room, 7-4
 - source, 15-3
 - vent, 10-6
 - wall, 13-10
 - wire, 19-1
- cross-section
 - displaying results, 28-12
 - displaying the mesh, 26-36
- CSV/Excel files, 4-20, 4-25, 4-28
- Curve specification panel, 7-88, 9-15, 10-15, 11-11, 13-19, 20-17, 21-11
- custom assembly, 7-90
- customizing
 - materials, 7-75, 7-83
 - mesh parameters, 26-1
 - units, 5-4
- cylinder, 7-56
 - elliptical, 7-59
 - meshing, 26-23
 - meshing, 26-13, 26-16, 26-22
- cylinder plate macro, 25-53
- cylinder plate panel, 25-53
- cylindrical enclosure macro, 25-55
- Cylindrical enclosure panel, 25-55
- data files, 3-4
- date, 2-61
- default materials, 6-40
- Define point report panel, 29-13
- Define summary report panel, 27-13, 29-9
- defining
 - isosurface, 28-21

- object face, 28-9
- plane cut, 28-12
- point, 28-27
- postprocessing objects, 27-13
- project, 6-1
- summary report, 27-13, 29-11
- Delete object button, 2-30, 7-84, 7-98, 28-8
- deleting
 - assembly, 7-98
 - CAD geometry, 4-18
 - group, 7-72
 - isosurface, 28-26
 - local coordinate system, 7-30
 - materials, 7-84
 - named point, 20-26
 - object, 7-20
 - object face, 28-12
 - parameters, 24-10
 - plane cut, 28-19
 - point, 28-29
 - postprocessing objects, 28-7
 - project, 6-10
- density, 7-77, 7-79, 21-2
 - incompressible ideal gas law, 31-22
- derived variables, 29-1, 30-1
- deselecting
 - object, 7-19
- design
 - checking, 7-99
 - optimization, 24-1
- design variable
 - creating a new design variable, 24-9
- diagnostic
 - files, 3-4
 - tools, 27-52
- diffuser macro, 25-10
 - box method, 25-11
 - modeling methods, 25-10
 - momentum method, 25-11
 - types, 25-12
 - using, 25-12
- Diffuser panel, 25-10
- diffusion coefficient, 21-2, 31-3
- diffusion species, 31-3
- diffusivity, 7-79
- discrete ordinates (DO) radiation model, 31-26
 - angular discretization, 31-26
 - boundary conditions
 - flow inlets and exits, 31-30
 - symmetry boundaries, 31-30
 - walls, 31-29
 - pixelation, 31-26
 - using, 22-14
- discrete values
 - storage points, 31-39
- discretization, 27-4, 31-38, 31-44
 - first-order scheme, 31-39
 - pressure interpolation schemes, 27-5, 31-41
 - second-order scheme, 31-39
 - spatial, 31-38
 - temporal, 31-44
- displacement diffuser, 25-12
- DISPLAY, 1-16
- Display object names button, 2-26
- displaying
 - mesh, 26-32
 - cross-section, 26-36
 - individual objects, 26-32
 - residuals, 27-12
 - results, 28-1
 - view factors, 22-11
- divergence, 27-5
- DO radiation model, 31-26
- documentation, 1-17
- double precision, 31-53
- double-precision solvers, 27-7
- Download RSF project button, 2-28
- drop-down list, 2-45
- DWG files, 4-20, 4-23
- DXF files, 4-20, 4-21
- dynamic head, 17-1
- dynamic-q optimization method, 23-1, 31-30
- Edit button, 7-4

- Edit commands toolbar, 2-25
- Edit menu, 2-5
- Edit object button, 2-30
- Edit units button, 5-9
- Edit window, 1-13, 2-38
- Edit window
 - object, 7-14
 - room, 7-4
- Edit/Annotations, 2-6
- Edit/Current group, 2-6, 7-65
- Edit/Current group/Activate all, 2-7, 7-73
- Edit/Current group/Add to group, 2-6, 7-69
- Edit/Current group/Copy, 2-7, 7-71
- Edit/Current group/Copy params, 2-7, 7-71
- Edit/Current group/Create, 2-6, 7-67
- Edit/Current group/Create assembly, 2-7, 7-73, 7-90
- Edit/Current group/Deactivate all, 2-7, 7-73
- Edit/Current group/Delete, 2-6, 7-72
- Edit/Current group/Delete all, 2-7, 7-73
- Edit/Current group/Edit, 2-7, 7-72
- Edit/Current group/Move, 2-7
- Edit/Current group/Remove from group, 2-6, 7-70
- Edit/Current group/Rename, 2-6, 7-67
- Edit/Current group/Save as project, 2-7, 7-73
- Edit/Current object, 2-6
- Edit/Current object/Active, 2-6
- Edit/Current object/Copy, 2-6
- Edit/Current object/Copy from, 2-6
- Edit/Current object/Delete, 2-6
- Edit/Current object/Edit, 2-6
- Edit/Current object/Modify, 2-6
- Edit/Current object/Move, 2-6
- Edit/Current object/Remove from group, 2-6
- Edit/Current object/Reset, 2-6
- Edit/Preferences, 2-6
- Edit/Redo, 2-6
- Edit/Undo, 2-6
- editing
 - assembly, 7-91, 7-96
 - interactively, 6-23
 - isosurface, 28-22
 - local coordinate system, 7-29
 - materials, 7-75
 - named point, 20-25
 - object, 7-19
 - object face, 28-11
 - plane cut, 28-15
 - point, 28-29
 - postprocessing objects, 28-7
 - view factors, 22-13
- element
 - count, 26-18
 - height, 26-18
 - quality, 26-39
 - volume, 26-42
- ellipsoid, 7-58
 - meshing, 26-5, 26-23
- elliptical cylinder, 7-59
 - meshing, 26-5, 26-23
- Email Project panel, 27-53
- emailing a project, 27-53
- emissivity, 7-80, 22-2, 22-14
- enclosure, *see also* object
- energy equation, 31-2
- environment variables, 1-15
 - Airpak, 1-16
 - system, 1-15
- Epsilon, 29-3
- error reduction, 31-45, 31-46
 - rate, 31-50
- examining
 - mesh, 26-32, 26-39
 - hexahedral, 26-5
 - tetrahedral, 26-7
 - results, 28-1
- exhaust hood, 18-1
- Exhaust species concentration panel, 25-45
- exiting, 2-64
- exporting
 - CAD geometry, 4-1, 4-28
 - CSV/Excel files, 4-28
 - IGES files, 4-28
 - mesh, 26-44
 - STEP files, 4-28

- tetin files, 4-28
- exporting solution data, 27-21
- external assembly, 7-92
- F1 key, 2-49
- F9 key, 2-62
- face alignment, 26-41
- face flow rate, 31-42
- families, 4-6
 - converting into Airpak objects, 4-10
 - moving CAD geometry between, 4-7
 - visibility, 4-17
- fan, *see also* object, 9-1
 - characteristic curve, 9-1, 9-9, 9-14, 9-15
 - circular, 9-3
 - combination
 - parallel, 9-6
 - series, 9-6
 - creating, 9-11
 - exhaust, 9-2, 9-4, 9-9, 9-12
 - fixed flow, 9-1, 9-8
 - flow, 9-1
 - flow direction, 9-4, 9-12
 - flow rate, 9-12
 - hub, 9-1, 9-3
 - intake, 9-2, 9-4, 9-9, 9-12
 - internal, 9-2, 9-4, 9-9, 9-13
 - located on a block, 9-7
 - meshing, 26-24
 - operating point, 29-19
 - pressure drop, 9-9
 - rectangular, 9-3
 - RPM, 9-8
 - species, 21-6
 - static pressure, 9-9
 - swirl magnitude, 9-8
 - transient simulations, 9-2, 9-14, 20-9
 - turbulence parameters, 9-14
- Fan curve window, 9-14
- Fans panel, 9-11
- File commands toolbar, 2-24, 6-3
- File menu, 2-2, 6-1
- File selection dialog box, 2-39
- File/Cleanup, 2-5, 3-17, 6-3
- File/Command prompt, 2-5, 6-3
- File/Create image file, 2-5, 6-3
- File/Email project, 2-5, 6-3, 27-53
- File/Export, 2-4, 6-2
- File/Import, 2-4, 4-1, 6-2
- File/Merge project, 2-4, 3-5, 6-2
- File/New project, 2-3, 6-1, 6-7
- File/Open project, 2-4, 6-1, 6-8
- File/Pack, 27-53
- File/Pack project, 2-5, 3-16, 6-2
- File/Print screen, 2-5, 6-3
- File/Quit, 2-5, 6-3
- File/Reload main version, 2-4, 6-2, 6-9
- File/Save project, 2-4, 3-9, 6-2
- File/Save project as, 2-4, 3-9, 6-2
- File/Shell window, 2-5, 6-3
- File/Unpack project, 2-4, 3-16, 6-2
- files
 - Airpak, 3-3
 - AVI, 28-17, 28-24
 - batch, 27-26
 - case, 3-3, 27-20
 - CSV/Excel, 4-20, 4-25, 4-28
 - data, 3-4
 - diagnostic, 3-4
 - DWG, 4-20, 4-23
 - DXF, 4-20, 4-21
 - exporting, 4-1, 4-28
 - Flash, 28-18, 28-25
 - FLI, 28-17, 28-24
 - GIF
 - animated, 28-17, 28-24
 - grid, 3-3
 - HTML, 1-18
 - IFC, 4-23
 - IGES, 4-2, 4-20, 4-21
 - image, 3-10
 - format, 3-14
 - frame, 3-15
 - label, 3-15
 - landscape, 3-12
 - portrait, 3-12
 - PostScript options, 3-14

