



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:



- 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/m³ 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.

DE2023_partA	× +	📒 U_1.0	× +
$\leftarrow \rightarrow \uparrow$	C □ → OE2023_partA →	\leftarrow \rightarrow \uparrow	C □ → OE2023_partA → U_1.0
🕂 New - 🐰	(□ (〕 (〕 (〕 (〕 (〕)	🕂 New - 🏑	(□ (Ē) 🖄 îī î\ s
A Home	Name	A Home	Name
Callery	U_1.0	Rallery	🖹 fluent.msh
> OneDrive - Perso	🖹 fluent.msh	> 🔷 OneDrive - Perso	🖹 prepare.jou
	📄 prepare.jou		🕲 run_fluent.bat
🛄 Desktop 🔹 🖈	💿 run_fluent.bat	🛄 Desktop 🔹 🖈	Ensure that the input files
🚽 Downloads 🖈	loout filoo	🛓 Downloads 🖈	are located within the
Documents *	input mes	📑 Documents 🖈	working directory.
🔀 Pictures 🔹 🖈		🔀 Pictures 🔹 🖈	



Step 2. Initialize Fluent case

📄 prepar	ejou 🔀
1	(define U 1.0)
2	;
3	;You should NOT make any changes under these linesYou 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 yes
12	<pre>/define/operating-conditions/reference-pressure-location -4.90 0</pre>
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
10	/aclus/act n w counting 34

- 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

University of Texas

at Austin

- 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.

⋈	File Edit	Selection	View Go	o Run		$\leftarrow \ \rightarrow$			s کر	earch			α
பு	≡ force	e.dat X											
P	C: > Us 1 2	ers > Thomas	> Deskto	p > OE20)23_partA >	U_1.0 > ≡ "For	force.dat ce Report"						
မို		Forces			Forces	s [N]							T -+-1
å		channel duct			(0 0 0) (12.8	ure 0) 35676 0 0)			(217.36064 0 0) (21.439784 0 0)			(217.36 (34.275
₿	8 9 10	Net			(12.8	35676 0 0)			(238.80042 0 0)			(251.63
Ľ⊘	11 12	Forces -	Directi	on Vect	or (1 0) Force	0) s [N]				Coefficients			
.	13 14 15	Zone channel duct			Press 0 12.83!	ure 5676 (Viscous 217.36064 21.439784	Total 217.3606) (34.27545	54	Pressure 0 0.0040879378	Viscous 0. 069225557 0.0068281956	Total 0.069225557 0.010916133	
	16 17 18	Net			12.83	5676	238.80042	251.6361	L	0.0046879378	0.076053752	0.08014169	
ç	19		pr	essure	drag	frictio	n drag	total drag	pre	essure drag coeffic	cient frie	ction drag coeffi	cient



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.
- For a two-dimensional simulation, select '2D' from Dimension.
- 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.

🔗 Windows Security Alert		×	NSYS Product Improvement Program
Windows Defender app	Firewall has blocked some features of this		ANSYS Product Improvement Program ANSYS Product Improvement Program helps improve ANSYS products.
Windows Defender Firewall has blocke networks. Name:	d some features of fl2310 on all public, private and domain		Participating in this program is like filling out a survey. Without interrupting your work, the software reports anonymous usage information such as errors, machine and solver statistics, features used, etc. to ANSYS. We never use the data to identify or contact you.
Publisher: U	Jnknown		The data does NOT contain:
Path: (C: \program files\ansys inc\v231\fluent \fluent23.1.0\win64\2ddp_host\fl2310.exe petworks:		 Any personally identifiable information including names, IP address, file names, part names etc. Any information about your geometry or design specific inputs
Domain networks, such as a wo	rkplace network		You can stop participation at any time. To change your selection go to Help >> ANSYS Product Improvement Program.
Private networks, such as my h	ome or work network		Yes, I am willing to participate in the ANSYS Product Improvement Program.
Public networks, such as those because these networks often l	in airports and coffee shops (not recommended have little or no security)		No, I would not like to participate. For more information about the ANSYS Privacy Policy, please check:
What are the risks of allowing an app	through a firewall?		http://www.ansys.com/privacy
	Allow access Cancel		Ok



