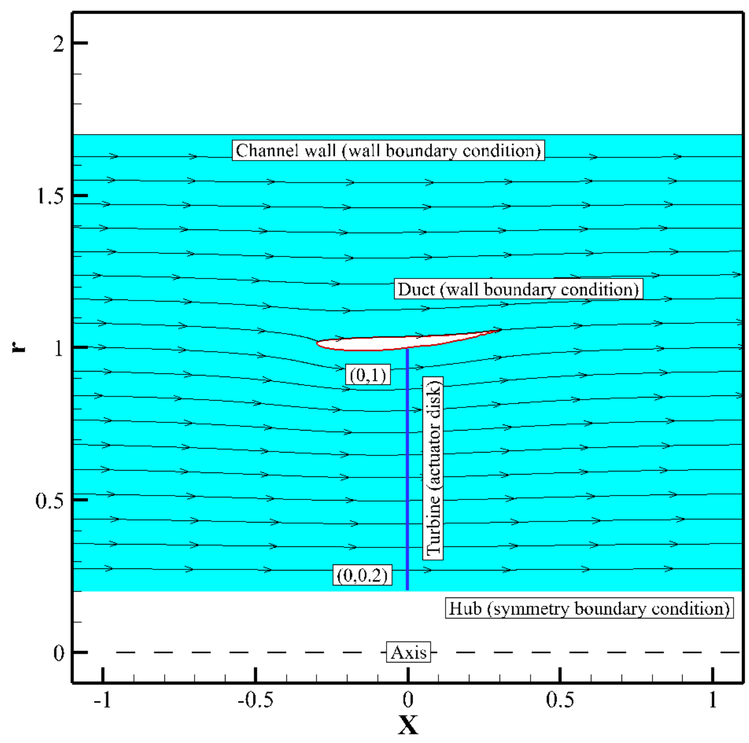


A Complementary Material for CE358 Introductory Ocean Engineering

ANSYS/Fluent v231:

A Quick Step into CFD

Spyros A. Kinnas (Fall 2023)

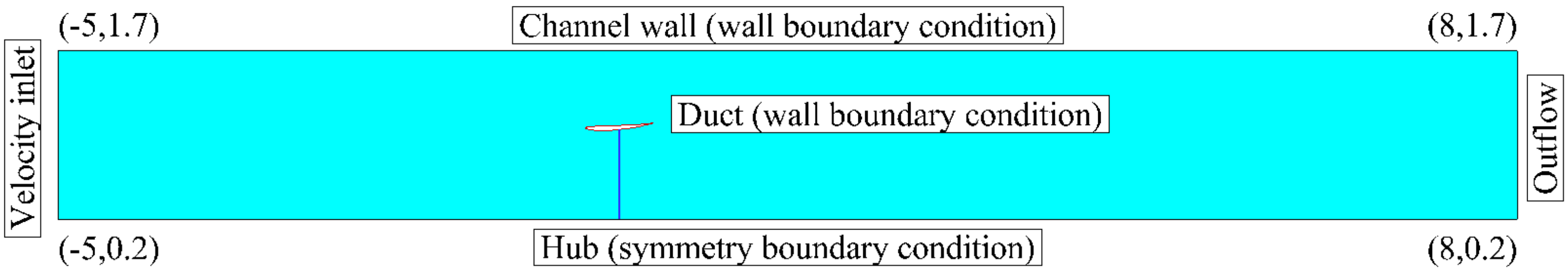
What is *ANSYS/Fluent*

- A commercial CFD code with many applications in many fields
- The CFD simulation pipeline:

PREPROCESSOR: GRID GENERATION → *PERFORM CFD SIMULATION* → *ANALYZE RESULTS*

- The software features:
  - Options for turbulence, acoustics, reaction flow, heat transfer, phase change, radiation and more
  - Multiphase flow and Parallel processing

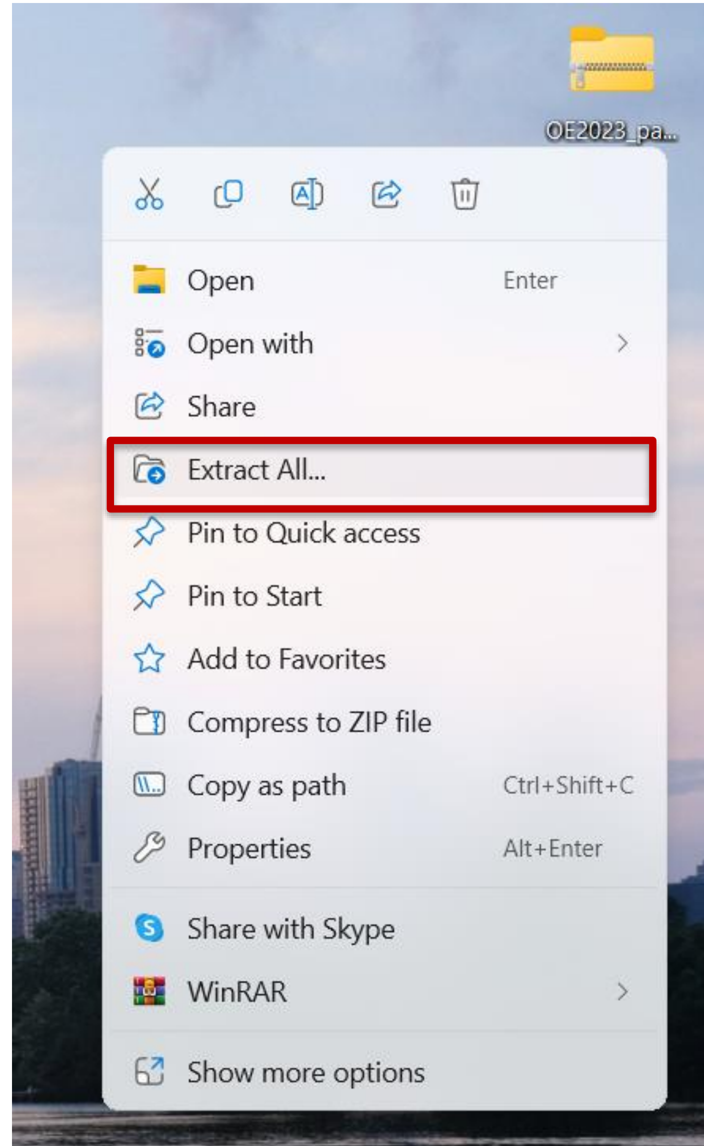
Computational Domain



- The sample run in this power point presentation is for the above duct at $U = 1$ m/s, with $\rho = 999.965$ Kg/m^3 and $\mu = 0.001519$Pa sec.
- The CAEE's water flume is 30 cm * 30 cm, and the radius of the turbine blade is 10 cm. In the CFD model, the radius of the turbine blade is 1 m, and the equivalent cylindrical channel has a radius of 1.7 m (with the same section area).

Step 1. Unzip the files

- Download the package from the term project website and unzip the files.
- Right-click the “OE2023_PartA.zip” file and click “Extract all” as shown in the screenshots.

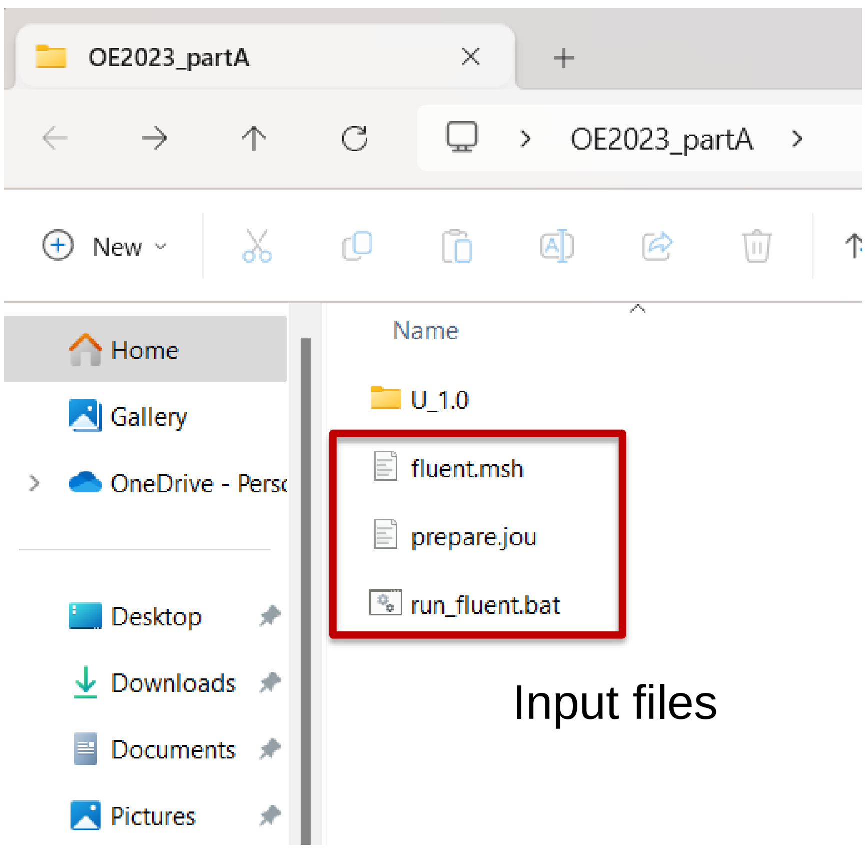


File descriptions:

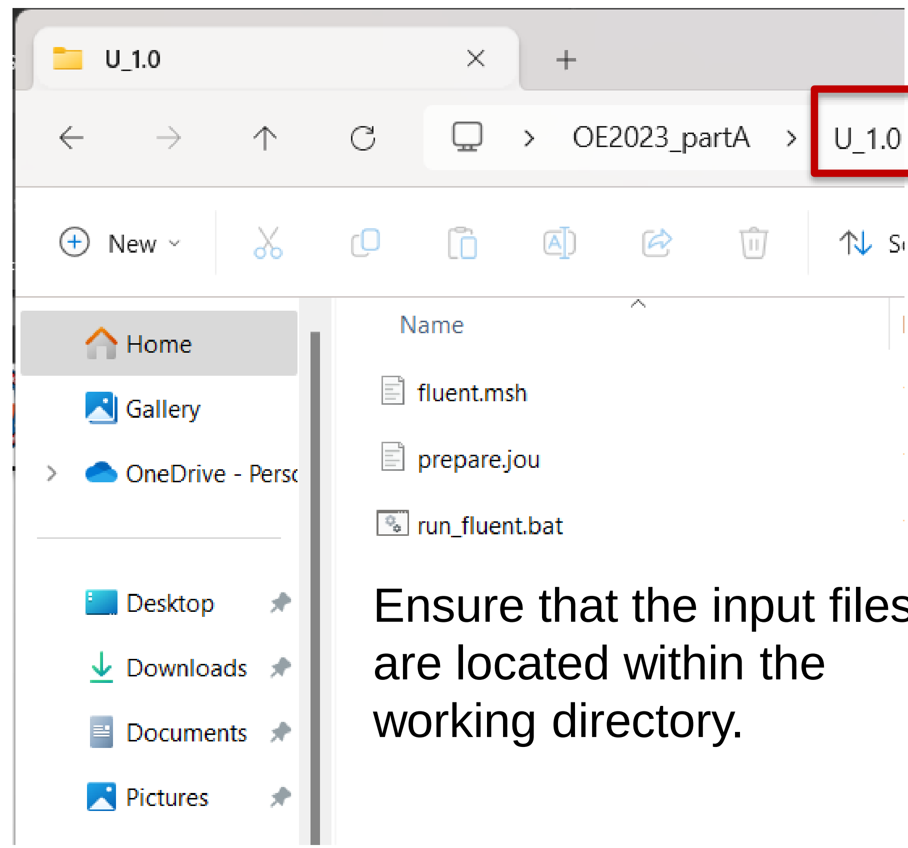
- fluent.msh: fluent mesh file
- prepare.jou: journal file to set up and run the CFD case automatically
- run_fluent.bat: a DOS batch file to automatically run the case

Step 2. Initialize Fluent case

- In order to maintain an organized file structure, it is recommended to create folders with descriptive names and subsequently duplicate the specified files into these folders.



Input files



Ensure that the input files are located within the working directory.

Step 2. Initialize Fluent case

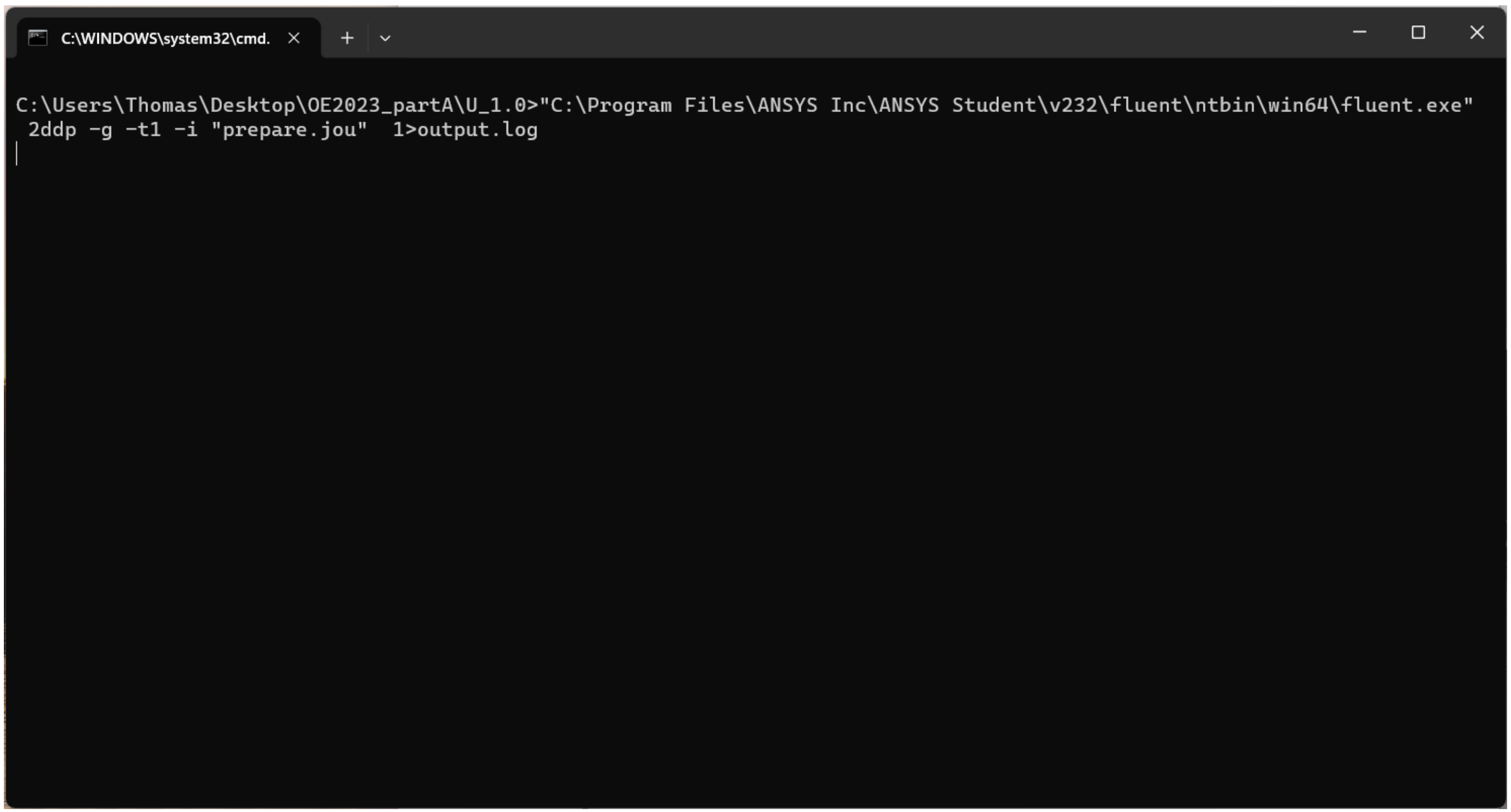
prepare.jou

```
1   (define U 1.0)
2   ;-----------------------------------------------------------------------------------------
3   ;-------------You should NOT make any changes under these lines-------------------------
4   ;-----------------------------------------------------------------------------------------
5   (define A 6.28 )
6   /file/read-case fluent.msh
7   /define/models/axisymmetric yes
8   /define/models/viscous/kw-sst yes
9   /define/materials/change-create/air water yes constant 999.965 no no yes
10  constant 0.001519
11  no no no yes
12  /define/operating-conditions/reference-pressure-location -4.90 0
13  /define/boundary-conditions/velocity-inlet inlet no yes yes no 0 no U no 0
14  no yes 2 0.07
15  /define/boundary-conditions/zone-type 18 interior
16  /report/reference-values area A density 999.965 enthalpy 0 length 1 pressure 0
17  /report/reference-values temperature 278.15 velocity U viscosity 0.001519
18  ;24 is for coupled p-v-coupling
19  /solve/set p-v-coupling 24
```

- Edit the input journal file to change the velocity
  - Right-click on **prepare.jou**
  - Edit with Notepad++
  - On line 1, change the inflow velocity
  - Very important: don’t change anything else in this journal file unless you are perfectly sure what you are doing.

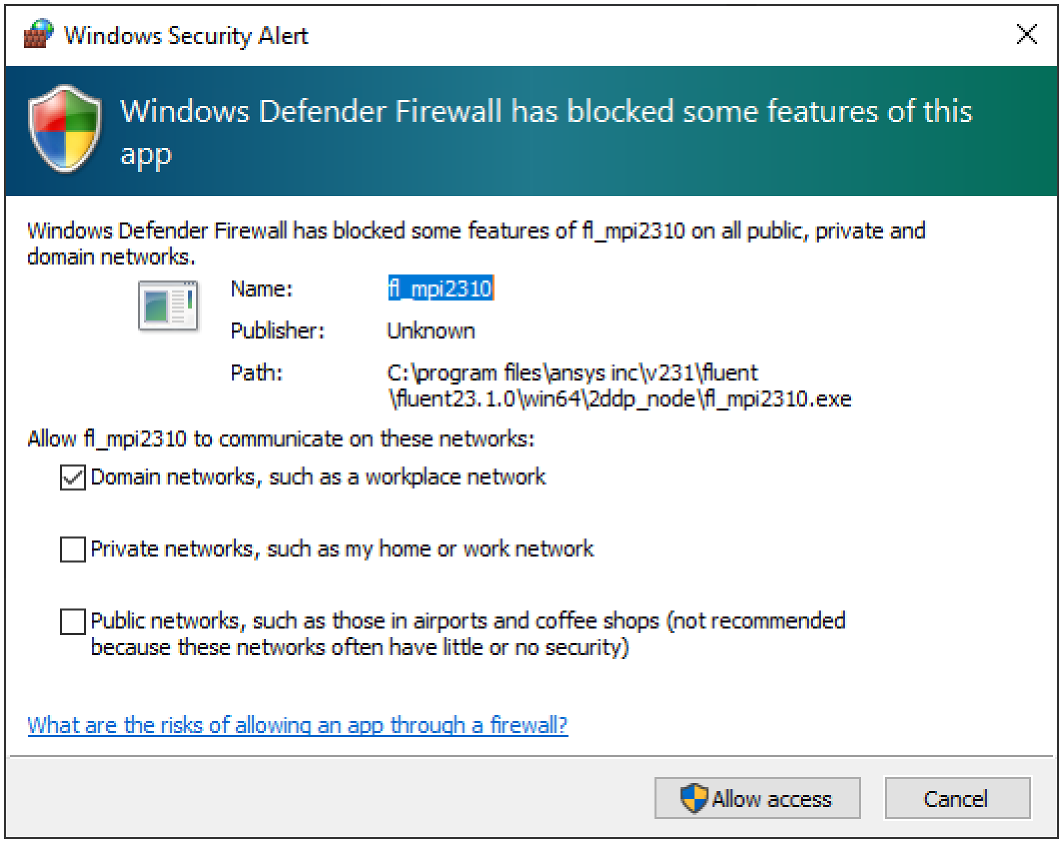
Step 3. Run Fluent case

- Double-click on "run_fluent.bat" will start the simulation. This DOS window means that Fluent is running. **Wait until it is automatically closed**. It might take about a few minutes to run one case.