- scaling, 3-12
- importing, 4-1
- job, 3-3
- log, 2-38
- mesh, 3-3
- model, 3-3
- MPEG, 28-17, 28-24
- node, 27-23
- opening, 2-39
- optimization, 3-4
 - input, 3-4
 - log, 3-4
 - output, 3-4
 - postprocessing, 3-4
 - tab, 3-4
- overview, 3-1
- PDF, 1-23
- postprocessing, 3-4
- PostScript, 3-14, 6-16
- problem, 3-3
- problem setup, 3-3
- reading, 2-39
- residual, 3-4, 27-53
- saving, 2-39
 - transient simulations, 20-4
- scratch, 3-18
- script, 3-4, 27-20
- selecting, 2-39
- solution, 3-18
- solver, 3-3
 - input, 3-4
 - output, 3-4
- STEP, 4-2
- tet, 4-2, 4-3
- version, 3-18
- writing, 2-39
- filter efficiency, 21-9
- Find in tree panel, 2-16
- finite-volume scheme, 31-38
- first-order accuracy, 27-5, 31-39
- Fix shapes, 12-5, 18-3
- Fix values, 5-6
- flex-cycle multigrid, 31-49
- floating toolbars, 2-47
- flow
 - heat, 29-3, 30-3
 - mass, 29-3, 30-2
 - radiative heat, 29-3
 - regime, 6-32
 - variables, 6-29
 - postprocessing, 6-31
 - volume, 29-4, 30-2
- Flow dependent heat transfer panel, 13-16
- FLUENT, 1-1, 1-7
- fluid material
 - default, 6-40
 - properties, 7-78
- flux
 - heat, 29-3, 30-3
 - radiation, 22-2
- font, 6-22
- forced convection, 6-29, 6-36
- forces
 - inertial, 6-39
 - viscous, 6-39
- Form factors panel, 22-8
- Four viewing windows button, 2-26
- fraction menu, 21-5
- frame
 - image, 3-15
- Full report panel, 29-15
- gas law, 31-22
- Gauss-Seidel method, 31-45
- Generate mesh button, 2-27, 5-2, 26-8, 26-10
- geometric transformations
 - for merging model data, 3-6
 - object, 7-22, 7-39
 - room, 7-8, 7-10
- geometry
 - object, 7-45
 - parameterization, 24-1
- governing equations, 31-1
 - discretization, 31-38
 - integral form, 31-41
- Graph editor, 7-86, 9-14, 10-13, 20-15
- graphical displays, 28-5

- color, 28-5
- graphical styles
 - editing, 6-20
- graphical user interface (GUI), 2-1
 - configuring, 6-11
- graphics display and control window, 2-34
- graphics driver, 1-15
- Graphics file options panel, 3-14
- graphics tools for postprocessing, 28-1
- Graphics window, 1-13
- graphics window, 2-33
 - adding annotations, 2-36
 - adding arrows, 2-36
 - adding lines, 2-37
 - adding markers, 2-8, 2-37
 - adding text, 2-37
 - background color, 2-9
 - isometric view, 2-14, 2-27
 - lighting options, 2-12
 - scale to fit, 2-14, 2-26
 - views, 2-13
 - zooming, 2-14, 2-26, 2-64
- Grashof number, 31-21
- gravity, 6-37
- grid, *see also* mesh, 26-1
 - files, 3-3
 - storage points, 31-39
- grille diffuser, 25-12
- group, 7-65
 - activating, 7-73
 - adding objects, 7-69
 - color, 7-68
 - copying, 7-71
 - creating, 7-67
 - assembly, 7-73, 7-90
 - from imported IGES geometry, 4-16
 - deactivating, 7-73
 - deleting, 7-72
 - displaying results, 28-9
 - editing object properties, 7-72
 - including objects, 7-43
 - line width, 7-68
 - moving, 7-72
 - removing objects, 7-70
 - renaming, 7-67
 - saving, 7-73
 - shading, 7-68
 - texture, 7-69
 - transparency, 7-69
- Group parameters panel, 7-67
- Groups node, 2-51, 6-6
- GUI, 2-1
 - configuring, 6-11
- heat exchanger, *see also* object, 17-1
 - creating, 17-3
 - heat transfer, 17-2
 - heat transfer coefficient, 17-2
 - meshing, 26-29
 - pressure drop, 17-1
- Heat exchangers panel, 17-3
- heat flow, 29-3, 30-3
- heat flux, 29-3, 30-3
 - radiation, 22-2
- heat sink, *see also* object
- heat sources, 17-1, 31-2
- Heat tr. coeff parameters panel, 30-4
- Heat tr. coeff, 29-4, 30-4
- heat transfer, 22-1, 31-2
 - conducting solids, 31-2
- heat transfer coefficient, 17-2, 29-4, 30-4
- heaters, 11-1
- help, 1-17, 2-48
 - bubble, 2-49
 - context-specific, 2-49
 - F1 key, 2-49
 - on-line, 2-48
- Help menu, 2-24
- Help/Airpak on the Web, 2-24
- Help/Help, 2-24
- Help/List shortcuts, 2-24
- Help/User Services Center, 2-24
- hemisphere macro, 25-63
- Hemisphere panel, 25-63
- hex-dominant mesh, 26-4
- hexahedral mesh, *see also* mesh, 26-4
- hidden line removal, 2-10

- high end, 26-19
- high side, 26-19
- High side surface properties panel, 14-12
- History plot button, 2-28, 20-23, 28-4
- history plot for transient simulations, 20-22
- History plot panel, 20-23
- HOME, 1-16
- Home position button, 2-26
- hood, 18-1
 - creating, 18-3
 - exhaust, 18-1
 - Fix shapes option, 18-3
 - flow rate, 18-3, 18-7
 - meshing, 26-29
- Hoods panel, 18-3
- Hoods shape data panel, 18-5
- HTML files, 1-18
- HTML report, 29-5
- HTML report panel, 29-5
- hub, 9-3
- humidity, 29-3, 30-6, 30-8
- humidity ratio, 21-5
- hydraulic diameter, 6-35
- I-deas, 27-21
- IAQ, 6-30
 - variables, 30-6
 - postprocessing, 6-32
- ideal gas law, 6-37, 31-22
- IFC files, 4-23
- IGES files, 4-2, 4-20, 4-21
 - exporting, 4-28
- IGES geometry
 - blanking surfaces, 4-17
 - converting families into Airpak objects, 4-10
 - converting regions into Airpak objects, 4-11
 - converting selected CAD geometry into Airpak objects, 4-9
 - deleting CAD geometry, 4-18
 - families, 4-6
 - moving CAD geometry between, 4-7
 - visibility, 4-17
 - fixing small gaps, 4-19
 - groups, 4-16
 - importing into Airpak, 4-3
 - manipulating with mouse, 4-19
 - minimum feature size, 4-16
 - options, 4-16
 - overview, 4-2
 - selecting CAD geometry, 4-7
 - shading CAD surfaces, 4-19
 - unblanking surfaces, 4-17
 - unused CAD geometry, 4-17
 - used CAD geometry, 4-17
- image, 3-10, 3-12
 - file format, 3-14
 - frame, 3-15
 - label, 3-15
 - landscape, 3-12
 - options, 3-10
 - portrait, 3-12
 - PostScript options, 3-14
 - scaling, 3-12
- importing
 - CAD geometry, 4-1
 - CSV/Excel files, 4-20, 4-25
 - DWG files, 4-20, 4-23
 - DXF files, 4-20, 4-21
 - IDF files, *see also* IDF files
 - IFC files, 4-23
 - IGES files, *see also* IGES geometry, 4-2, 4-20, 4-21
 - overview, 4-1
 - STEP files, 4-2
 - tetin files, 4-2, 4-3
- improving the mesh
 - hexahedral, 26-6
 - tetrahedral, 26-7
- Inactive node, 2-54, 6-6
- inclined object, 7-48
 - meshing, 26-26
- Index button, 1-20
- individual plate macro, 25-65
- Individual polygonal blocks panel, 25-65, 25-68

