



Tutorial 1

Example Workflow

Flat Plate Laminar Boundary Layer

AE 462: Applied CFD
26 Jan 2026

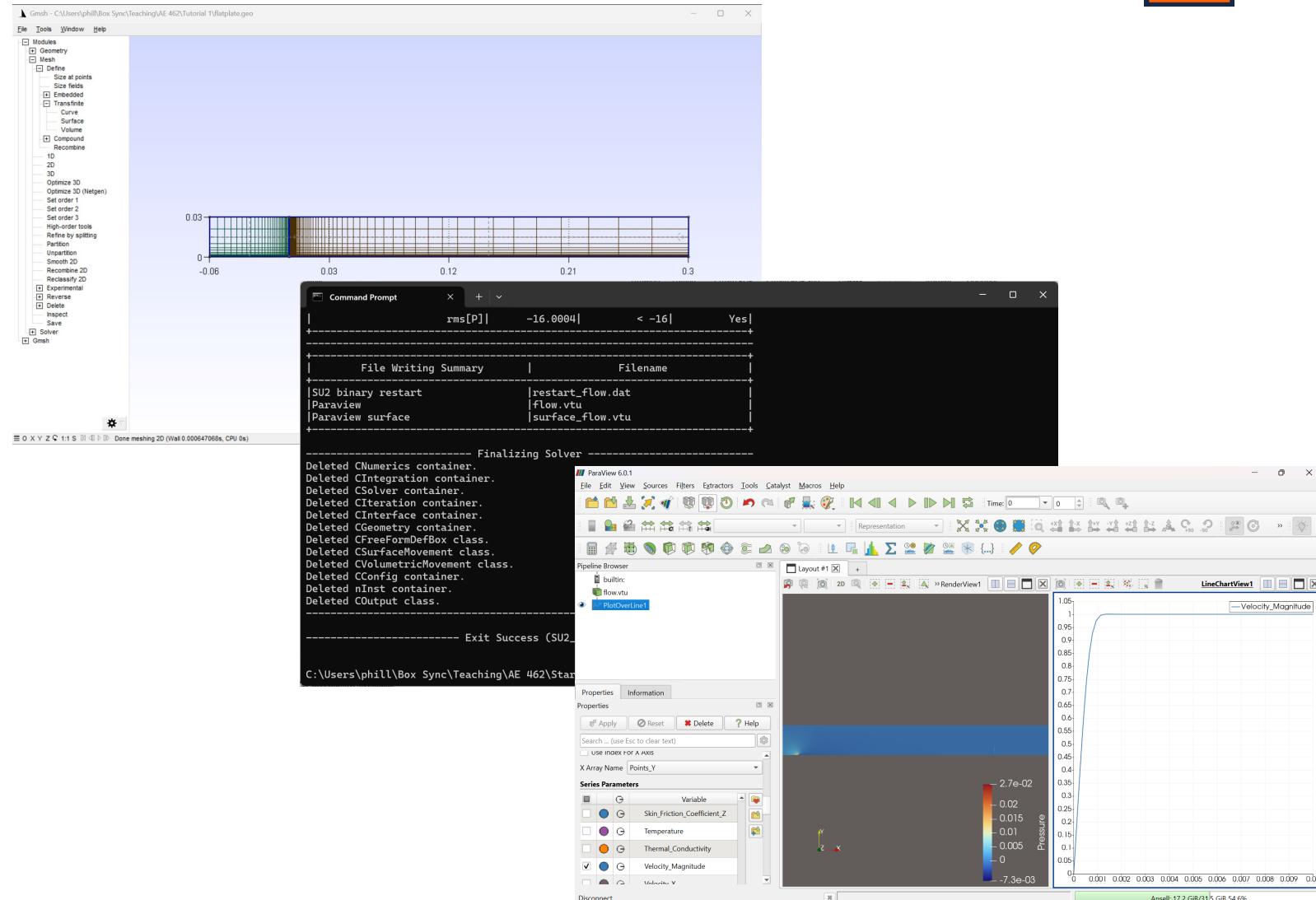


Workflow Example



Overview

- This example considers a basic flat plate laminar boundary layer



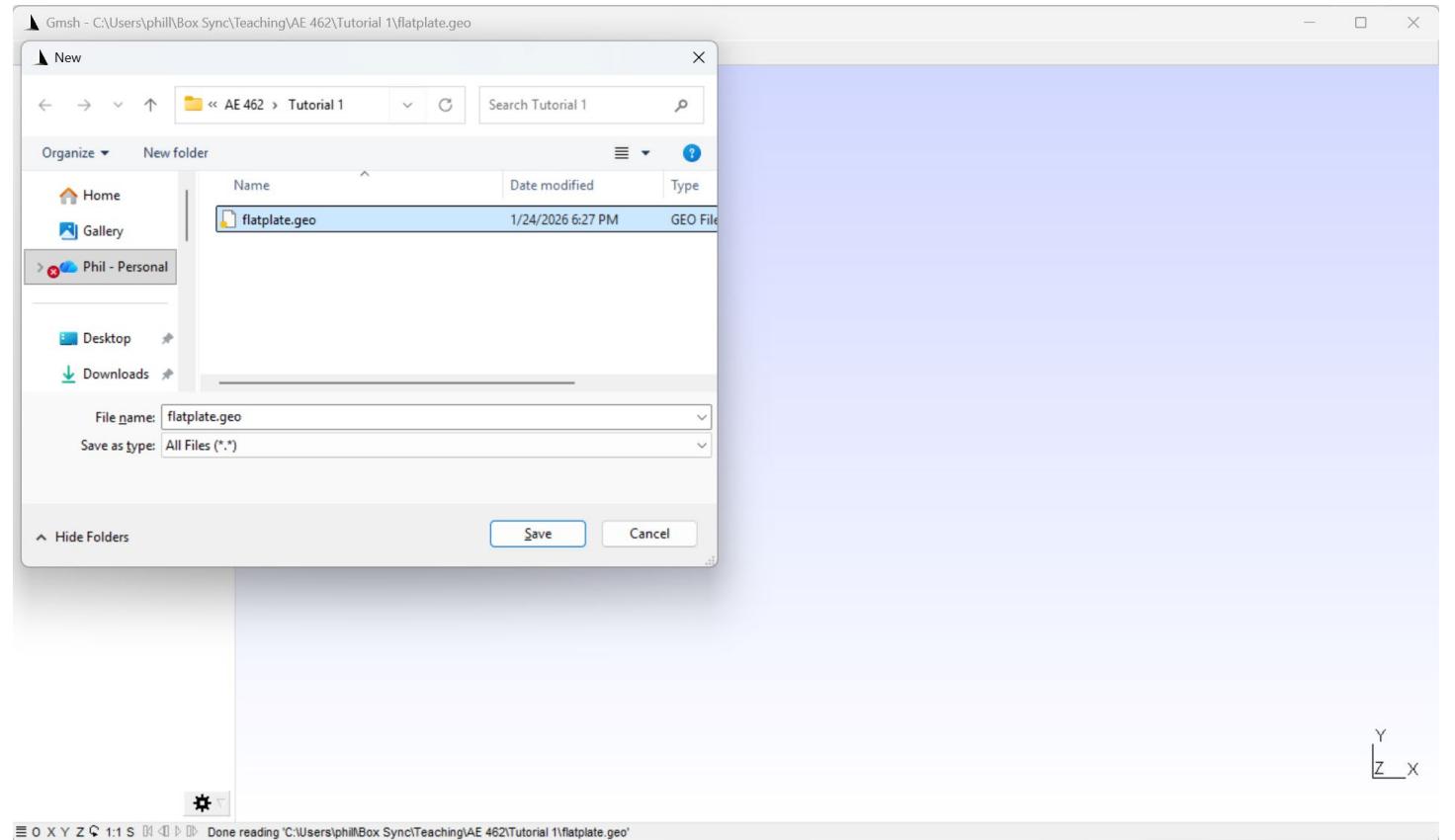


Geometry

- 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

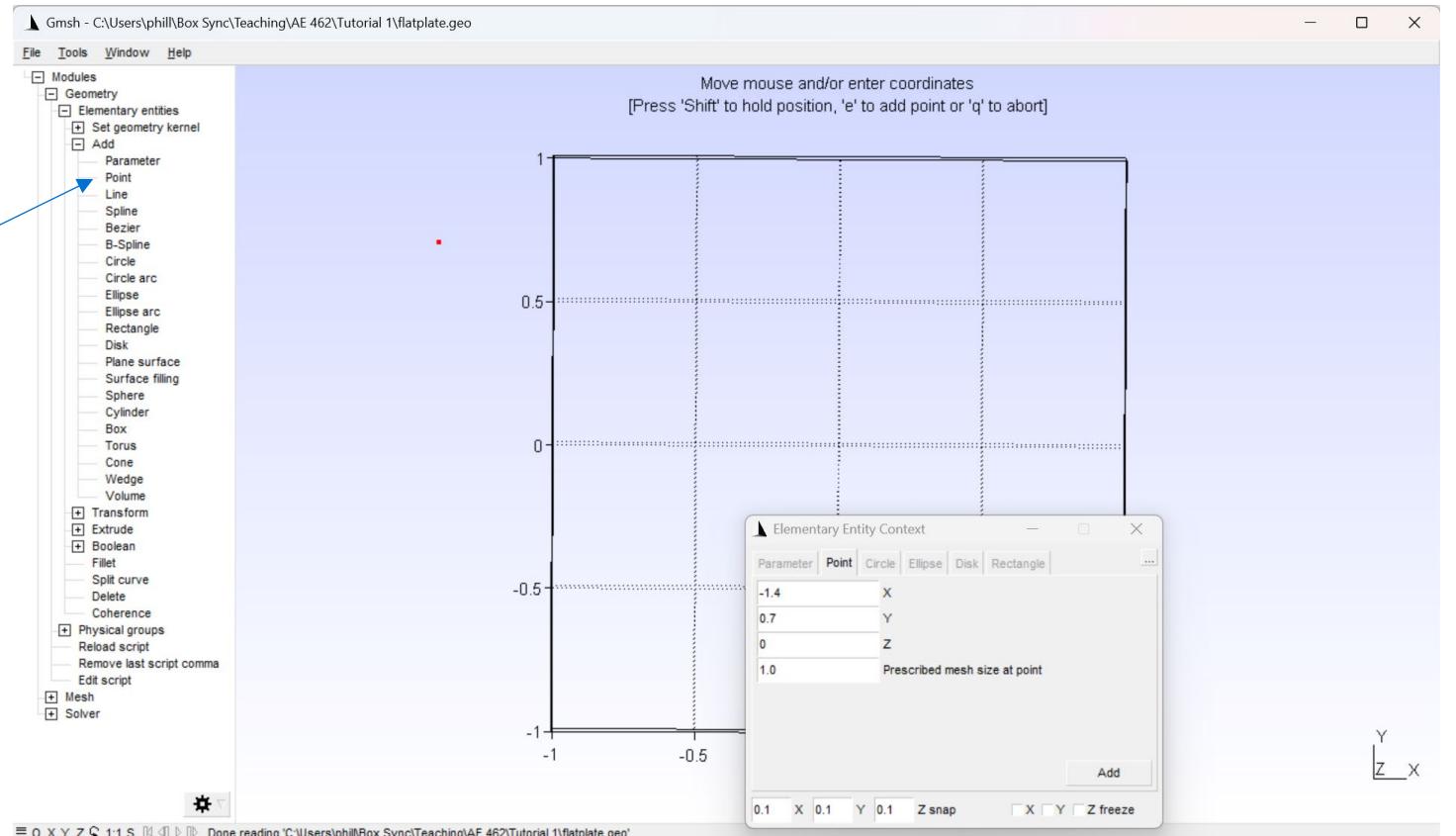
- We will begin by establishing the physical dimensions of the domain

- 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”