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

Task Page	<		N AN
General	?		
Scale Check Report Quality Display Units Solver Type • Pressure-Based • Density-Based • Relative			
Time 2D Snace			
Mesh Display			×
Options Edge Type Nodes All Edges Feature Faces Outline Partitions Outline Overset Shrink Factor Feature Angle Outline Interior Interior	xt		
Adjacency	- J		
2 Display Colors.	Close	e) [Help	

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.

fluent Parallel Fluent@ONGLAISOO [axi, dp, pbns, sstkw, single-process]							
<u>F</u> ile Domain Physics User-De	efined Solution Results V	iew Parallel Design	Parametric 🔺		Q Quick Search (Ct 🕜 📑	/\nsys	
Mesh Image: Scale of the second s	Zones Separate → □ Delete Data → Adjacency → Adjacency → Activate	Interfaces Mesh Models Image: State St	Turbomachinery Turbo Models Turbo Workflow Turbo Create Turbo Topology L. Spectral Content Spectral Content.	Adapt Manual Manual Automatic Controls Manage	Surface + Create _ & Manage		
Outline View < Case View Filter Text Setup General	Task Page < General ⑦ Mesh Scale Check Report Quality Display Units		,	/lesh		X SS SR1	
 ♥ Models ● ▲ Materials ● □ Cell Zone Conditions ● ■ Boundary Conditions 	Solver Velocity Formulation • Pressure-Based • Absolute Density-Based Relative					8 8 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
 Dynamic Mesh Reference Values 1/2, Reference Frames Solution Methods 	Ime 2D Space • Steady Planar • Transient • Axisymmetric • Gravity Gravity						
 Controls Report Definitions Call Registers Automatic Mesh Adaption Initialization Calculation Activities 		Image: Constraint of the second secon	ion to be zoomed	Release			
 ⇒ Run Calculation → Results → Surfaces → Graphics → U Plots ⇒ Scene 					0 selected all		



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.

fluent Parallel Fluent@ONGLAISOO [axi, dp, pbns, sstkw, single-process] — Ō X						
<u>F</u> ile Domain Physics User-Do	efined Solution Results V	iew Parallel Design	Parametric 🔺		Q Quick Search (Ct	🕐 📕 🖊 nsys
Mesh Image: State of the	Zones Combine E Delete Delete Append Separate E Deactivate Append Append	Interfaces Mesh Models Image: State of the state of	Turbo Models ∯ Turbo Workflow ✓ □ Turbo Create Image: Turbo Topology Image: Turbo Topology ↓ Spectral Content Image: Turbo Topology Image: Turbo Topology	Adapt Manual 📢 Automatic X Controls 🔬 Manage	Surface + Create & Manage	
Outline View <	Task Page <		N	1esh		×
Case View	General (7) Mesh					
 Setup General Models Image: Conditions Cell Zone Conditions Electronic Conditions Boundary Conditions Mesh Interfaces Auxiliary Geometry Definitions Image: Conditions Auxiliary Geometry Definitions Oynamic Mesh Reference Values Lefference Frames Named Expressions Solution Methods Controls Report Definitions Cell Registers Automatic Mesh Adaption Cell Registers Automatic Mesh Adaption Calculation Atvitites Results Surfaces Surfaces 	Display Units Solver Ype Ype Oelocity Formulation Opensity-Based Absolute Relative Relative Time 2D Space Steady Planar Transient Axisymmetric Swirl Gravity Gravity		Release			
					0 selec	ted all 💌



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.