- Individual prismatic blocks panel, 25-65
- Individual side specification panel, 8-11
- indoor air quality, 30-6
- inertial forces, 6-39
- initial conditions, 6-41
- initializing the solution, 27-7
- interactive editing, 6-23
- Interactive editing panel, 7-9
- internal assembly, 7-93
- interpolation, 31-39, 31-41
- introduction, 1-1
- isometric view, 2-14, 2-27
- Isometric view button, 2-27
- isosurface
 - clipping, 28-25
 - displaying results, 28-19
- Isosurface button, 2-28, 28-4, 28-21
- Isosurface contours panel, 28-32
- Isosurface panel, 28-21
- Isosurface particles panel, 28-40
- Isosurface vectors panel, 28-35
- iterations, 27-7
 - transient simulations, 20-11
- iterative procedure, 31-36
- job files, 3-3
- keyboard
 - shortcuts, 2-62
- label
 - image, 3-15
- laminar flow, 6-32
- landscape, 3-12
- Lewis number, 31-3
- Libraries node, 6-6
- Library name and info panel, 6-19
- library path, 6-17
- Library path panel, 6-17
- Lighting options panel, 2-12
- line solvers, 31-45
- line width, 6-22
 - contour plot, 28-33
 - group, 7-68
- object, 7-43
- room, 7-13
- linearization, 31-37
- list
 - drop-down, 2-45
 - single selection, 2-45
- Load solution ID button, 2-28, 28-4, 28-51
- loading
 - assembly, 7-97
 - existing mesh, 26-45
 - postprocessing objects, 28-8
 - postprocessing views, 28-8
 - report format, 29-7
 - trials plot, 28-51
 - variation plot, 28-47
- Local coord systems panel, 7-28
- local coordinate systems, 7-28
 - creating, 7-28
 - deleting, 7-30
 - editing, 7-29
 - viewing definition of, 7-30
- Local coords panel, 7-29
- log file, 2-38
- loss coefficient, 17-1
- low end, 26-19
- low side, 26-19
- Low side surface properties panel, 14-12
- low-Reynolds-number flows, 31-16
- Lower pri button, 27-51
- macros, 25-1
 - 1/4 polygonal cylinder, 25-51
 - atmospheric boundary layer, 25-4, 25-6
 - northward direction, 6-39, 25-6
 - wind direction, 25-6
 - wind speed, 25-8
 - closed box, 25-48
 - cylinder plate, 25-53
 - cylindrical enclosure, 25-55
 - diffuser, 25-10
 - group of prism blocks, 25-70
 - hemisphere, 25-63
 - individual plate, 25-65
 - polygonal block, 25-68

- polygonal circle, [25-58](#)
- polygonal cylinder, [25-62](#)
- polygonal duct, [25-46](#)
- prism block, [25-67](#)
- rotating
 - group of prism blocks, [25-70](#)
 - polygonal blocks, [25-68](#)
 - prism blocks, [25-67](#)
- solar flux, [25-2](#)
- Macros menu, [2-17](#), [25-1](#)
- Macros/1/4 Polygonal cylinder, [25-51](#)
- Macros/Atmospheric boundary layer, [25-6](#)
- Macros/Boundary conditions/Atmospheric boundary layer, [2-18](#)
- Macros/Boundary conditions/Diffuser, [2-18](#)
- Macros/Closed box, [25-48](#)
- Macros/Cylinder plate, [25-53](#)
- Macros/Cylindrical enclosure, [25-55](#)
- Macros/Diffuser, [25-10](#)
- Macros/Ducts/Polygonal ducts, [25-46](#)
- Macros/Groups of prism blocks, [25-70](#)
- Macros/Hemisphere, [25-63](#)
- Macros/Individual Plates, [25-65](#)
- Macros/Individual polygonal blocks, [25-65](#), [25-68](#)
- Macros/Individual prism blocks, [25-67](#)
- Macros/Individual prismatic blocks, [25-65](#)
- Macros/Polygonal circle, [25-58](#)
- Macros/Polygonal cylinder, [25-62](#)
- Macros/Quick geometry approximations/1/4 Polygonal cylinder, [2-18](#)
- Macros/Quick geometry approximations/Closed box, [2-18](#)
- Macros/Quick geometry approximations/Cylinder plates, [2-18](#)
- Macros/Quick geometry approximations/Cylindrical enclosure, [2-18](#)
- Macros/Quick geometry approximations/Hemisphere, [2-18](#)
- Macros/Quick geometry approximations/Polygonal circle, [2-18](#)
- Macros/Quick geometry approximations/Polygonal cylinder, [2-18](#)
- Macros/Quick geometry approximations/Polygonal ducts, [2-18](#)
- Macros/Rotate objects/Groups of prism blocks, [2-18](#)
- Macros/Rotate objects/Individual plates, [2-18](#)
- Macros/Rotate objects/Individual polygonal blocks, [2-18](#)
- Macros/Rotate objects/Individual prism blocks, [2-18](#)
- Macros/Solar Flux, [25-2](#)
- Macros/Solar flux, [2-18](#)
- Main library node, [2-50](#)
- Main window, [1-12](#), [2-2](#)
- managing
 - postprocessing objects, [28-6](#)
- manuals, [1-17](#)
 - printing, [1-23](#)
 - using, [1-18](#), [UTM-1](#)
- mass diffusion, [31-3](#)
- mass diffusion coefficients, [21-2](#)
- mass flow, [29-3](#), [30-2](#)
- mass fraction, [29-3](#), [30-5](#)
 - profiles, [21-11](#)
- Match edges button, [7-38](#)
- Match faces button, [7-37](#)
- materials, [7-74](#)
 - copying, [7-81](#)
 - creating, [7-82](#)
 - default, [6-40](#)
 - deleting, [7-84](#)
 - editing, [7-75](#)
 - fluid, [7-78](#)
 - parameterization, [24-1](#)
 - saving, [7-83](#)
 - solid, [7-77](#)
 - conductivity, [7-77](#), [24-3](#)
 - summary, [7-99](#)
 - surface, [7-80](#)
 - temperature dependence, [7-74](#), [7-85](#)
 - velocity dependence, [7-84](#)
 - viewing properties, [7-81](#)
- Materials node, [2-57](#)

- Materials panel, 7-75, 7-78
- mean age of air, 29-3, 30-6
- mean radiant temperature, 6-30, 29-3, 30-6, 30-8
- Merge project panel, 3-5
- merging
 - assembly, 7-97
 - model data, 3-5
 - projects, 3-5
- mesh, 26-1
 - checking, 26-39
 - element quality, 26-39
 - element volume, 26-42
 - face alignment, 26-41
 - controlling order for objects, 26-30
 - copying, 3-10
 - copying parameters, 26-19
 - displaying, 26-32
 - cross-section, 26-36
 - individual objects, 26-32
 - exporting, 26-44
 - files, 3-3, 3-17
 - guidelines, 26-4
 - hex-dominant, 26-4
 - hexahedral, 26-4
 - coarse, 26-5
 - examining, 26-5
 - improving, 26-6
 - minimum-count, 26-5, 26-8
 - object-specific controls, 26-5
 - parameters, 26-12
 - procedure, 26-5
 - refining, 26-5, 26-11, 26-17
 - loading, 26-45
 - object-specific controls, 26-17
 - parameters
 - 2D polygon, 26-26
 - 3D polygon, 26-22
 - assembly, 26-29
 - block, 26-21
 - circular object, 26-24
 - cylinder, 26-22
 - ellipsoid, 26-23
 - elliptical cylinder, 26-23
 - fan, 26-24
 - heat exchanger, 26-29
 - hood, 26-29
 - inclined object, 26-26
 - opening, 26-26
 - partition, 26-28
 - person, 26-27
 - prism, 26-21
 - rectangular object, 26-25
 - resistance, 26-28
 - room, 26-20
 - source, 26-28
 - vent, 26-24
 - wall, 26-27
 - partitioning, 27-23
 - planar resistance, 10-3
 - quality, 26-3, 26-39
 - resistance, 16-1
 - skew, 26-3
 - tetrahedral, 26-4
 - coarse, 26-7
 - coarsening, 26-14
 - examining, 26-7
 - improving, 26-7
 - minimum-count, 26-7, 26-10
 - parameters, 26-15
 - procedure, 26-7
 - refining, 26-7, 26-13, 26-17
 - smoothing, 26-14
 - units, 5-2
- Mesh control panel, 5-2, 26-8, 26-10
- mesh type, 6-24
- Message window, 1-13, 2-37
- metabolic rate, 12-3, 30-8
- METIS, 27-23
- Min/max locations panel, 28-30
- minimum-count mesh
 - hexahedral, 26-5, 26-8
 - tetrahedral, 26-7, 26-10
- mirroring
 - merged model data, 3-7
 - object, 7-22, 7-41

