

A Complementary Material for CE358 Introductory Ocean Engineering

ANSYS/Fluent v252:

A Quick Step into CFD

Spyros A. Kinnas (Fall 2025)

For running Fluent on Student PCs

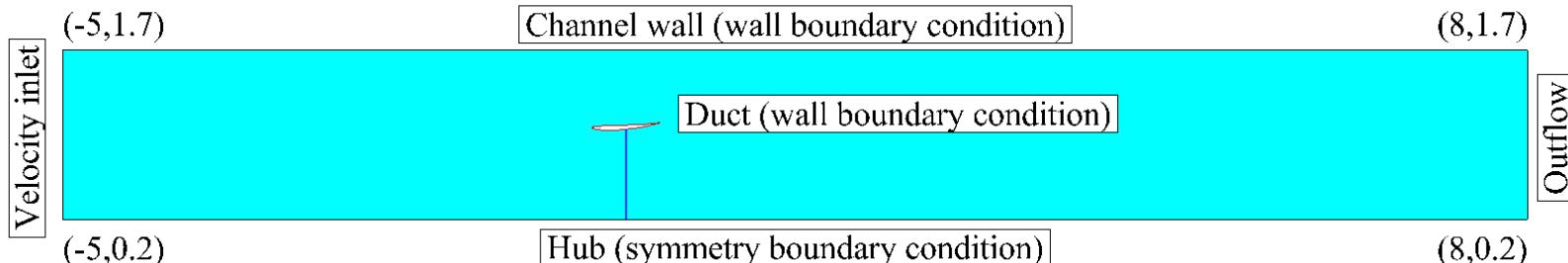
What is *ANSYS/Fluent*

- ◆ 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



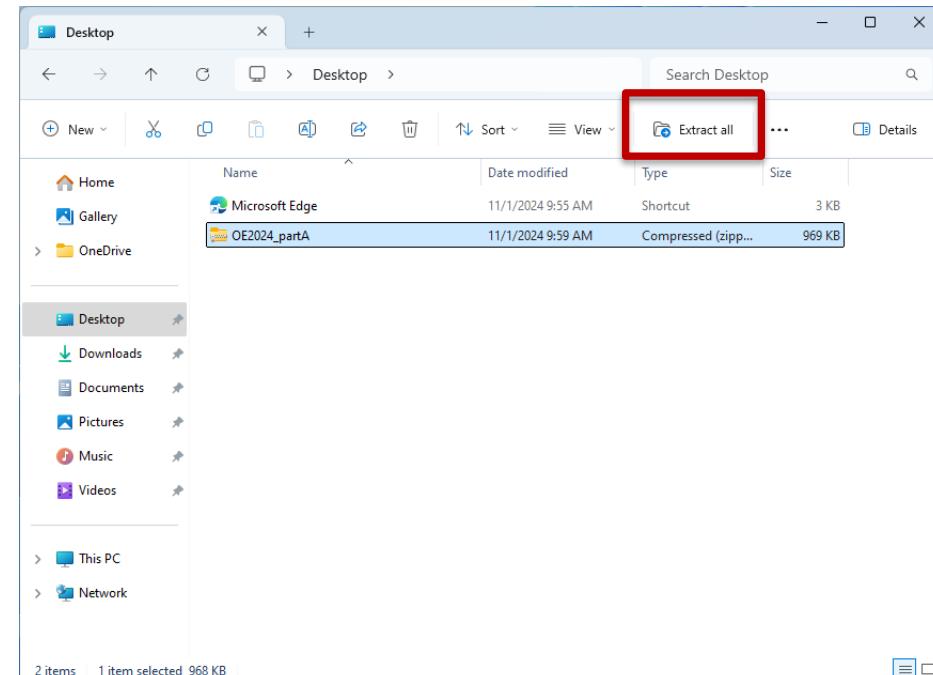
- The sample run in this power point presentation is for the above duct at $U = 1 \text{ m/s}$, with $\rho = 999.965 \text{ Kg/m}^3$ and $\mu = 0.001519 \text{ Pa sec}$.
- The CAEE's water flume is 30 cm * 30 cm, and the radius of the turbine blade is 10 cm. In the CFD model, the radius of the turbine 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 Project Package:
 - Location: Term project website
 - File name: "OE2025_PartA.zip"

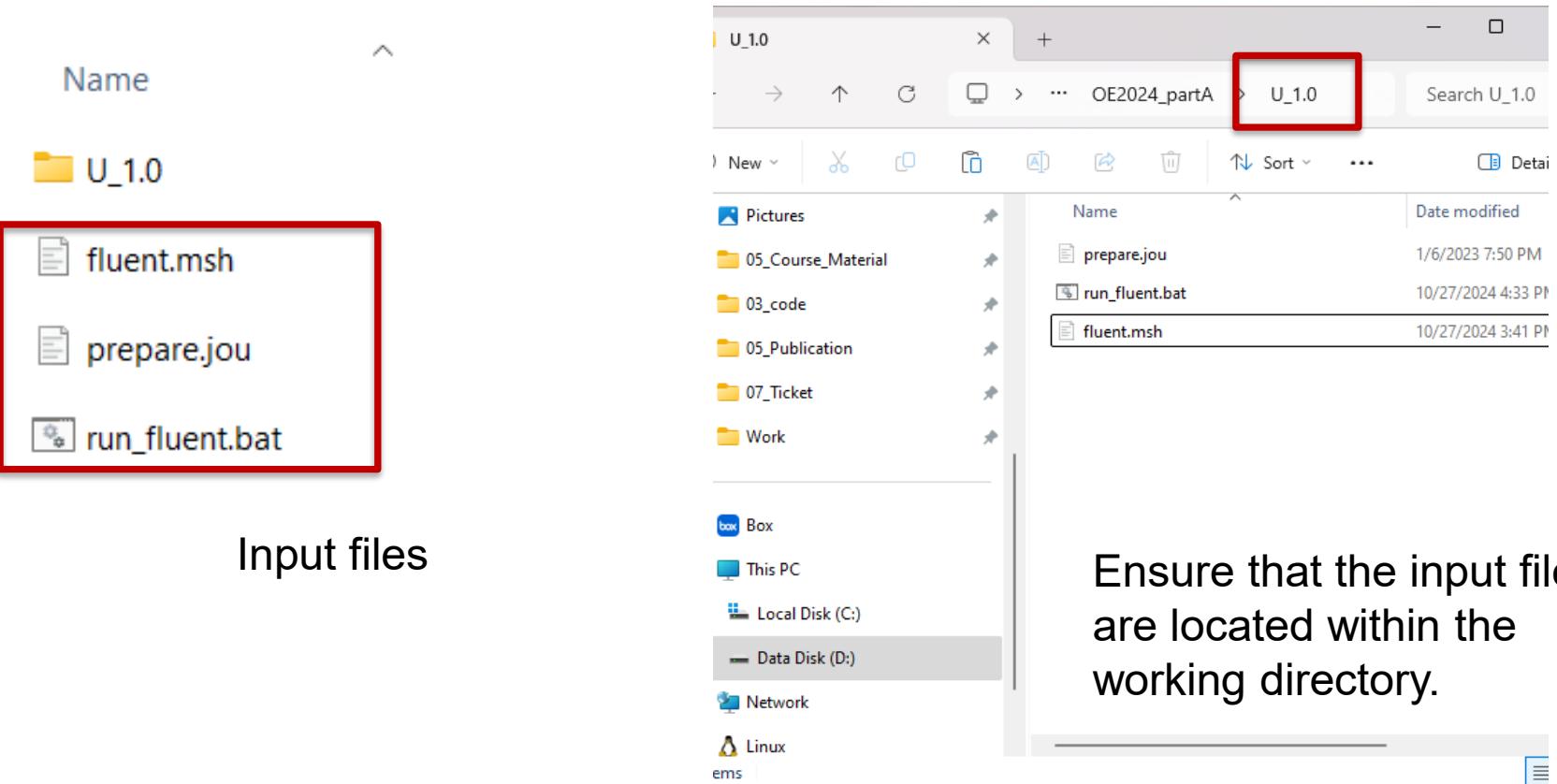
- ◆ Extract Files:
 - Select "Extract All..."

