Tutorial 1
Example Workflow
Flat Plate Laminar Boundary Layer

AE 462: Applied CFD
## 26 Jan 2026
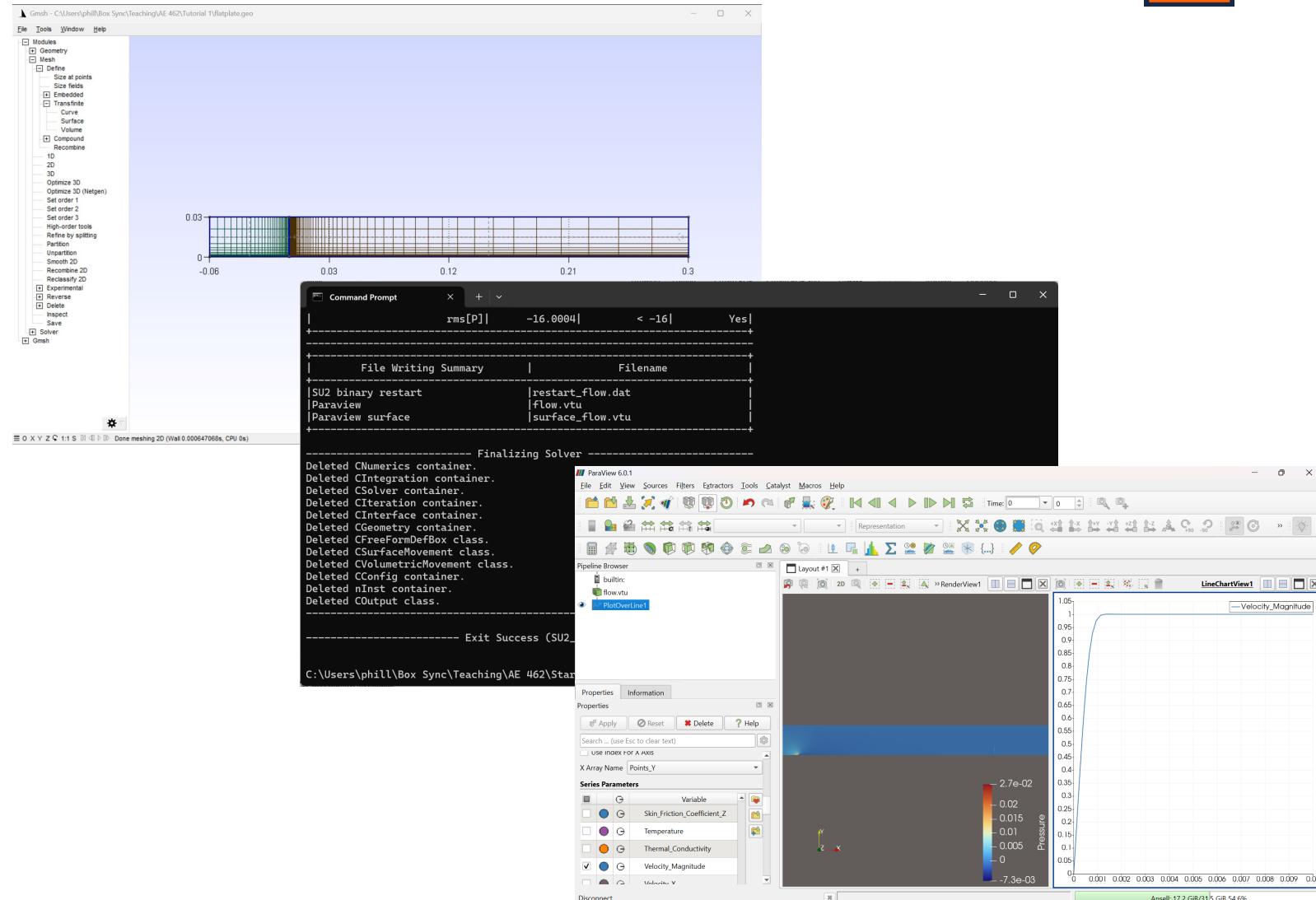
Workflow Example

Overview

- This example considers a basic flat plate laminar boundary layer
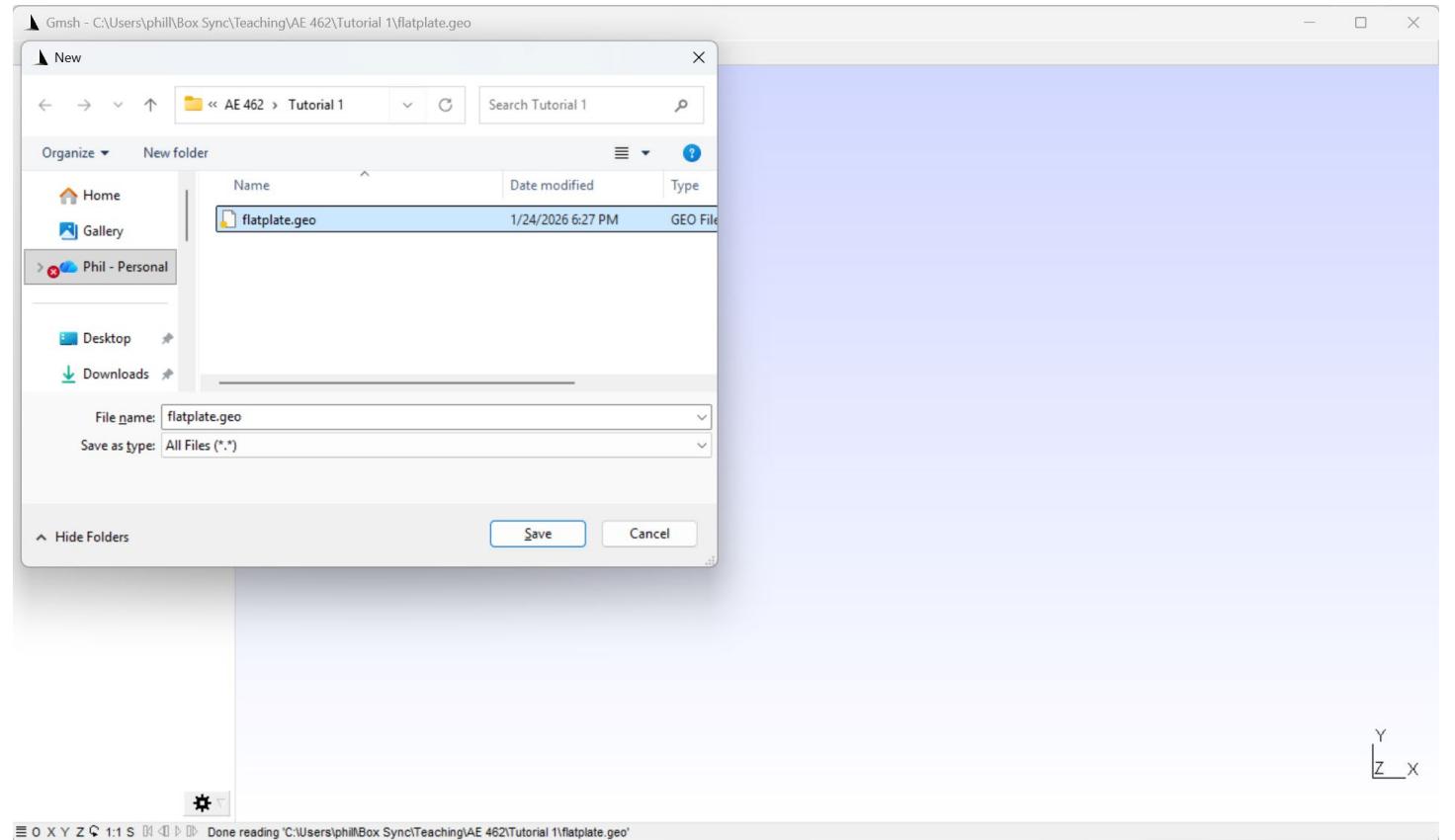
Geometry

- We will revisit this later…

Meshing

- Welcome to Gmsh!
- Gmsh uses .geo file extensions
  - Create a new folder in an easily accessible directory
  - Go to File → New
  - Name your new project "flatplate.geo"
  - If prompted to select a geometry kernel, choose "Built-in"

Meshing
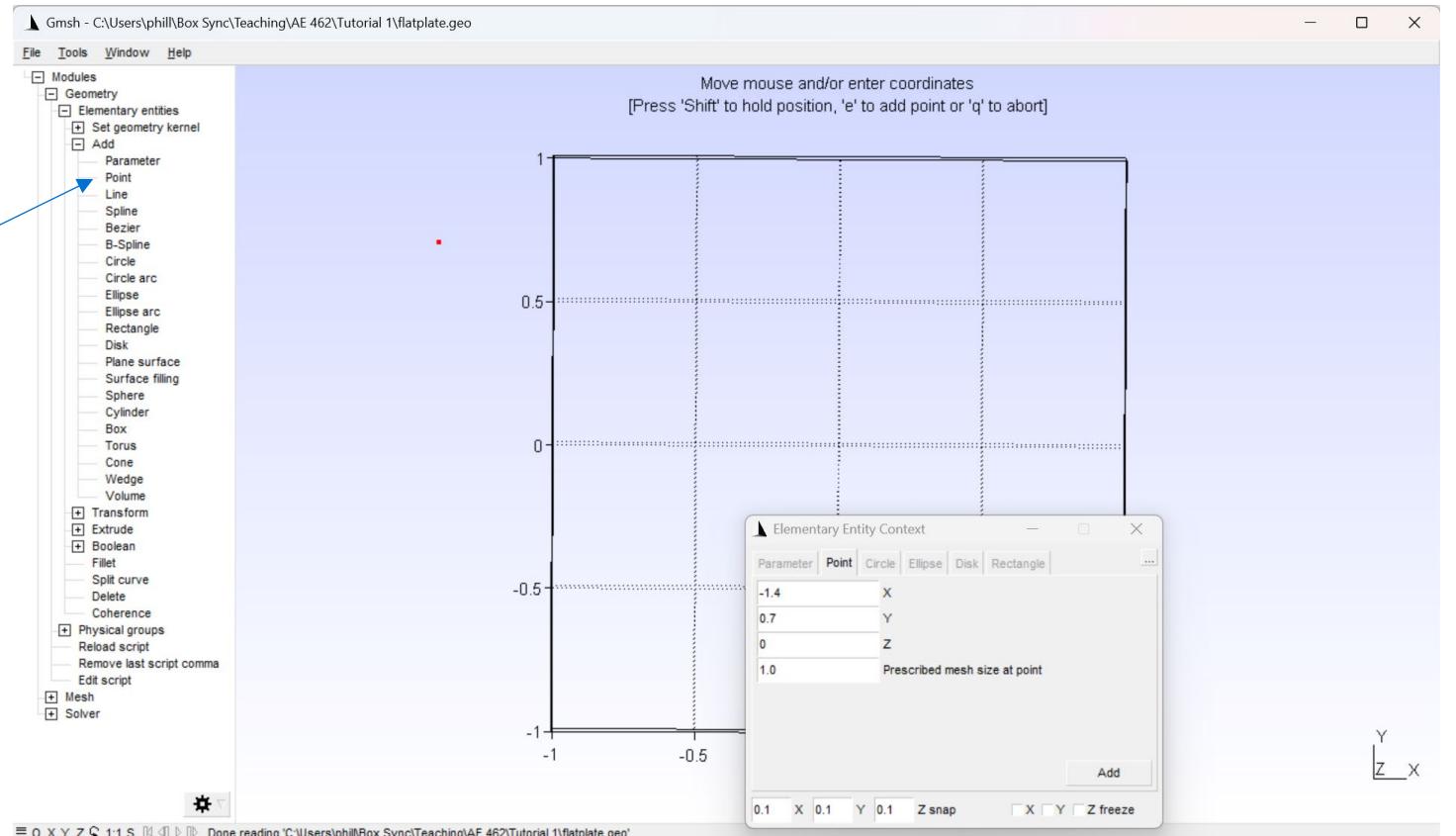
- We will begin by establishing the physical dimensions of the domain
- Go to:
  - Modules
  - → Geometry
  - → Elementary entities
  - → Add
 - → Point