- room, 7-10
- model
 - adding objects, 7-64
 - building, 7-1
 - merging, 3-5
 - northward direction, 6-39, 25-6
 - parameterization, 24-1
- Model and solve toolbar, 2-27
- Model Display window, 2-33
- model files, 3-3
- Model manager window
 - Groups node, 6-6
 - Libraries node, 6-6
 - Problem setup node, 6-4
 - Solution settings node, 6-6
- Model manager window, 1-13, 2-31, 6-4
- Model manager window
 - collapsing nodes, 6-4
 - expanding nodes, 6-4
 - Inactive node, 6-6
 - Model node, 6-6
 - Points node, 6-6
 - Postprocessing node, 6-6
 - postprocessing objects, 28-6
 - Trash node, 6-6
- Model menu, 2-15, 7-4
- Model node, 2-55, 6-6
- Model/CAD data, 2-15, 4-3
- Model/Check model, 2-15, 7-99
- Model/Create material library, 2-16
- Model/Edit priorities, 2-15, 26-30
- Model/Generate mesh, 2-15, 5-2, 26-8, 26-10
- Model/Radiation, 2-15, 22-8
- Model/Snap to grid, 2-15
- Modify form factors panel, 22-13
- Modify point panel, 27-10
- mole fraction, 29-3, 30-5
- molecular weight, 7-79, 21-2
- momentum equation, 31-2
- momentum method, 25-11
- Monitor window, 27-9
- monitoring
 - convergence, 31-53
 - residuals, 31-53
 - solution, 27-8, 27-49, 27-51
- Morph edges button, 2-31
- Morph faces button, 2-31
- mouse, 2-49
 - adding objects, 2-60
 - changing color spectrum, 2-61
 - changing controls, 2-61
 - controlling panel inputs, 2-49
 - manipulating graphics, 2-59
- Model manager window, 2-50
 - context menus, 2-50
- moving color legend, 2-61
- moving coordinate axes, 2-61
- moving date, 2-61
- moving title, 2-61
- resizing objects, 2-61
- rotating the model, 2-59
- selecting objects, 2-60
- switching modes, 2-62
- translating objects, 2-60
- translating the model, 2-60
- zooming the model, 2-60
- Move all objects in model panel, 7-8, 7-10
- Move assembly panel, 7-97
- Move group panel, 7-72
- Move object button, 2-31, 7-8, 7-97
- moving
 - assembly, 7-97
 - group, 7-72
 - object, 7-21
 - point, 28-28
 - room, 7-9
- MPEG file, 28-17, 28-24
- multigrid solver, 27-6, 31-45
 - cycles, 31-47
 - flex cycle, 31-49
 - prolongation, 31-47
 - residual reduction rate, 31-49, 31-50
 - restriction, 31-47
 - termination criteria, 31-51
 - V cycle, 31-47, 31-49
 - W cycle, 31-47, 31-49

- Multiple regions panel, 4-13
- multiprocessor workstations, 27-22
- name
 - object, 7-43
 - room, 7-12
- named point, 20-24
 - creating, 20-25
 - deleting, 20-26
 - editing, 20-25
- Named point panel, 20-24
- NASTRAN, 27-21
- natural convection, 6-29, 6-36, 31-42
- Navier-Stokes equations, 31-1
- near-wall flows, 31-16
- network, *see also* object
- network parallel solver, 27-24
- New directory button, 2-41
- New project button, 2-24, 6-3
- New project panel, 1-14
- New unit name panel, 5-7
- New/existing panel, 1-12
- Next button, 1-19
- node file, 27-23
- node values, 31-39
- northward direction, 6-39, 25-6
- notes, 2-42, 6-8
 - object, 7-43
 - room, 7-13
- nozzle diffuser, 25-12
- numerical scheme, 31-36
- object
 - adding to group, 7-69
 - aligning
 - centers, 7-35
 - edges, 7-33
 - face centers, 7-36
 - faces, 7-32
 - matching edges, 7-38
 - matching faces, 7-37
 - vertices, 7-35
 - aligning with another object, 7-31
 - Edit window, 7-31
 - Object modification toolbar, 7-31
 - assembly, *see also* assembly, 7-90
 - attributes, 7-42
 - block, *see also* block, 8-1
 - blower, *see also* blower
 - circular, 7-47
 - color, 7-43
 - controlling meshing order, 26-30
 - copying, 7-38, 7-41
 - copying mesh parameters, 26-19
 - creating, 7-19
 - cylindrical, 7-56
 - deleting, 7-20
 - deselecting, 7-19
 - displaying results, 28-9
 - displaying the mesh, 26-32
 - Edit window, 7-14
 - editing, 7-19
 - editing properties in a group, 7-72
 - ellipsoid, 7-58
 - elliptical cylinder, 7-59
 - emissivity, 22-14
 - enclosure, *see also* enclosure
 - excluding, 7-43
 - fan, *see also* fan, 9-1
 - geometry, 7-45
 - grouping, 7-65
 - heat exchanger, *see also* heat exchanger, 17-1
 - heat sink, *see also* heat sink
 - high side, 26-19
 - hood, 18-1
 - inclined, 7-48
 - including
 - in a group, 7-43
 - in model, 7-43
 - line width, 7-43
 - low side, 26-19
 - minimum separation, 6-24
 - mirroring, 7-22, 7-41
 - moving, 7-21
 - name, 7-43
 - network, *see also* network

- notes, 7-43
 - opening, *see also* opening, 11-1
 - package, *see also* package
 - parameterization, 24-1
 - partition, *see also* partition, 14-1
 - PCB, *see also* PCB
 - person, 12-1
 - physical characteristics, 7-63
 - polygon
 - 2D, 7-51
 - 3D, 7-52
 - position, 7-44
 - postprocessing
 - activating, 28-7
 - deactivating, 28-7
 - managing, 28-6
 - prism, 7-55
 - radiation
 - modeling, 22-2
 - surface-to-surface model, 22-5
 - rectangular, 7-45
 - removing from group, 7-70
 - report
 - full, 29-17
 - summary, 29-12
 - resistance, *see also* resistance, 16-1
 - resizing, 7-20
 - rotating, 7-22, 7-41
 - scaling, 7-22, 7-39
 - selecting, 7-19
 - shading, 2-9, 7-44
 - size, 7-44
 - source, *see also* source, 15-1
 - summary, 7-99
 - texture, 7-44
 - translating, 7-22, 7-41
 - transparency, 7-44
 - vent, *see also* vent, 10-1
 - wall, *see also* wall, 13-1
 - wire, *see also* wire, 19-1
- Object creation toolbar, 2-29, 7-1
 - Object face button, 2-28, 28-4, 28-9
 - Object face contours panel, 28-32
 - Object face panel, 28-9
 - Object face particles panel, 28-40
 - Object face vectors panel, 28-35
 - Object modification toolbar, 2-30, 7-2
 - Object panel, 7-14
 - Object priority panel, 26-30
 - Object selection panel, 7-41
 - object-specific meshing controls, 26-17
 - hexahedral, 26-5
 - tetrahedral, 26-7
 - Objects outside panel, 7-22
 - on-line help, 2-48
 - One viewing window button, 2-26
 - Open project button, 2-25, 6-4, 6-8
 - Open project panel, 1-14, 2-39, 6-8
 - opening, *see also* object, 11-1
 - creating, 11-6
 - flow direction, 11-4, 11-14
 - free, 11-1, 11-2, 11-8
 - heat flow, 11-4
 - mass flow rate, 11-4, 11-14
 - meshing, 26-26
 - pressure, 11-2
 - profiles, 11-11
 - recirculation, 11-1, 11-3, 11-4, 11-12
 - mass flow rate, 11-1
 - report, 29-12
 - species, 21-9
 - species filters, 11-5
 - thermal specification, 11-4
 - species, 11-9, 11-15, 21-6
 - augmentation, 21-9
 - filtering, 21-9
 - species filters, 11-5
 - temperature, 11-2
 - thermal specification, 11-4, 11-15
 - transient simulations, 11-9, 11-10, 20-10
 - turbulence parameters, 11-9
 - opening a file, 2-39
 - opening a project, 6-8
 - Openings panel, 11-6
 - operating density, 6-37, 31-22
 - operating point for blower, 29-19