 Named Expressions Solution Methods Controls 	Density-Based Relative		×
 Report Definitions Monitors Cell Registers Automatic Mesh Adaption Initialization Calculation Activities Run Calculation Results 	XY Plot Name xy-plot-1 Options ✓ Node Values ✓ Position on X Axis Position on Y Axis Write to File Oueler Points	Plot Direction Y Axis Function X 1 Pressure Y 0 Pressure Coefficient A Axis Function	
Surfaces Graphics C Plots Plots C Y Plot C Y Plot From Profile Data C Interpolated Data W FFT		Surfaces Filter Text Load File Free Data Free Data Surfaces Filter Text adisk Surfaces Filter Text adisk Surfaces Filter Text adisk Surfaces	
Cumulative Plot Comparis Cumulative Plot Cumulative Plot Animations Parameters & Customization	4 Save/Plot	Axes) Curves) Close Help	se

Fall 2023

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'.

Solution XY Plot	:			×
XY Plot Name				
xy-plot-1				
Options		Plot Direction	Y Axis Function	
✓ Node Values		X1	Pressure	•
🖌 Position on X Axis		YO		
Position on V Axis		Z O	Pressure Coefficient	-
✓ Write to File	7		X Axis Function	
Order Points	1		Direction Vector	-
File Data [0/0]	₹ ₹	Load File Free Data	Surfaces Filter Text	
	Write	Axes) Curves] Close Help	

? \times 🛸 Select File 🗈 🖓 🎉 🗉 🗐 C:\Users\Thomas\Desktop\OE2023_partA\U_1.0 - G Look in: Size **Date Modified** Type Name My Computer cleanup-fluent-ONGLAISOO-4988.bat 575 bytes bat File 11/7/2...:29 PM Thomas debug.log 11/7/2...:29 PM 182 bytes log File 🛸 fluent.cas 2.39 MiB cas File 11/6/2...:58 PM fluent.dat 5.25 MiB dat File 11/6/2...:58 PM fluent.msh 3.89 MiB msh File 11/6/2...:02 PM 11/6/2...:58 PM fluent-20231106-235745-20252.trn 22.15 KiB trn File fluent-20231107-132912-16444.trn 11/7/2...:45 PM 6.68 KiB trn File force.dat 2.59 KiB dat File 11/7/2...:08 AM output.le 21.89 KiB log File 11/7/2...:00 AM prepate 1.53 KiB jou File 1/6/20...:50 PM 🔏 run_fluent.bat 120 bytes bat File 11/6/2...:50 PM XY File cp.dat ОК Files of type: All Files (*) \mathbf{v} Cancel Filter String Filter

The resulting file is in plain text format.

📄 cp.d	lat 🗵
1	(title "Pressure Coefficient")
2	(labels "Position" "Pressure Coefficient")
3	
4	((xy/key/label "duct")
5	0.298357 -0.0545595
6	0.298256 -0.0471698
7	0.298139 -0.0471209
8	0.298002 -0.0469199
9	0.297842 -0.0466514

19

 \times

| ₽_×



Step 6 Plot the Velocity Profile Along y Direction

Surfaces

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

- 1. Go to '**Results**' -> '**Create**' -> '**Iso-Surface**'.
- 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'.

解 fluent Parallel Fluent@ONGLAISOO [axi,	dp, pbns, sstkw, single-proces	ss]	
Eile Domain Physics Surface 2 Gr. + Create Mesh 2 Zone Partition tours Partition tors Image: HSF	User-Defined Solu aphics ines • cle Tracks • 🏠 Dashboard File	Ition Results Pi [☆, XY Plot ↓] Data Sou] Histogram W FFT W Residuals 문 Profile D	1
Ou Imprint Point Line/Rake Plane Quadric Iso-Surface Iso-Clip Transform Structural Point Expression-Volume Muxiliary Geometry Definitions	 Task Page General Mesh Scale Display Solver Type Pressure-B Density-Ba Time 	Check Report Quality Units Units Units Uelocity Formulation Absolute Relative 2D Space	
Iso-Surface	From Surface Fi	lter Text	
5 8 5-Values [m] 6	Compute From Zones Filt	er Text) 🖥 🗮 🏹

8

20

Create

Help

Close



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'.







Step 7 Exit

1

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.



2



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.





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* !