- ◆ Important Files Included:
 - fluent.msh - Fluent mesh file - Contains the pre-made computational grid
 - prepare.jou - Journal file - Automates CFD case setup and execution
 - run_fluent.bat - DOS batch file - Automates the case running process

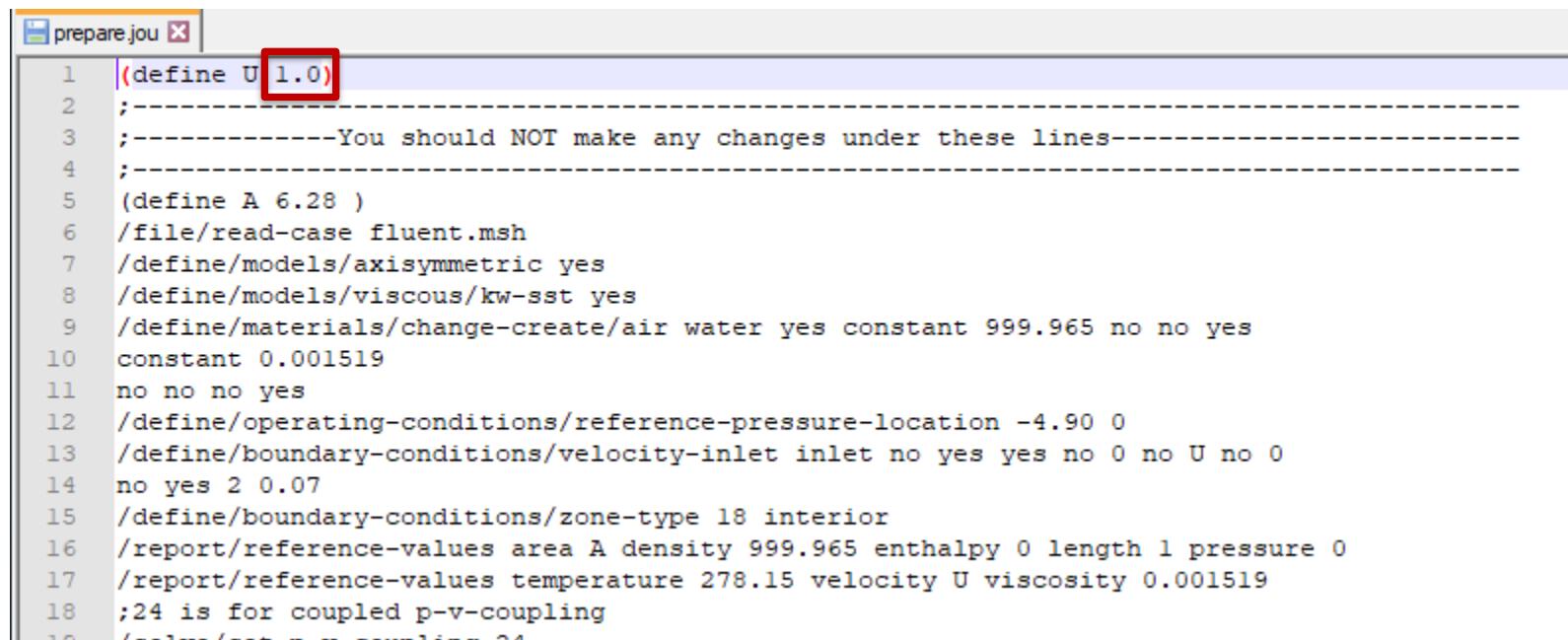


Step 2. Initialize Fluent case

- Before starting your analysis, create a new folder with a clear name (e.g., U_1.0) and copy all the extracted files into it. This will help keep your project files organized and easily accessible.



Step 2. Initialize Fluent case



```
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 /coupling/set n n coupling 24
```

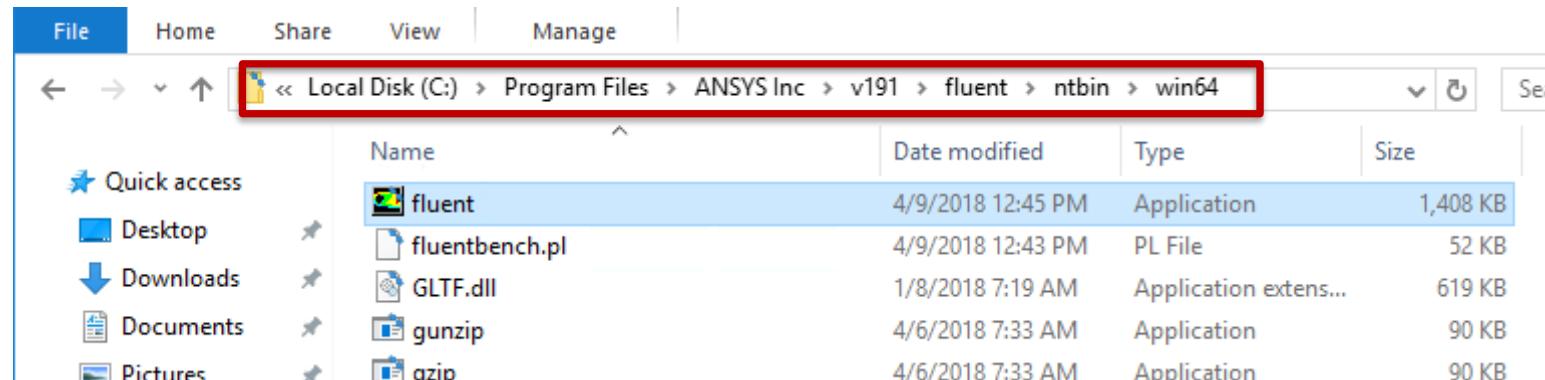
- ◆ Edit the input journal file to change the velocity
 - Right-click on **prepare.jou**
 - Edit with Notepad++
 - On line 1, change the inflow velocity
 - Very important: don't change anything else in this journal file unless you are perfectly sure what you are doing.

Step 3. Run Fluent case (Important!!)

The “run_fluent.bat” file can be accessed and modified by right-clicking on it and selecting “Edit.” Inside the “run_fluent.bat” file, you will find an address indicating the location of the Fluent executable installed on your computer. This address acts as a convenient shortcut for launching the software. For users who have installed the student version of Fluent in the default directory, please copy the following line and use it to replace the existing line in the "run_fluent.bat" file.

```
"C:\Program Files\ANSYS Inc\ANSYS Student\v252\fluent\ntbin\win64\fluent.exe" 2ddp -g -t1 -i "prepare.jou" > output.log
```

In cases where different machines may have varying versions of Fluent installed, it's important to note that the path to the executable may differ from the default one specified. A good check is to find the location of the fluent executable and check the path before you run the case. For instance, if you found the path to the fluent executable to be like the following,