- operating point for fan, 29-19
- operating pressure, 6-37, 31-22
- optimization, 23-1
 - dynamic-q method, 31-30
 - files, 3-4
 - input, 3-4
 - log, 3-4
 - output, 3-4
 - postprocessing, 3-4
 - tab, 3-4
 - specification, 23-2
- Option parameter panel, 24-8
- Orient menu, 2-13
- Orient negative Z button, 2-26
- Orient positive X button, 2-26
- Orient positive Y button, 2-26
- Orient/Clear user views, 2-14, 28-8
- Orient/Home position, 2-14
- Orient/Isometric view, 2-14
- Orient/Nearest axis, 2-14
- Orient/Orient negative X, 2-14
- Orient/Orient negative Y, 2-14
- Orient/Orient negative Z, 2-14
- Orient/Orient positive X, 2-14
- Orient/Orient positive Y, 2-14
- Orient/Orient positive Z, 2-14
- Orient/Reverse orientation, 2-14
- Orient/Save user view, 2-14, 28-8
- Orient/Scale to fit, 2-14
- Orient/Zoom in, 2-14
- Orientation commands toolbar, 2-26
- Output units button, 5-8
- package, *see also* object
 - radiation, 22-2
- packing up a project, 1-15, 3-16, 27-53
- panels, 2-42
- parallel processing, 27-21
 - efficiency, 27-22
 - on a dedicated parallel machine, 27-22
 - on a multiprocessor workstation, 27-22
 - on a workstation cluster, 27-23
 - partitioning, 27-23
- Parallel settings panel, 27-22
- parallel solver, 27-21
- Param value panel, 24-4
- parameterization, 24-1
 - creating a new parameter, 24-2, 24-9
 - deleting parameters, 24-10
 - trials
 - defining, 24-11
 - running, 24-13
 - saving, 24-18
 - selecting, 24-13
- Parameters and optimization panel, 23-2, 24-9
- Parameters and optimization panel, 24-9
- Parameters and trials panel
 - selecting trials, 24-11
- particle trace, 28-39
 - animation, 28-41
- partition, *see also* object, 14-1
 - additional resistance, 14-12
 - adiabatic thin, 14-1, 14-3, 14-11
 - block surface coincident, 8-6
 - conducting thick, 14-1, 14-3, 14-7
 - conducting thin, 14-1, 14-3, 14-10
 - contact resistance, 14-1, 14-3, 14-9
 - creating, 14-4
 - fluid, 14-1, 14-11
 - high side, 14-2, 14-11
 - hollow thick, 14-1, 14-3, 14-8
 - intersecting block, 8-7
 - low side, 14-2, 14-11
 - meshing, 26-28
 - radiation, 14-13, 22-2
 - emissivity, 22-7
 - specification, 22-6
 - surface roughness, 14-4
 - temperature dependence, 14-7, 14-10
 - thermal model, 14-3, 14-7
 - thickness, 14-2
 - transient simulations, 14-8, 20-10
 - using with other objects, 14-4
- partitioning the mesh, 27-23
- Partitions panel, 14-4
- PATH, 1-16

- PATRAN, [27-21](#)
- PCB, *see also* object
- PDF files, [1-23](#)
- Peclet number, [6-39](#), [27-8](#)
- Per-object mesh parameters panel, [26-17](#)
- Per-object params, [26-17](#)
- Performance data panel, [25-36](#)
- person
 - creating, [12-3](#)
 - Fix shapes option, [12-5](#)
 - location and dimensions, [12-2](#)
 - meshing, [26-27](#)
 - metabolic rate, [12-3](#)
 - position, [12-2](#), [12-5](#)
 - proportions, [12-2](#), [12-5](#)
 - radiation, [12-7](#)
 - emissivity, [22-7](#)
 - specification, [22-6](#)
 - thermal conditions, [12-3](#), [12-6](#)
- person object, [12-1](#)
- Person shape data panel, [12-6](#)
- Persons panel, [12-3](#)
- picture file, [3-17](#)
 - saving, [2-42](#), [3-10](#)
- pixelation, [31-26](#)
- planar resistance, [10-2](#)
- plane cut
 - clipping, [28-18](#)
- Plane cut button, [2-28](#), [28-4](#), [28-12](#)
- Plane cut contours panel, [28-32](#)
- Plane cut panel, [28-12](#)
- Plane cut particles panel, [28-40](#)
- Plane cut vectors panel, [28-35](#)
- planning the analysis, [1-9](#)
 - computational model, [1-9](#)
 - modeling goals, [1-9](#)
 - physical models, [1-9](#)
 - solution procedure, [1-9](#)
- PMV, [6-30](#), [29-3](#), [30-6](#)
- point
 - displaying results, [28-26](#), [28-29](#)
 - named, [20-24](#)
 - report, [29-13](#)
- Point button, [2-28](#), [28-4](#), [28-27](#)
- Point panel, [28-27](#)
- Point particles panel, [28-40](#)
- Point vectors panel, [28-35](#)
- Points node, [2-53](#), [6-6](#)
- polygon
 - 2D, [7-51](#)
 - 3D, [7-52](#)
 - meshing, [26-13](#)
 - 2D, [26-26](#)
 - 3D, [26-22](#)
- polygonal circle macro, [25-58](#)
- Polygonal circle panel, [25-58](#)
- polygonal cylinder macro, [25-62](#)
- Polygonal cylinder panel, [25-62](#)
- polygonal duct macro, [25-46](#)
 - meshing, [26-5](#)
 - wire, [19-1](#)
- Polygonal ducts panel, [25-46](#)
- portrait, [3-12](#)
- position
 - object, [7-44](#)
 - variables, [30-5](#)
- Post menu, [2-21](#), [28-1](#)
- Post-processing node, [2-53](#), [6-6](#)
- Post/Convergence plot, [2-21](#), [27-12](#)
- Post/Create zoom-in model, [2-22](#), [28-52](#)
- Post/Download RSF project, [2-22](#), [28-4](#)
- Post/History plot, [2-22](#), [20-23](#)
- Post/Isosurface, [2-21](#), [28-21](#)
- Post/Load post objects from file, [2-22](#), [28-8](#)
- Post/Load solution ID, [2-22](#), [28-51](#)
- Post/Min/max locations, [2-21](#)
- Post/Object face, [2-21](#), [28-9](#)
- Post/Plane cut, [2-21](#), [28-12](#)
- Post/Point, [2-21](#), [28-27](#)
- Post/Postprocessing units, [2-22](#), [5-8](#)
- Post/Rescale vectors, [2-22](#), [28-6](#), [28-8](#)
- Post/Save post objects to file, [2-22](#), [28-8](#)
- Post/Surface probe, [2-21](#)
- Post/Time average, [2-22](#), [20-19](#)
- Post/Transient settings, [2-22](#), [20-18](#)
- Post/Trials plot, [2-22](#), [28-49](#)

- Post/Variation plot, 2-22, 28-44
- postprocessing, 28-1
 - animation, 20-21, 28-41
 - cascade modeling, 28-52
 - color, 28-5
 - contour attributes, 28-31
 - cross-section, 28-12
 - files, 3-4, 3-17
 - graphical displays, 28-5
 - isosurface, 28-19
 - clipping, 28-25
 - min/max points, 28-30
 - object face, 28-9
 - defining, 28-9
 - objects
 - activating, 28-7
 - deactivating, 28-7
 - managing, 28-6
 - Model manager window, 28-6
 - saving, 3-10
 - particle trace, 28-39
 - plane cut, 28-12
 - clipping, 28-18
 - defining, 28-12
 - point, 28-26
 - solution ID, 28-51
 - solution variables, 6-31
 - species transport, 21-12
 - time averaged results, 20-19
 - transient simulations, 20-18
 - animation, 20-21
 - time averaged results, 20-19
 - trials plot, 28-47
 - units, 5-8
 - variables, 30-1
 - variation plot, 28-43
 - vectors, 28-35
 - XY plot, 28-43
 - zoom-in modeling, 28-52
- postprocessing objects
 - define, 27-13
- Postprocessing toolbar, 2-28, 28-4
- Postprocessing units panel, 5-8
- PostScript files, 3-14, 6-16
- Power and temperature limits button, 2-27
- PPD, 6-30, 29-3, 30-6
- Prandtl number, 6-39, 27-8
- precision, 27-7, 31-53
- predicted mean vote, 30-6
- predicted percentage dissatisfied, 30-6
- preferences
 - display options, 6-12
 - editing options, 6-14
 - meshing options, 6-24
 - miscellaneous options, 6-17
 - postprocessing options, 6-25
 - printing options, 6-15
 - solution options, 6-25
- Preferences and settings panel, 6-11
- Preferences panel, 5-3
- pressure, 29-3, 30-2
 - drop, 17-1
 - interpolation schemes, 27-5, 31-41
 - variable, 30-2
 - postprocessing, 6-31
- pressure-correction equation, 31-43
- pressure-velocity coupling, 31-43
- Previous button, 1-20
- primary variables, 29-1, 30-1
- Print options panel, 2-5, 2-25, 6-3
- print region, 3-15
- Print screen button, 2-25, 6-4
- printed circuit board, *see also* PCB
- printing images, 3-10, 3-12
 - file format, 3-14
 - frame, 3-15
 - label, 3-15
 - landscape, 3-12
 - portrait, 3-12
 - PostScript options, 3-14
 - print region, 3-15
 - scale, 3-12
- printing text files, 6-15
- printing trials plots, 28-51
- printing variation plots, 28-47
- prism, 7-55