Step 3. Run Fluent case

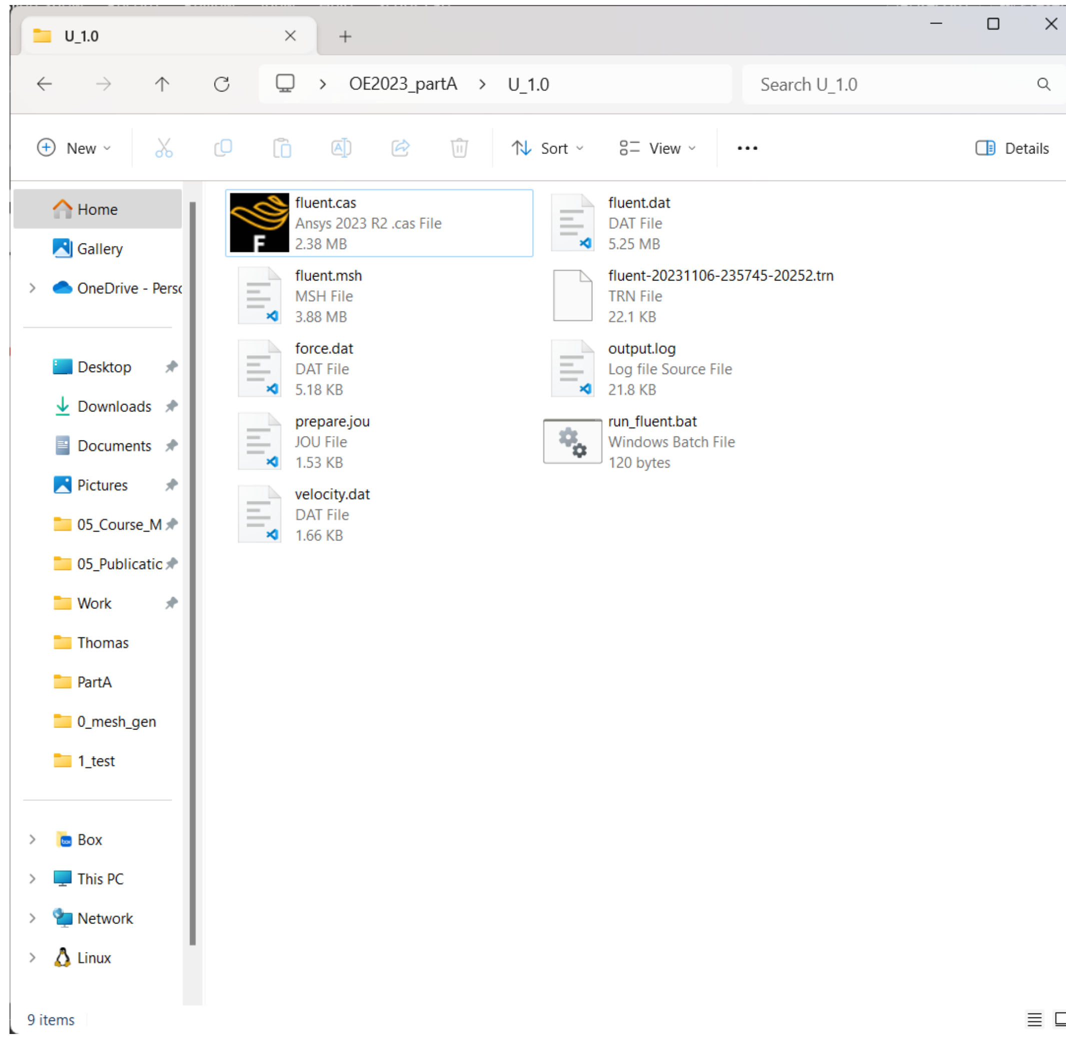
- Left click "Cancel" if you see any prompt



Step 3. Run Fluent case

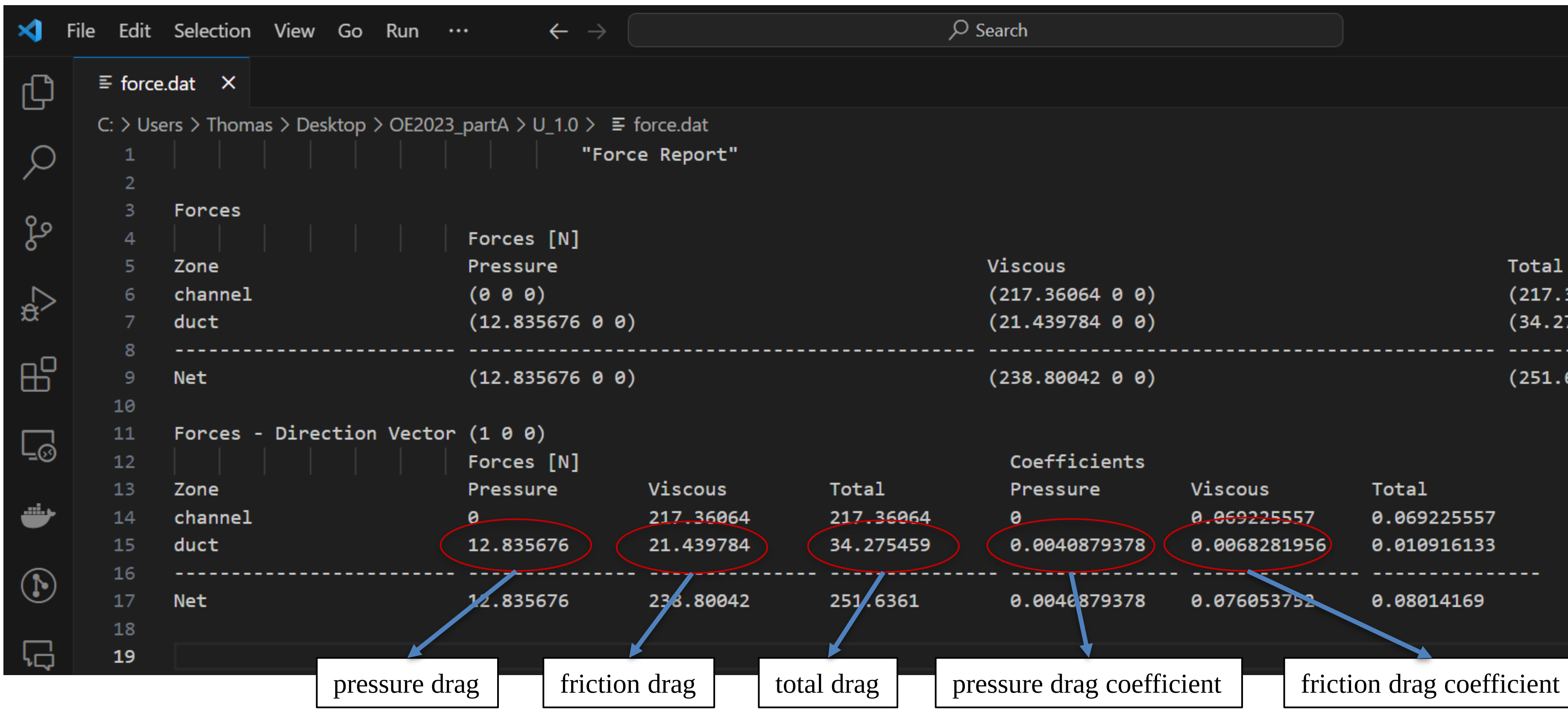
- Once the simulation is complete, several files will be generated, including:

1. fluent.cas (or fluent.cas.h5)
2. fluent.dat (or fluent.dat.h5)
3. force.dat
4. velocity.dat
5. output



Step 4. Check the results

- The force.dat file contains the forces on both the duct and the channel, as the boundary condition for the channel is also set to "wall." In your report, you should include only the drag coefficients and forces on the duct. These results were obtained by setting the inflow velocity at 1.0 m/s.



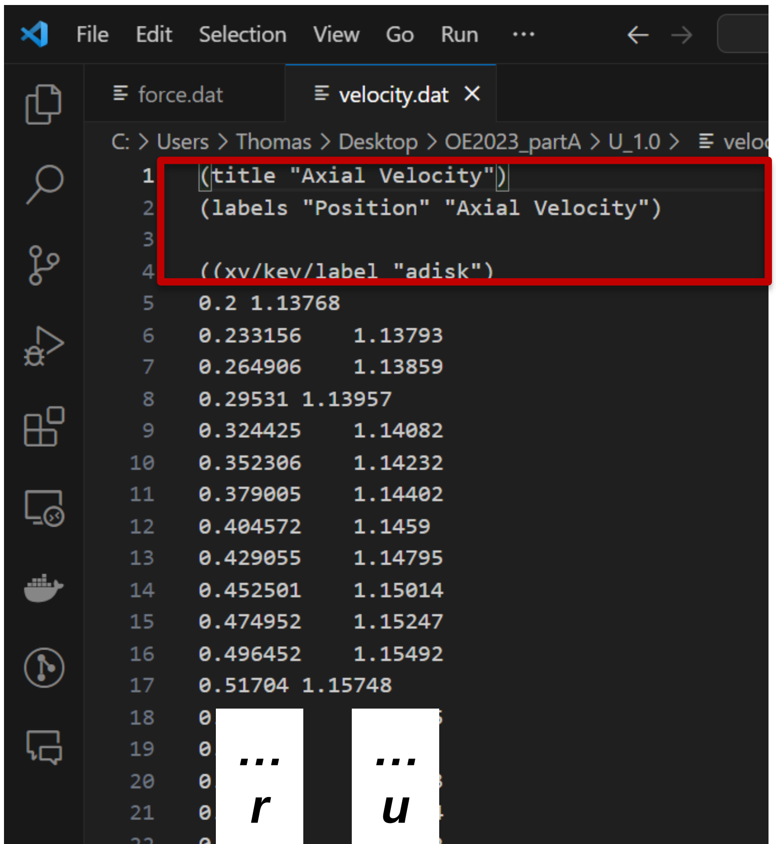
```
"Force Report"

Forces
                         Forces [N]
Zone                     Pressure                                     Viscous                                      Total
channel                  (0 0 0)                                      (217.36064 0 0)                              (217.36
duct                     (12.835676 0 0)                              (21.439784 0 0)                              (34.275
------------------------ -------------------------------------------- -------------------------------------------- ------
Net                      (12.835676 0 0)                              (238.80042 0 0)                              (251.63

Forces - Direction Vector (1 0 0)
                         Forces [N]                                     Coefficients
Zone                     Pressure        Viscous         Total          Pressure        Viscous         Total
channel                  0               217.36064       217.36064      0               0.069225557     0.069225557
duct                     12.835676       21.439784       34.275459      0.0040879378    0.0068281956    0.010916133
------------------------ --------------- --------------- -------------- --------------- --------------- ---------------
Net                      12.835676       238.80042       251.6361       0.0040879378    0.076053752     0.08014169
```

Step 4. Check the results

- The velocity.dat file includes the velocity profile at the actuator disk, which represents the turbine. You will need to perform numerical integration to determine the average velocity and flow rate using the data from this file. The first column lists the radial locations, while the second column provides the corresponding velocity at each grid point.