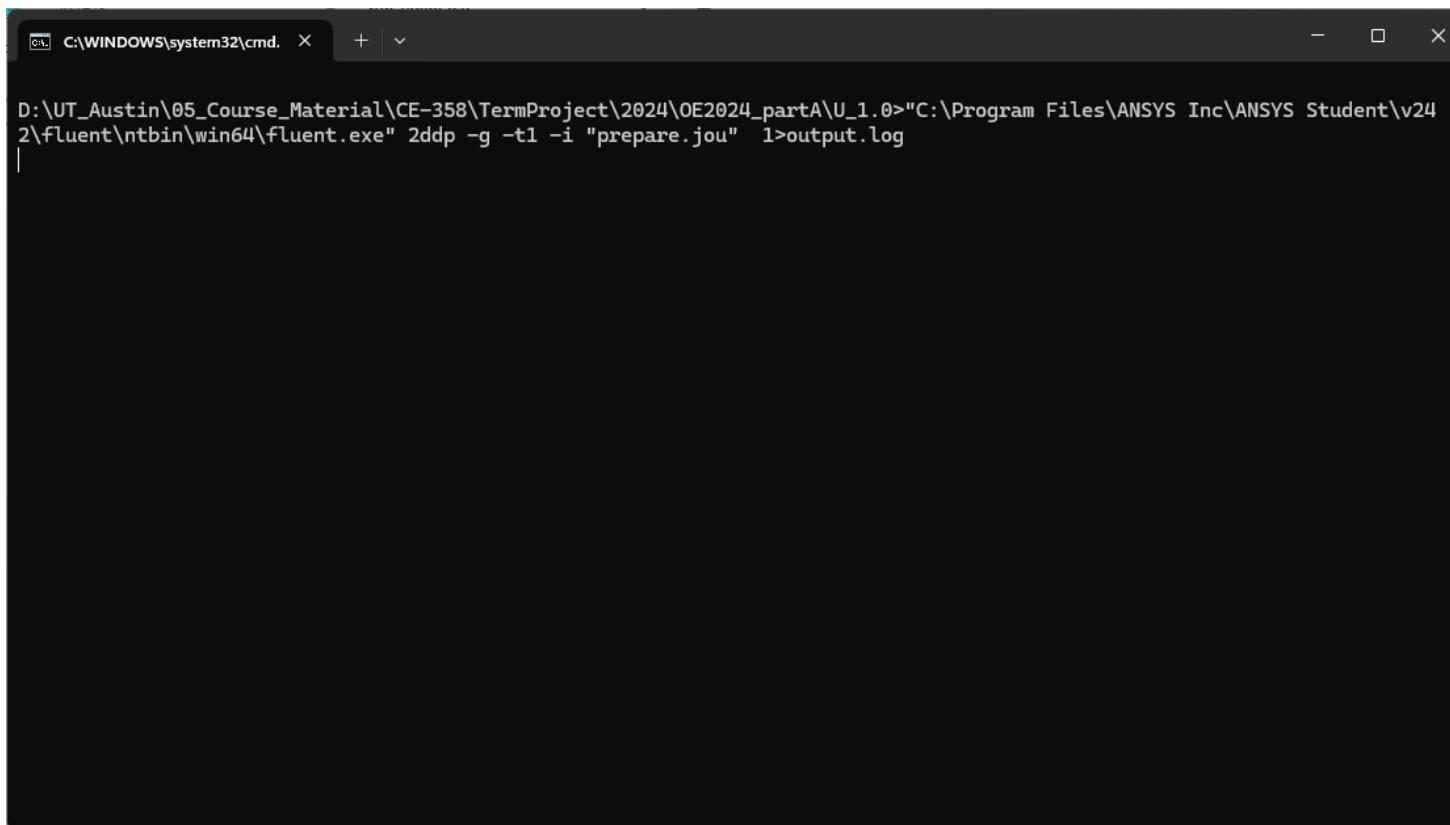


then you should change the path in the run_fluent.bat accordingly as

```
"C:\Program Files\ANSYS Inc\v191\fluent\ntbin\win64\fluent.exe" 2ddp -g -t1 -i "prepare.jou" > output.log
```

Step 3. Run Fluent case

- ◆ 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.



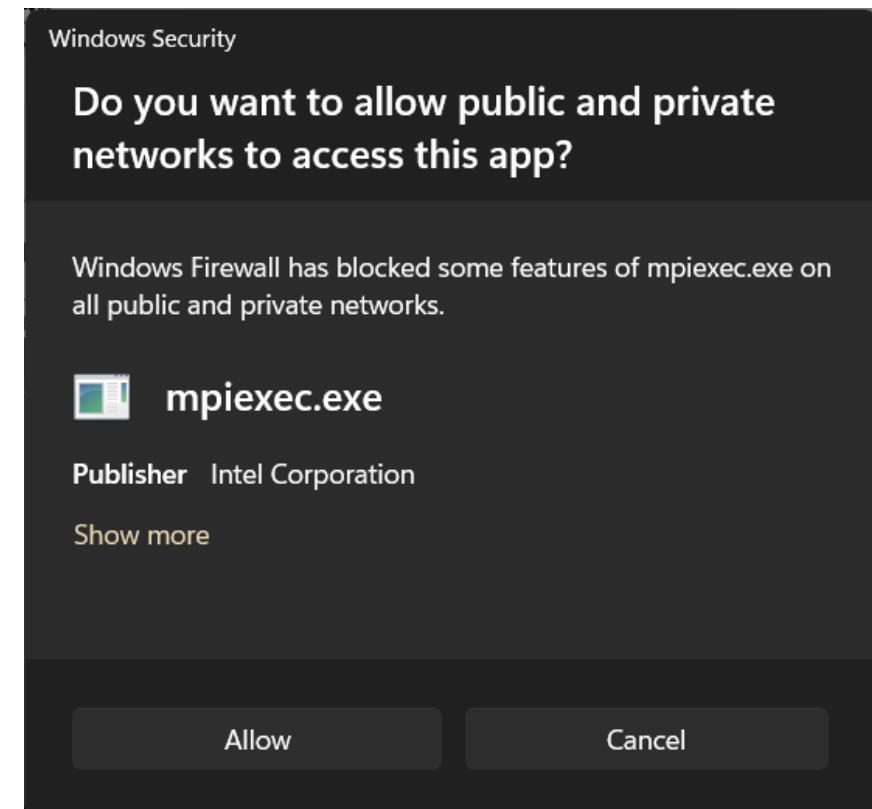
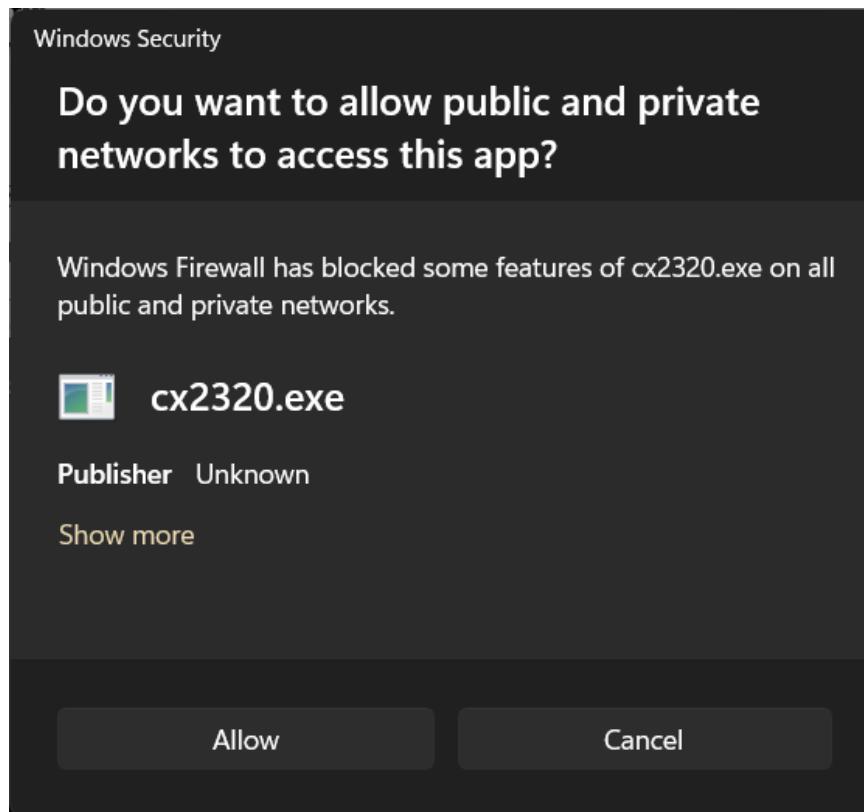
A screenshot of a Windows Command Prompt window titled "C:\WINDOWS\system32\cmd.". The window contains the following command:

```
D:\UT_Austin\05_Course_Material\CE-358\TermProject\2024\0E2024_partA\U_1.0>"C:\Program Files\ANSYS Inc\ANSYS Student\v242\fluent\ntbin\win64\fluent.exe" 2ddp -g -t1 -i "prepare.jou" 1>output.log
```

The command is intended to run the Fluent solver with specific parameters: 2ddp, -g, -t1, -i "prepare.jou", and 1>output.log. The window has a dark theme and is set against a white background.