- meshing, 26-21
- problem files, 3-3
- problem parameters, 6-26
- problem setup, 1-10
- Problem setup node
 - Basic parameters, 6-4
 - Local coords, 6-6
 - Title/notes, 6-4
- profiles, 11-11, 13-19
 - species mass fraction, 21-11
- program structure, 1-2
- project
 - compressing data, 3-16
 - configuring, 6-11
 - creating, 6-7
 - defining, 6-1
 - deleting, 6-10
 - emailing, 27-53
 - merging, 3-5
 - notes, 6-8
 - opening, 6-8
 - packing up, 1-15, 3-16, 27-53
 - parameters, 6-26
 - reloading, 6-9
 - removing data, 3-16
 - saving, 3-9, 6-7
 - at specified frequency, 27-21
 - mesh data, 3-10
 - picture files, 3-10
 - postprocessing objects, 3-10
 - solution data, 3-10
 - transient simulations, 20-4
 - scaling, 6-14
 - title, 6-8
 - unpacking, 1-15, 3-16
- prolongation, 31-47
- push buttons, 2-43
- quality of mesh, 26-3, 26-39
- radiation, 22-1
 - choosing a model, 22-1
 - discrete ordinates (DO) model, 31-26
 - using, 22-14
 - heat flow, 29-3, 30-4
 - heat flux, 22-2
 - solar load model, 22-14
 - specification, 22-4
 - surface-to-surface model, 22-2, 31-24
 - for individual objects, 22-5
 - objects to include in calculation, 22-10
 - using Form factors panel, 22-8
 - view factors, 22-10
 - temperature, 22-3
 - variables, 6-30, 30-4
 - postprocessing, 6-32
- Radiation button, 2-27, 22-8
- Radiation object selection panel, 22-6
- Radiation specification panel, 22-6
- radiator, 17-1
- radio buttons, 2-44
- Rayleigh number, 6-39, 27-8, 31-21
- reading a file, 2-39
- real number entry, 2-44
- rectangular object, 7-45
 - meshing, 26-25
- Redo button, 2-25
- reference temperature, 30-4
- refining the mesh
 - globally, 26-11
 - hexahedral, 26-5, 26-11, 26-17
 - locally, 26-16
 - tetrahedral, 26-7, 26-13, 26-17
- refractive index, 31-24
- refrigeration circuits, 11-1
- relative humidity, 21-5, 29-3, 30-6, 30-8
- relaxation scheme, 31-45, 31-49
- relaxation sweeps, 31-47
- reloading a project, 6-9
- Remote execution parameters (via rsh) panel, 27-18
- Remote Simulation Facility (RSF), 27-29
- removing
 - object from group, 7-70
 - project data, 3-16
- renaming

- group, 7-67
- renormalization group (RNG) theory, 31-16
- report
 - solution overview, 29-8
- Report menu, 2-22, 29-1
- Report point data panel, 29-14
- Report summary data panel, 29-10
- Report window, 29-17
- Report/ADPI, 29-20
- Report/ADPI report, 2-23
- Report/Comfort level, 2-23, 30-7
- Report/Fan operating points, 2-23, 29-19
- Report/Full report, 2-23, 29-15
- Report/HTML report, 2-22, 29-5
- Report/Point report, 2-23, 29-13
- Report/Show optimization/param results, 2-23
- Report/Solution overview/Create, 2-22, 29-8
- Report/Solution overview/View, 2-22, 29-8
- Report/Summary report, 2-23, 29-9
- reports, 29-1
 - ADPI, 29-20
 - comfort level, 30-6
 - fan operating point, 29-19
 - full, 29-15
 - transient simulation, 29-17
 - HTML, 29-5
 - loading format, 29-7
 - point, 29-13
 - transient simulation, 29-14
 - saving format, 29-7, 29-12, 29-15
 - summary, 27-13, 29-9
 - transient simulation, 29-10
 - transient simulations, 20-22
 - variables, 29-3, 30-1
- rescaling vectors, 28-8
- Reset button, 2-43
- residual files, 3-4
- residual reduction rate criteria, 31-50
- residuals, 27-7, 27-9, 27-53
 - definition of, 31-53
 - monitoring, 31-53
 - plotting, 27-12
- reduction rate, 31-51
- scaling of, 31-54
- resistance, *see also* object, 16-1
 - approach velocity, 16-1, 16-2, 16-6
 - creating, 16-4
 - device velocity, 16-1, 16-2, 16-6
 - flow
 - pressure drop, 16-2
 - free area ratio, 16-1, 16-3
 - heat transfer coefficient, 17-5
 - loss coefficient, 16-1, 16-2, 16-5, 17-2
 - meshing, 16-1, 26-28
 - power law, 16-1, 16-2, 16-6
 - pressure drop, 16-1, 16-2
 - species, 21-7
 - transient simulations, 16-7, 20-11
- Resistance curve window, 10-13
- Resistances panel, 16-4
- resizing
 - object, 2-61, 7-20
 - room, 7-5
- restarting the solution, 27-15
- restriction, 31-47
- Return to Home button, 1-20
- Reverse orientation button, 2-27
- Reynolds averaging, 31-6
- Reynolds number, 6-39, 27-8, 31-21
- room
 - color, 7-13
 - creating, 7-4
 - default, 7-4
 - Edit window, 7-4
 - line width, 7-13
 - meshing, 26-20
 - mirroring, 7-10
 - moving, 7-9
 - name, 7-12
 - notes, 7-13
 - resizing, 7-5
 - to fit objects, 7-9
 - rotating, 7-11
 - scaling, 7-8
 - snap to grid, 7-11

- translating, 7-11
- wall, 7-12, 13-1
- Room node, 2-56
- Room panel, 7-5
- Rotate about screen normal button, 2-26
- Rotate groups of prism blocks panel, 25-70
- rotate polygonal block macro, 25-68
- rotate prism block macro, 25-67
- Rotate prism blocks panel, 25-67
- Rotate prism plates panel, 25-65
- rotated group of prism blocks macro, 25-70
- rotating
 - merged model data, 3-7
 - model, 2-59
 - object, 7-22, 7-41
 - room, 7-11
- roughness, 7-80
- round-off error, 31-53
- RSF
 - running independent of Airpak, 27-38
 - solve using, 27-29
- RSF running from Airpak session, 27-29
- Run optimization button, 2-27
- Run solution button, 2-27, 27-8, 27-14
- sample session, 1-25
- Save image panel, 3-10
- Save object panel, 4-28
- Save picture file, 2-42
- Save project button, 2-25, 3-9, 6-4
- Save project panel, 2-39, 3-9
- saving
 - assembly, 7-97
 - contour plot, 28-35
 - fan curve, 9-13, 9-15
 - files, 2-39
 - group, 7-73
 - log files, 2-38
 - materials, 7-83
 - picture file, 2-42, 3-10, 3-17
 - postprocessing objects, 3-10, 28-8
 - postprocessing views, 28-8
 - project, 3-9, 6-7
 - at specified interval, 27-21
 - mesh data, 3-10
 - picture file, 3-10
 - postprocessing objects, 3-10
 - solution data, 3-10
- report
 - full, 29-17
 - point, 29-14
 - summary, 27-17, 29-10
- report format, 29-7, 29-12, 29-15
- resistance curve, 16-6
- solution data, 27-21
- trials plot, 28-51
- variation plot, 28-47
- vent curve, 10-10, 10-13
- view factors, 22-13
- scalar variables, 30-1
- scale (in a GUI panel), 2-46
- Scale to fit button, 2-26
- scaling
 - merged model data, 3-6
 - object, 7-22, 7-39
 - project, 6-14
 - room, 7-8
 - vectors, 28-38
- Schmidt number, 31-3
- scratch files, 3-18
- screens, 10-2
- script files, 3-4, 27-20
- Search button, 1-20
- second-order accuracy, 27-5, 31-39
- Select family panel, 4-8
- selecting
 - assembly, 7-96
 - files, 2-39
 - object, 2-60, 7-19
- Selection panel, 7-83
- sequential solution, 31-36
- serial solver, 27-22
- Set range panel, 7-88, 9-15, 10-13, 20-17
- setting up a problem, 1-10
- shading, 2-9, 6-22
 - CAD surfaces, 4-19
 - contour plot, 28-33