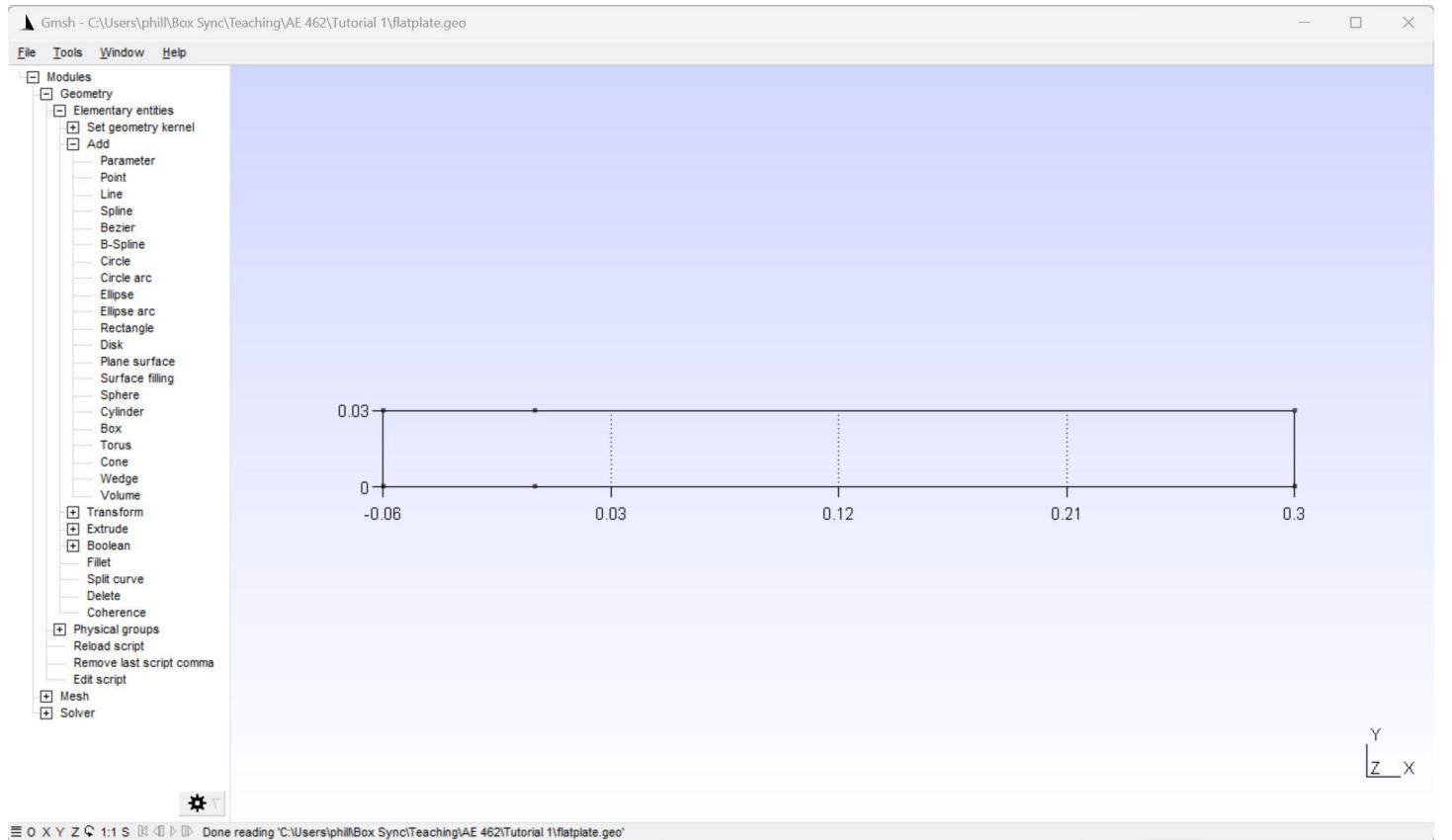
Meshing

- 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



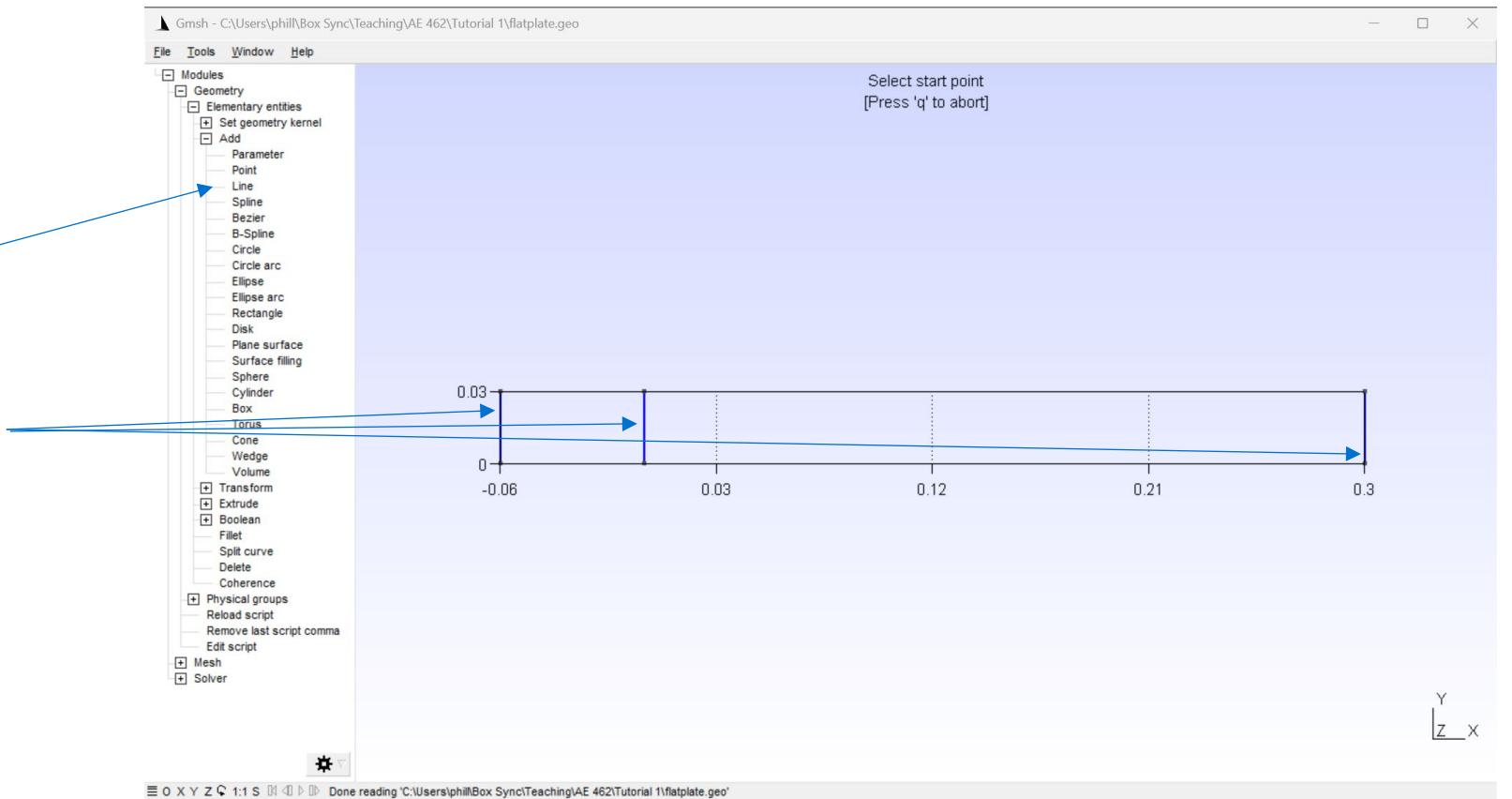
Meshing

- Select

- Modules
- → Geometry
- → Elementary entities
- → Add
- → Line

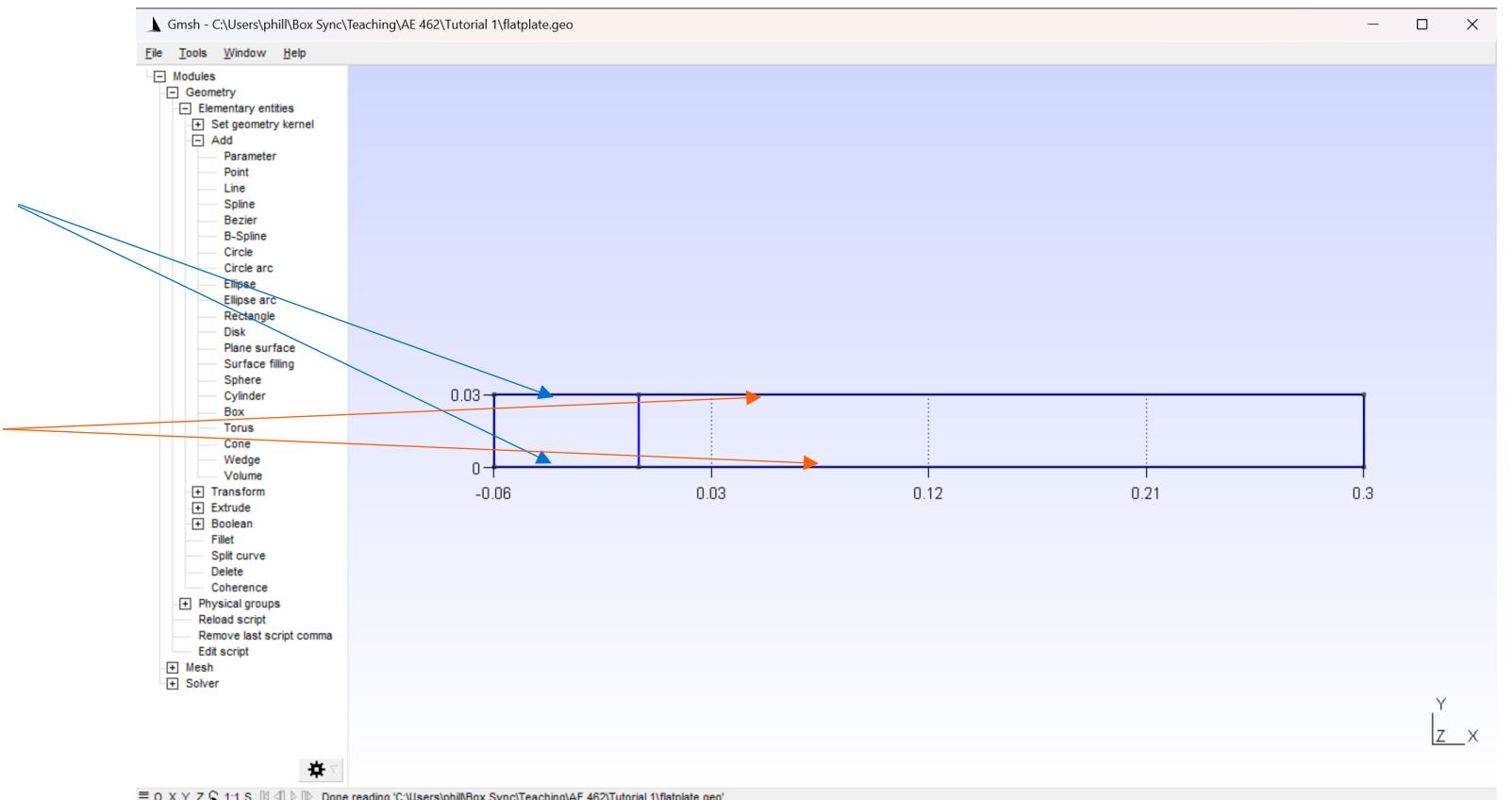
• Add three vertical lines, from bottom ($y = 0$) to top ($y = 0.03$) on the upstream and downstream end of the domain