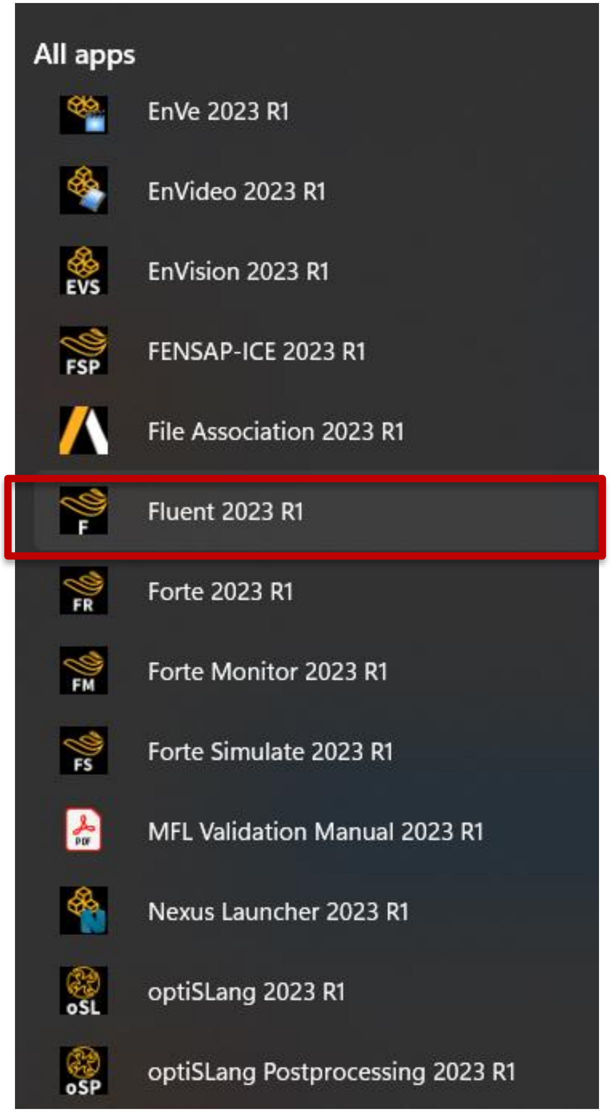


Ignore the header lines.

Step 4. Check the results

- To check additional results, you need to open the ANSYS/Fluent graphical user interface (GUI). You can find Fluent under ANSYS 2023 R1 in the start menu by following these steps:

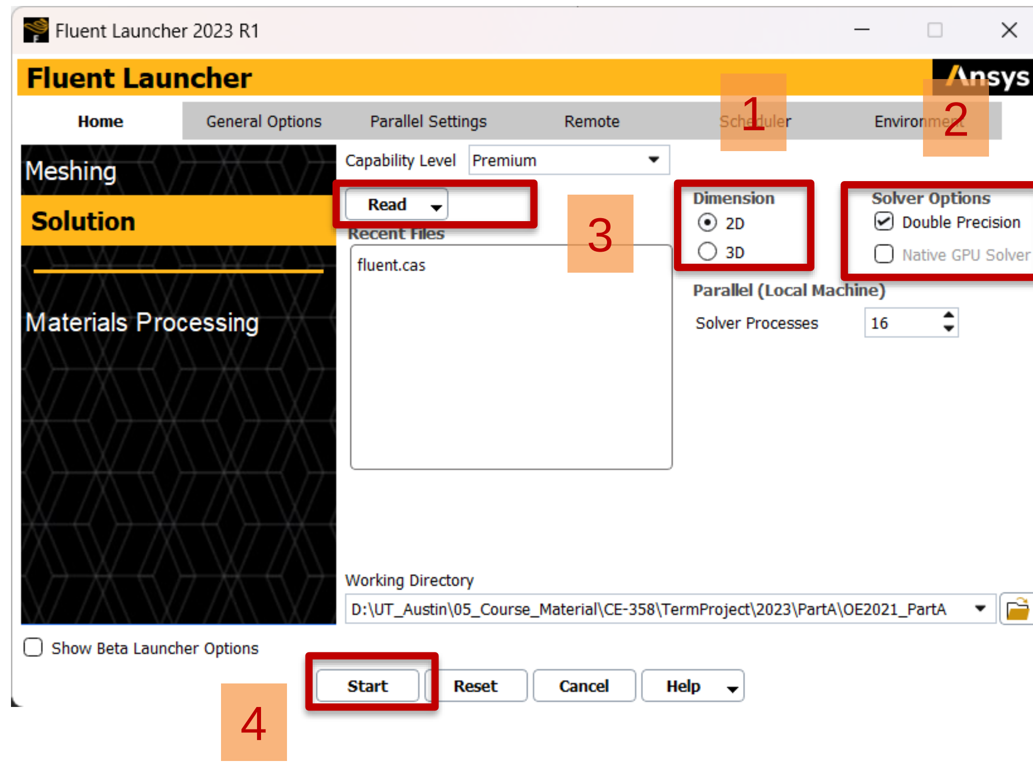
- 1. Go to the Start Menu.
- 2. Navigate to ANSYS 2023 R1.
- 3. Select Fluent 2023 R1 to launch the program.



Step 4 Start ANSYS FLUENT (V23.1)

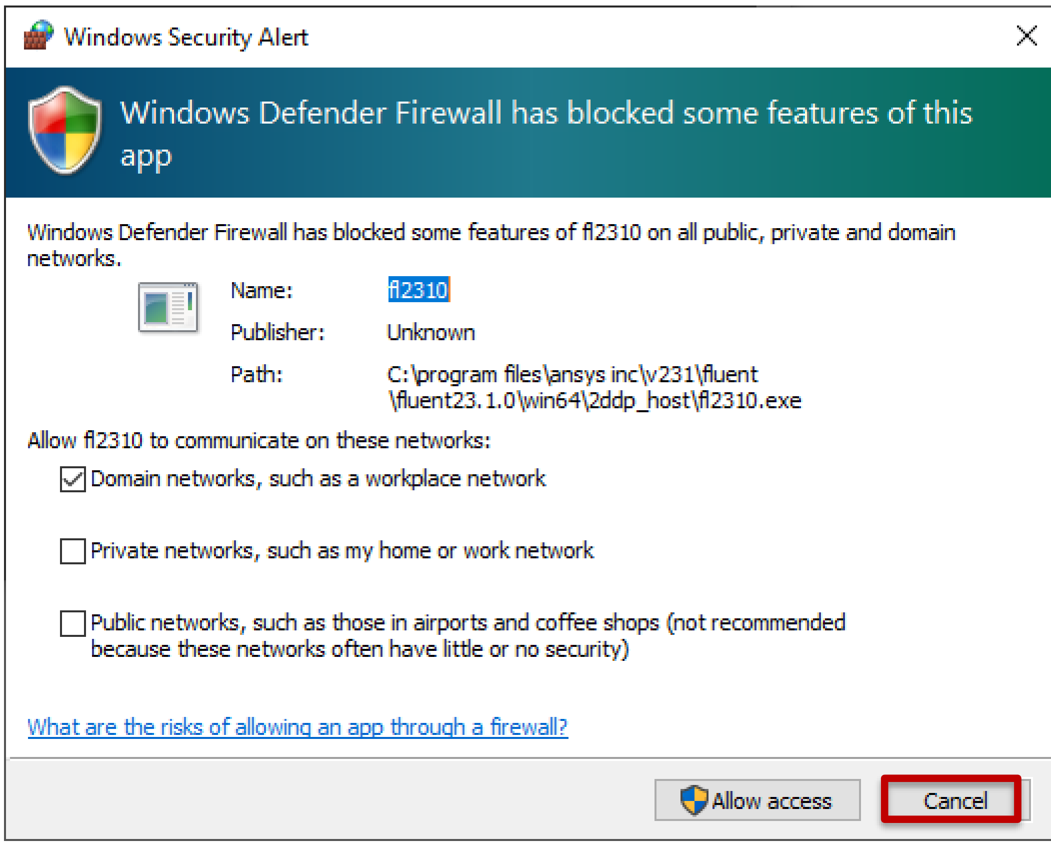
- The first thing that you see is the FLUENT Launcher.