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.log
 6. .trn file (Fluent console log file.
Not used in this study)

Name	Date modified	Type
fluent.cas.h5	11/3/2025 7:21 PM	H5 File
fluent.dat.h5	11/3/2025 7:21 PM	H5 File
fluent.msh	11/3/2025 7:19 PM	MSH Fi
fluent-20251103-191952-12088.trn	11/3/2025 7:21 PM	TRN Fil
fluent-20251103-192125-25792.trn	11/3/2025 9:07 PM	TRN Fil
force.dat	11/3/2025 7:21 PM	DAT Fil
output.log	11/3/2025 7:21 PM	Text Dc
prepare.jou	11/3/2025 6:40 PM	JOU Fil
run_fluent.bat	11/3/2025 6:40 PM	Window
velocity.dat	11/3/2025 7:21 PM	DAT Fil

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					
Coefficients					
Zone	Pressure	Viscous	Viscous	Total	Total
Pressure channel (0 0 0)	(0 0 0)	(0.069256349 0 0)	(217.45733 0 0)	(0.069256349 0 0)	(217.45733 0 0)
duct (0.0035388036 0 0)	(11.111454 0 0)	(0.0068894455 0 0)	(21.632102 0 0)	(0.010428249 0 0)	(32.743556 0 0)
<hr/>					
<hr/>					
Net (0.0035388036 0 0)	(11.111454 0 0)	(0.076145795 0 0)	(239.08943 0 0)	(0.079684598 0 0)	(250.20088 0 0)
<hr/>					
Forces - Direction Vector (1 0 0)					
Forces [N]					
Zone	Pressure	Viscous	Total	Coefficients	
channel	0	217.45733	217.45733	Pressure	0.069256349
duct	11.111454	21.632102	32.743556	Viscous	0.0068894455
Net	11.111454	239.08943	250.20088	Total	0.010428249
<hr/>					

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.

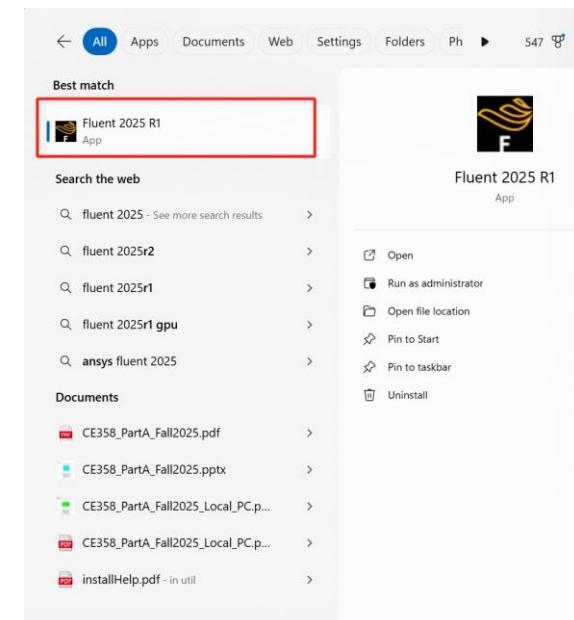
```
(title "Axial Velocity")
(labels "Position" "Axial Velocity")
((xy/key/label "adisk")
 0.2      1.13043
 0.233169  1.13067
 0.264931  1.1313
 0.295347  1.13223
 0.324473  1.13343
 0.352364  1.13485
 0.379072  1.13648
 0.404648  1.13828
 0.42914   1.14023
 0.452593  1.14233
 0.475052  1.14456
 0.496559  1.14691
 0.517154  1.14936
 0.536875  1.15191
 0.555761  1.15455
 0.573846  1.15728
 0.591164  1.16007
 0.607747  1.16293
 0.623628  1.16585
 0.638835  1.16882
 0.653398  1.17182
 0.667343  1.17486
 0.680697  1.17792
 0.693485  1.18099
 0.705731  1.18408
 0.717457  1.18719
 0.728686  1.19029
 0.73944   1.19333
```

Ignore the header lines.

... ...
r *u*

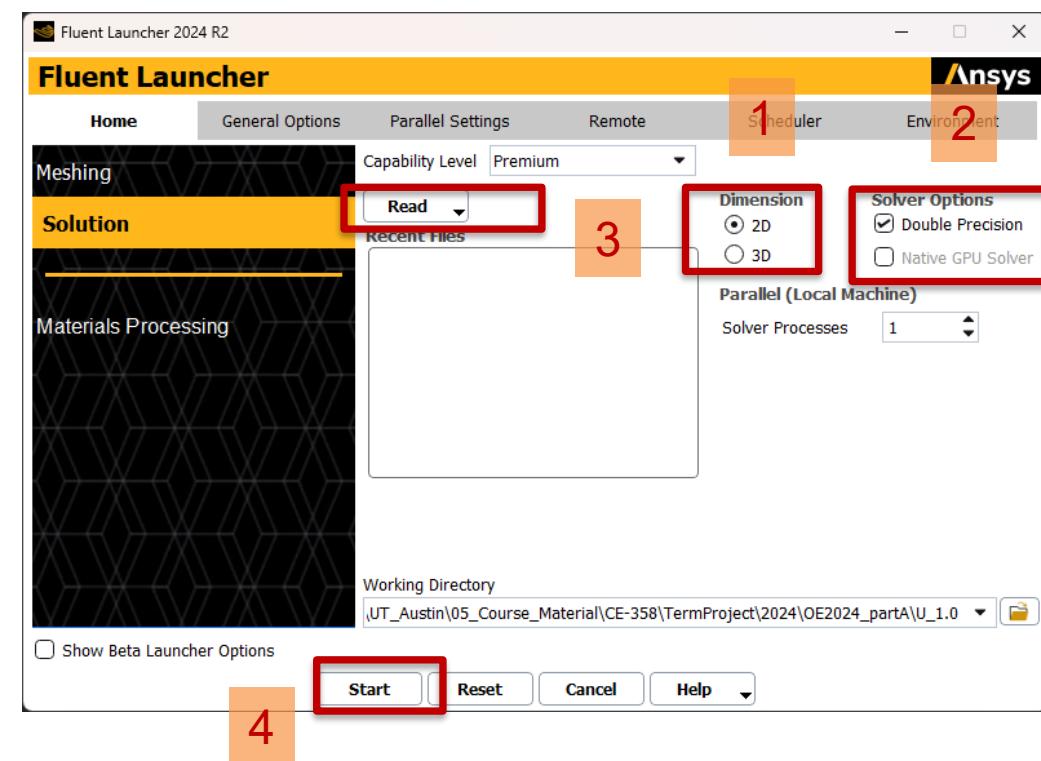
Step 4. Check the results

- ◆ To check additional results, you need to open the ANSYS/Fluent graphical user interface (GUI). You can find Fluent under ANSYS 2025 R2 in the start menu by following these steps:
 - ◆ 1. Go to the Start Menu.
 - ◆ 2. Navigate to ANSYS 2025 R2.
 - ◆ 3. Select Fluent 2025 R2 to launch the program.



Step 4 Start ANSYS FLUENT (V25.2)

- 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.
 - Then select '**Case and Data**' to load your case file.
 - Click **Start**



Step 4 Getting started with ANSYS Fluent (V25.2)

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

The image shows two windows side-by-side. On the left is a 'Windows Security Alert' window from Windows Defender Firewall. It displays a message about a blocked app (fl2310) and provides details like Name, Publisher, and Path. It also asks if the user wants to allow the app to communicate on different network types. On the right is an 'ANSYS Product Improvement Program' window. It explains the program's purpose, the data it collects (which is anonymous), and participation options. A red box highlights the 'X' button in the top right corner of this window, indicating it should be closed.

Windows Security Alert

Windows Defender Firewall has blocked some features of this app

Windows Defender Firewall has blocked some features of fl2310 on all public, private and domain networks.

Name: fl2310
Publisher: Unknown
Path: C:\program files\ansys inc\v231\fluent\fluent23.1.0\win64\2ddp_host\fl2310.exe

Allow fl2310 to communicate on these networks:

Domain networks, such as a workplace network

Private networks, such as my home or work network

Public networks, such as those in airports and coffee shops (not recommended because these networks often have little or no security)

[What are the risks of allowing an app through a firewall?](#)

ANSYS Product Improvement Program

ANSYS Product Improvement Program helps improve ANSYS products. 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.

The data does NOT contain:

- Any personally identifiable information including names, IP address, file names, part names etc.
- Any information about your geometry or design specific inputs.

You can stop participation at any time. To change your selection go to Help >> ANSYS Product Improvement Program.