- group, 7-68
- object, 7-44
- SIMPLE algorithm, 31-43
- single-precision solvers, 27-7
- single-selection list, 2-45
- size of an object, 7-44
- skew, 26-3
- slider bar, 2-46
- slot diffuser, 25-12
- smoothing the mesh, 26-14
- snap to grid, 7-11
- Snap to grid panel, 7-11**
- solar flux calculator panel, 25-2
- solar flux macro, 25-2
- Solar Load Model parameters panel, 22-15**
- solid material
 - default, 6-40
 - properties, 7-77
- solution, *see also* solver
 - accuracy, 31-39
 - batch file, 27-26
 - calculating, 27-1, 27-3, 27-49
 - on another computer, 27-18
 - cascade modeling, 28-52
 - continuation of, 27-15
 - control, 27-14
 - advanced options, 27-18
 - convergence, 27-52, 31-53
 - criteria, 27-7
 - copying data, 3-10
 - diagnostic tools, 27-52
 - discretization scheme, 27-4
 - ending, 27-52
 - examining results of, 28-1
 - initial conditions, 6-41
 - initializing, 27-7
 - monitoring, 27-8, 27-49, 27-51
 - multigrid, 27-6
 - network parallel, 27-24
 - numerical scheme, 31-36
 - overview, 29-8
 - parallel, 27-21
 - parameters, 27-14
 - under-relaxation, 27-5
 - plotting residuals, 27-12
 - precision, 27-7
 - procedure, 27-5
 - restarting, 27-15, 27-20
 - running on RSF, 27-29
 - script file, 27-20
 - serial, 27-22
 - setup procedure, 27-3
 - under-relaxation, 27-5, 31-40
 - variables, 6-29
 - postprocessing, 6-31
 - version, 28-51
- Solution monitor definition panel, 27-12, 27-51**
- Solution monitor parameters panel, 27-9**
- Solution panel, 29-8**
- Solution residuals window, 2-34, 27-9, 27-49**
- Solution settings node
 - Advanced settings, 6-6
 - Basic settings, 6-6
 - Parallel settings, 6-6
- solve
 - RSF, 27-29
- Solve menu, 2-19, 27-2**
- Solve panel, 27-14, 27-26**
- Solve/Define report, 2-20, 27-13**
- Solve/Define trials, 2-19, 24-11**
- Solve/Diagnostics, 2-19, 27-52**
- Solve/Run optimization, 2-19**
- Solve/Run solution, 2-19, 27-8, 27-14**
- Solve/Settings, 2-19**
- Solve/Settings/Advanced, 2-19, 27-4**
- Solve/Settings/Basic, 2-19, 27-7**
- Solve/Settings/Parallel, 2-19, 27-22**
- Solve/Solution monitor, 2-19, 27-12, 27-51**
- solver, *see also* solution, 1-1, 1-7
 - control, 27-14
 - advanced options, 27-18
 - discretization, 31-38
 - double-precision, 27-7
 - files, 3-3
 - input, 3-4

- output, 3-4
- linearization, 31-37
- multigrid, 31-40, 31-45
- network parallel, 27-24
- numerical scheme, 31-36
- overview of, 31-36
- parallel, 27-21
- serial, 27-22
- single-precision, 27-7
- solver precision, 27-7
- source, *see also* object, 15-1
 - creating, 15-3
 - heat parameters, 15-2, 15-5
 - meshing, 26-28
 - radiation, 15-5, 22-2
 - emissivity, 22-7
 - specification, 22-6
 - species, 21-8
 - temperature, 15-2, 15-5
 - transient simulations, 15-5, 20-10
 - using, 15-2
- Sources panel, 15-3
- Spalart-Allmaras model, 31-7
- species, 21-1
 - adding, 21-3
 - augmentation, 21-9
 - block, 21-6
 - boundary conditions, 21-6, 21-8
 - concentration, 21-6–21-8
 - concentrations, 21-6
 - diffusion, 31-3
 - fan, 21-6
 - filtering, 21-9
 - initial concentrations, 21-5
 - mass fraction, 29-3, 30-5
 - profiles, 21-11
 - material properties, 21-1
 - mole fraction, 29-3, 30-5
 - opening, 11-9, 11-15, 21-6
 - recirculation, 21-9
 - order of, 21-6
 - profiles, 21-11
 - properties, 21-1
 - resistance, 21-7
 - source, 21-8
 - specification, 21-3
 - transient simulations, 21-9
 - transport, 21-1
 - inputs for, 21-3
 - postprocessing, 21-12
 - transport equations, 31-3
 - variables, 30-5
 - postprocessing, 6-31
 - vent, 10-11, 21-6
 - volume source, 21-7, 21-9
- Species concentrations panel, 21-6, 21-7
- Species definitions panel, 21-3
- Species filter efficiency panel, 21-9
- species transport
 - transient simulations, 20-1
- species variables, 6-29
- specific heat, 7-77, 7-79
- specific heat capacity, 21-2
- speed, 29-3, 30-2
- Square Wave Time-Step Parameters panel, 20-4
- stability, 31-40
- starting **Airpak**, 1-10
 - on a UNIX system, 1-11
 - on a Windows system, 1-11
- startup options for UNIX systems, 1-14
- startup screen, 1-12
- static pressure, 30-2
- steady-state calculation, 6-29
- STEP files, 4-2
 - exporting, 4-28
- STEP geometry
 - importing into **Airpak**, 4-3
 - minimum feature size, 4-16
 - overview, 4-2
- summary
 - assembly, 7-98
 - material, 7-99
 - object, 7-99
- summary report, 27-13, 29-9
- Summary report button, 2-28, 27-13, 28-4,

- 29-9
- Supply species concentration panel, 25-41
- surface
 - high side, 26-19
 - low side, 26-19
 - material
 - default, 6-40
 - properties, 7-80
 - plane cut, 28-12
- Surface probe button, 2-28, 28-4, 28-29
- surface roughness, 7-80, 8-2, 13-3, 14-4
- surface-to-surface radiation model, 22-1, 22-2, 31-24
 - adaptive method, 22-11
 - objects to include in calculation, 22-10
 - specification for individual objects, 22-5
 - using Form factors panel, 22-8
 - view factors
 - computing, 22-10
 - displaying, 22-11
 - editing, 22-13
 - saving, 22-13
- swirling flow, 31-42
 - turbulence modeling in, 31-8
- symmetry boundary
 - discrete ordinates (DO) radiation model, 31-30
- system environment variables, 1-15
- tabs (in a GUI panel), 2-46
- temperature, 29-3, 30-3
 - reference, 30-4
 - variables, 30-3
- Temperature dependent panel, 7-85
- Temperature dependent power panel, 8-15, 14-7, 15-5
- Temperature or velocity dependent fluid conductivity panel, 7-85
- temperature variables, 6-29
 - postprocessing, 6-31
- temperature-dependent material parameters, 7-85
- Temperature/value curve window, 7-86
- temporal discretization, 31-44
- TERM, 1-16
- termination criteria, 31-51
- tetin files, 4-2, 4-3
- Tetin geometry
 - overview, 4-2
- Tetra params panel, 26-15
- tetrahedral mesh, *see also* mesh, 26-4
- Text editor, 7-88, 9-15, 10-15, 20-17
- text entry, 2-44
- text window, 6-17
- texture
 - group, 7-69
 - object, 7-44
- thermal conductivity, 7-77, 7-79, 21-2
 - anisotropic, 7-78
 - parameterization, 24-3
 - biaxial, 7-78
 - orthotropic, 7-77
- This project button, 6-12, 6-19
- Time average button, 2-28, 20-19, 28-4
- Time averaged data panel, 20-19
- time variation, 6-29
- time-dependent calculations, *see also* transient simulations, 20-1
- time-dependent variables, 20-11
- Time/value curve window, 20-15
- title, 2-42, 6-8
 - moving, 2-61
- Title/notes panel, 6-8
- TKE, 29-3
- total area
 - assembly, 7-99
- total volume
 - assembly, 7-99
- transformations
 - merging model data, 3-6
 - object, 7-22, 7-39
 - room, 7-8, 7-10
- Transient animation panel, 20-21
- Transient fan strength panel, 20-9
- Transient heat tr coeff panel, 20-10
- Transient parameters panel, 20-1
- Transient power panel, 20-9

- Transient power/area panel, 20-10
- Transient pressure panel, 20-10
- Transient settings button, 2-28, 20-18, 28-4
- transient simulations, 6-29, 20-1
 - animation, 20-21
 - Basic parameters panel, 20-1
 - Basic settings panel, 20-11
 - block, 20-9
 - fan, 20-9
 - history plot, 20-22
 - initial conditions, 20-9
 - iterations, 20-11
 - opening, 20-10
 - partition, 20-10
 - postprocessing, 20-18
 - report, 20-22
 - full, 29-17
 - point, 29-14
 - summary, 29-10
 - resistance, 20-11
 - saving, 20-4
 - solving, 20-1
 - source, 20-10
 - species, 21-9
 - specifying variables as a function of time, 20-11
 - time averaged results, 20-19
 - user inputs, 20-1
 - variation of temperature with time, 20-8
 - vent, 20-9
 - wall, 20-10
- Transient temperature panel, 20-8
- Transient X velocity panel, 20-10
- Transient Y velocity panel, 20-10
- Transient Z velocity panel, 20-10
- Transients panel, 20-13
- translating
 - assembly, 7-93
 - merged model data, 3-7
 - model, 2-60
 - object, 2-60, 7-22, 7-41
 - room, 7-11
- transparency
 - group, 7-69
 - object, 7-44
- Trash node, 2-54, 6-6
- Tree menu, 2-16
- Tree/Close all model nodes, 2-17
- Tree/Close all tree nodes, 2-17
- Tree/Find in tree, 2-16
- Tree/Open all model nodes, 2-17
- Tree/Open all tree nodes, 2-17
- Tree/Organize objects, 2-17
- Tree/Organize objects/Flat, 2-17
- Tree/Organize objects/Types, 2-17
- Tree/Organize objects/Types / subtypes, 2-17
- Tree/Organize objects/Types / subtypes / shapes, 2-17
- Tree/Sort, 2-17
- Tree/Sort/Alphabetical, 2-17
- Tree/Sort/Creation order, 2-17
- Tree/Sort/Meshing priority, 2-17
- trials, 24-1
 - defining, 24-11
 - multiple, 27-17
 - reporting and plotting, 24-18
 - running, 24-13
 - multiple trials, 24-16
 - single trial, 24-15
 - saving, 24-18
 - selecting, 24-13
- trials plot, 28-47
 - set range, 28-50
- Trials plot button, 2-28, 28-4, 28-49
- Trials plot panel, 28-49
- Trials plot window, 28-47
- turbulence, 6-32, 6-33, 31-4
 - boundary conditions
 - computing, 6-35
 - input methods, 6-33
 - relationships for deriving, 6-35
 - buoyancy effects, 31-19
 - computational expense, 31-10
 - enhanced two-equation model, 6-33, 31-