- 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

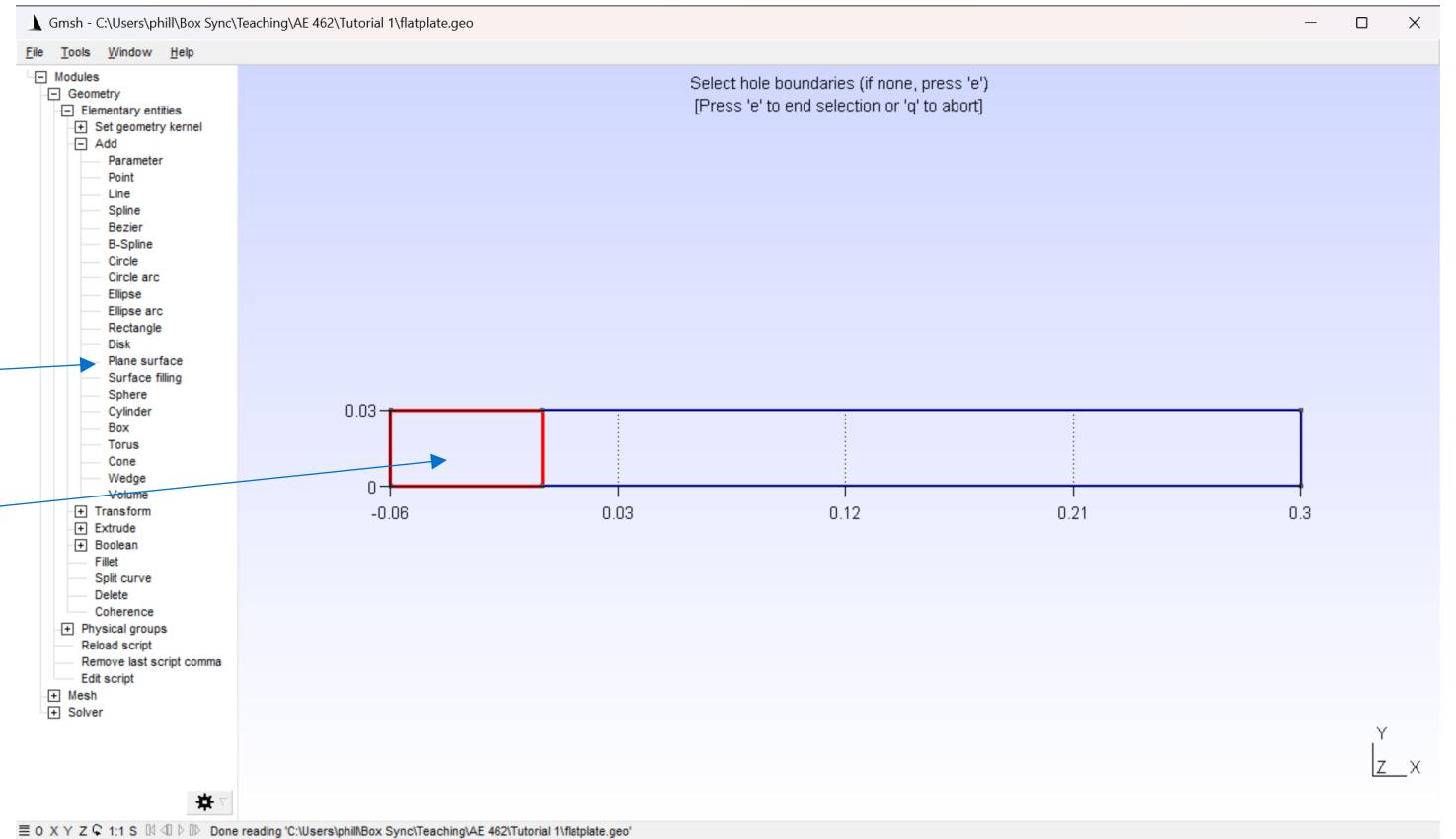
- Go to:

- Modules
- → Geometry
- → Elementary entities
- → Add
- → Plane Surface

- Select the box on the upstream end of the domain created by our line segments

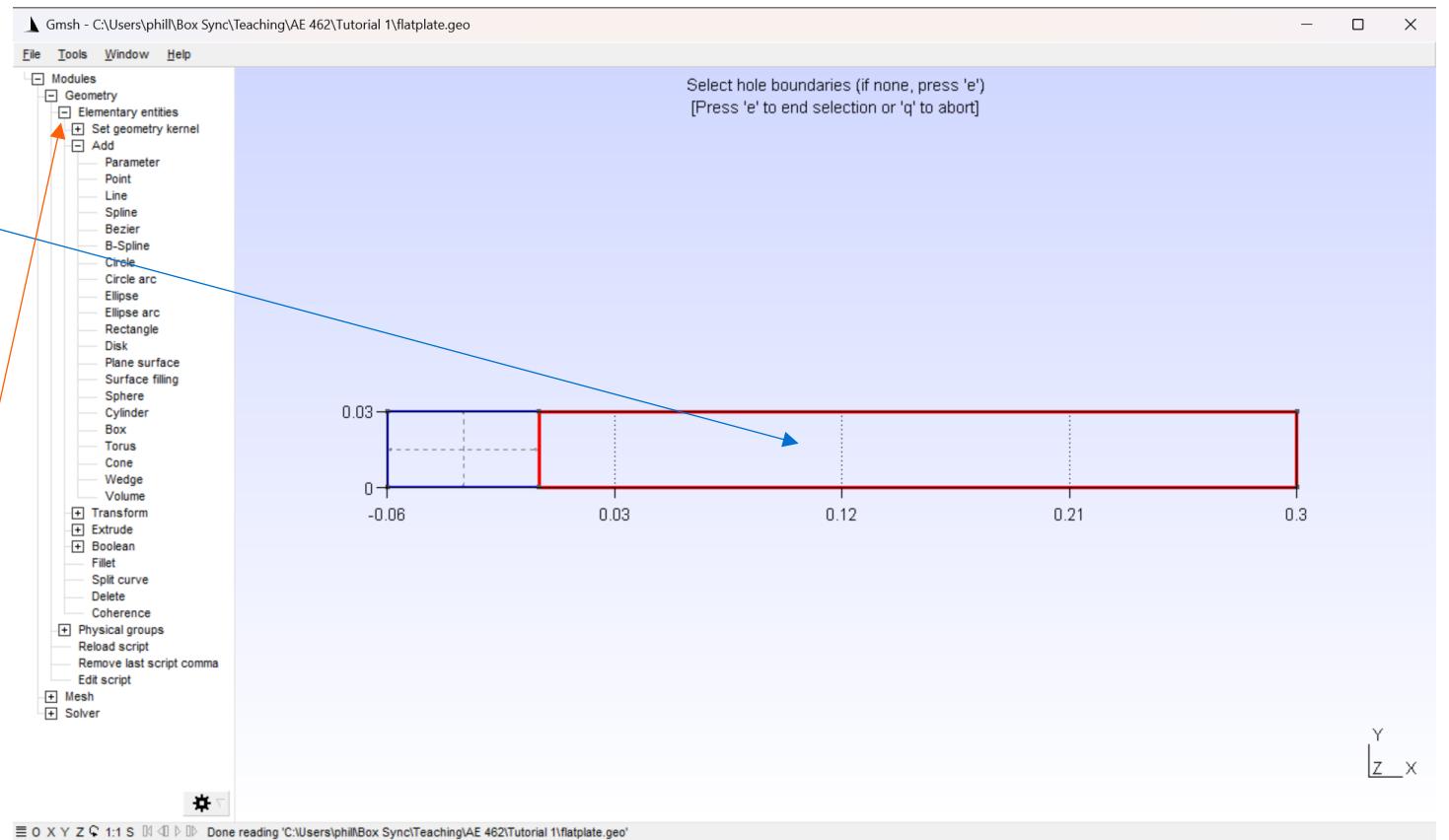
- Selecting the top, left, or bottom will select all three of the outside segments. Also click the inside vertical line.

- 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

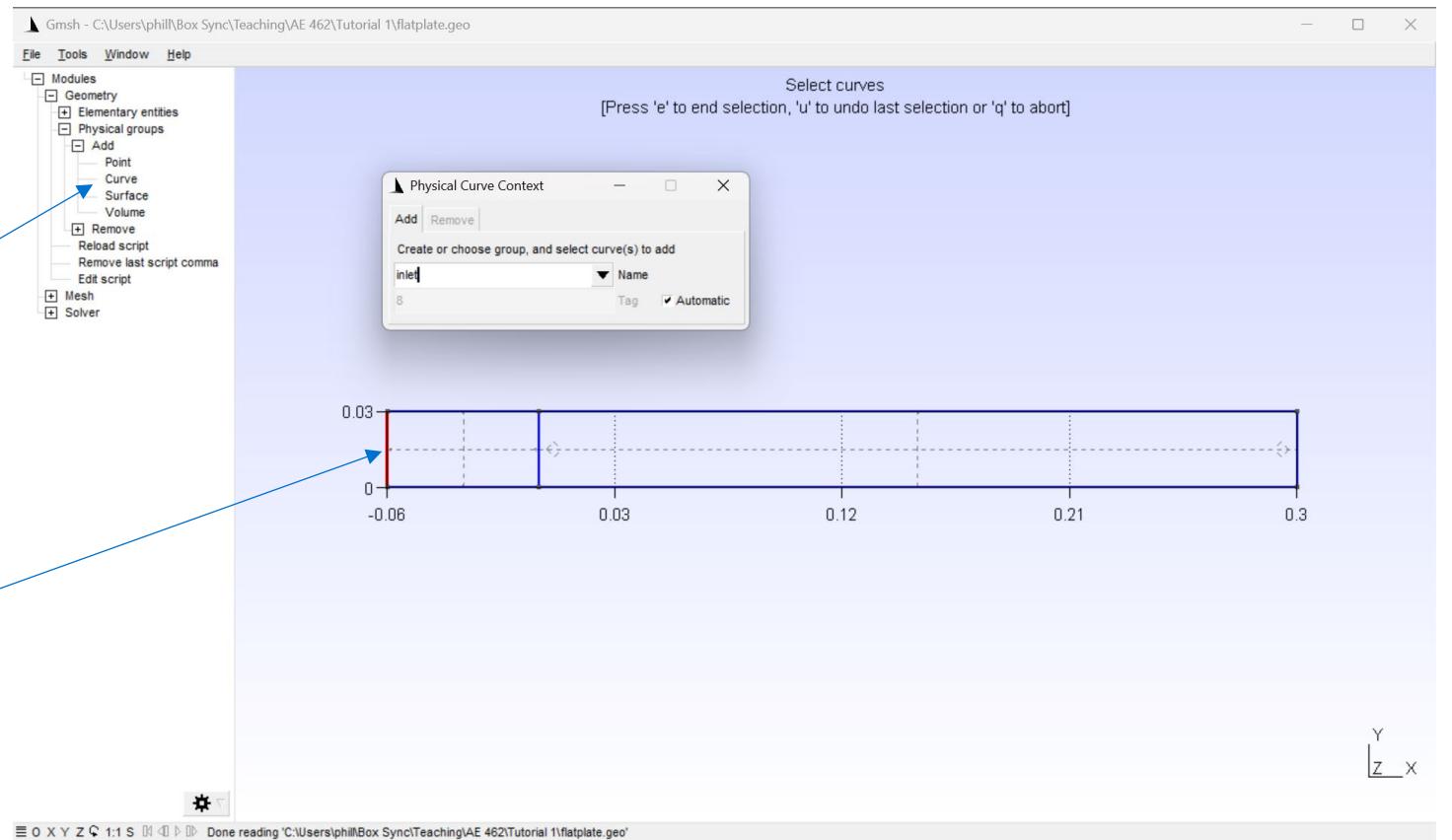
- We will now provide variable names for the different boundaries of the flow domain