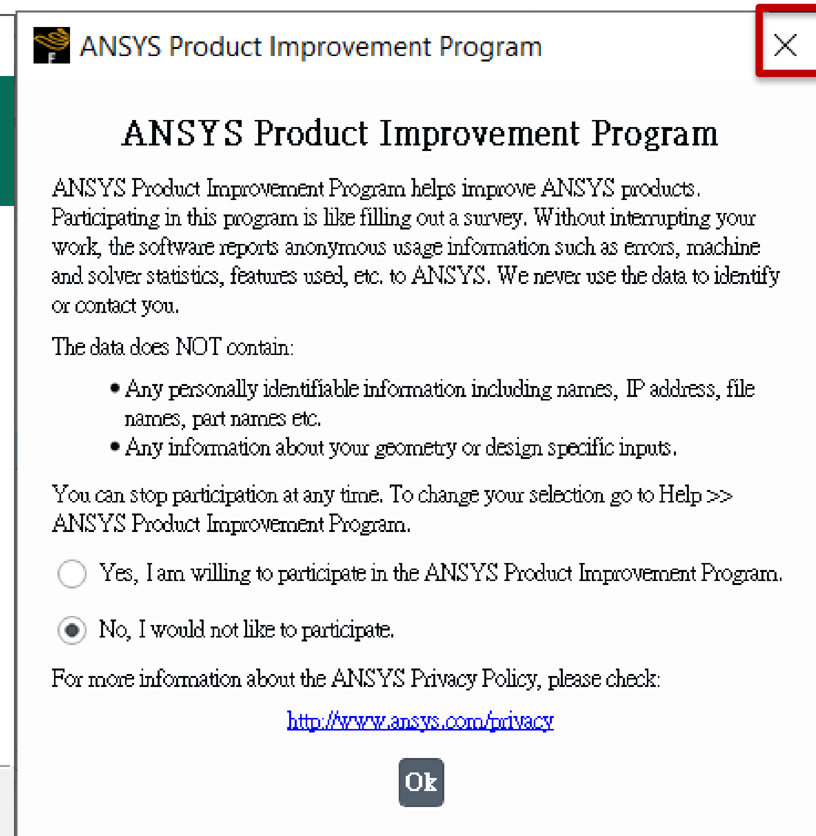
1) For a two-dimensional simulation, select '**2D**' from Dimension.
2) Ensure you also choose '**Double Precision**' in the Solver options.
3) Then select '**Case and Data**' to load your case file.
4) Click **Start**



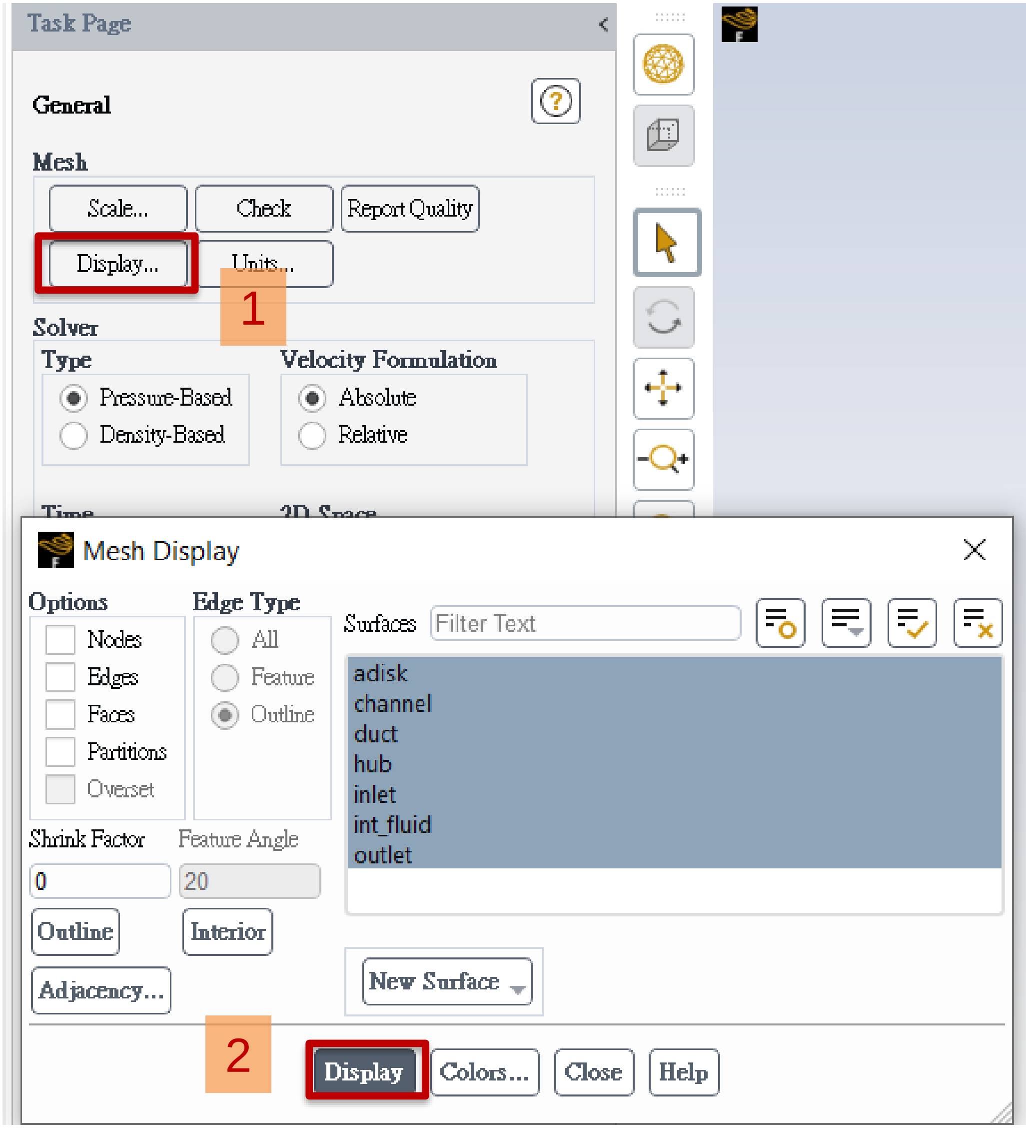
Step 4 Getting started with ANSYS Fluent (V23.1)

- Again, click “Cancel” and close other prompts if you need to.





Step 4 Getting started with ANSYS Fluent (V23.1)

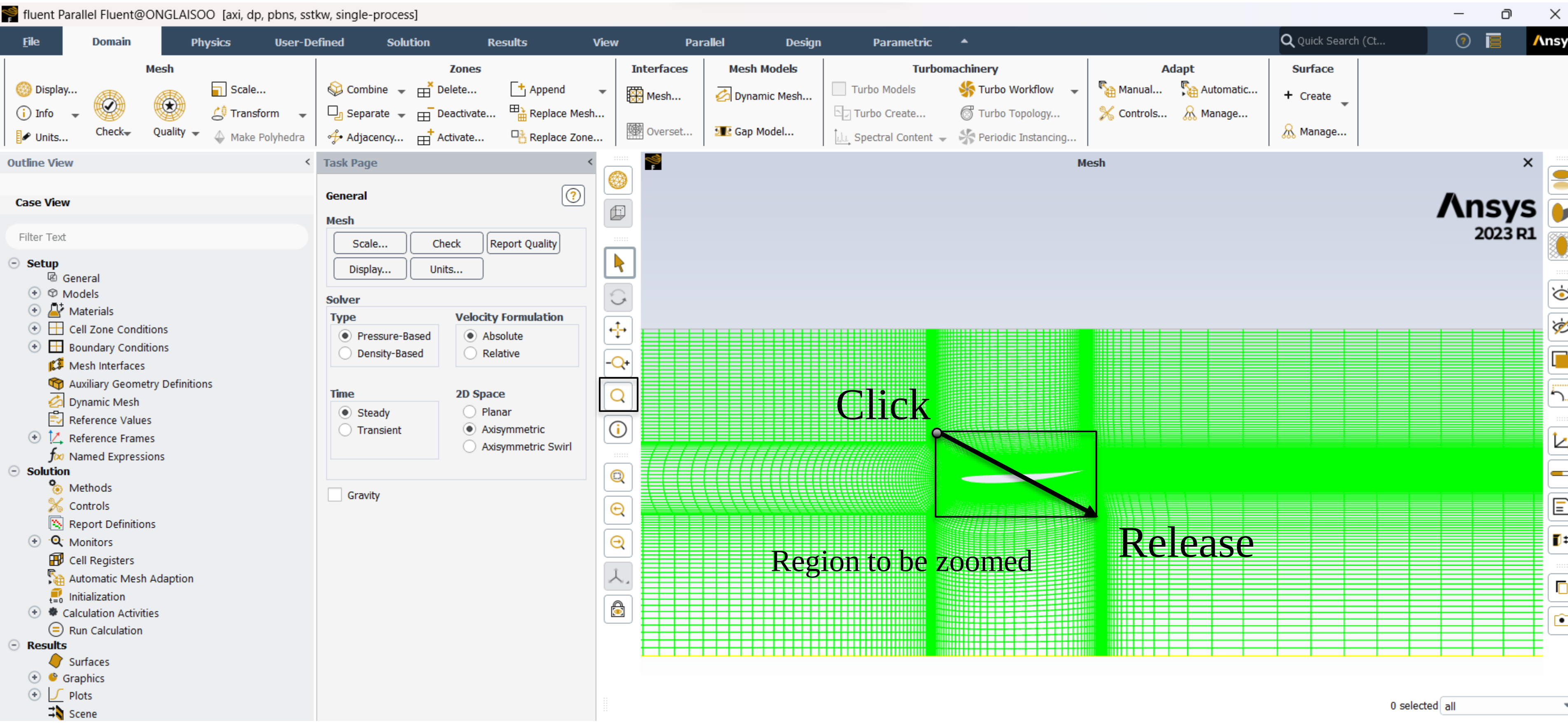


To visualize the mesh and your results:

1) Click on '**Display**' found under the 'General' tab.
2) In the prompt window that appears, click on '**Display**' to proceed.