- 5, 31-8
- indoor zero-equation model, 31-4
- intensity, 6-34
- length scale, 6-34
- mixing-length model, 31-4
- modeling, 31-7
- parameters
 - fan, 9-14
 - opening, 11-9
 - vent, 10-12
- production, 31-11, 31-18
- RNG k - ϵ model, 31-5, 31-8, 31-16
- RNG k - ϵ model, 6-33
- Spalart-Allmaras model, 6-33, 31-7, 31-10
- standard k - ϵ model, 6-33, 31-5, 31-7, 31-14
- two-equation model, 6-33, 31-5, 31-7, 31-14
- variables, 30-5
- zero-equation model, 6-33, 31-4
 - indoor, 31-4
 - mixing-length, 31-4
- Turbulence equation parameters panel, 9-14, 10-12, 11-9, 25-42
- turbulent dissipation rate, 29-3, 30-5
- turbulent kinetic energy, 29-3, 30-5
- turbulent viscosity, 6-33, 31-15
- turbulent viscosity ratio, 29-3
- tutorial, 1-25, 2-48
- under-relaxation, 27-5, 31-40
- Undo button, 2-25, 7-73
- units, 5-1
 - conversion factors, 5-7
 - creating, 5-7
 - customizing, 5-4
 - deleting, 5-8
 - Fix values option, 5-6
 - limitations, 5-1
 - meshing, 5-2
 - postprocessing, 5-8
- UNIX systems
 - starting Airpak on, 1-11
 - startup options, 1-14
- unpacking a project, 1-15, 3-16
- Up button, 1-20
- upwind schemes, 31-39
 - first-order, 31-39
 - second-order, 31-39
- user interface, 2-1
- using the manual, UTM-1
- V-cycle multigrid, 31-47, 31-49
- valve diffuser, 25-12
- variables, 30-1
 - reports, 29-3
 - solution, 6-29
 - value at a point, 28-29
- variation plot, 28-43
 - set range, 28-46
- Variation plot button, 2-28, 28-4, 28-44
- Variation plot panel, 28-44
- Variation plot window, 28-43
- vector variables, 30-1
- vectors, 28-35
 - scaling, 28-8, 28-38
- velocity, 29-4
 - variables, 30-2
- velocity-dependent material parameters, 7-84
- vent, *see also* object, 10-1
 - approach velocity, 10-5, 10-10
 - creating, 10-6
 - device velocity, 10-5, 10-10
 - external, 10-10
 - flow direction, 10-3, 10-11
 - free area ratio, 10-5, 10-10
 - internal 2D resistance, 10-10
 - loss coefficient, 10-4, 10-5, 10-10
 - meshing, 26-24
 - 2D polygon, 26-26
 - circular, 26-24
 - inclined, 26-26
 - rectangular, 26-25
 - planar resistance, 10-2
 - approach velocity, 10-2
 - device velocity, 10-2

- free area ratio, [10-2](#)
- loss coefficient, [10-2](#)
- meshing, [10-3](#)
- pressure drop, [10-1](#), [10-3](#)
- resistance curve, [10-13](#), [10-15](#)
- species, [10-11](#), [21-6](#)
- stagnation pressure, [10-11](#)
- static pressure, [10-11](#)
- transient simulations, [20-9](#)
- turbulence parameters, [10-12](#)
- type, [10-10](#)
- Vents panel, [10-6](#)
- Version selection panel, [28-51](#), [29-9](#), [29-14](#)
- versions, [2-41](#)
- view factors, [22-2](#)
 - computing, [22-10](#)
 - adaptive method, [22-11](#)
 - hemicube method, [22-11](#)
 - displaying, [22-11](#)
 - editing, [22-13](#)
 - saving, [22-13](#)
- View menu, [2-7](#)
- View/Action, [2-12](#)
- View/Angle, [2-8](#)
- View/Bounding box, [2-8](#)
- View/Display, [2-10](#)
- View/Display/Construction lines, [2-11](#)
- View/Display/Construction points, [2-11](#)
- View/Display/Coord axes, [2-10](#)
- View/Display/Current date, [2-11](#)
- View/Display/Depthcue, [2-11](#)
- View/Display/Mesh, [2-11](#)
- View/Display/Mouse position, [2-11](#)
- View/Display/Object names, [2-10](#)
- View/Display/Object names/Current assembly, [2-10](#)
- View/Display/Object names/None, [2-10](#)
- View/Display/Object names/Selected, [2-10](#)
- View/Display/Origin marker, [2-10](#)
- View/Display/Project title, [2-11](#)
- View/Display/Rulers, [2-10](#)
- View/Display/Visible grid, [2-10](#)
- View/Distance, [2-8](#)
- View/Edit toolbars, [2-9](#)
- View/Lights, [2-12](#)
- View/Location, [2-8](#)
- View/Markers, [2-8](#)
- View/Markers/Add, [2-8](#)
- View/Markers/Clear, [2-9](#)
- View/Rubber bands, [2-9](#)
- View/Rubber bands/Add, [2-9](#)
- View/Rubber bands/Clear, [2-9](#)
- View/Set background, [2-9](#)
- View/Shading, [2-9](#)
- View/Shading/Hidden line, [2-10](#)
- View/Shading/Selected solid, [2-9](#)
- View/Shading/Solid, [2-9](#)
- View/Shading/Wireframe, [2-9](#)
- View/Summary (HTML), [2-8](#), [7-99](#)
- View/Visible, [2-11](#)
- viewing
 - assembly, [7-96](#)
- viewing material properties, [7-81](#)
- Viewing options toolbar, [2-26](#)
- viscosity, [7-78](#), [21-2](#)
- viscosity ratio, [29-3](#)
- viscous forces, [6-39](#)
- volume flow, [29-4](#), [30-2](#)
- volumetric expansion coefficient, [7-78](#), [21-2](#)
- volumetric heat sources, [31-2](#)
- vortex diffuser, [25-12](#)
- W-cycle multigrid, [31-47](#), [31-49](#)
- wall, *see also* object, [13-1](#)
 - boundary conditions
 - discrete ordinates (DO) radiation model, [31-29](#)
 - creating, [13-10](#)
 - effective thickness, [13-2](#)
 - external thermal conditions, [13-6](#)
 - heat transfer, [13-15](#)
 - convective, [13-6](#)
 - radiative, [13-6](#), [13-7](#), [13-18](#), [13-19](#)
 - heat transfer coefficient, [13-6](#)
 - inside, [13-1](#)
 - meshing, [26-27](#)
 - moving, [13-3](#), [13-13](#)

- multifaceted, [13-9](#)
- outside, [13-1](#)
- profiles, [13-19](#)
- radiation, [22-2](#)
 - emissivity, [22-7](#)
 - specification, [22-6](#)
- room, [7-12](#)
- specified heat flux, [13-4](#)
- specified temperature, [13-5](#)
- stationary, [13-13](#)
- surface roughness, [13-3](#)
- symmetry, [13-13](#)
- thermal boundary conditions, [13-4](#)
- thermal specification, [13-15](#)
- thickness, [13-2](#), [13-13](#)
- transient simulations, [13-15](#), [20-10](#)
- velocity, [13-3](#), [13-14](#)
- Wall external thermal conditions panel, [13-15](#)
- Walls panel, [13-10](#)
- web browser, [1-18](#)
- wind direction, [25-6](#)
- wind speed, [25-8](#)
- Windows 2000 systems, [1-11](#)
- Windows menu, [2-23](#)
- Windows systems
 - batch processing, [27-26](#)
 - starting **Airpak** on, [1-11](#)
- wire, *see also* object, [19-1](#)
 - creating, [19-1](#)
 - polygonal ducts, [25-46](#)
- wireframe, [2-9](#)
- Wires panel, [19-1](#)
- workstation clusters, [27-23](#)
- Write assembly button, [7-97](#)
- writing
 - file, [2-39](#)
 - report
 - full, [29-17](#)
 - point, [29-14](#)
 - summary, [27-17](#), [29-10](#)
- XY plot, [28-43](#)
- Zoom in button, [2-26](#)
- Zoom-in modeling panel, [28-52](#)
- zooming the model, [2-60](#)