- 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

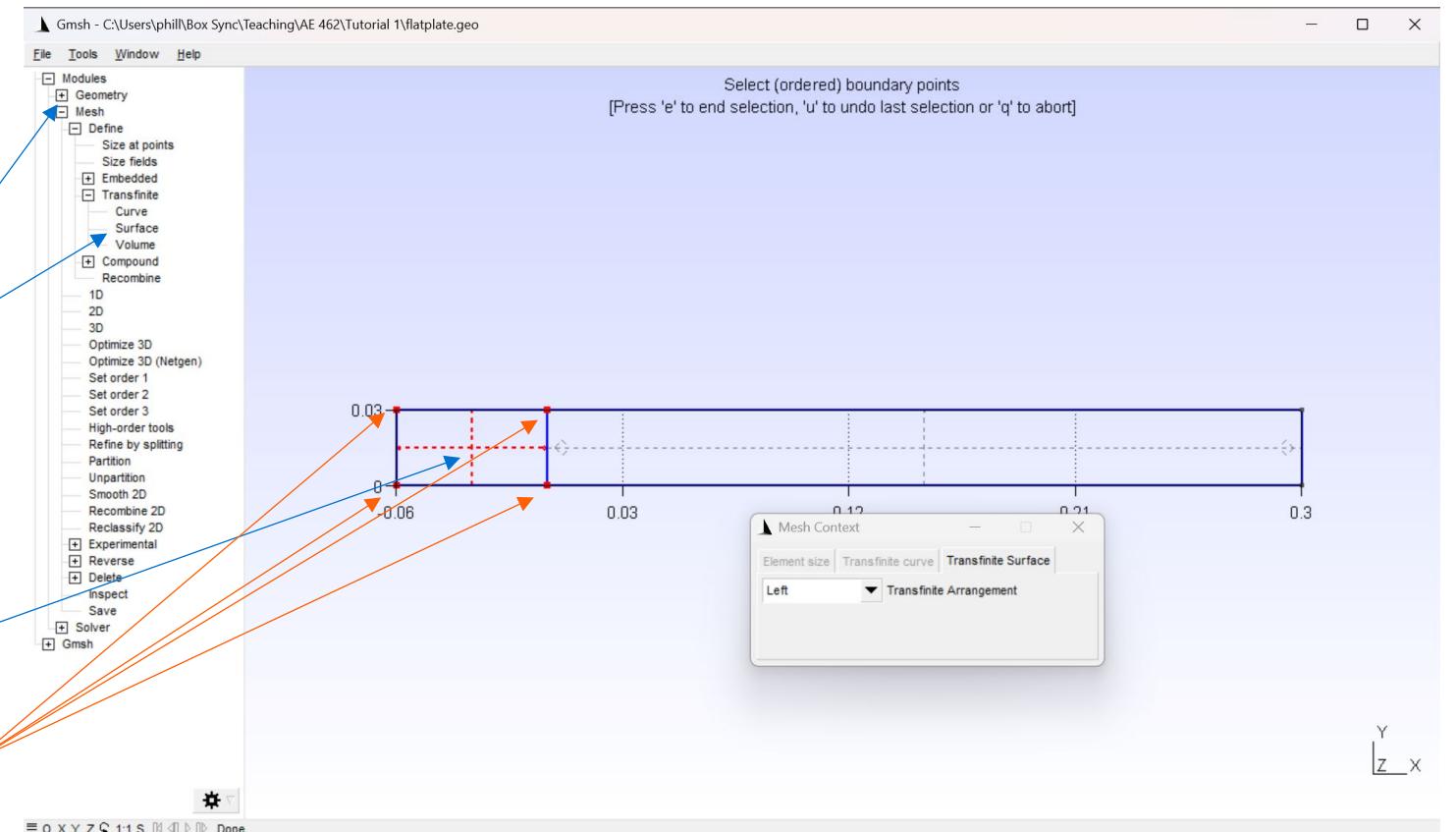
- Now that we are finished with defining the geometry, you can select the [-] next to the geometry entry in the left-side menu

- Select

- Modules
- → Mesh
- → Define
- → Transfinite
- → Surface

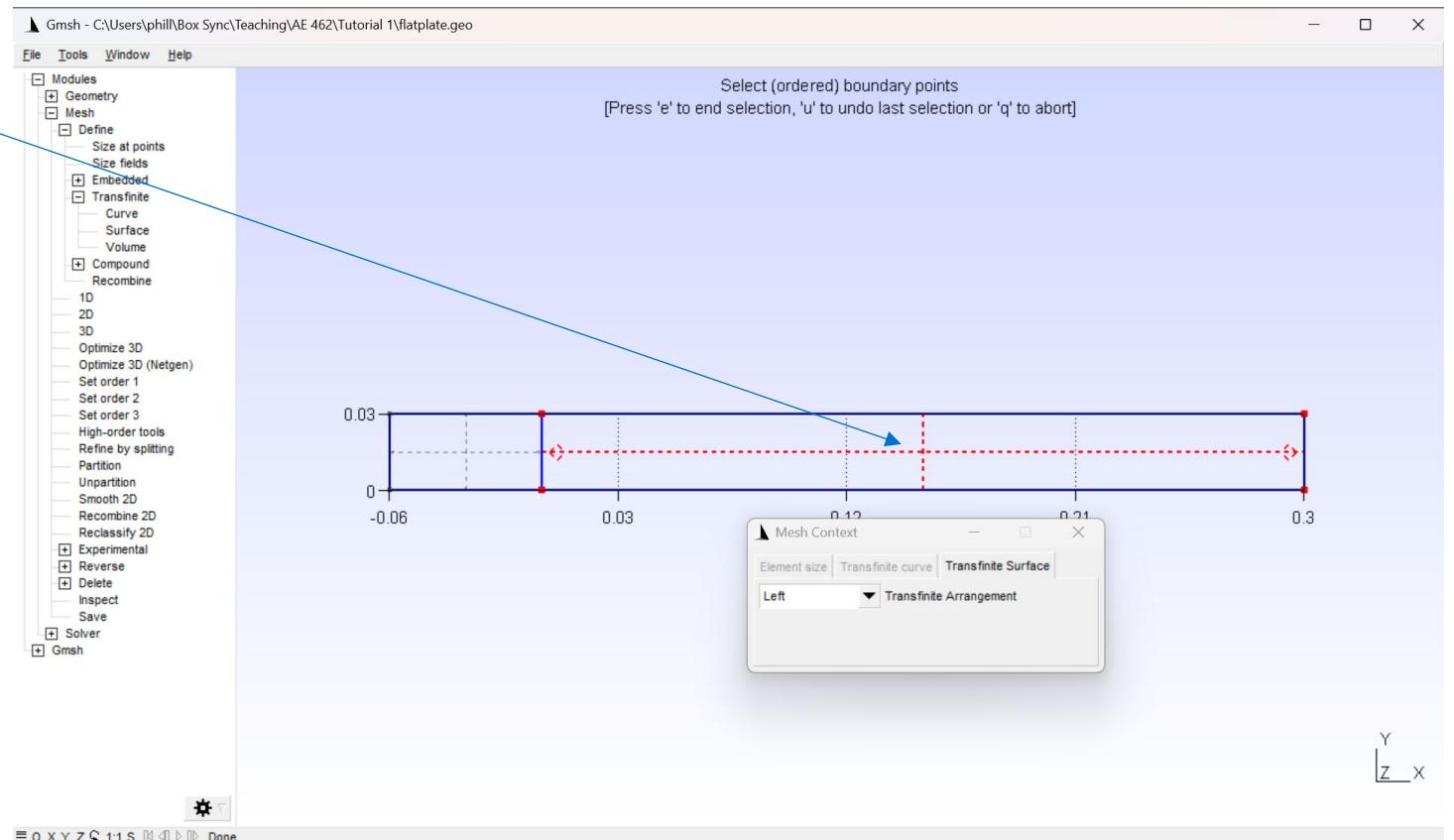
- Click on the dashed lines in the left-side box to highlight it in red

- 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

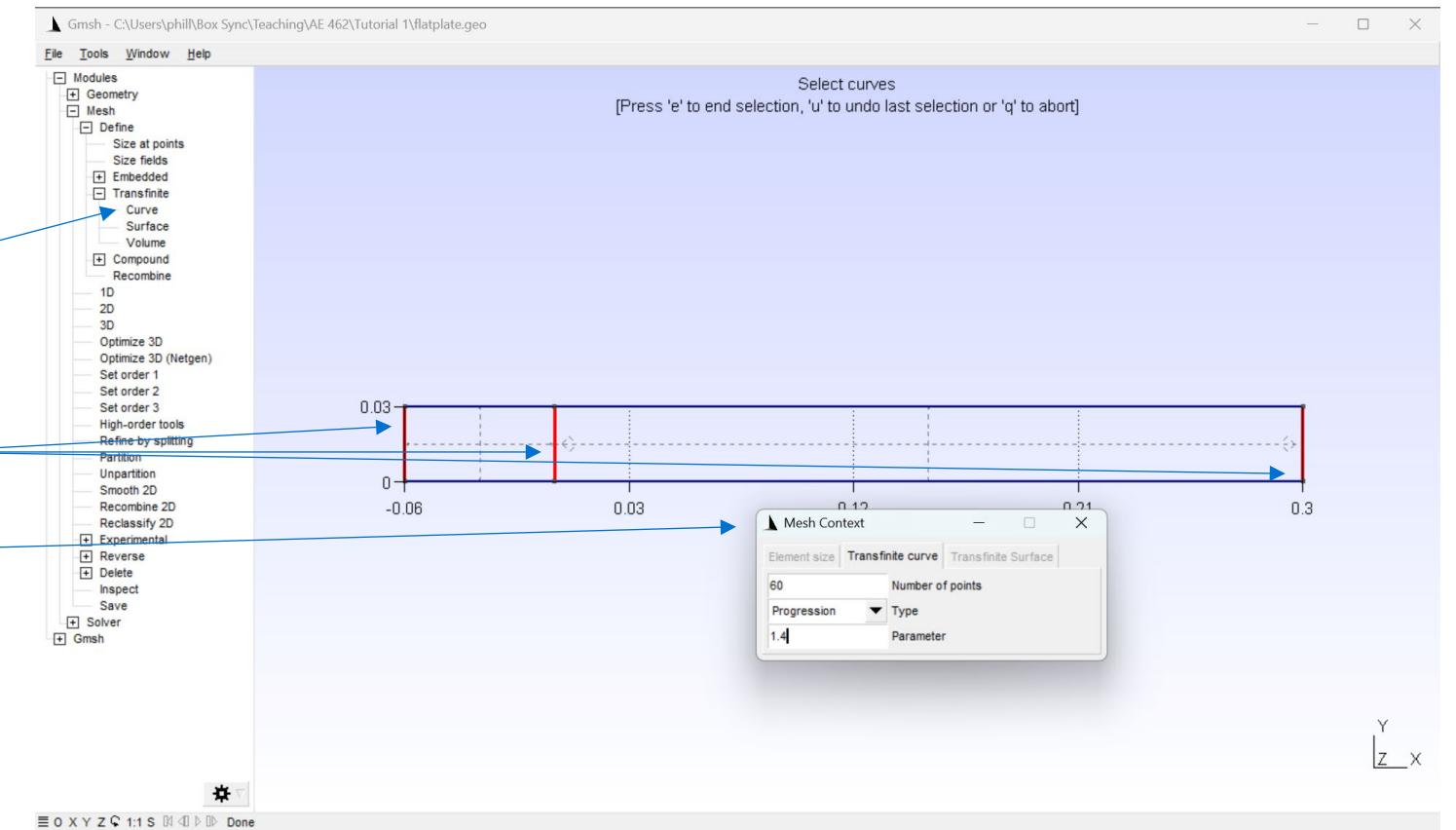
- Modules
- → Mesh
- → Define
- → Transfinite
- → Curve

- Click on the three vertical lines
($x = -0.06, 0$, and 0.3)

- Under the “Mesh Context” menu,
enter:

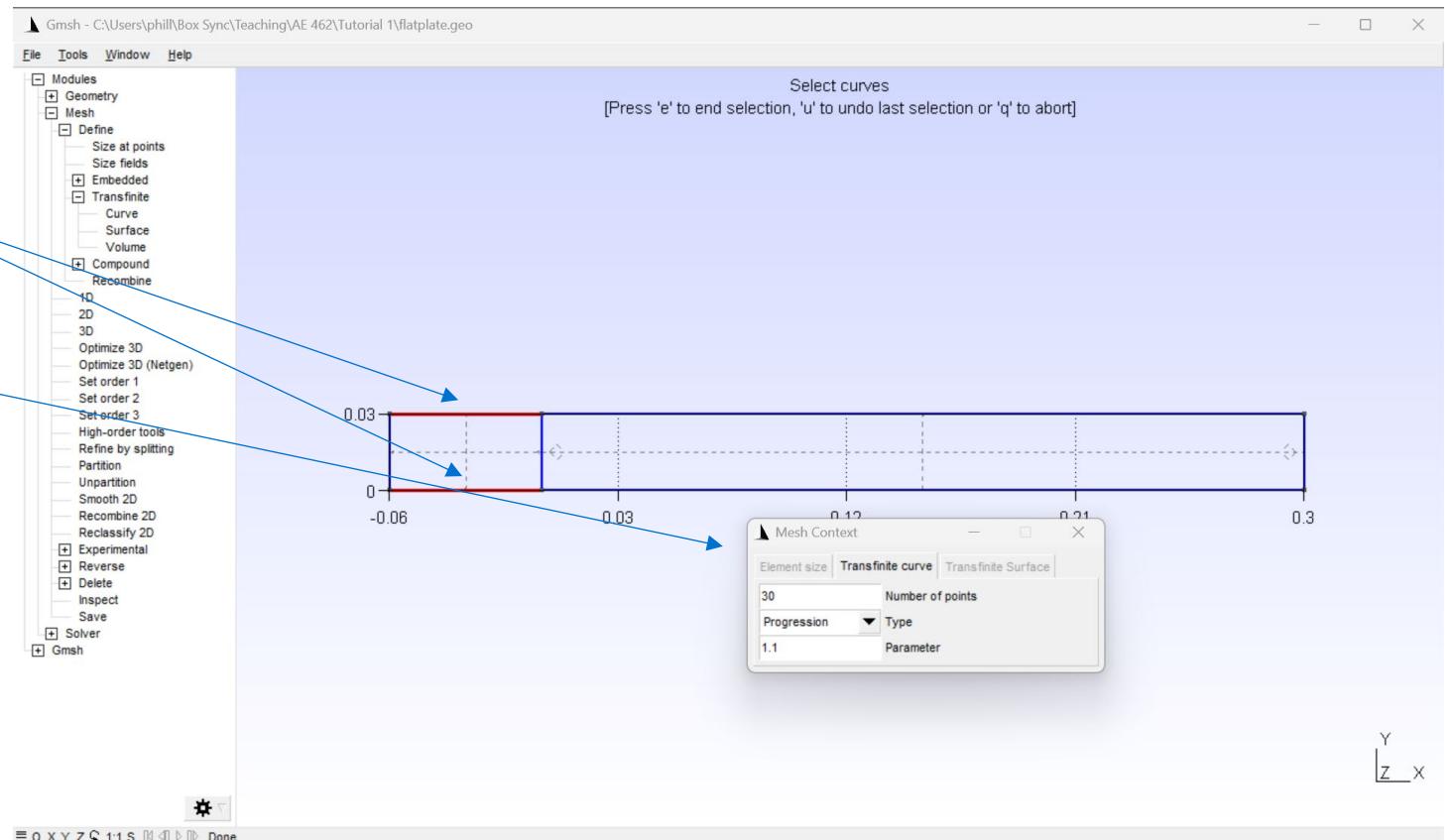
- Number of points = 60
- Type = Progression
- Parameter = 1.2

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



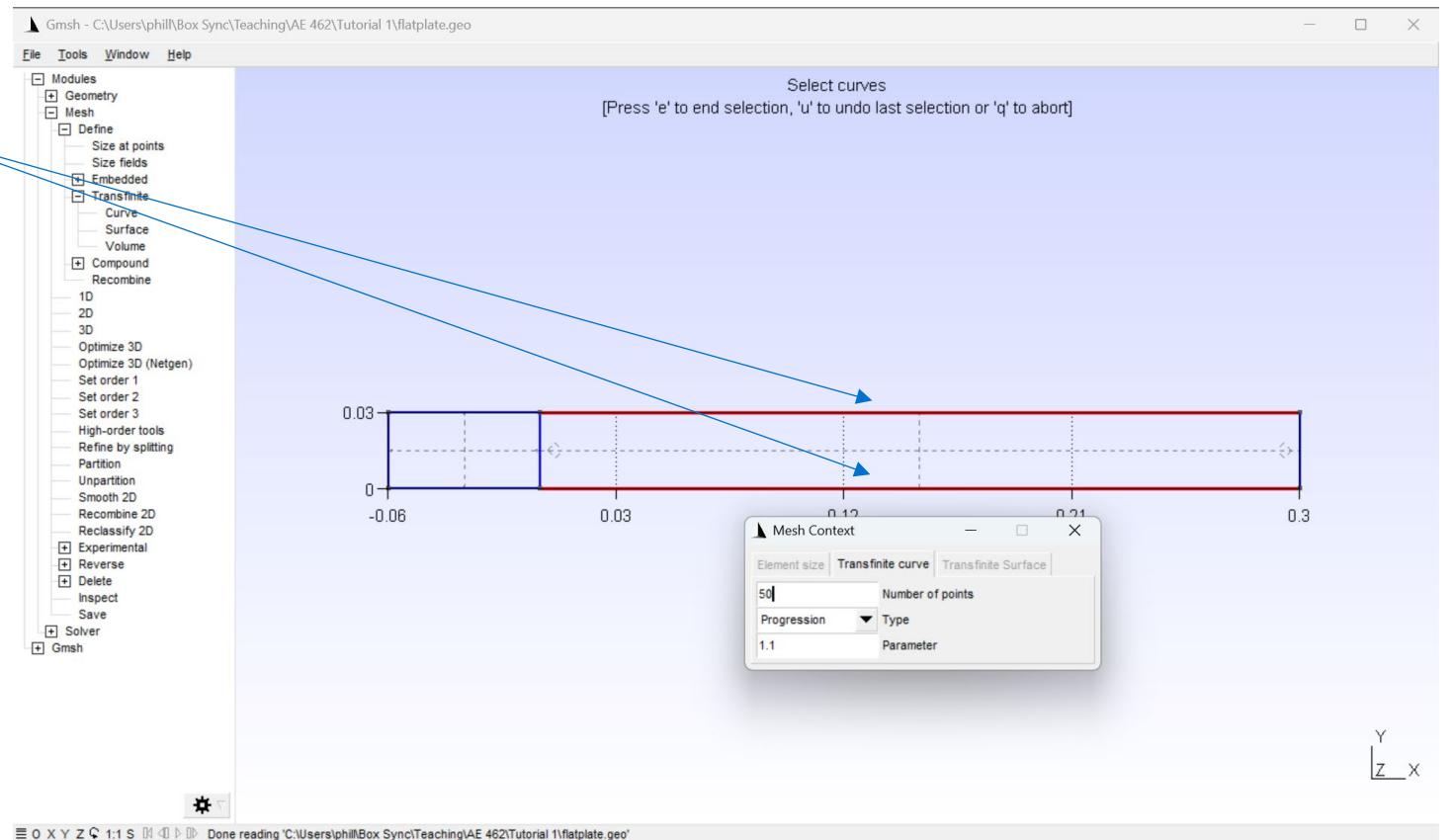
Meshing

- 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

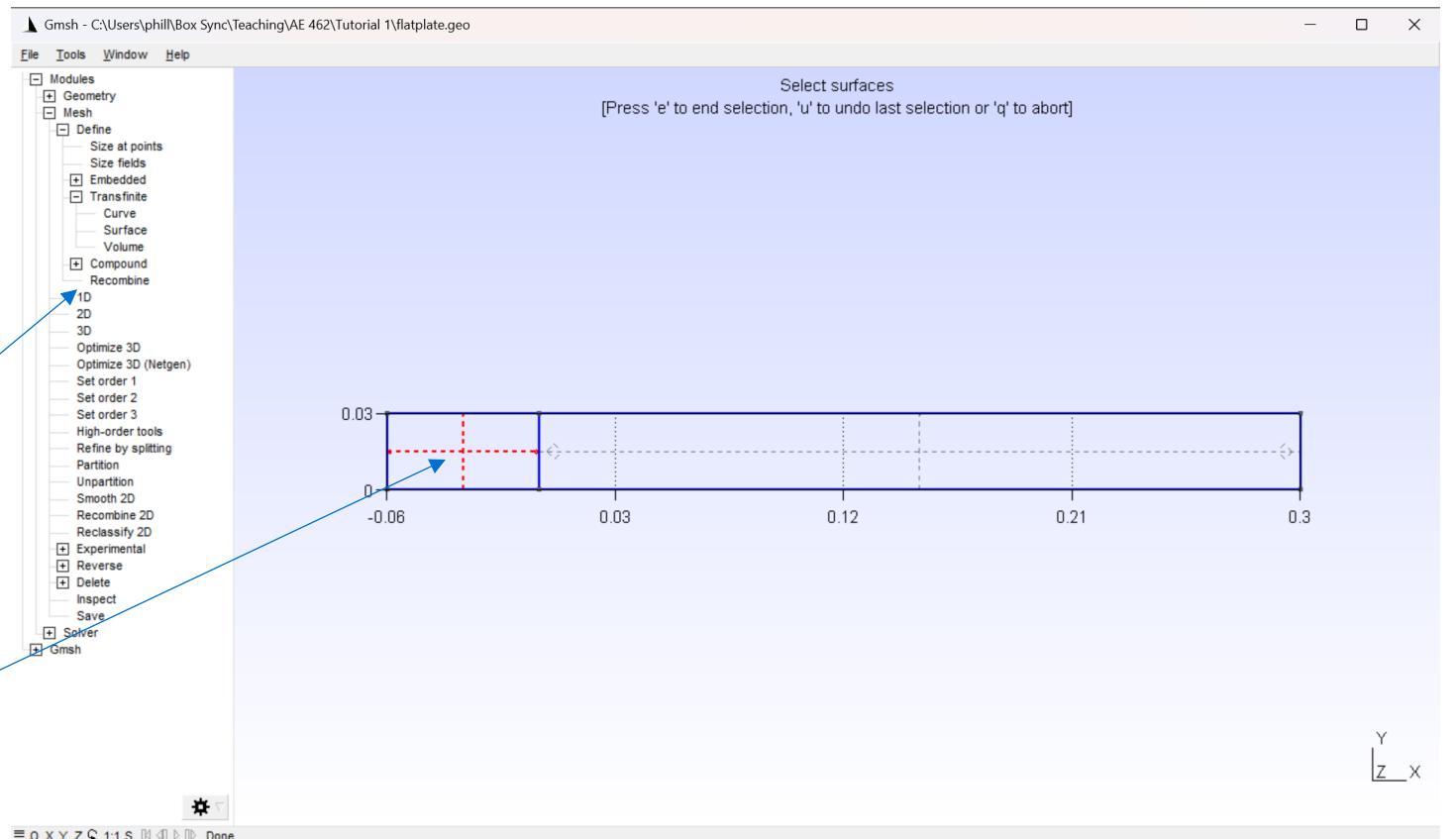
- We are now just about ready to generate a mesh