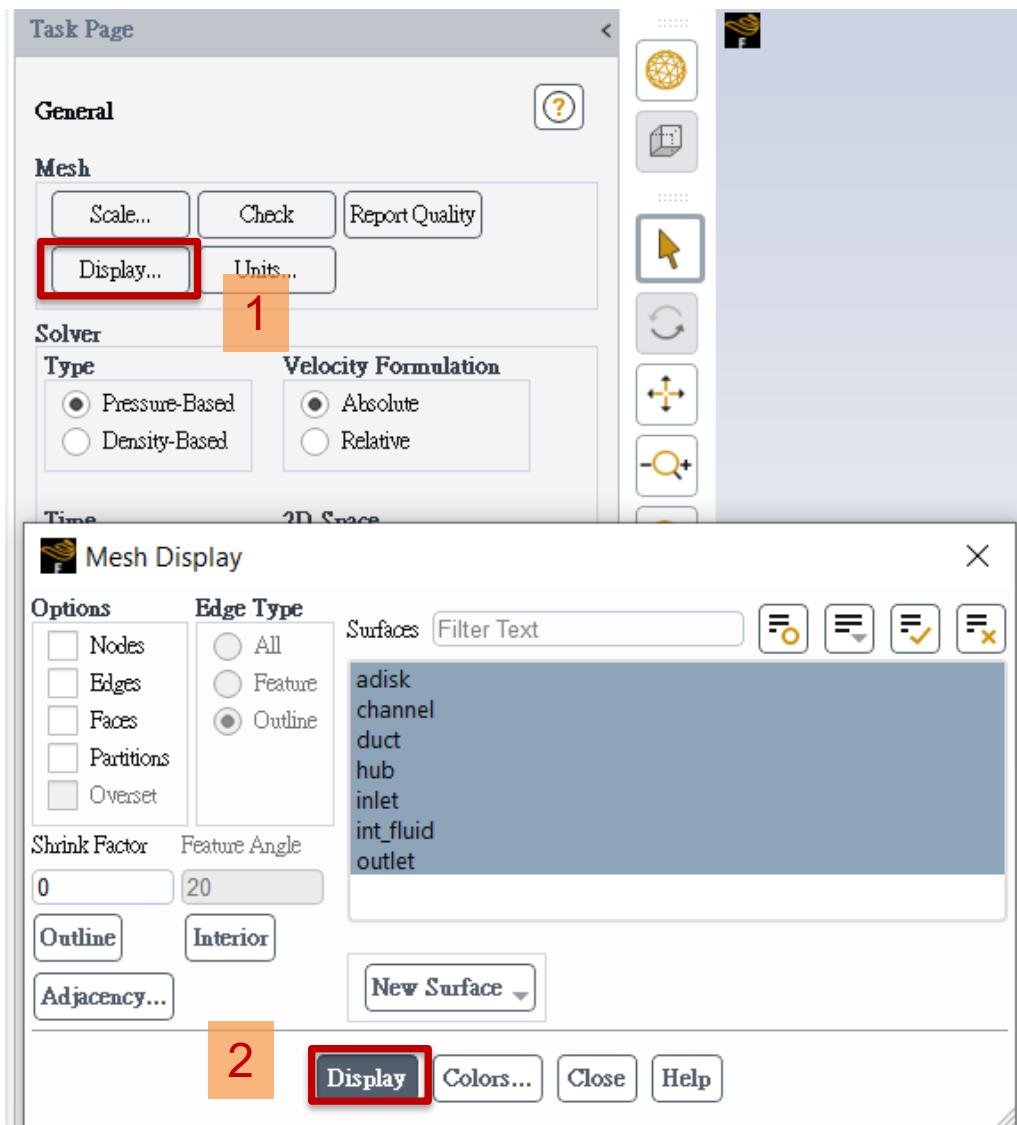
Yes, I am willing to participate in the ANSYS Product Improvement Program.

No, I would not like to participate.

For more information about the ANSYS Privacy Policy, please check:
<http://www.ansys.com/privacy>

Ok

Step 4 Getting started with ANSYS Fluent (V25.2)

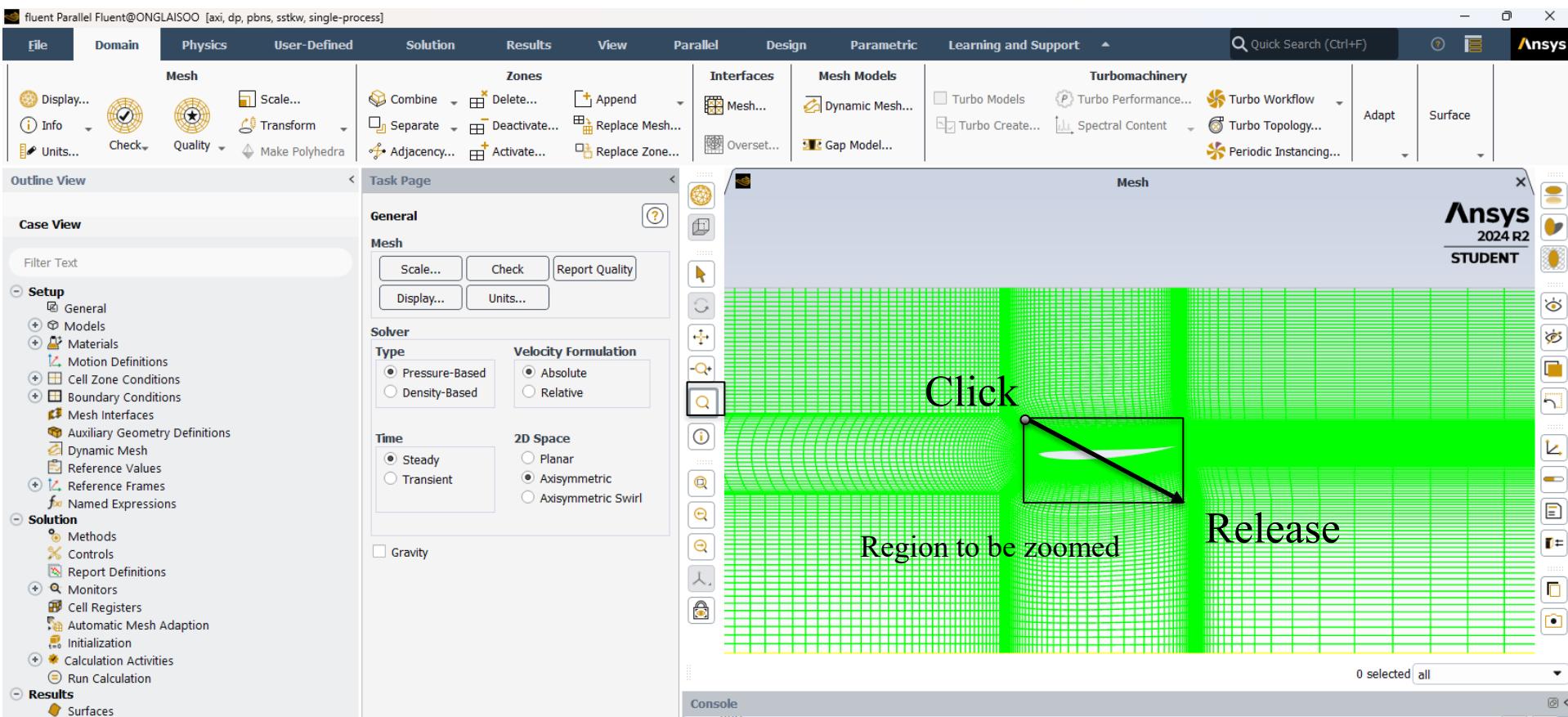


To visualize the mesh and your results:

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

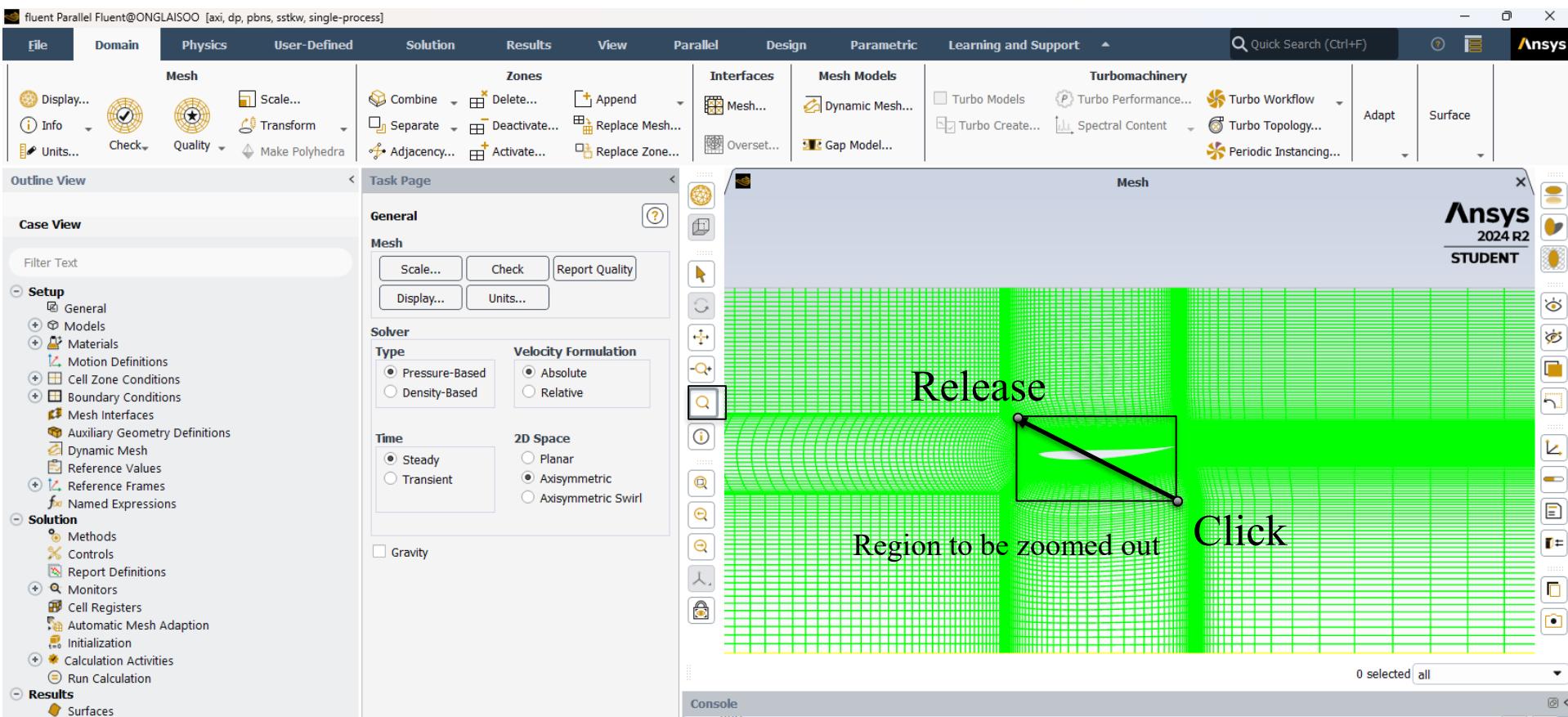
Step 4 Getting started with ANSYS Fluent (V25.2)

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



Step 4 Getting started with ANSYS Fluent (V25.2)

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



Step 5 Plot Pressure Coefficient

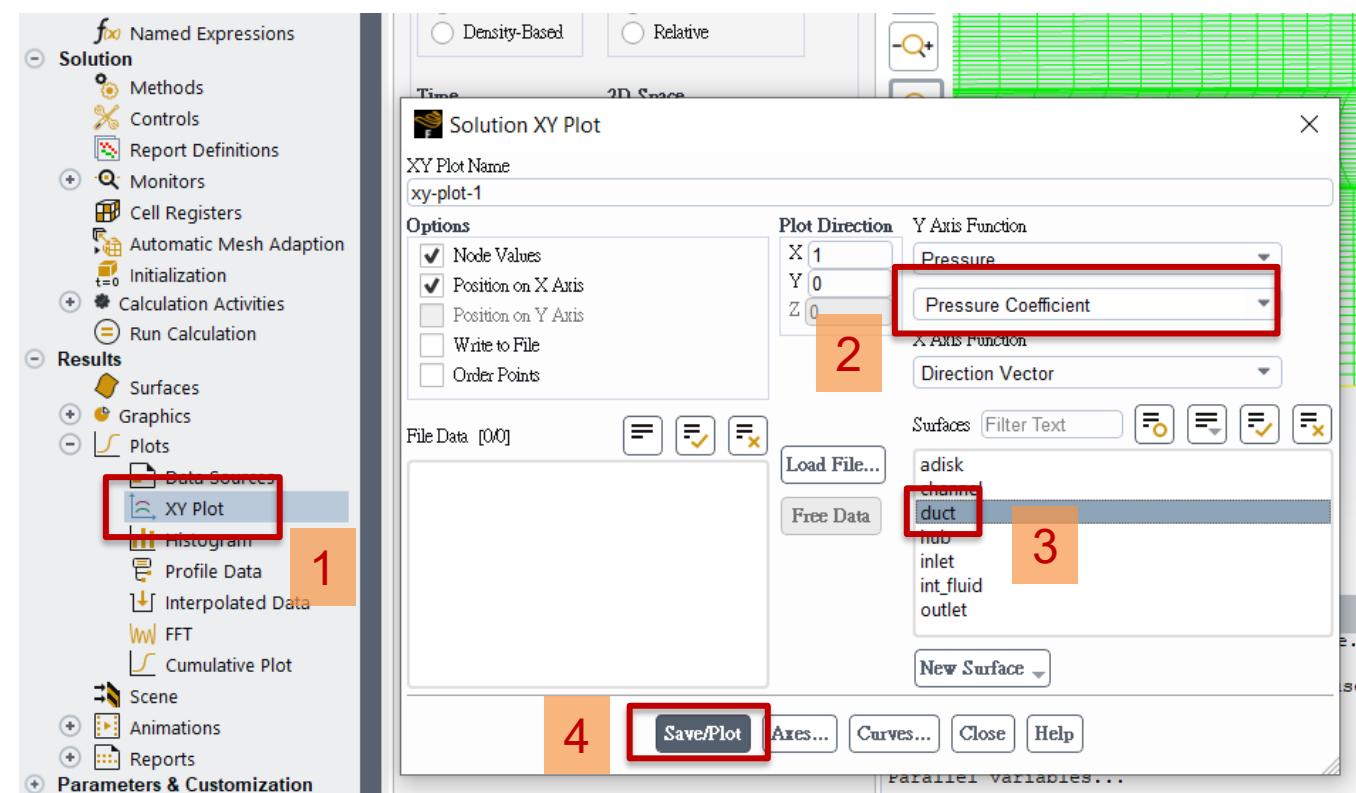
To generate an XY Plot of pressure and pressure coefficient distributions over the duct:

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

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

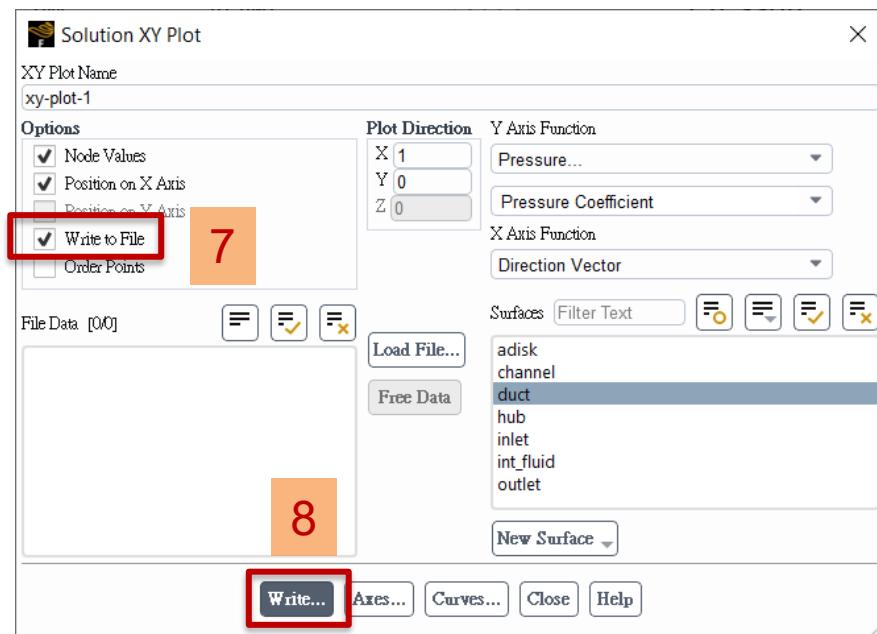
where p is the pressure, ρ is the density, p_∞ and u_∞ are the pressure and velocity far upstream.