- **Note:**
  - If you ever need to reset the view to default, press Alt+Z
  - The process used by Gmsh is all script based, and you can edit the commanded steps by opening the .geo file or by clicking "Edit script"
    - Following any edits, you will need to save and click "Reload script"
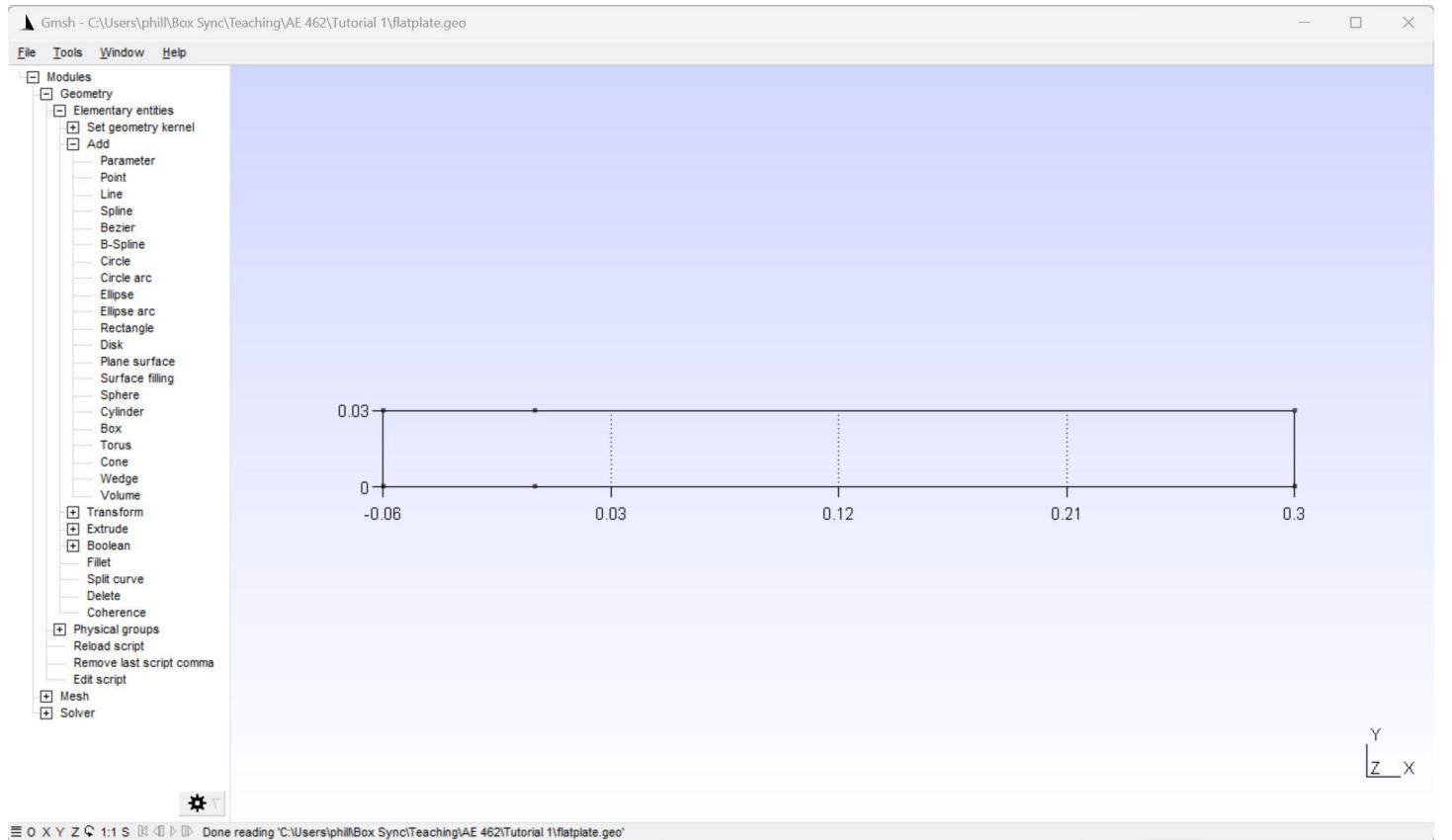
Meshing

- Add points corresponding to
  - For all points, set z = 0 and Prescribed mesh at point = 1.0

- Input x, y, z coordinates and click "Add" button
  - x = -0.06
    y = 0
  - x = -0.06
    y = 0.03
  - x = 0
    y = 0.03
  - x = 0.3
    y = 0.03
  - x = 0.3
    y = 0
  - x = 0
    y = 0

- After inserting all points, press "q" to exit the add point interface
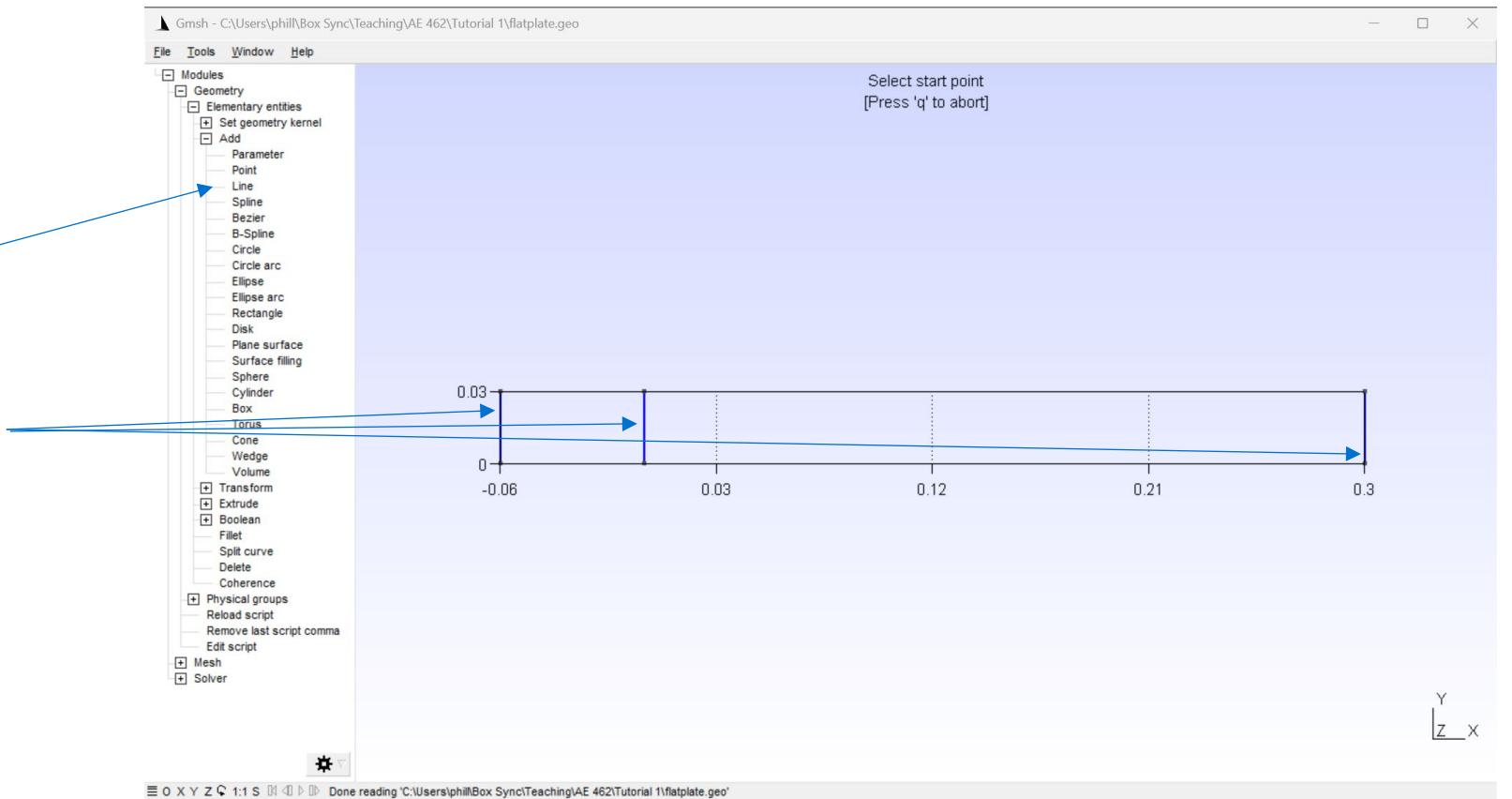
Meshing

- Select
  - Modules
  - → Geometry
  - → Elementary entities
  - → Add
  - → Line
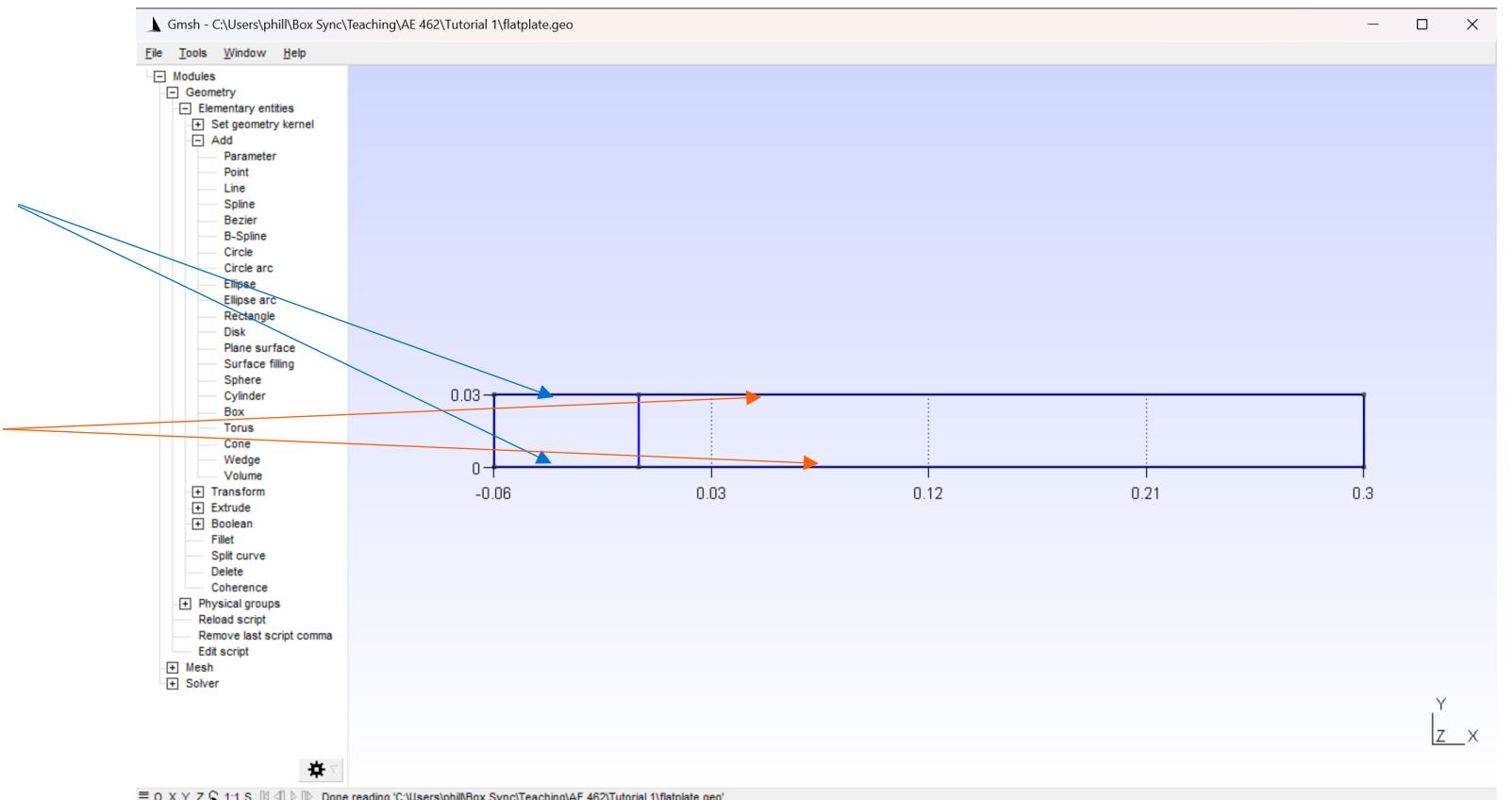
- Add three vertical lines, from bottom (y = 0) to top (y = 0.03) on the upstream and downstream end of the domain
  - Note that the order of first point → second point matters!
  - These lines are placed with your previous points located at x = -0.06, 0, and 0.3

Meshing

- Add horizontal lines across the top and bottom edges, along the upstream end of the domain, going from right (x = 0) to left (x = -0.06)

- Add horizontal lines across the top and bottom edges, along the downstream end of the domain, going from left (x = 0) to right (x = 0.3)