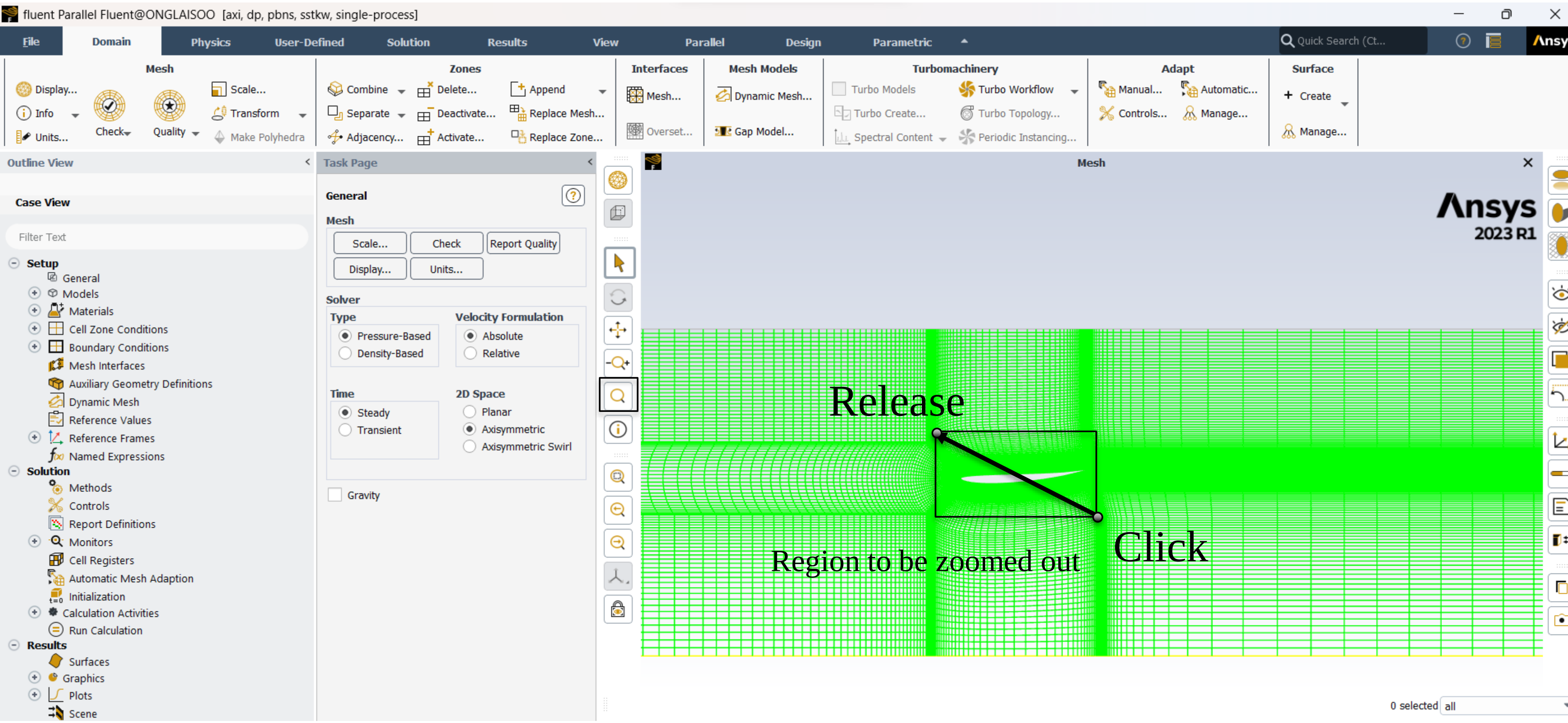
Step 4 Getting started with ANSYS Fluent (V23.1)

To zoom in, click and hold the left mouse button and drag downwards to the right.



Step 4 Getting started with ANSYS Fluent (V23.1)

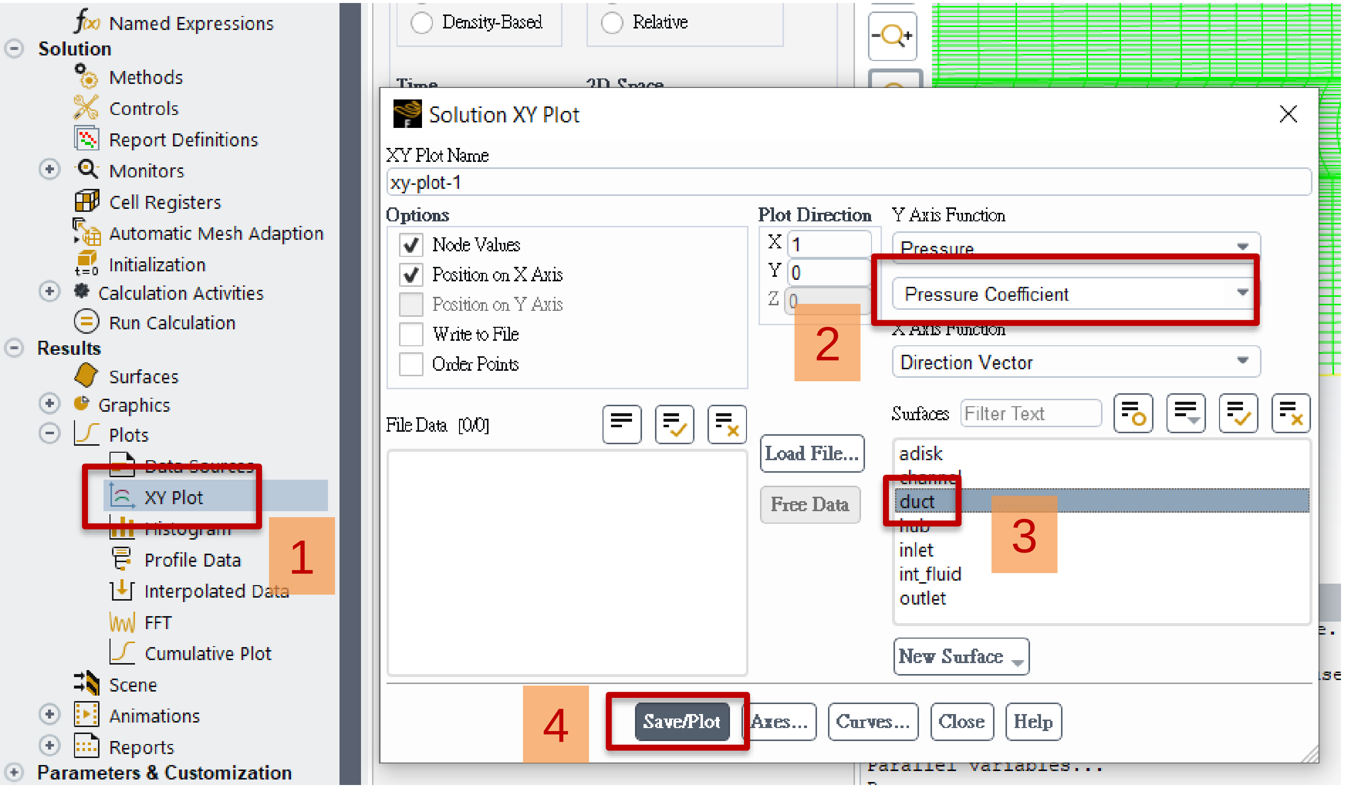
To zoom out, click and hold the left mouse button and drag upwards to the left.



Step 5 Plot Pressure Coefficient

To generate an XY Plot of pressure and pressure coefficient distributions over the duct:

1. Navigate to '**Results**' -> '**Plots**' -> '**XY Plot**' and double-click.
2. Under '**Y Axis Function**', select '**Pressure**' and '**Pressure Coefficient**'.
3. Choose 'duct' from the 'Surfaces' list.
4. Click on 'Save/Plot'.



The pressure coefficient is defined as

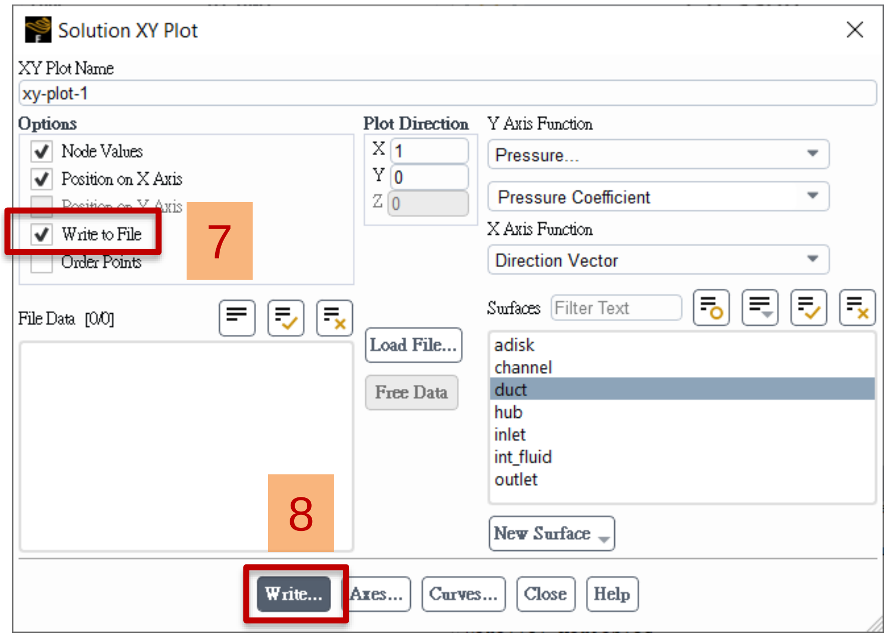
$$C_P = \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2}$$

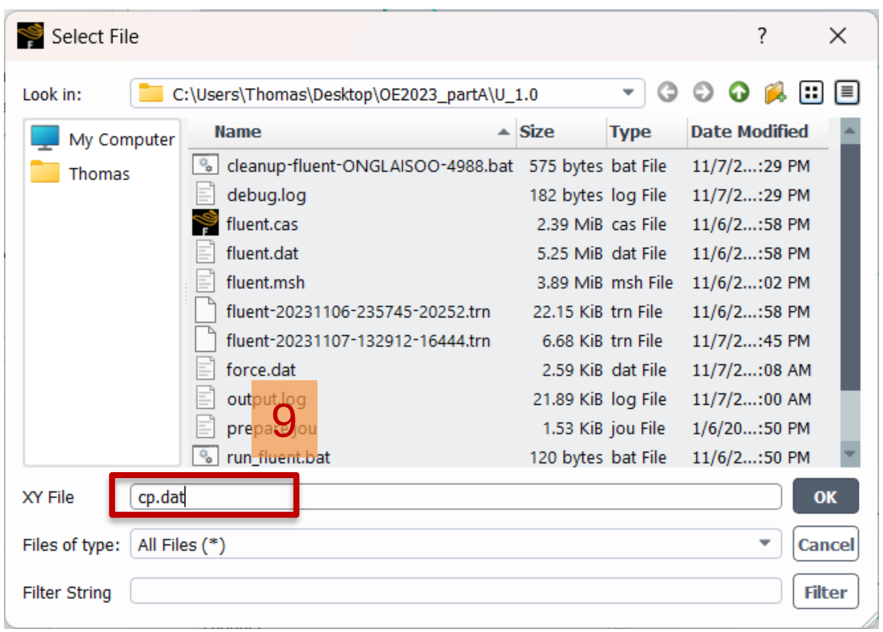
where p is the pressure, ρ is the density, p_∞ and u_∞ are the pressure and velocity far upstream.