- Select

- Modules
- → Mesh
- → Define
- → Recombine

- Select the dashed lines in the left-side box

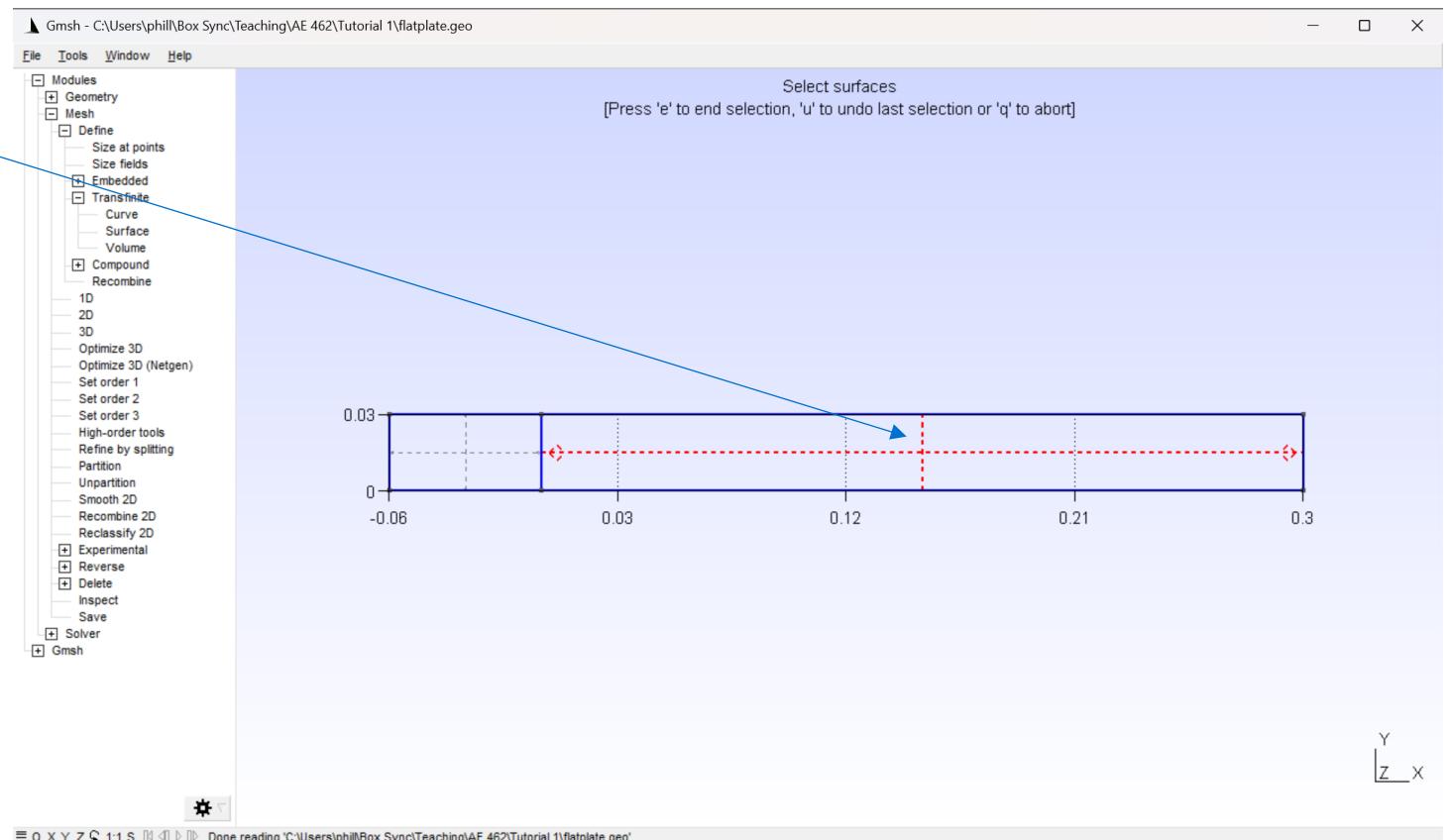
- Press “e” to end selection



Meshing



- Repeat for the right-size box
 - Press “e” to finalize selection
 - Press “q” to exit the selection menu



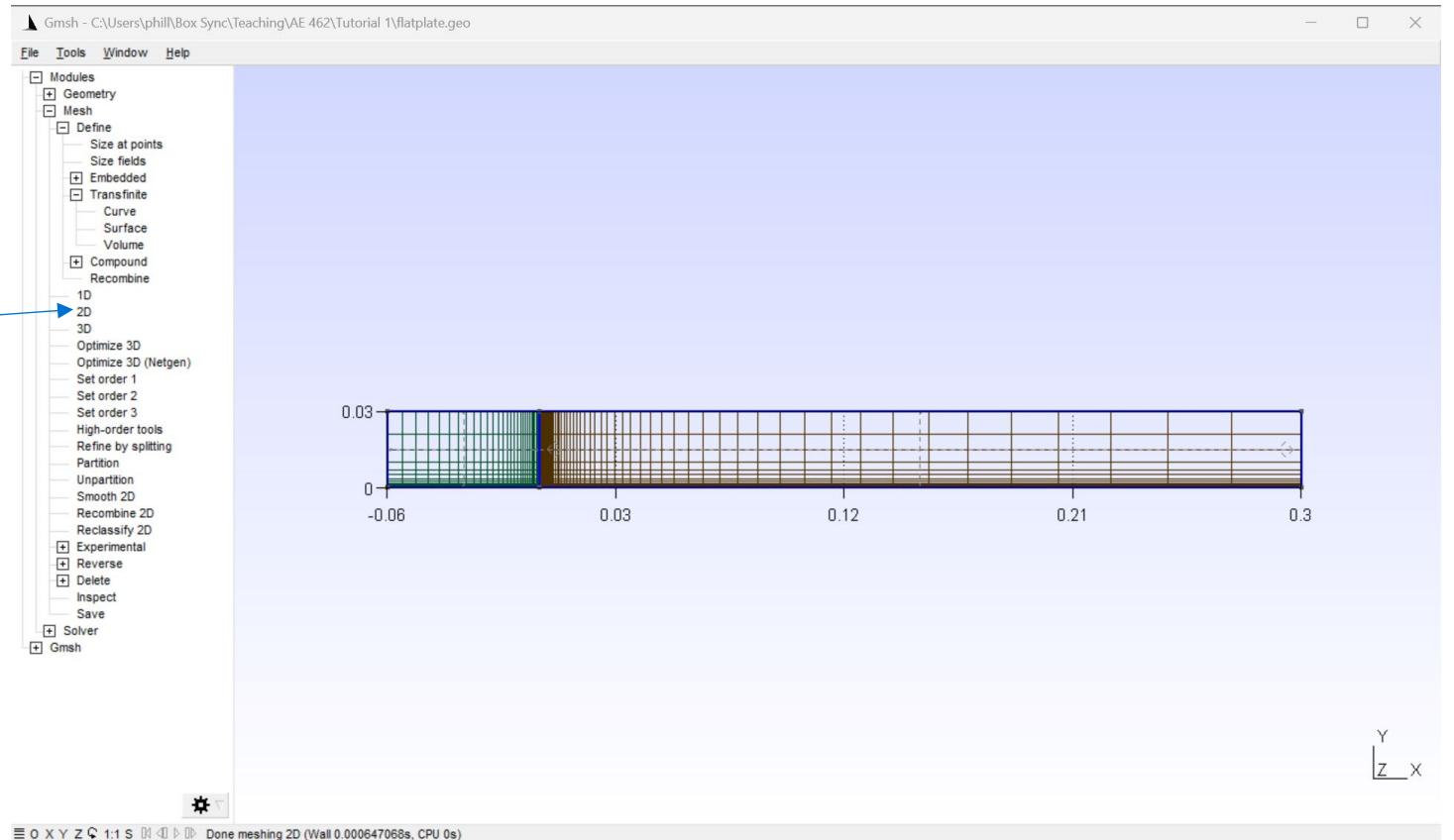
Meshing

- Select

- Modules
- → Mesh
- → 2D

- You will now see a visualization of the mesh

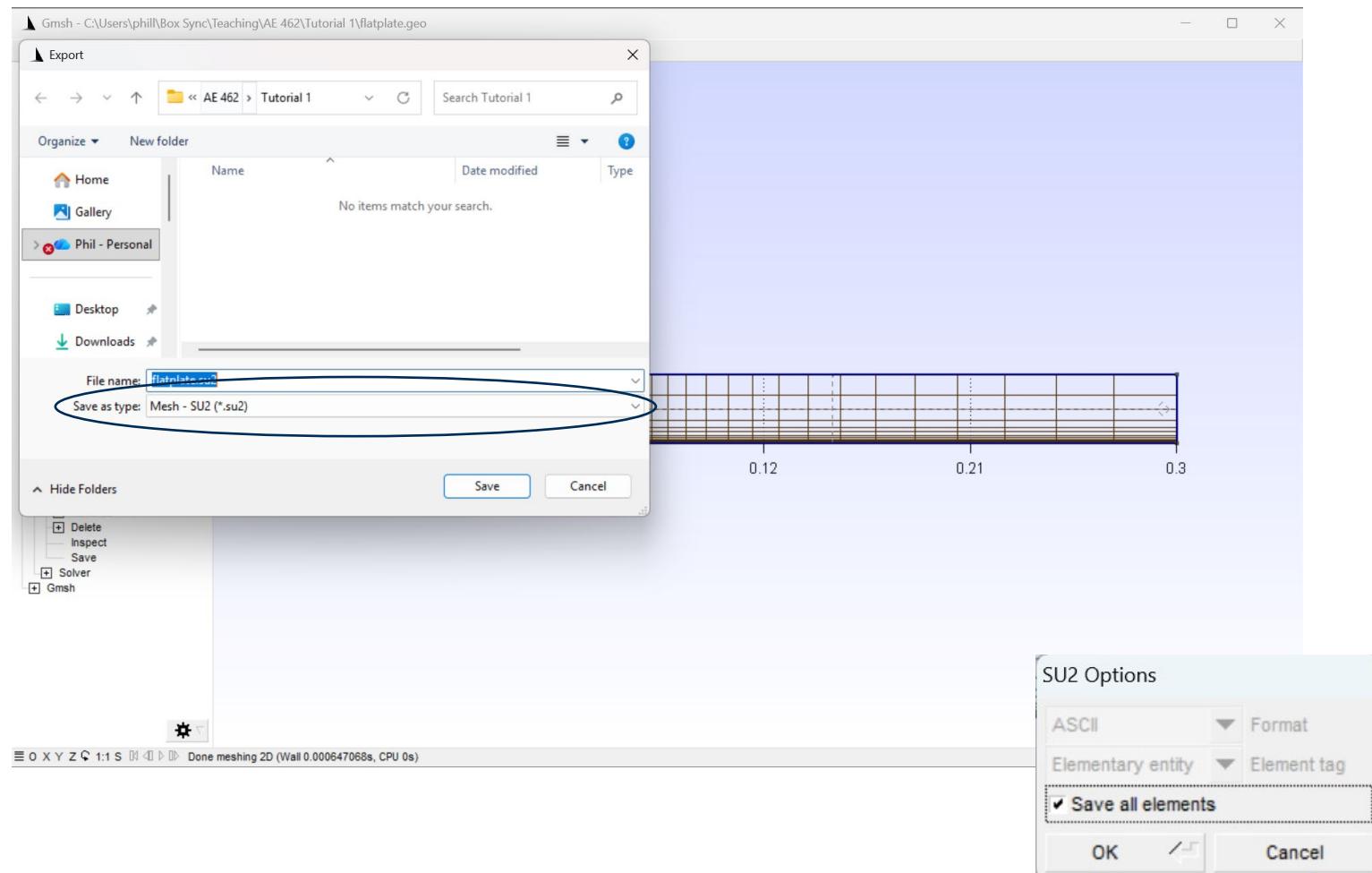
- Cells are concentrated to y-positions close to the plate and x-positions close to the leading edge ($x = 0$)