- After inserting all lines, press "q" to exit the add lines interface

Meshing
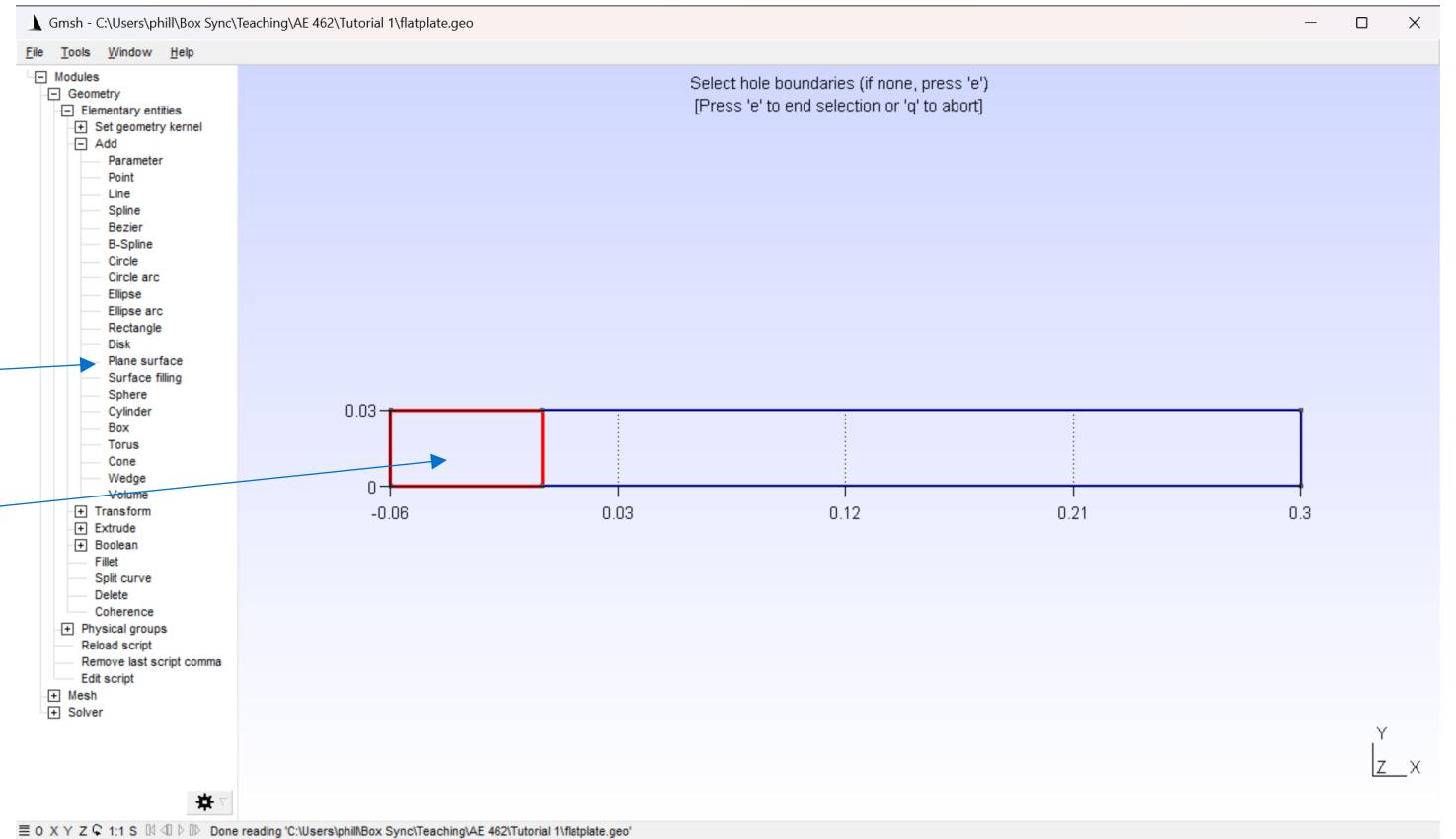
- Go to:
  - Modules
  - → Geometry
  - → Elementary entities
  - → Add
  - → Plane Surface
- Select the box on the upstream end of the domain created by our line segments
  - Selecting the top, left, or bottom will select all three of the outside segments. Also click the inside vertical line.
- Press "e" to end selection and create the plane surface
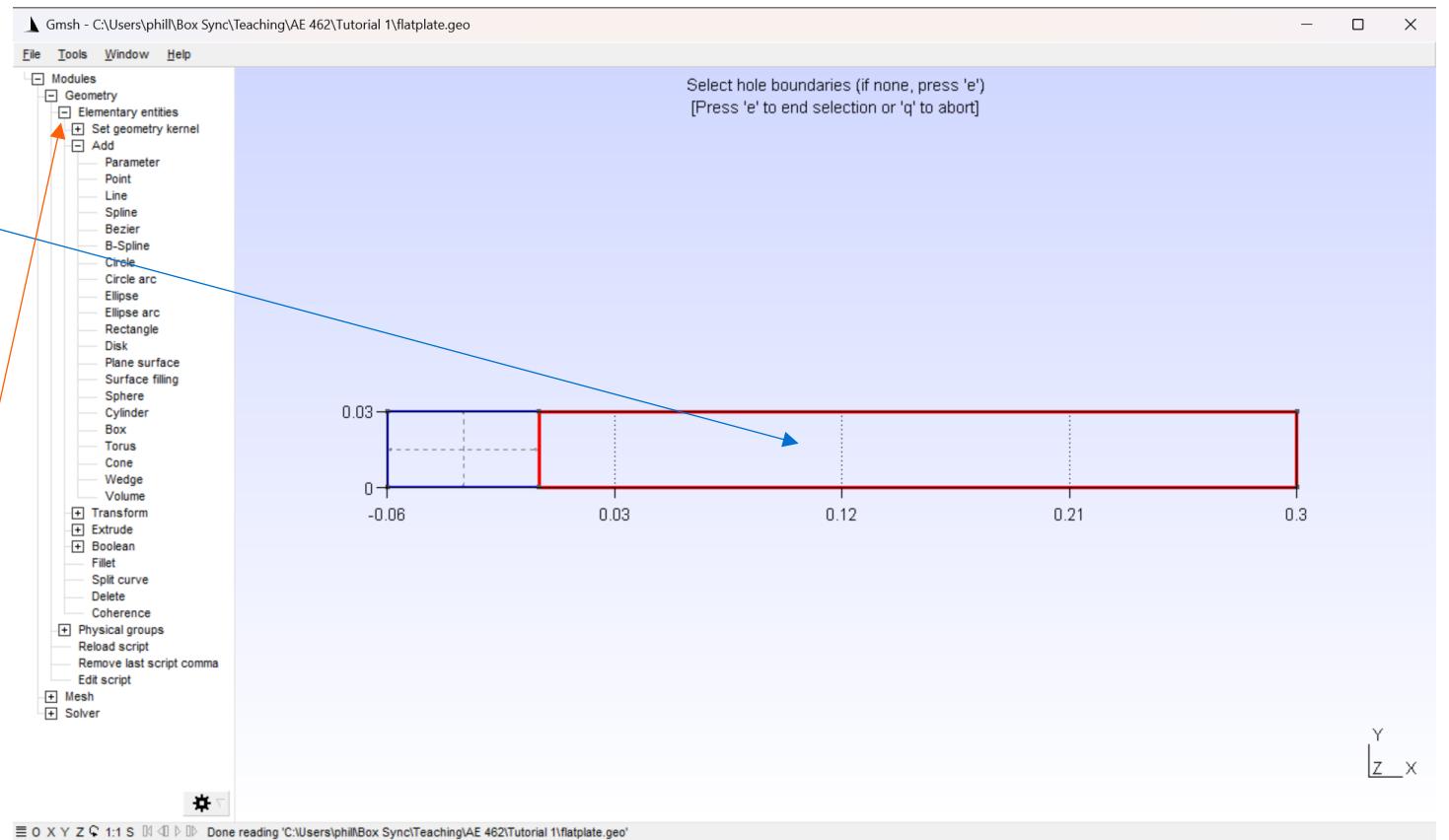
Meshing

- Do the same with the downstream box
  - Press "e" to end selection and create the plane surface
- Press "q" to exit the plane surface selection
- We are finished defining the Elementary Entities, so you can collapse this option [-] in the left-hand menu of options

Meshing
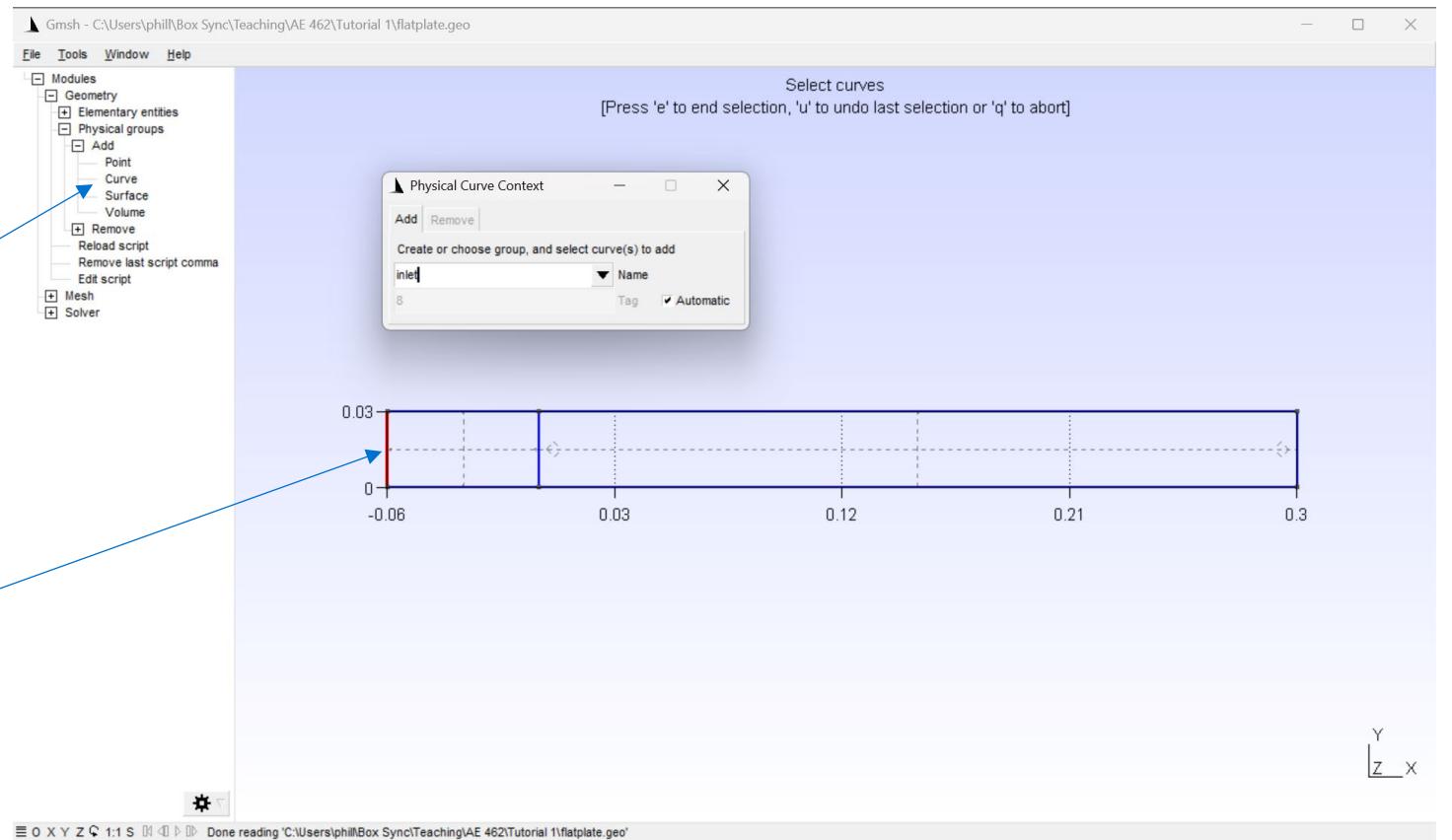
- We will now provide variable names for the different boundaries of the flow domain

- Select
  - Modules
  - → Geometry
  - → Physical Groups
  - → Add
  - → Curve

- Click on the upstream vertical line so it is selected (highlighted in red)
  - In the text box, type "inlet"
  - Click on the blue background
  - Press "e" to end the selection

Meshing

- Repeat this process for...
  - Lower-left horizontal line
    - Label this as "symmetry"
  - Bottom-right horizontal line
    - Label this as "wall"
  - Right-side vertical line and *both* upper horizontal lines
    - Label this as "outlet"
  - The vertical line in the middle of the flow domain will *not* have a label

- At each step, don't forget to select the blue background and press "e" to end selection