Step 5 Plot Pressure Coefficient

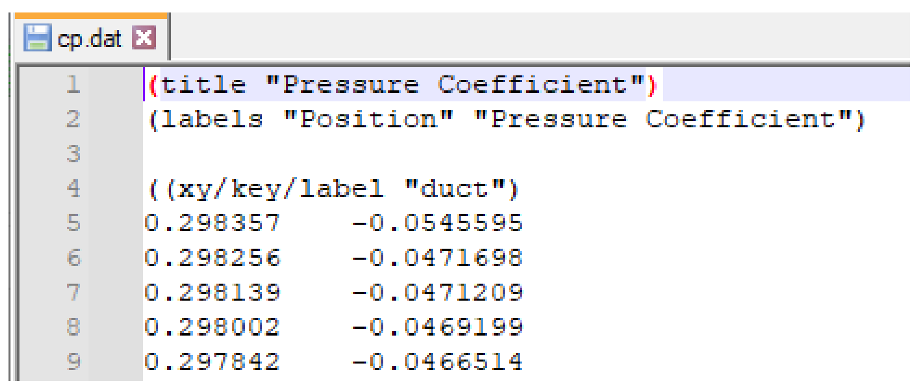
To export the plotted data for use with other plotting software:

1. Choose 'Write to File'.
2. Click 'Write…'.
3. Specify the directory and file name (for example, cp.dat), then click 'OK'.





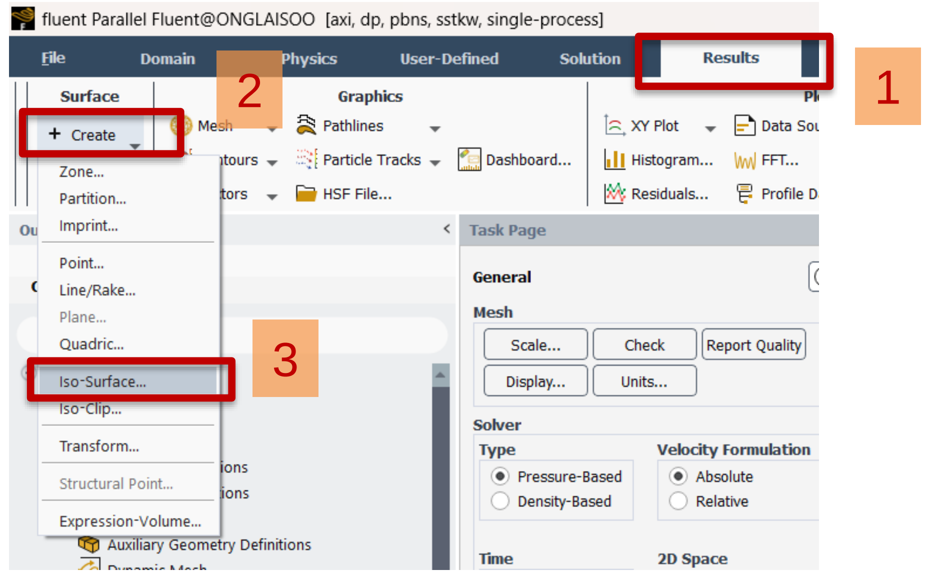
The resulting file is in plain text format.

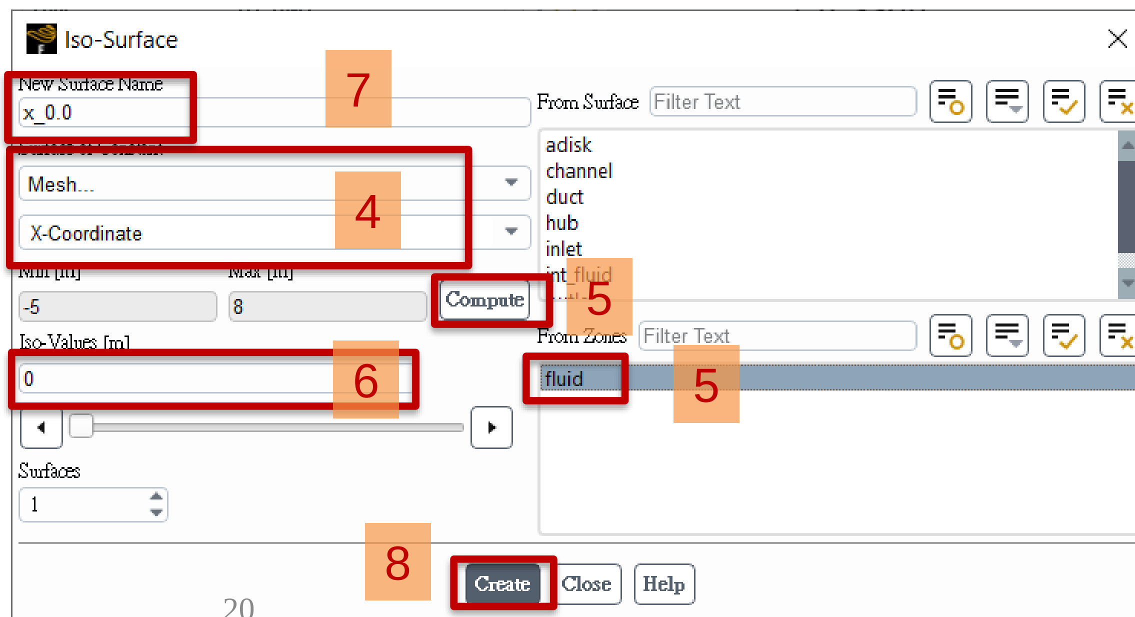


Step 6 Plot the Velocity Profile Along y Direction

To create an iso-surface at a specific location along the x-coordinate in the fluid zone:

1. Go to '**Results**' -> '**Create**' -> '**Iso-Surface**'.
2. Under 'Surface of Constant', select '**Mesh**' and then choose '**X-Coordinate**'.
3. From the “From Zones” dropdown, select '**fluid**' and then click '**Compute**' to display the maximum and minimum values of the x-coordinate.
4. To plot the profile at x=0.0, enter ‘**0**’ in the 'Iso-Values' field.
5. For 'New Surface Name', provide a different name for each new surface you create.
6. Click '**Create**'.





Step 6 Plot the Velocity Profile Along y Direction

To create an XY plot of velocity profiles at the surface created previously:

1. Go to '**Results**' -> '**Plots**' -> '**XY Plot**' and double-click.

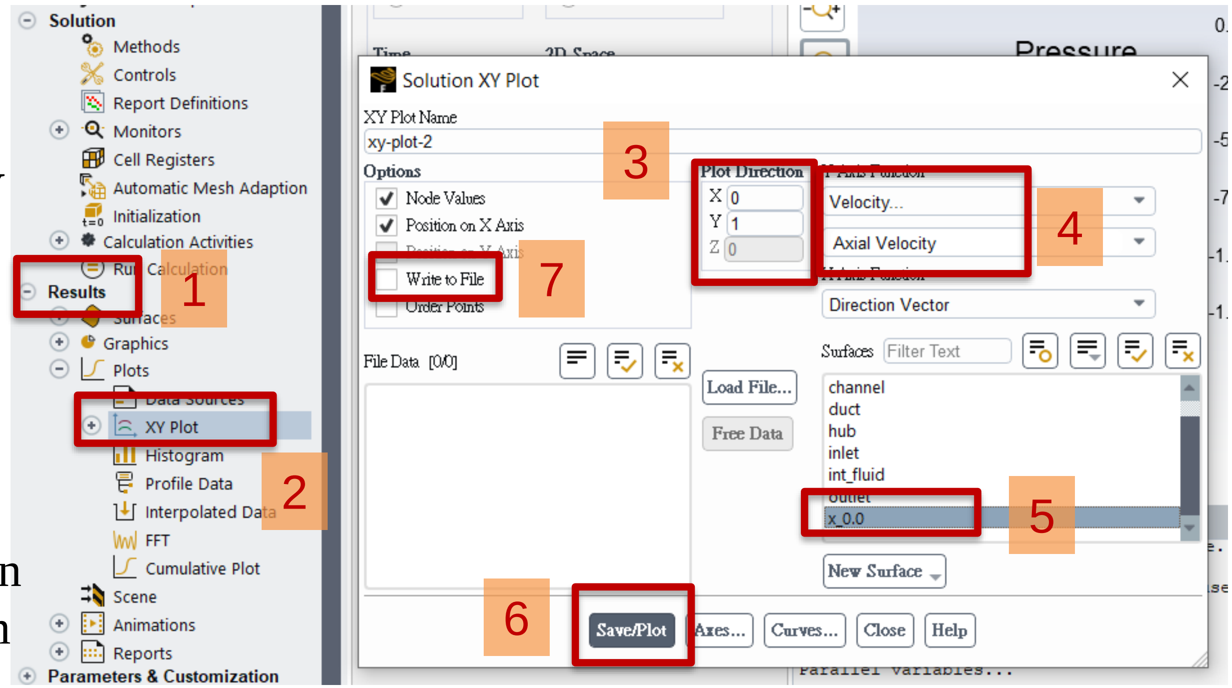
2. Set the Plot Direction to X=0, Y=1.