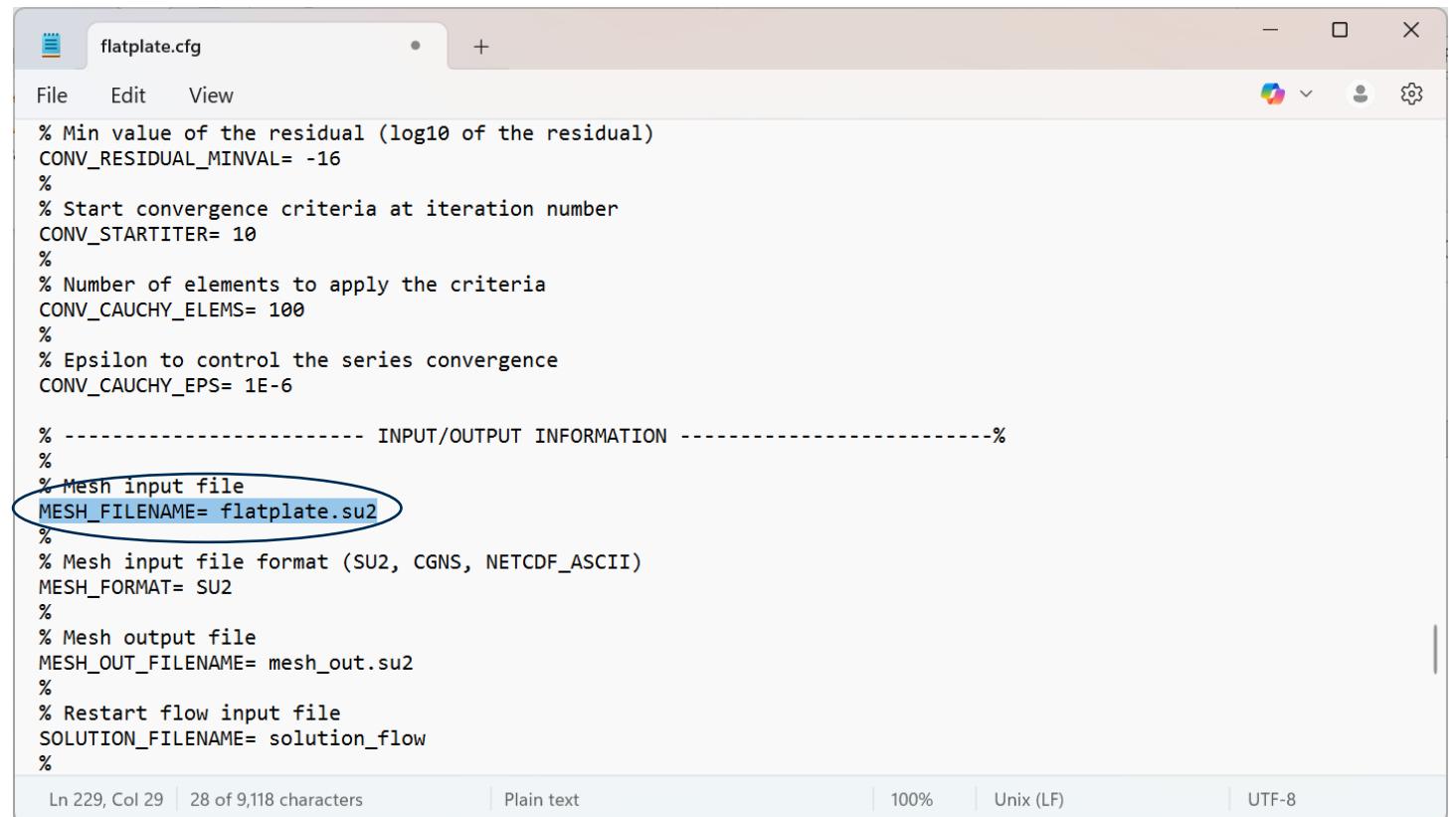
Meshering

- 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

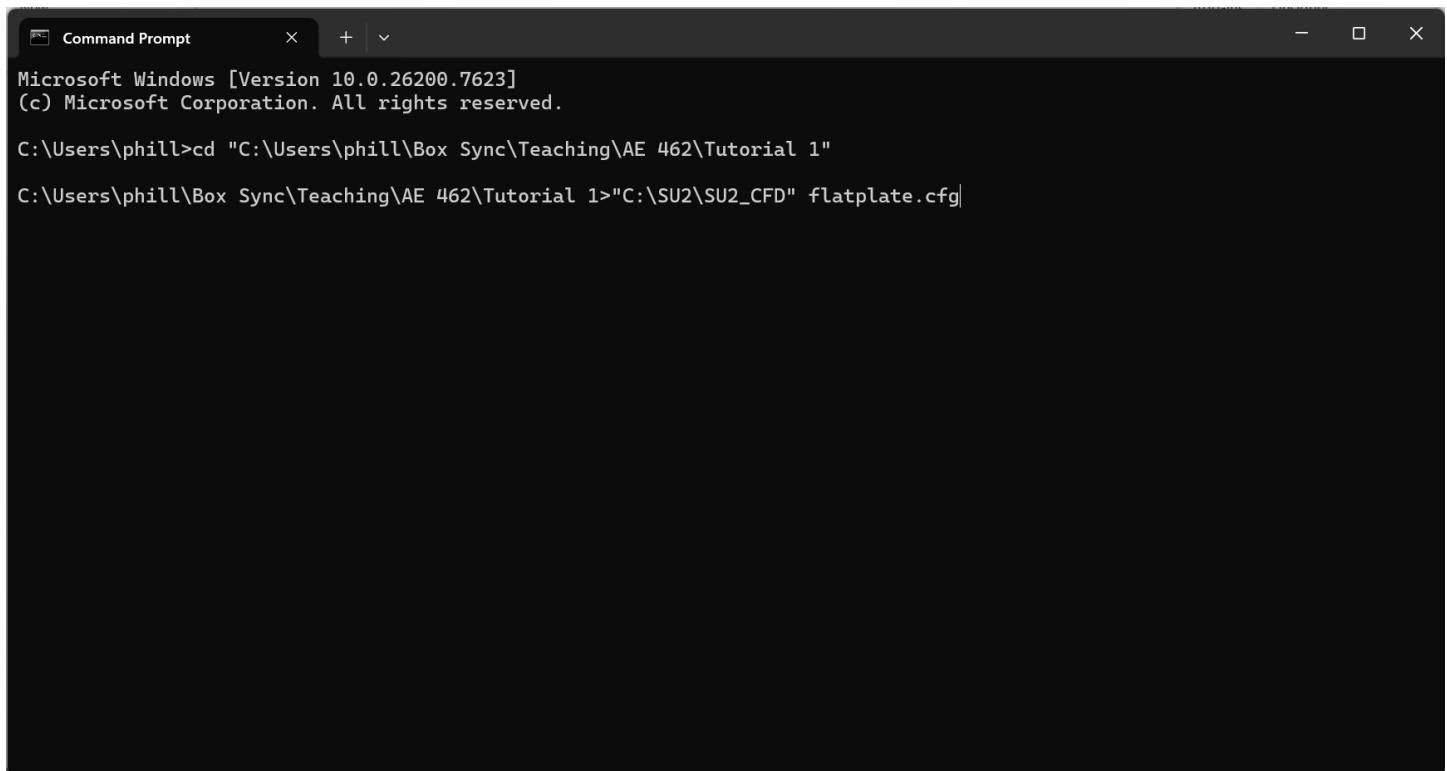


```
flatplate.cfg
File Edit View
% Min value of the residual (log10 of the residual)
CONV_RESIDUAL_MINVAL= -16
%
% Start convergence criteria at iteration number
CONV_STARTITER= 10
%
% Number of elements to apply the criteria
CONV_CAUCHY_ELEMS= 100
%
% Epsilon to control the series convergence
CONV_CAUCHY_EPS= 1E-6

% ----- INPUT/OUTPUT INFORMATION -----
%
% Mesh input file
MESH_FILENAME= flatplate.su2
%
% Mesh input file format (SU2, CGNS, NETCDF_ASCII)
MESH_FORMAT= SU2
%
% Mesh output file
MESH_OUT_FILENAME= mesh_out.su2
%
% Restart flow input file
SOLUTION_FILENAME= solution_flow
%
Ln 229, Col 29 | 28 of 9,118 characters | Plain text | 100% | Unix (LF) | UTF-8
```

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



The screenshot shows a Microsoft Windows Command Prompt window titled "Command Prompt". The window title bar includes standard icons for minimize, maximize, and close. The command prompt itself has a dark background and white text. It displays the following command and its output:

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

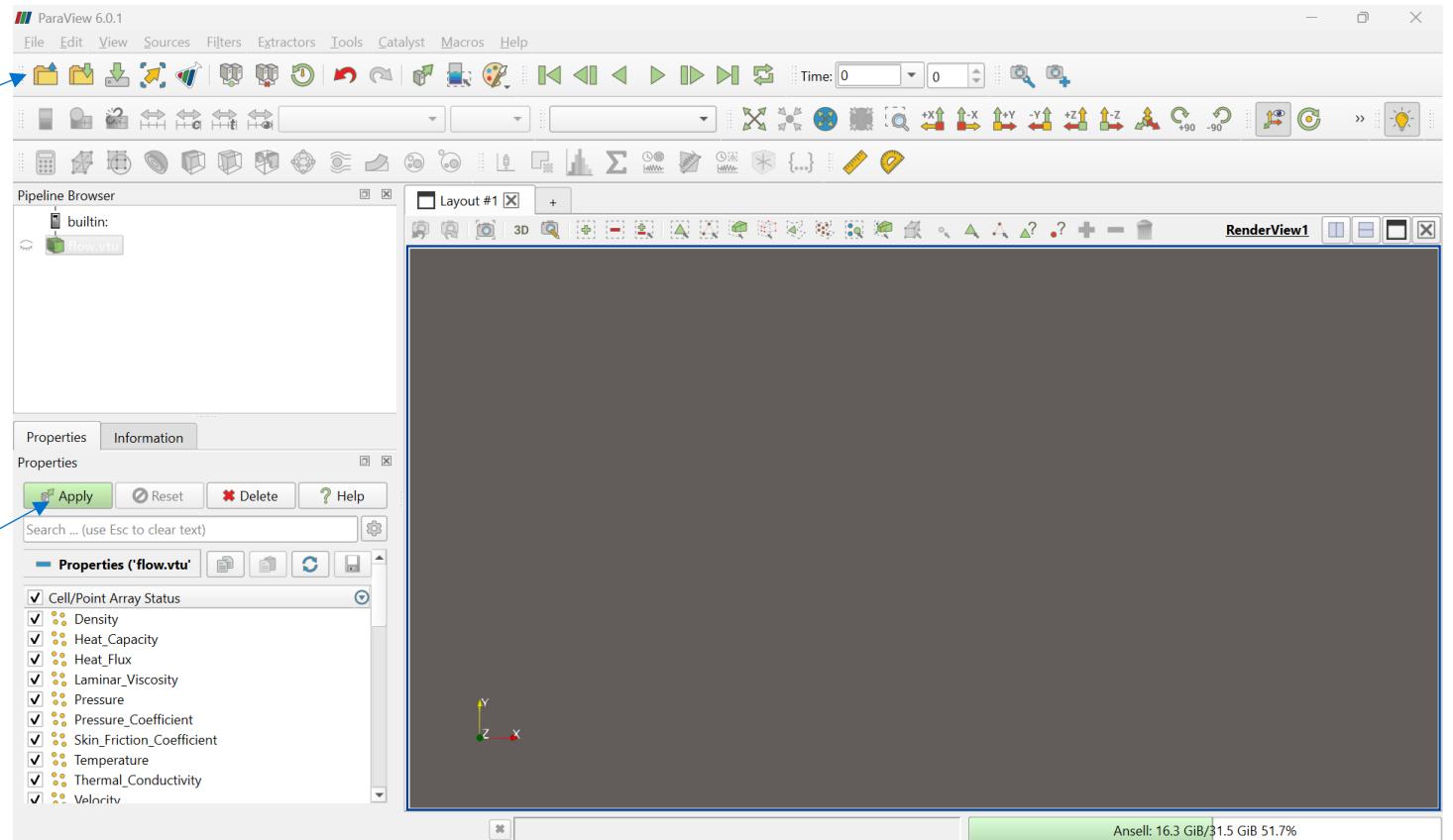
C:\Users\phill>cd "C:\Users\phill\Box Sync\Teaching\AE 462\Tutorial 1"