- After labeling each line, press "q" to exit the physical curve definition menu

Meshing

- We will now begin defining the meshing strategy for our flow domain
 - Now that we are finished with defining the geometry, you can select the [-] next to the geometry entry in the left-side menu
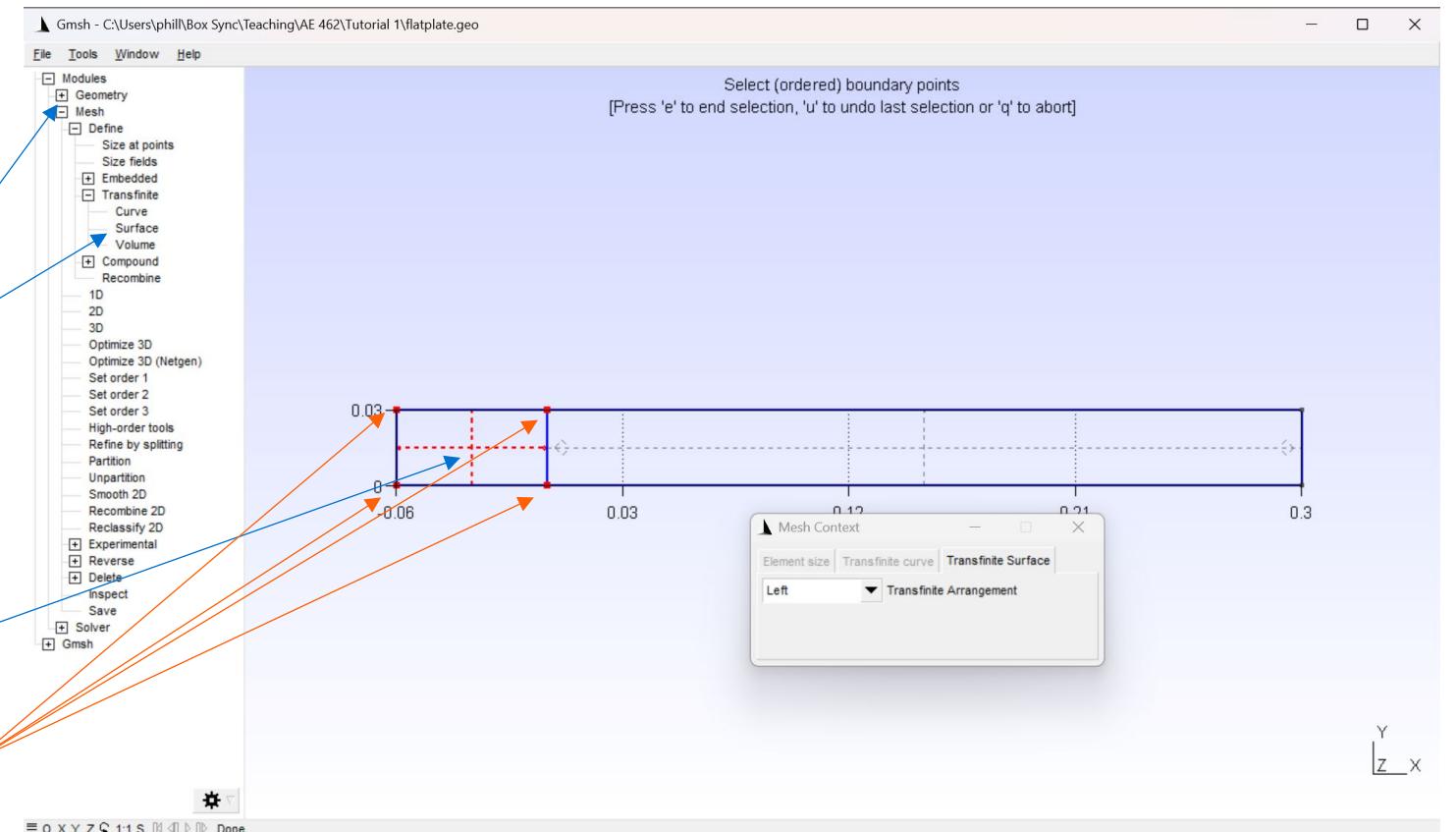
- Select
  - Modules
  - → Mesh
  - → Define
  - → Transfinite
  - → Surface
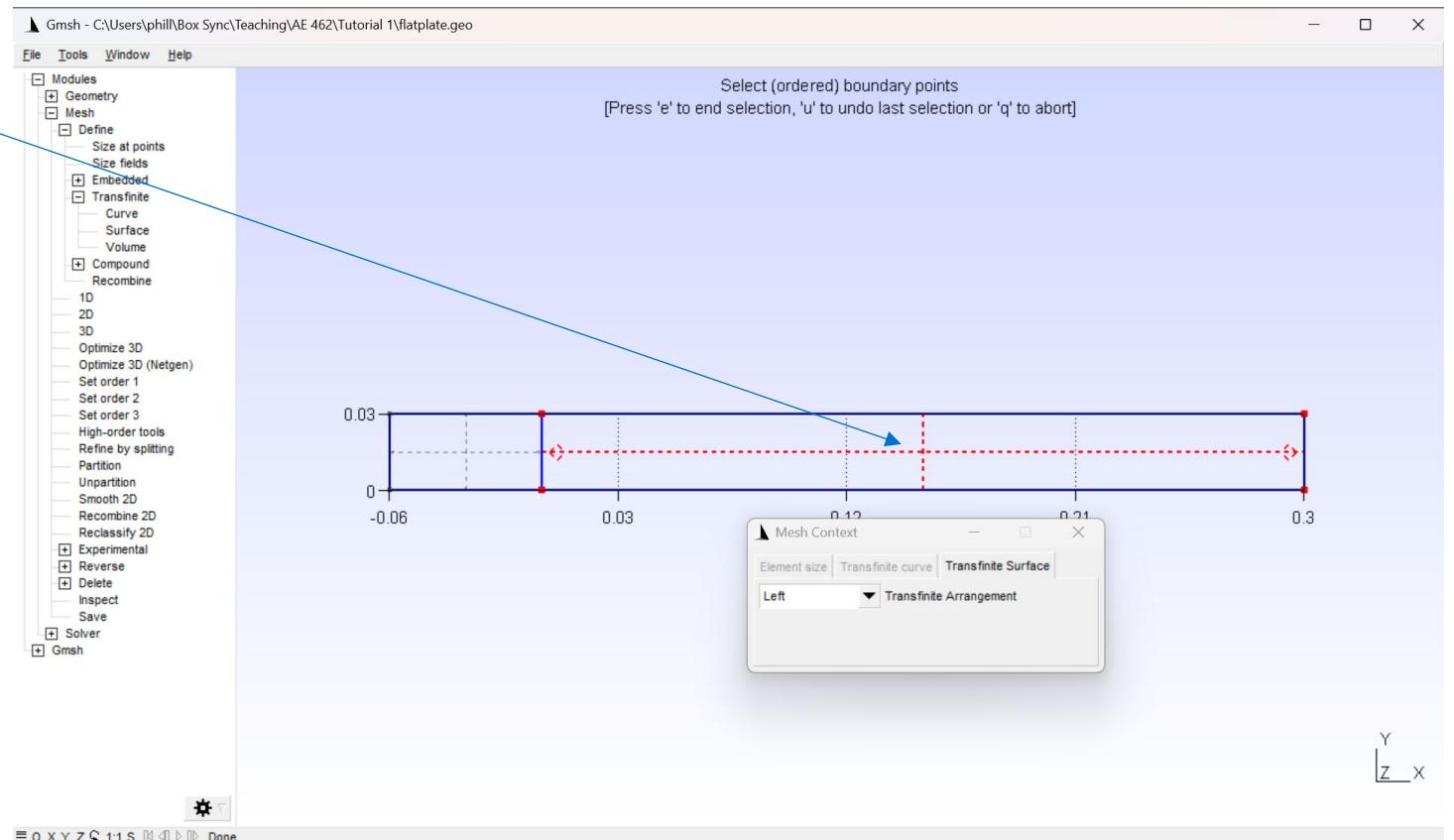
- Click on the dashed lines in the left-side box to highlight it in red
  - Next, click the four points that define the corners of the domain so they are also highlighted in red
  - Click on the blue background and press "e" to end the selection

Meshing

- Repeat for the right-side box
  - Select the dashed lines inside of the rectangular domain
  - Click the four corner points that define this rectangular domain
  - Click on the blue background and press "e" to end selection
- Press "q" to exit the transfinite surface definition menu
- Now that we have defined the surfaces where we *want* to define the mesh, we will next determine the number of nodes to place in the x and y dimensions