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'.

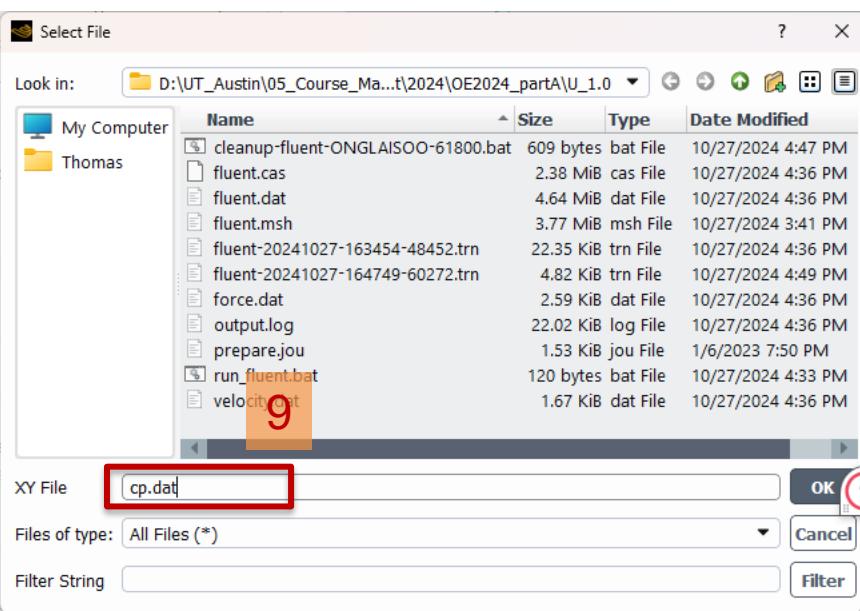


The resulting file is in plain text format.

```

cp.dat
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

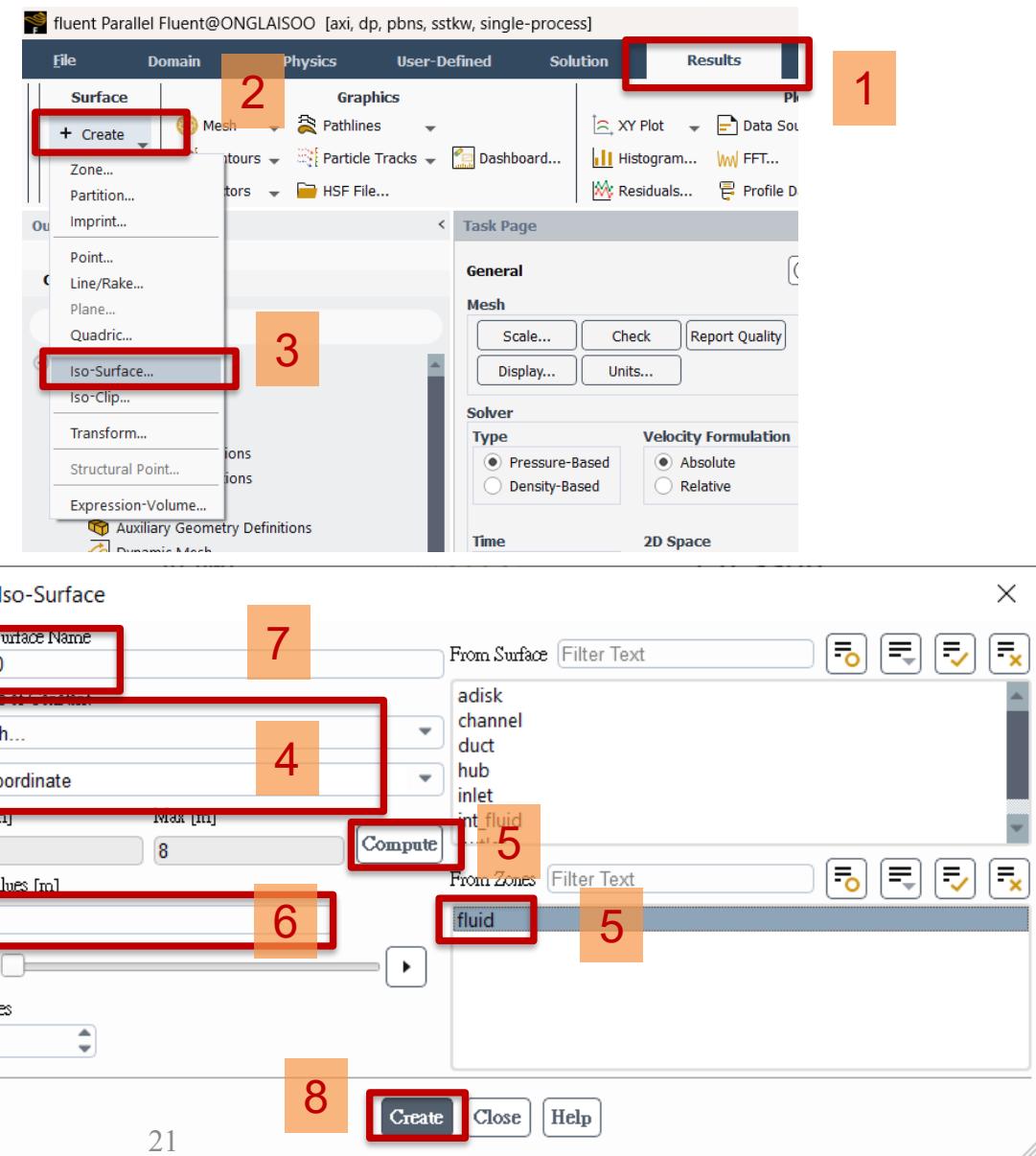
```



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:

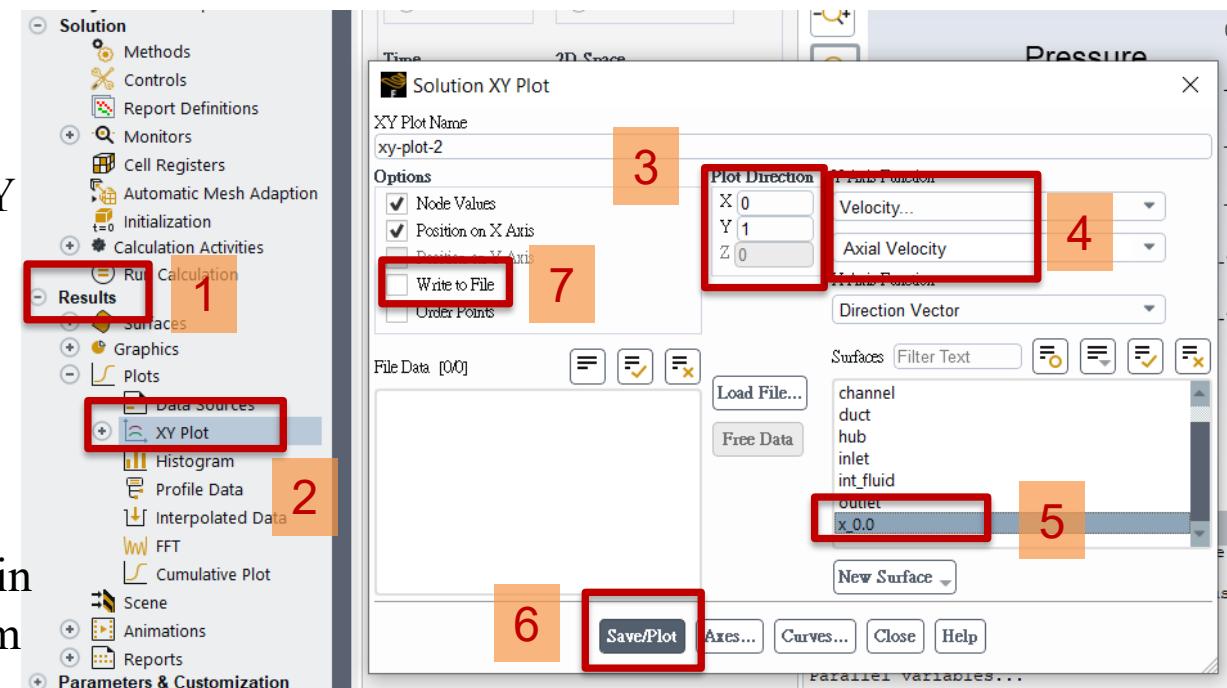
1. Go to 'Results' -> 'Create' -> 'Iso-Surface'.
2. Under 'Surface of Constant', select 'Mesh' and then choose 'X-Coordinate'.
3. From the “From Zones” dropdown, select 'fluid' and then click 'Compute' to display the maximum and minimum values of the x-coordinate.
4. To plot the profile at $x=0.0$, enter '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.