3. Under 'Y Axis Function', select '**Velocity**' and '**Axial Velocity**'.

4. Choose the surface you named in the previous step (e.g., x_0.0) from the 'Surfaces' list.

5. Click 'Save/Plot'.

6. If you wish to export the data, click 'Write to File', then 'Write'.

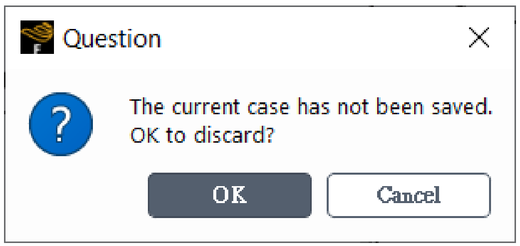


For 2D-axisymmetric simulations in Fluent where the "y" direction corresponds to the radial direction, repeat the steps outlined on pages 21 and 22 to plot velocity profiles at different axial locations.

Step 7 Exit

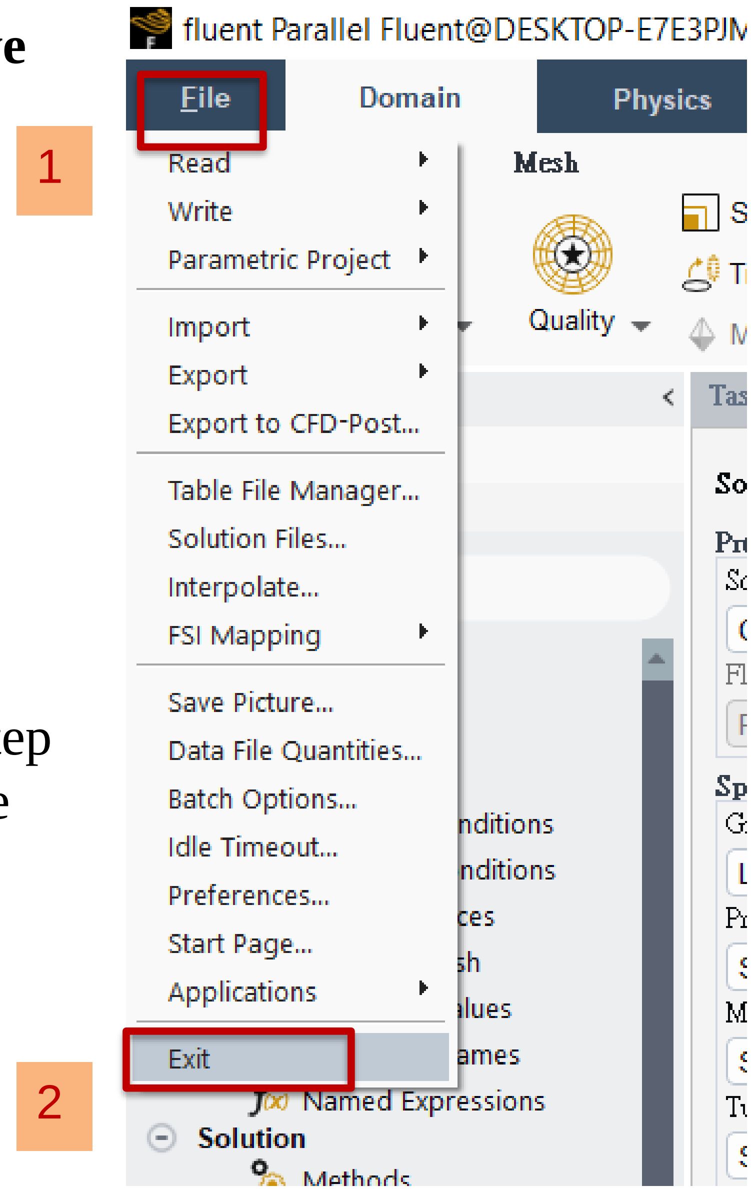
File -> Close Fluent or Close Without Save

Note: It may ask you



Just click **OK**

If you want to save the surfaces created in Step 6, you can save the case file to over-write the previous file.



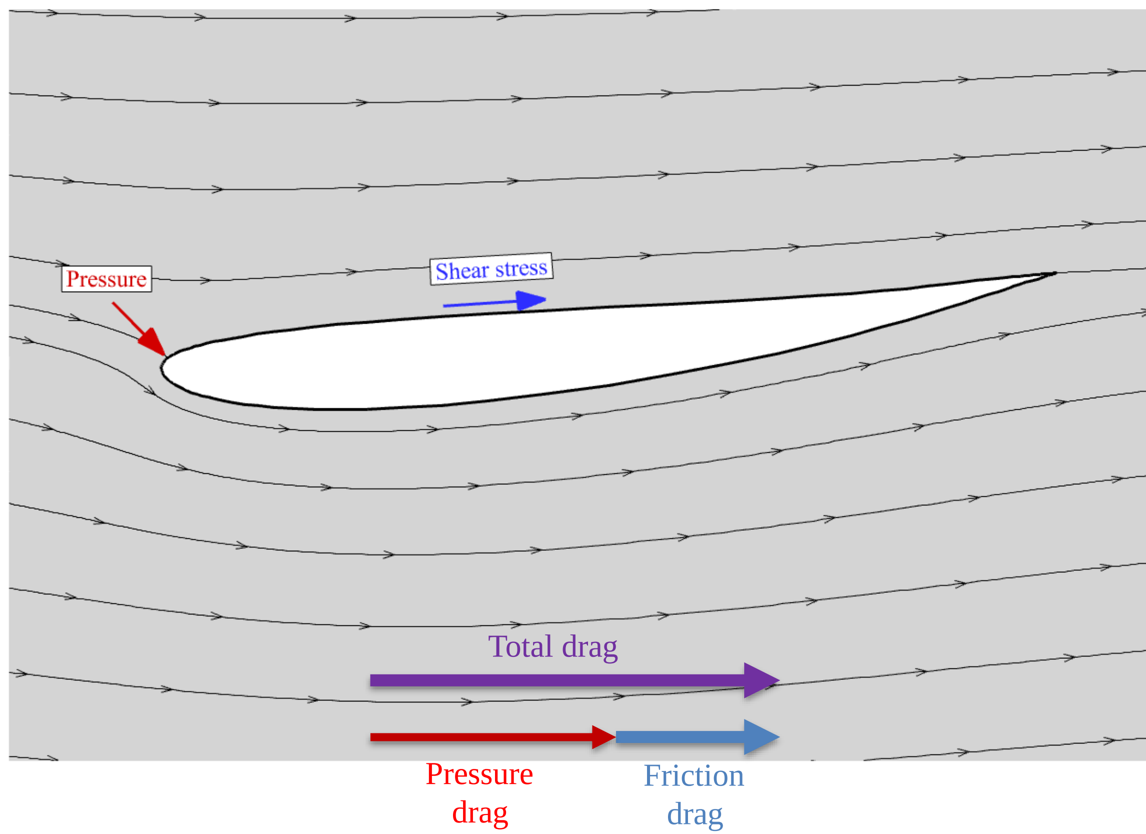
Running new velocities

1. Adjust the inflow velocity for your simulation by modifying the first line in the prepare.jou file, as detailed on page 6 of your instructions.
2. Before running a new simulation with a different inflow velocity, make sure that you delete the output files generated by the previous run. Failing to do so will cause Fluent to encounter errors when executing the journal file due to the presence of existing files.
3. To avoid this issue and to keep your results organized, **it is strongly recommended to create a new folder for each simulation run**. Naming the folder appropriately will make it more convenient to locate and analyze the results afterward.

Pressure and Friction Drag

- Pressure drag force is generated by the pressure acting on the duct surface in the normal direction. The different pressures around the surface result in drag.

- Due to the viscosity of the fluid, the frictional force is acting on the duct surface. This can be seen by observing the flow velocity slow down when the layer of fluid is close to the surface. (right on the duct wall, velocity equals zero.)



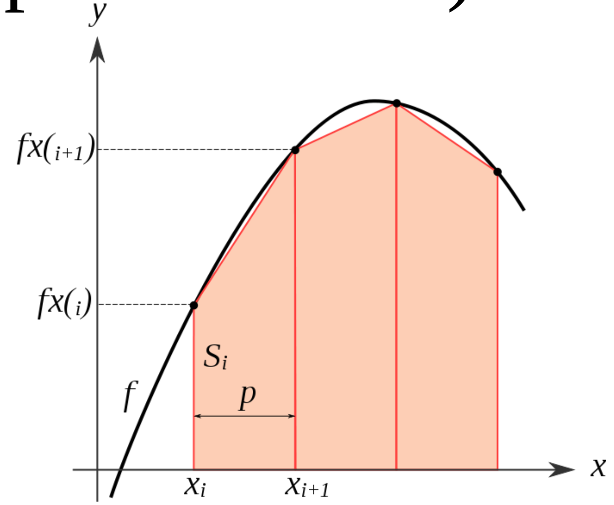
Pressure and Friction Drag Coefficient

In addition to the actual pressure, friction and total drag, the drag coefficients are also presented in the force.dat file. The drag coefficient is defined as the drag divided by $\frac{1}{2}\rho U^2 A$, where ρ, U, and A are the density, inflow velocity, and area.
The area is defined as

$$2\pi \times r \times 2C$$

Where r the turbine radius is equaled to 1, and C the chord of the duct is equaled to 0.5.

Numerical integration (trapezoidal rule)



Generally,

$$\int_a^b f(x)\,dx \approx \sum_{i=1}^{N-1} \frac{f(x_i) + f(x_{i+1})}{2} \Delta x_i$$

In this case, the flow rate Q is

$$Q = \int_{r_{hub}}^{r_{blade}} u(r)\,dA = 2\pi \int_{r_{hub}}^{r_{blade}} u(r)\,r dr \approx 2\pi \sum_{i=1}^{N-1} \left[\frac{u(r_i)r_i + u(r_{i+1})r_{i+1}}{2} \times (r_{i+1} - r_i)\right]$$

The average velocity is

$$ave(u) = \frac{Q}{A}$$

where A is the area of the hollow circular disk (don't forget to subtract the hub).

This ends the tutorial. Have fun using ***ANSYS/Fluent* !**