Meshing
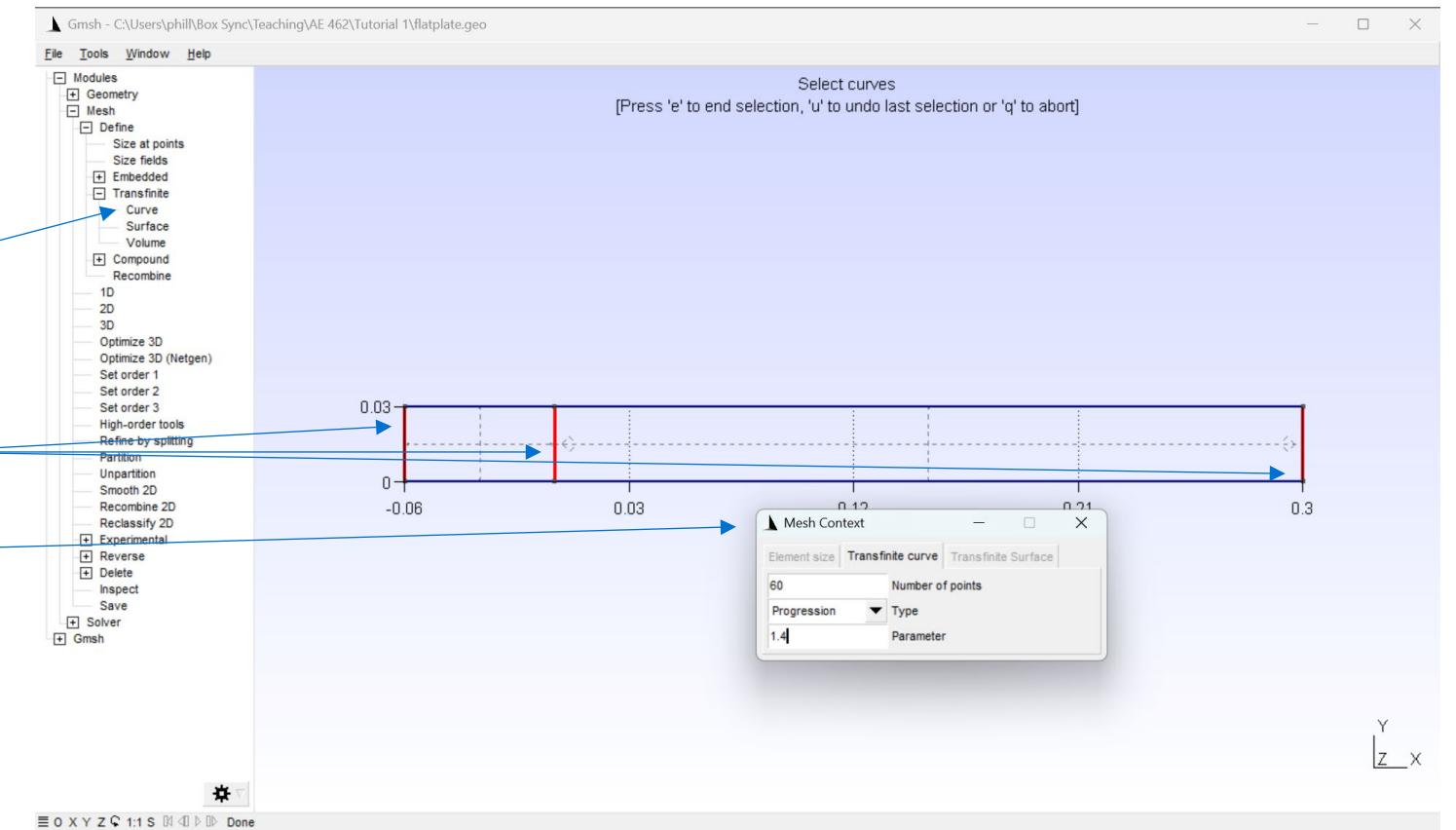
- Select
  - Modules
  - → Mesh
  - → Define
  - → Transfinite
  - → Curve
- Click on the three vertical lines (x = -0.06, 0, and 0.3)
- Under the "Mesh Context" menu, enter:
  - Number of points = 60
  - Type = Progression
  - Parameter = 1.2
- Click on the blue background and press "e" to end selection
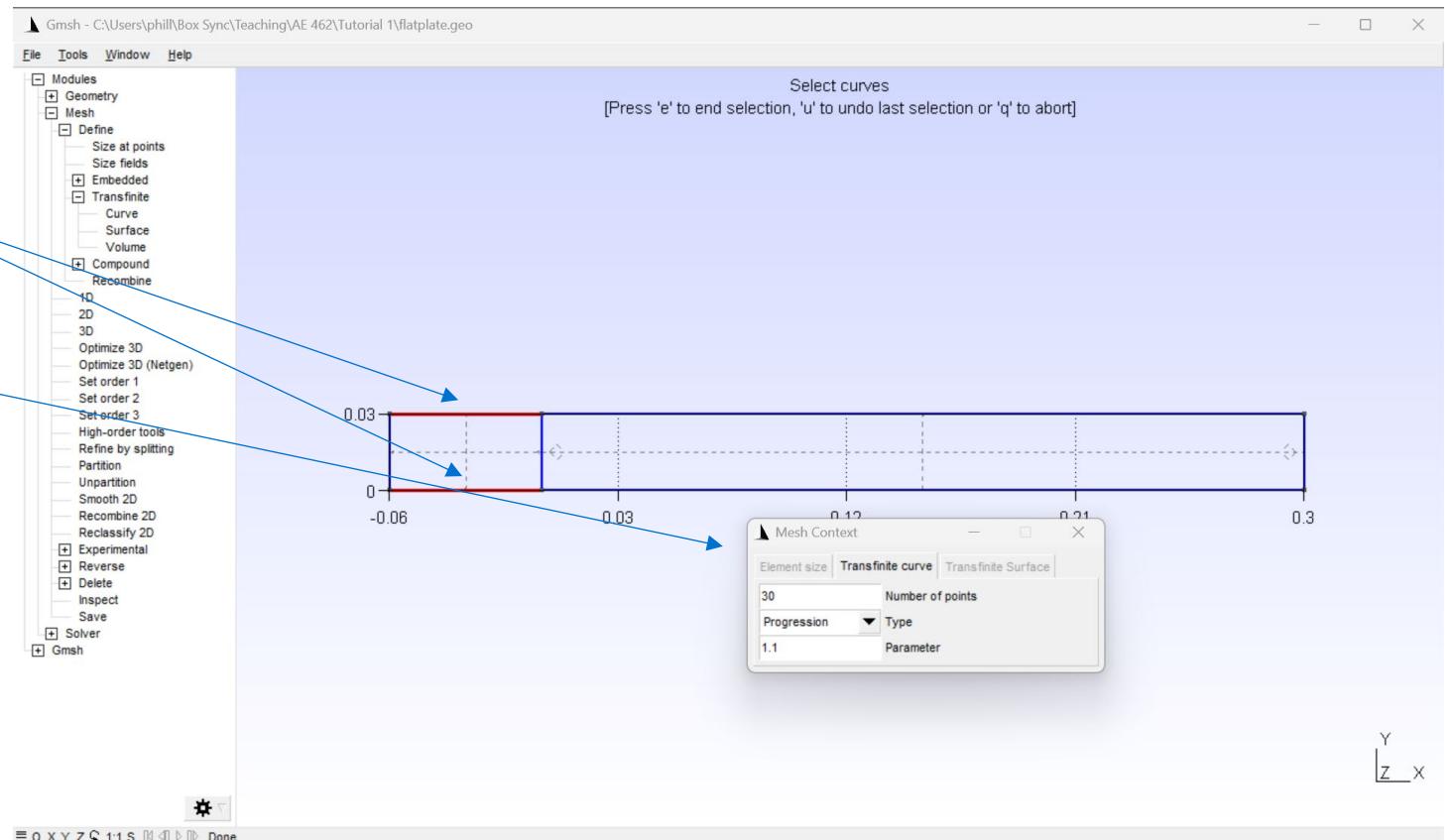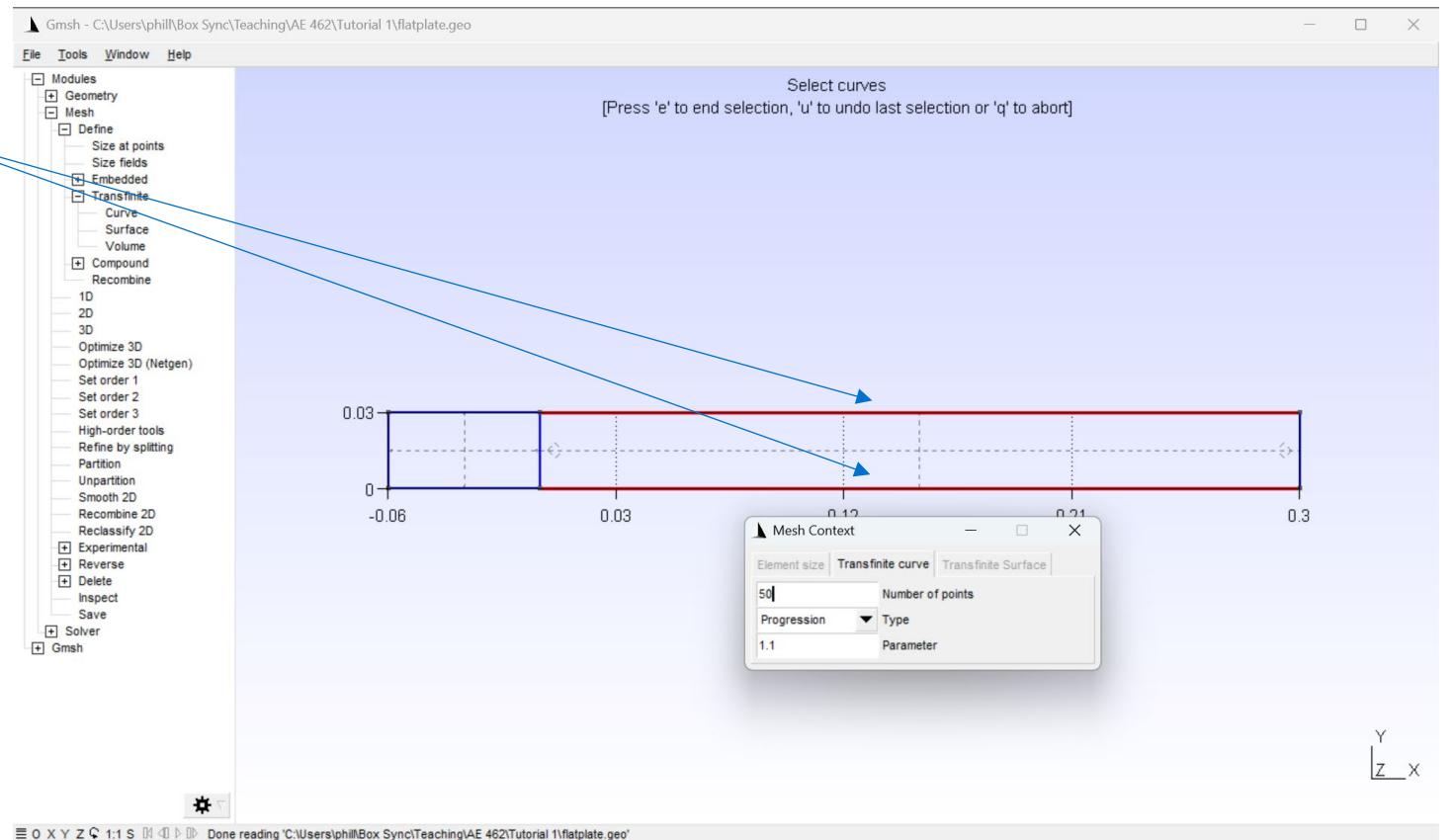
Meshing

- Click on the two left-side horizontal lines

- Under the "Mesh Context" menu, enter:
  - Number of points = 30
  - Type = Progression
  - Parameter = 1.1

- Click on the blue background and press "e" to end selection

Meshing

- Repeat for the two right-side horizontal lines with more points:
  - Number of points = 50
  - Type = Progression
  - Parameter = 1.1
- Click on the blue background and press "e" to end selection
- Press "q" to exit the mesh definition menu

Meshing
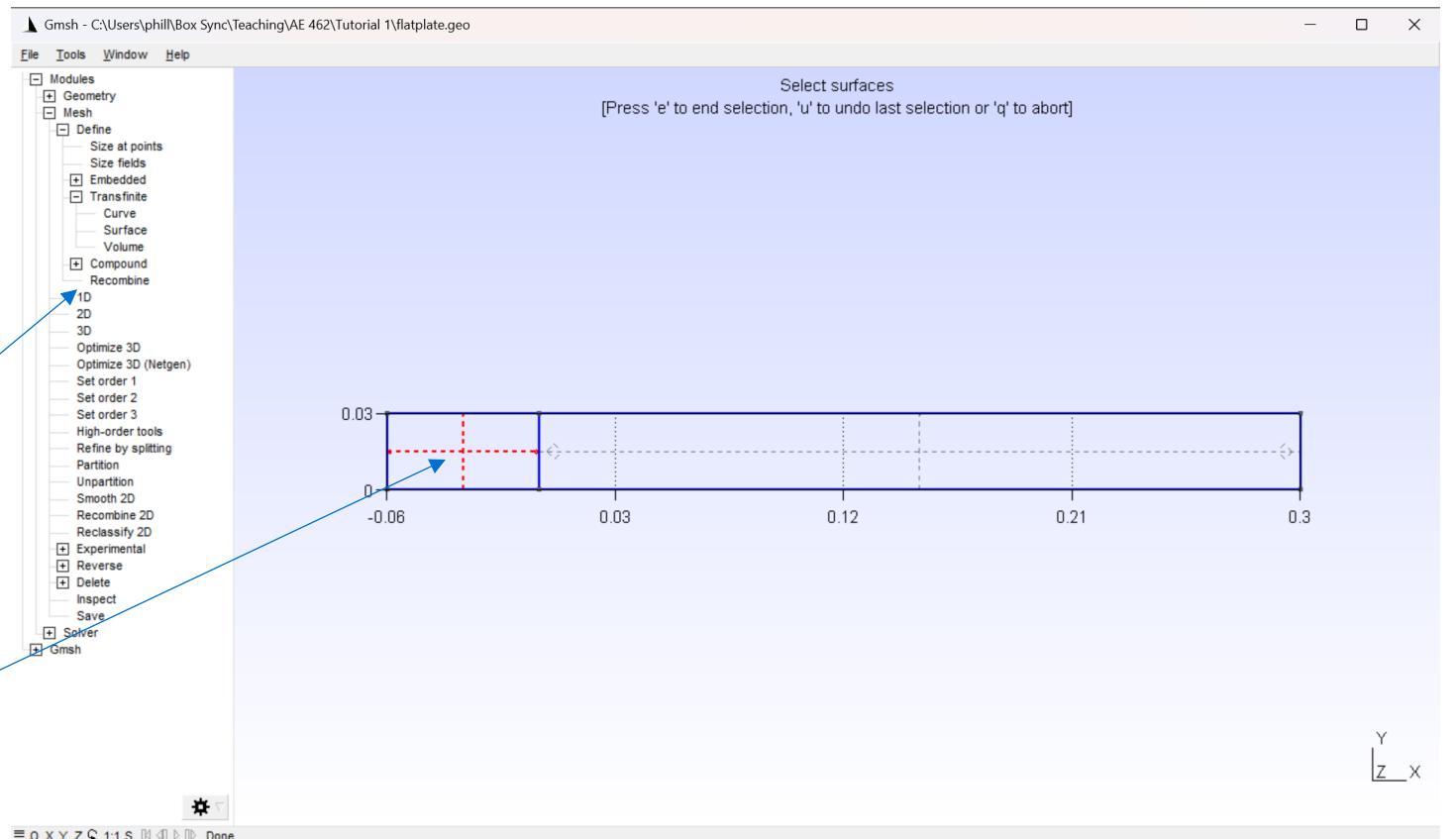
- We are now just about ready to generate a mesh