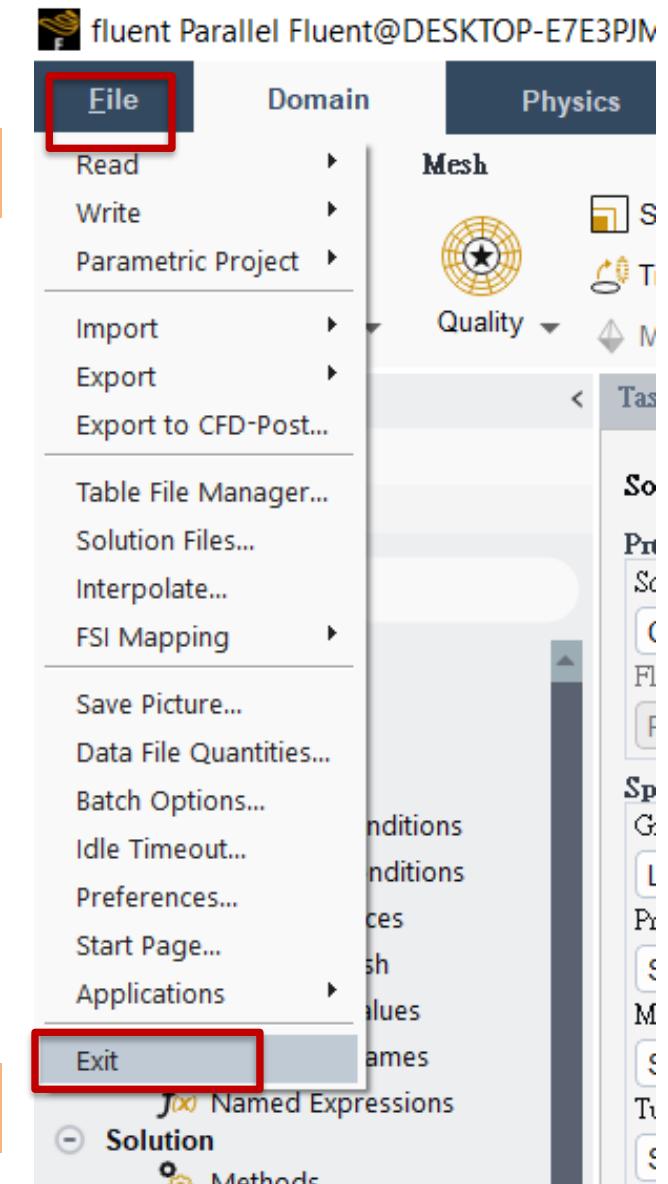
Step 7 Exit

File -> Close Fluent or Close Without Save

Note: It may ask you



1



2

Just click **OK**

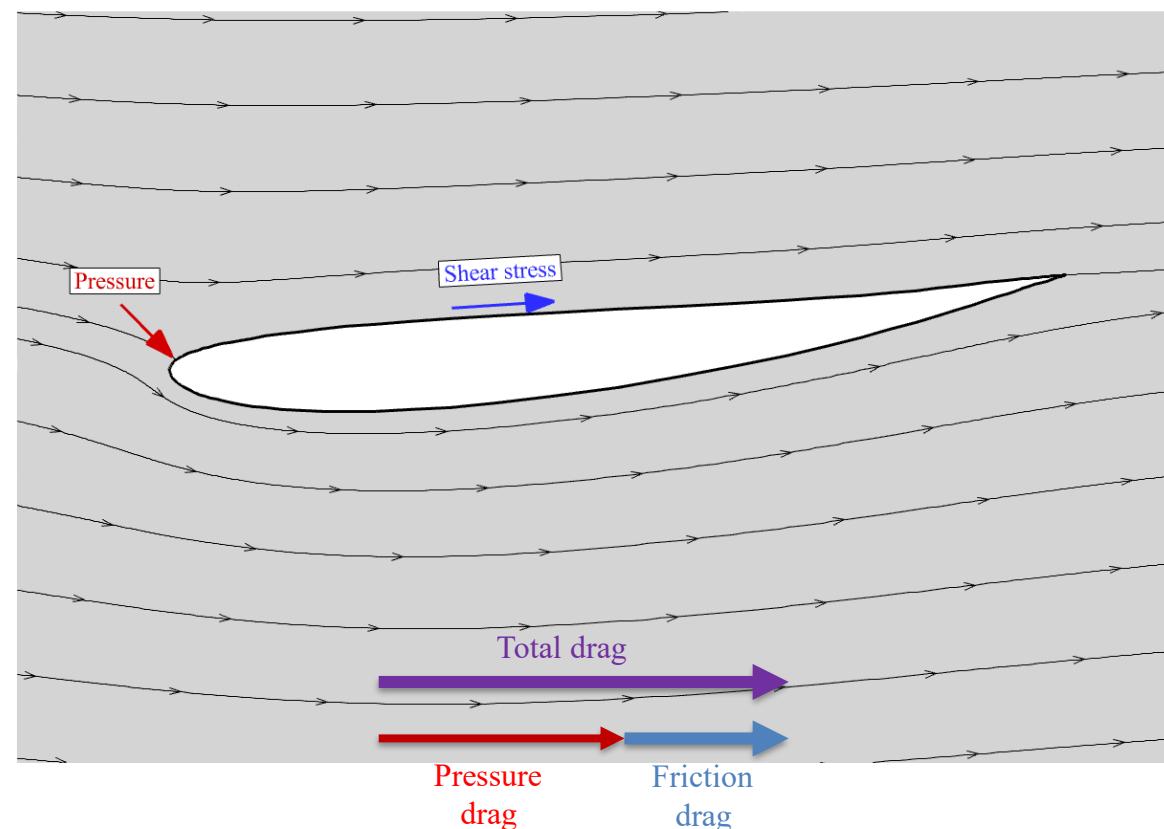
If you want to save the surfaces created in Step 6, you can save the case file to over-write the previous file.

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.

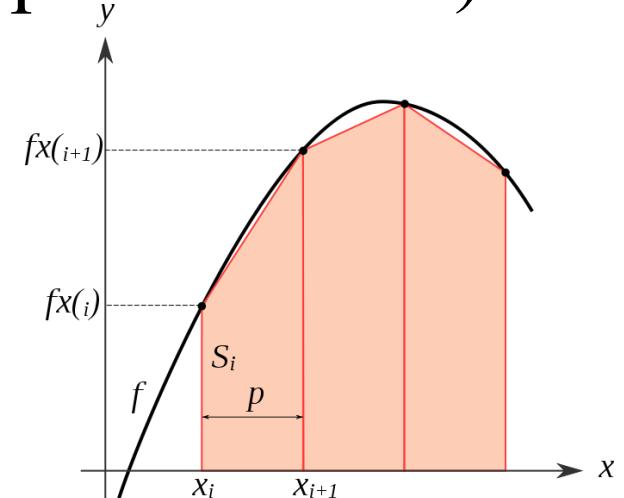
Numerical integration (trapezoidal rule)

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* !