- Select
  - Modules
  - → Mesh
  - → Define
  - → Recombine

- Select the dashed lines in the left-side box
  - Press "e" to end selection
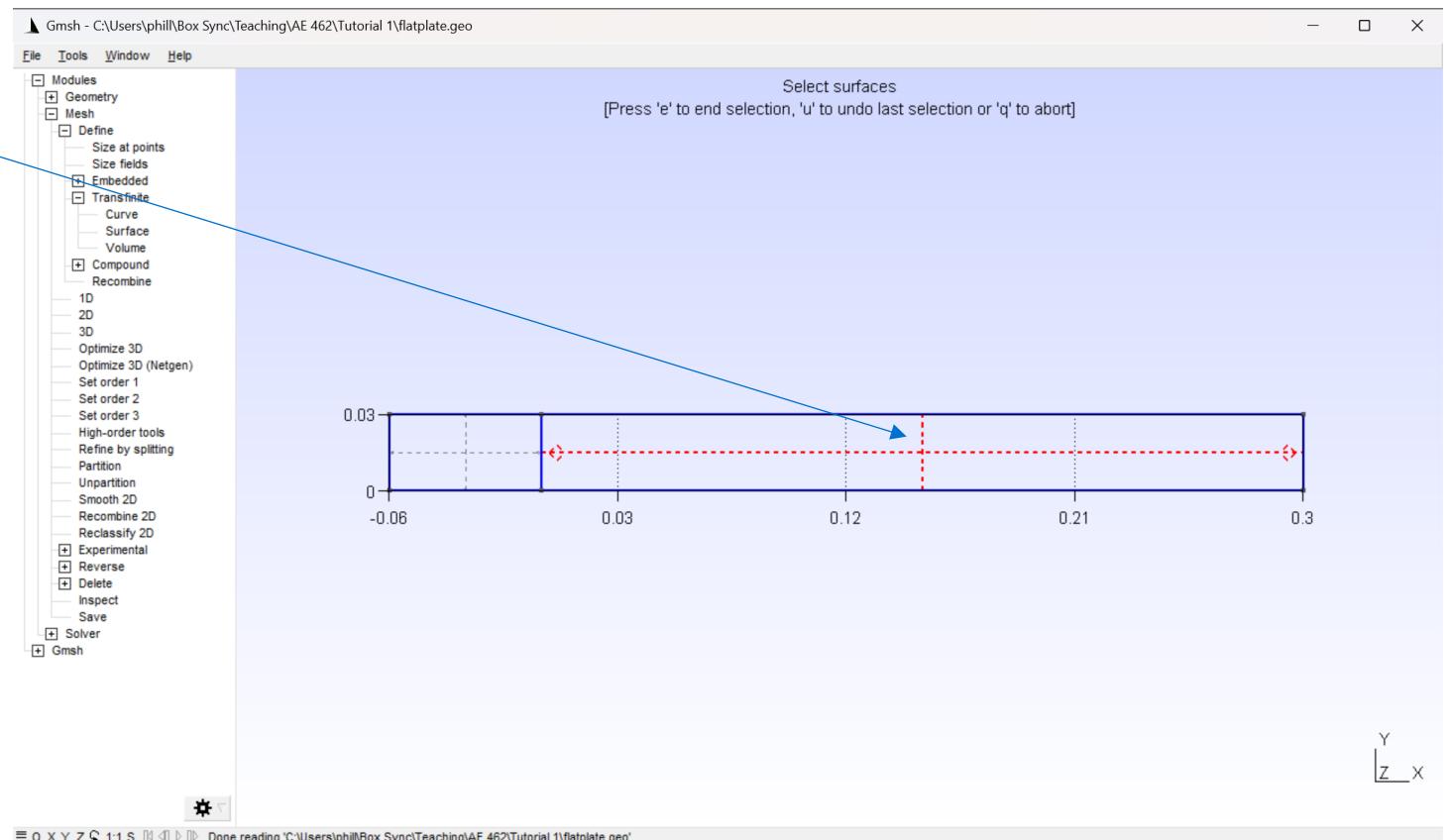
Meshing

- Repeat for the right-size box
  - Press "e" to finalize selection
- Press "q" to exit the selection menu
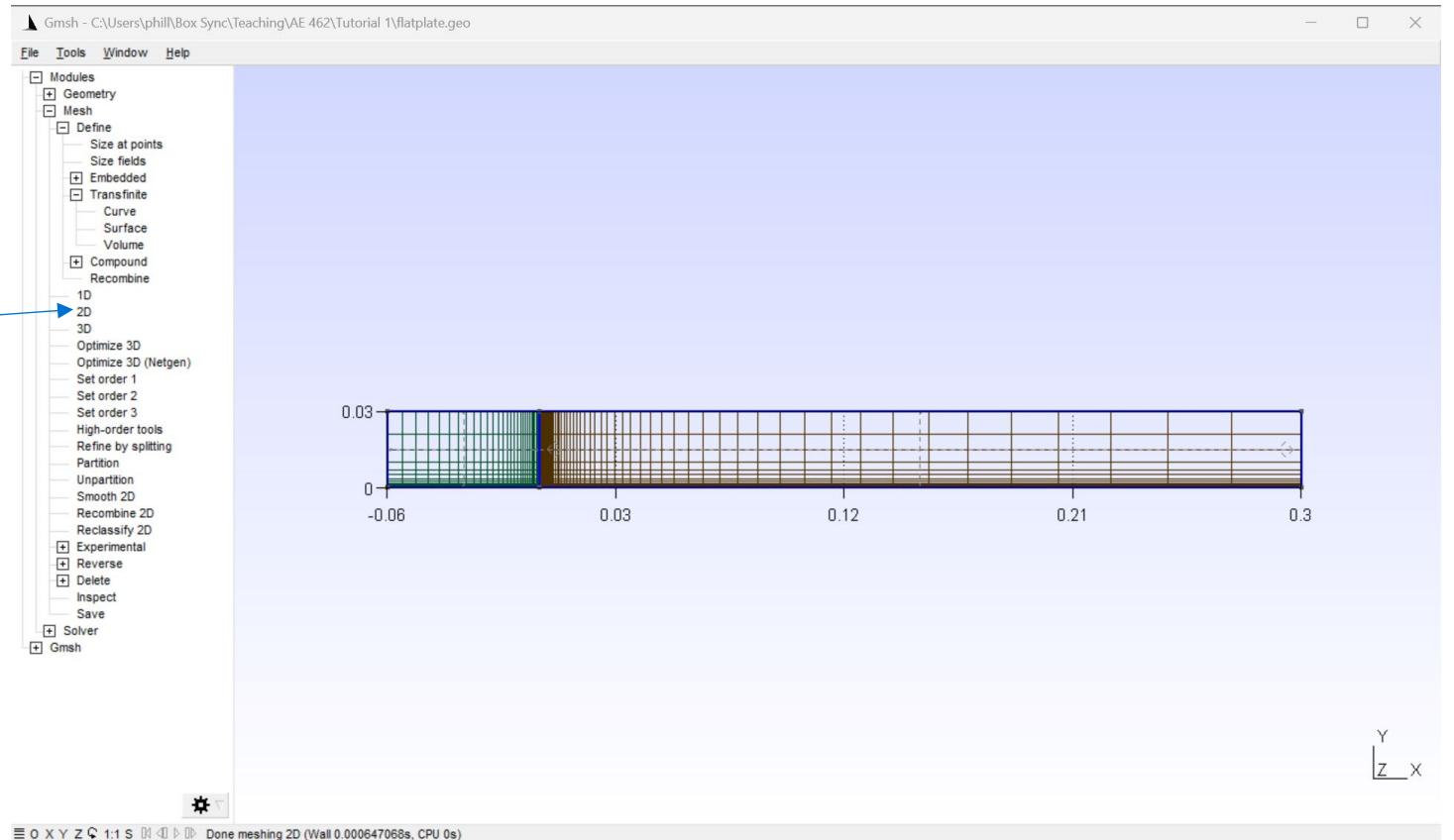
Meshing

- Select
  - Modules
  - → Mesh
  - → 2D

- You will now see a visualization of the mesh
  - Cells are concentrated to y-positions close to the plate and x-positions close to the leading edge (x = 0)
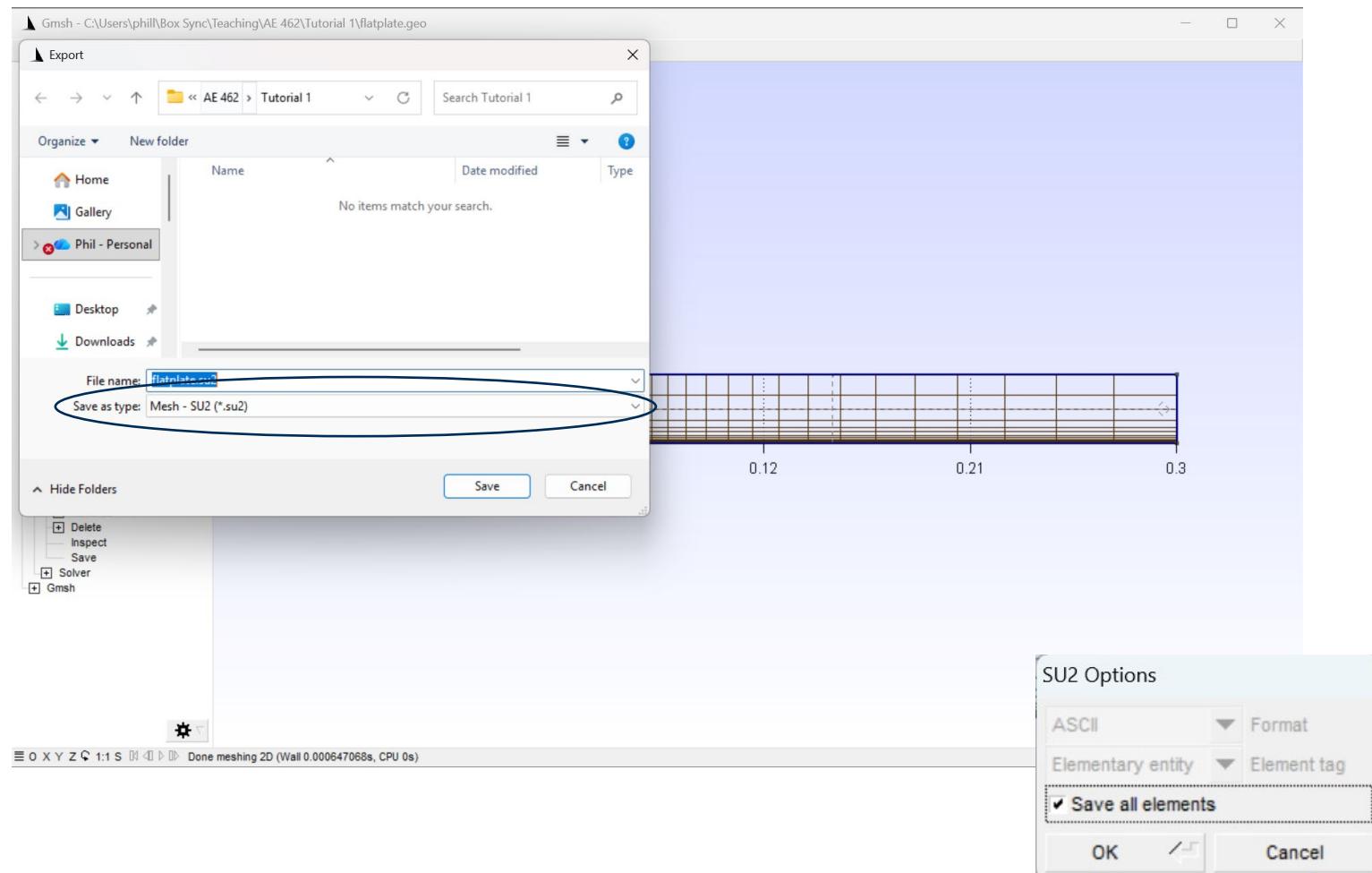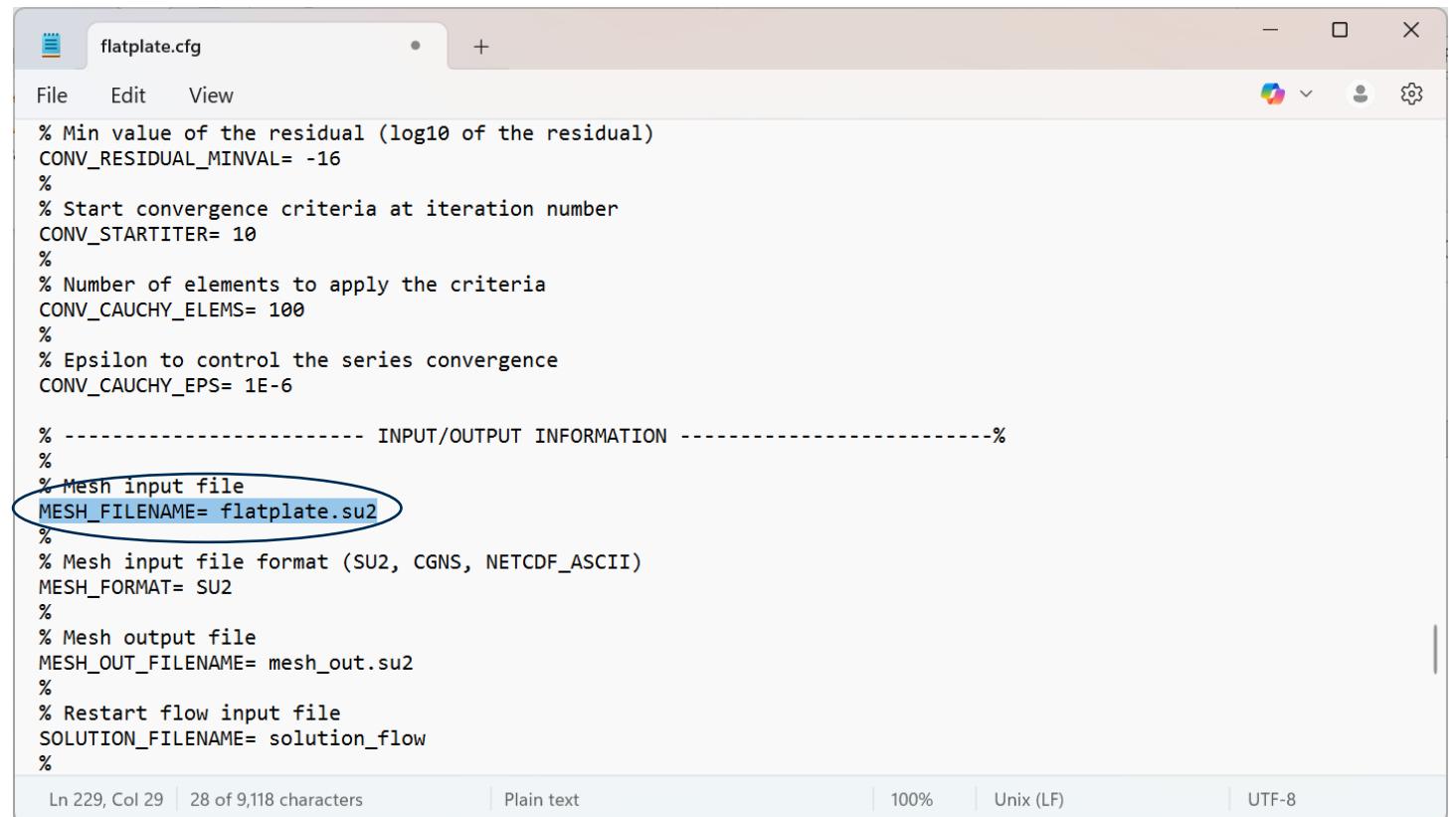
Meshing

•Save your flatplate.geo file, just to be safe

•In the top-level Gmsh menu, select:

- File
- →Export

•In the export directory, save the file as "flatplate.su2"

- When prompted with SU2 options, **be sure to enable** the "Safe all elements" checkbox before selecting "OK"

Input Parameters

- Download the "flatplate.cfg" file from the Canvas site
  - This file is configured to recognize "flatplate.su2" as the input mesh file, as well as the labels given for the inlet, outlet, symmetry, and wall boundaries defined earlier

Running the Code

- Open a new terminal window
  - For Windows, go to the start menu
  - Type "cmd"

- Navigate to the directory of your .su2 mesh and .cfg input file
  - For example: cd "C:\Users\phill\Box Sync\Teaching\AE 462\Tutorial 1"

- Call the SU2 CFD package, pointing to your .cfg file, with:
  - "C:\SU2\SU2_CFD" flatplate.cfg
  - If your SU2 executables are in a different directory, point towards that location instead
  - EWS machines have been configured to have all SU2 files in C:\SU2

Command Prompt

```
Microsoft Windows [Version 10.0.26200.7623]
(c) Microsoft Corporation. All rights reserved.

C:\Users\phill>cd "C:\Users\phill\Box Sync\Teaching\AE 462\Tutorial 1"

C:\Users\phill\Box Sync\Teaching\AE 462\Tutorial 1>"C:\SU2\SU2_CFD" flatplate.cfg
```
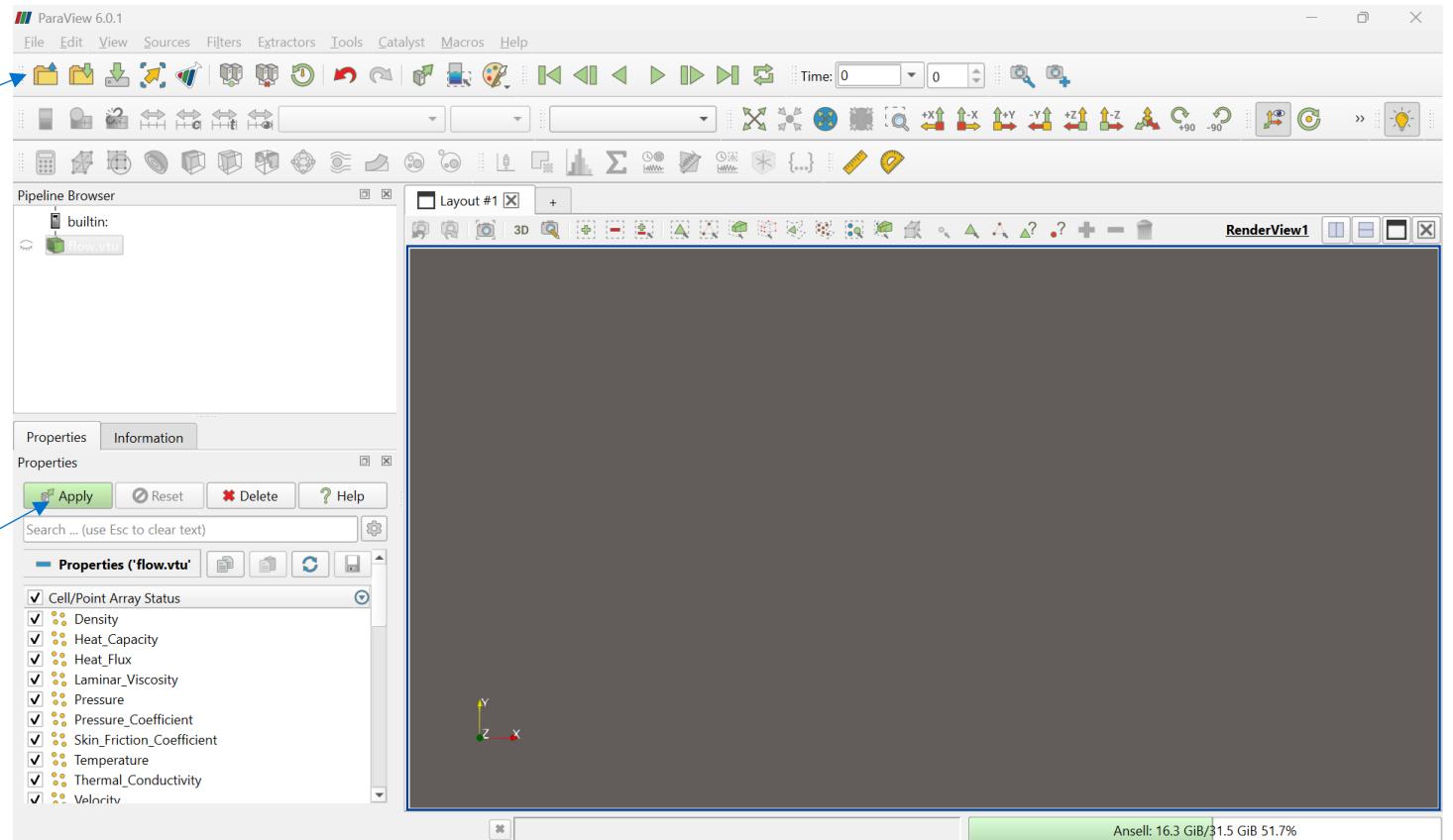
Loading the Field Data
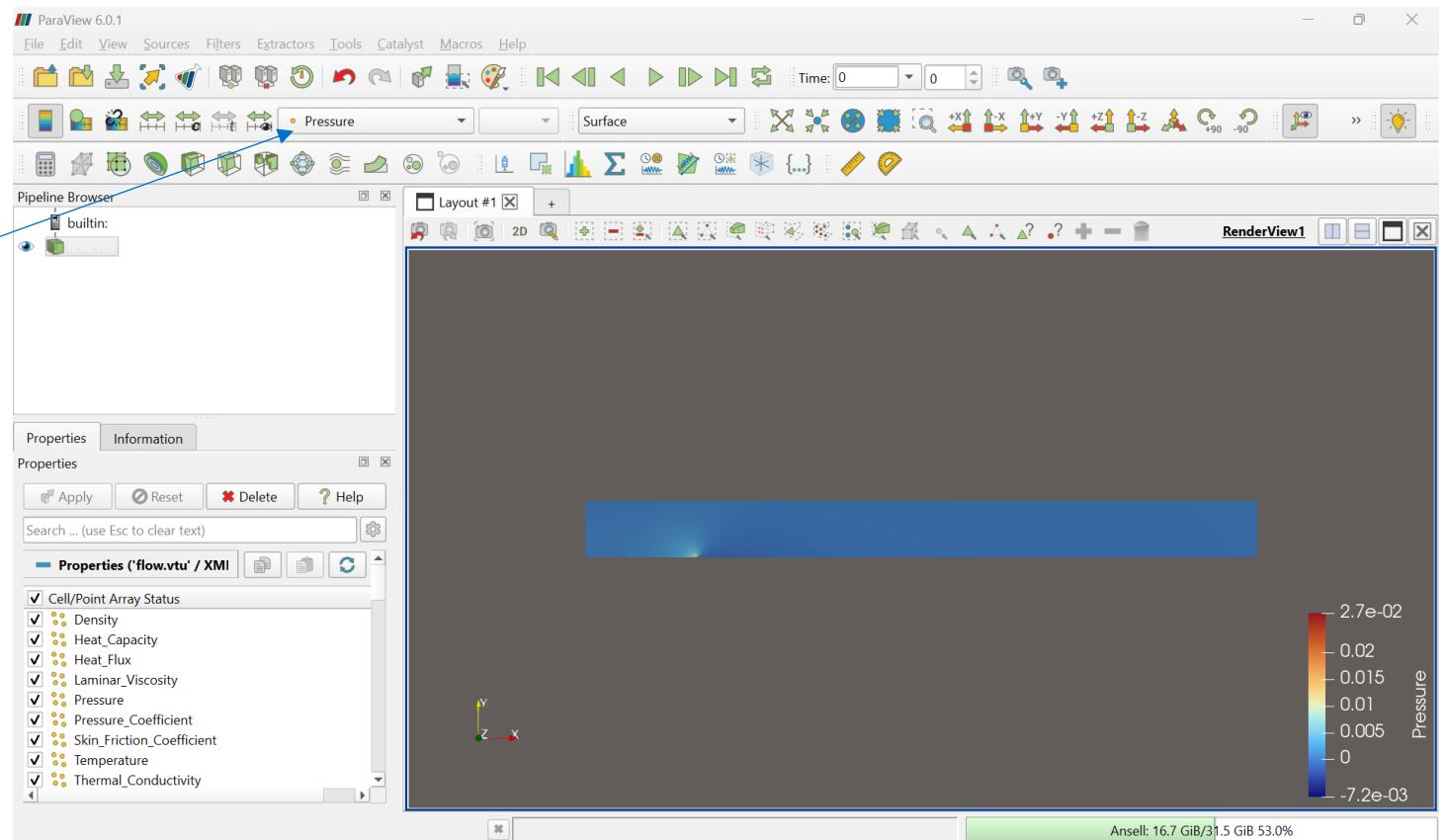
- Launch Paraview
- Click the "Open" button
  - Navigate to the directory of your .su2 mesh file and your .cfg input file
  - Load the "flow.vtu" file from this directory
  - Click the "Apply" button

Loading the Field Data

- You can now view selected flow variables across the flow field by selecting them from the drop-down menu

- However, most of the interesting interactions occur at the wall boundary

  - We can extract the boundary-layer velocity profile by applying a filter
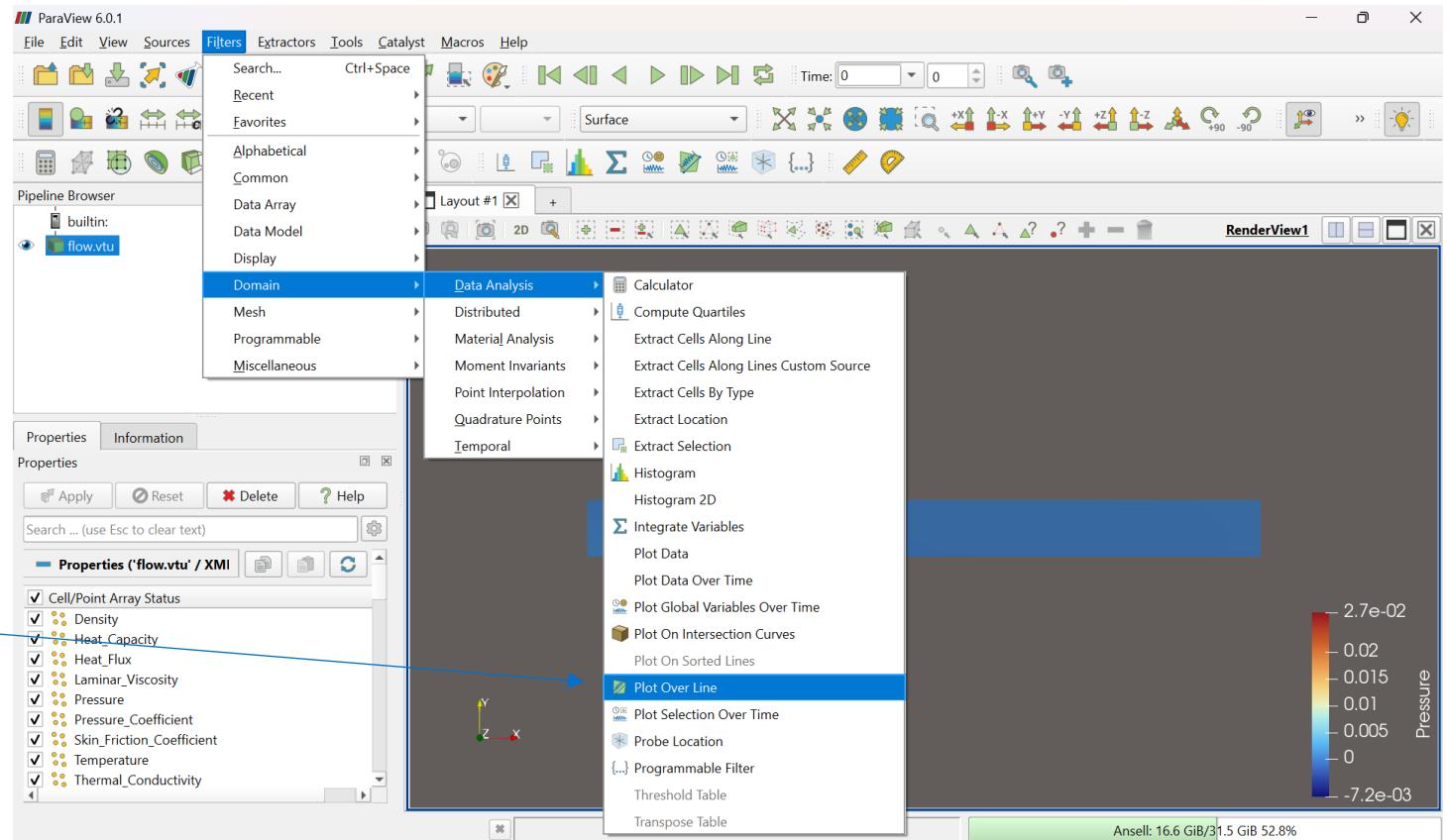
Extracting Data

- With "flow.vtu" selected in the Pipeline Browser, go to:
  - Filters
  - →Domain
  - →Data Analysis
  - →Plot Over Line
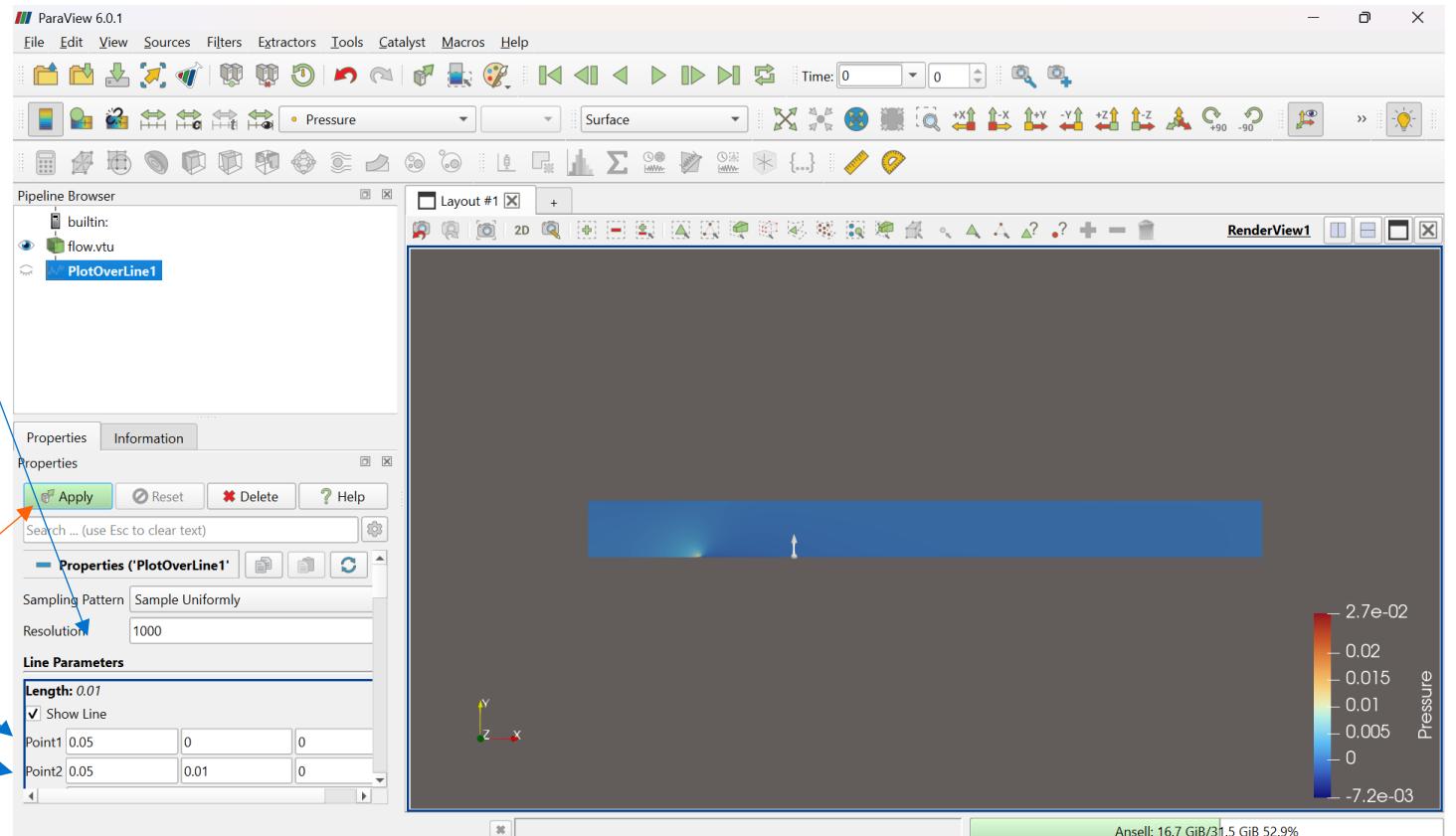
Extracting Data

- Under "Line Parameters" (inside the "Properties" tab on the left) you can place the endpoints of the line for extracting data
  - Change "Point 1" to $x = 0.2$, $y = 0$, $z = 0$
  - Change "Point 2" to $x = 0.2$, $y = 0.01$, $z = 0$
- Click "Apply"
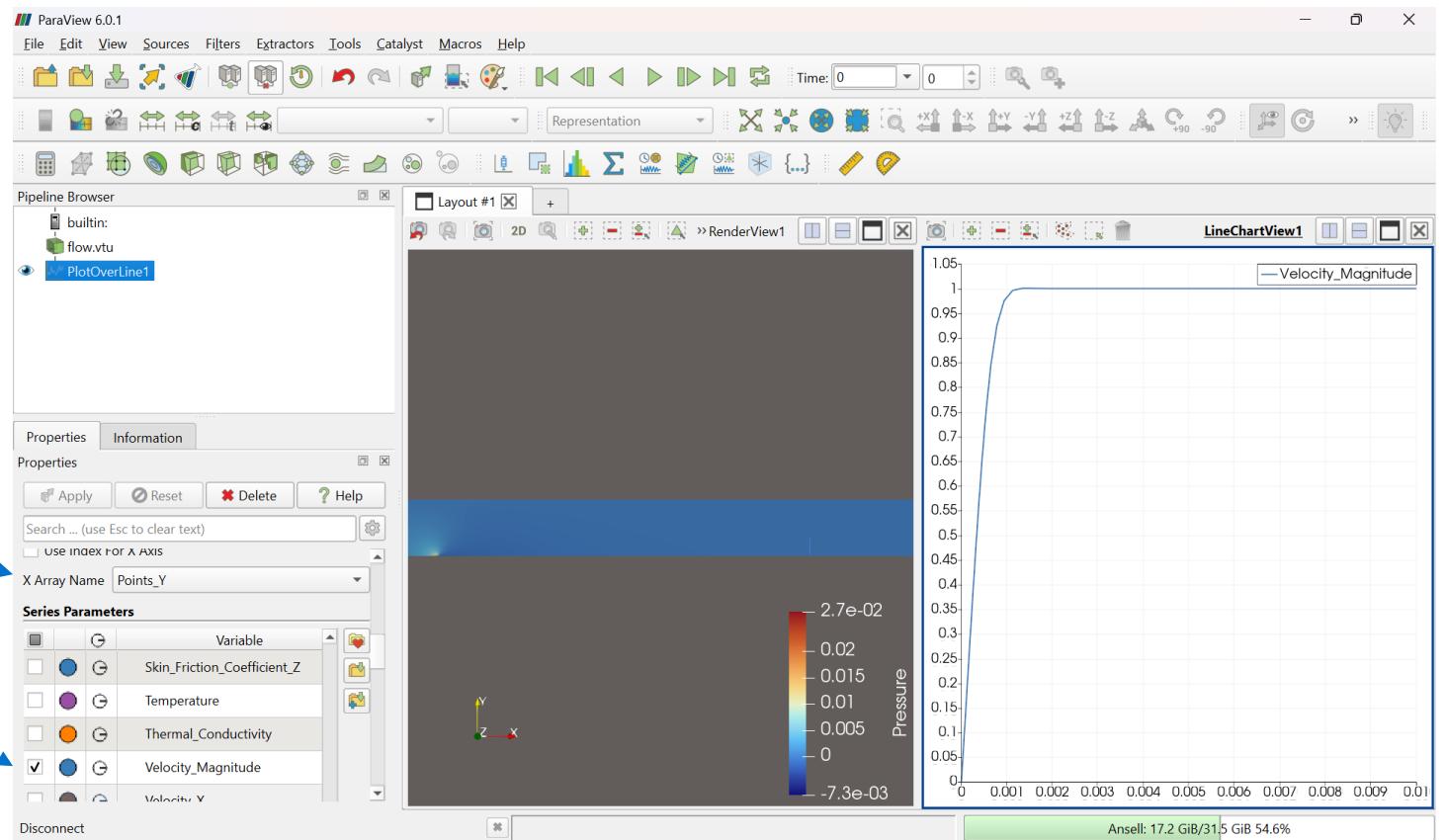
Extracting Data
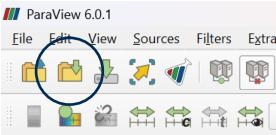
- A line chart will appear on the right side of the screen
  - Scroll down in the "Properties" tab on the left side of the screen
  - To plot *just* the velocity magnitude
    - Change "X Array Name" to "Points_Y"
    - Uncheck all variables except for "Velocity_Magnitude"
  - You can now see (a sideways plot of) the laminar boundary-layer profile

Extracting Data

- Select "PlotOverLine1" in the Pipeline Browser
  - Click the "Save Data" button
- In the save prompt, change "Files of type" to "*.csv, *.tsv, *.txt"
  - Give the data file a name and save (e.g., bldata.csv)
- Your extracted field data is now saved in the .csv file
  - You can open the data file in a spreadsheet editor (e.g., Excel) and/or use a programming script (e.g., Python) to import and plot results