C:\Users\phill\Box Sync\Teaching\AE 462\Tutorial 1>"C:\SU2\SU2_CFD" flatplate.cfg
```



Loading the Field Data

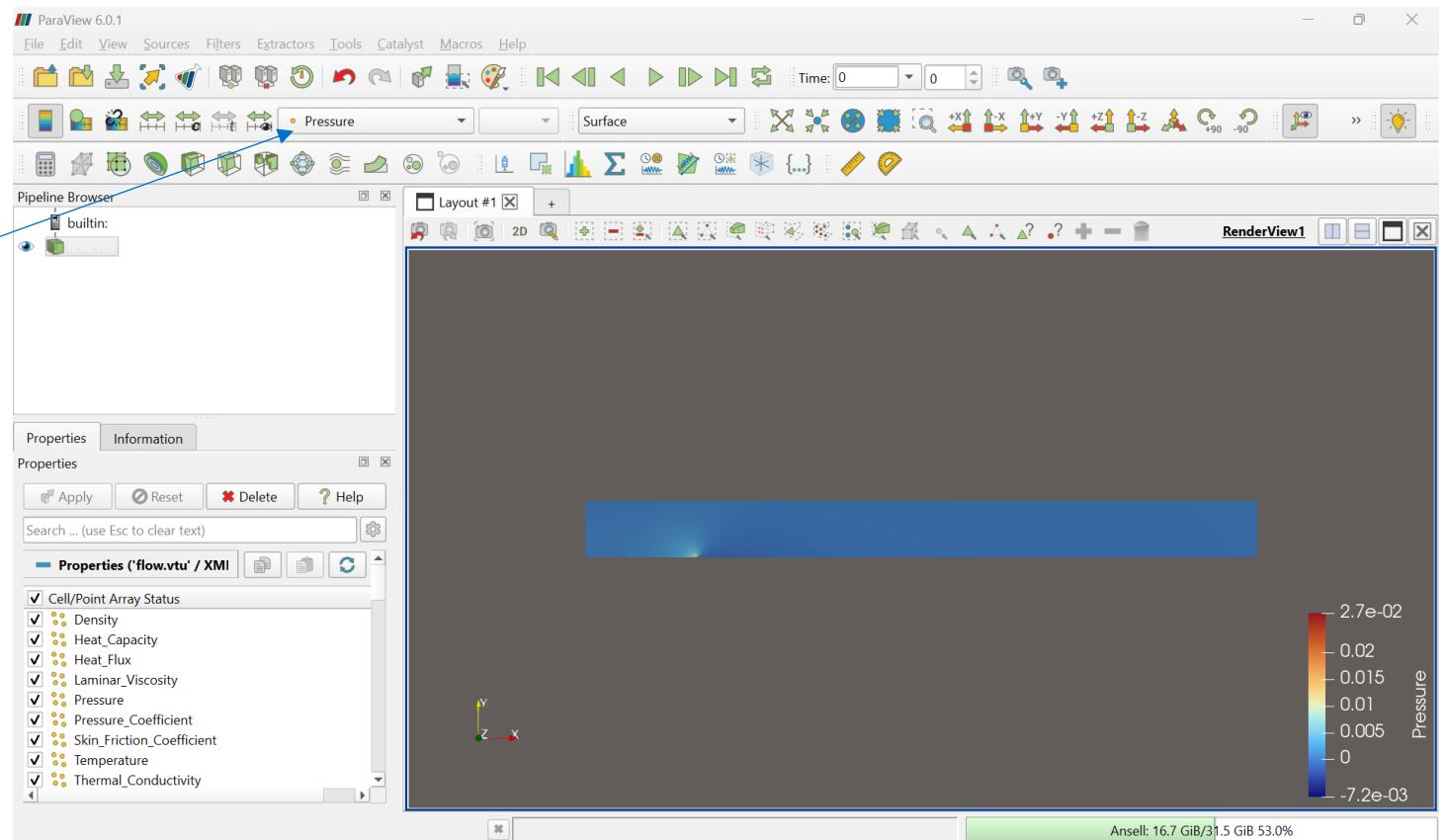
- Launch Paraview
- Click the “Open” button
 - Navigate to the directory of your .su2 mesh file and your .cfg input file
 - Load the “flow.vtu” file from this directory
 - Click the “Apply” button





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

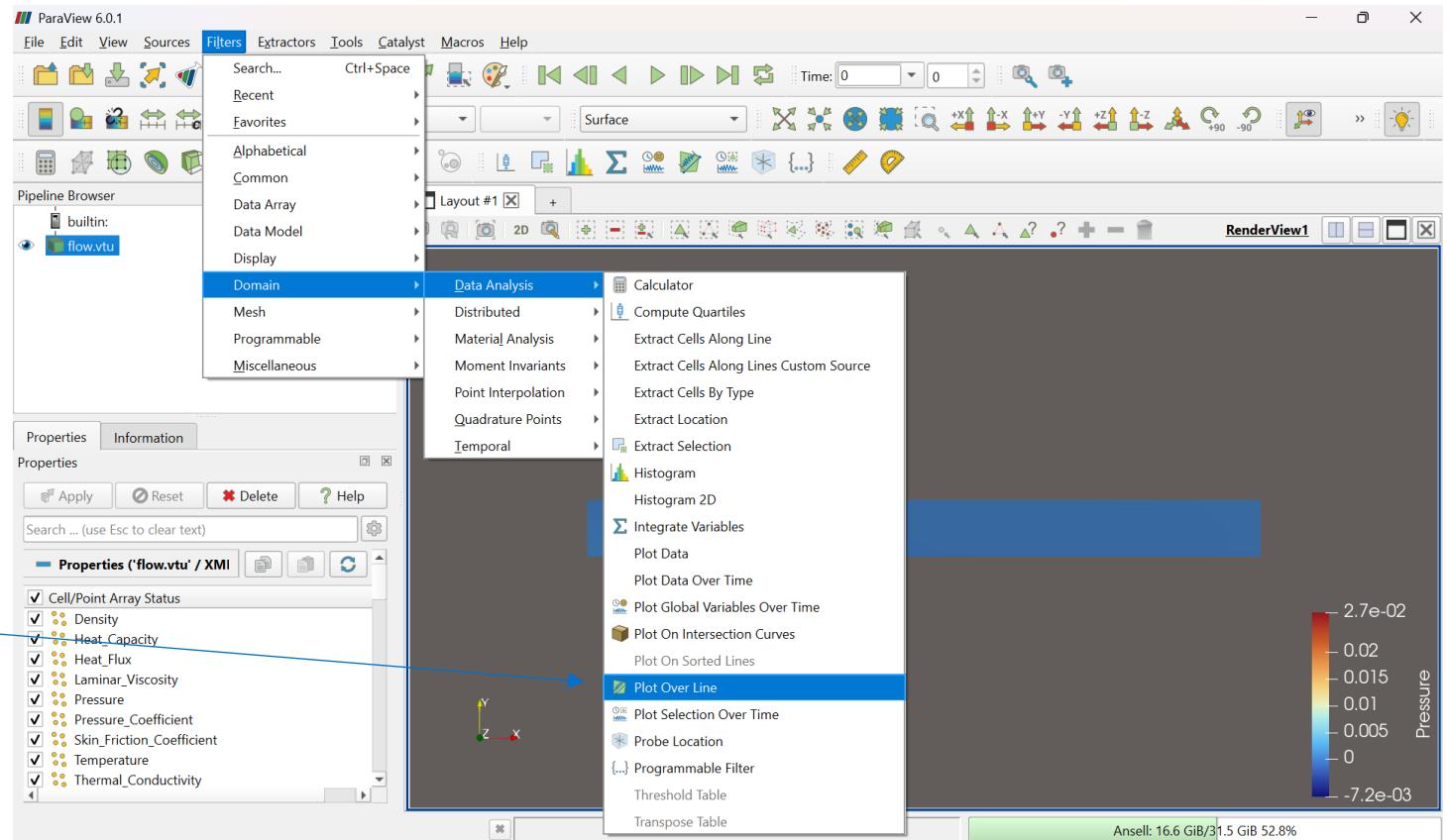


Extracting Data



- With “flow.vtu” selected in the Pipeline Browser, go to:

- Filters
- → Domain
- → Data Analysis
- → Plot Over Line



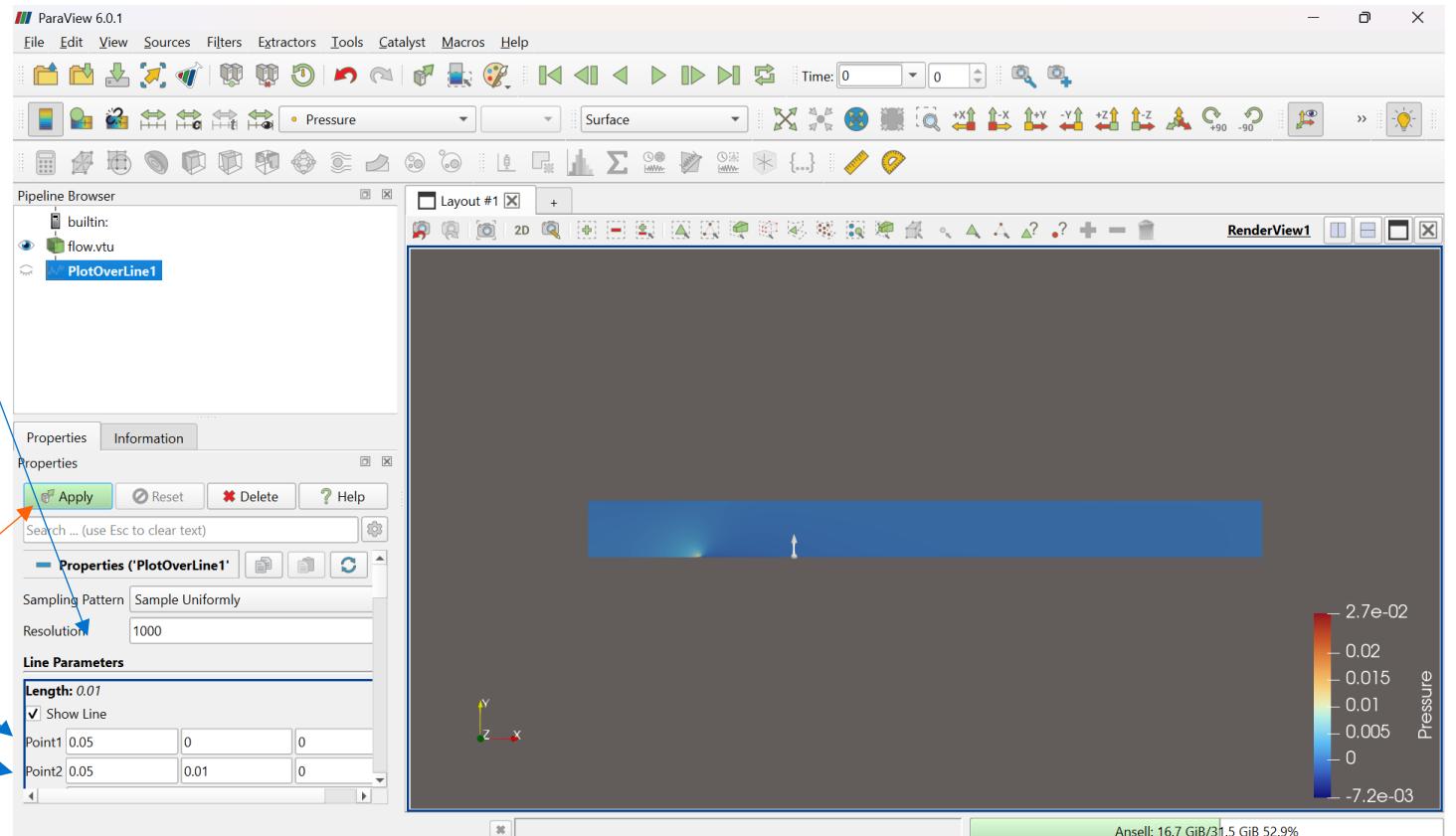


Extracting Data

- 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 $x = 0.2, y = 0.01, z = 0$

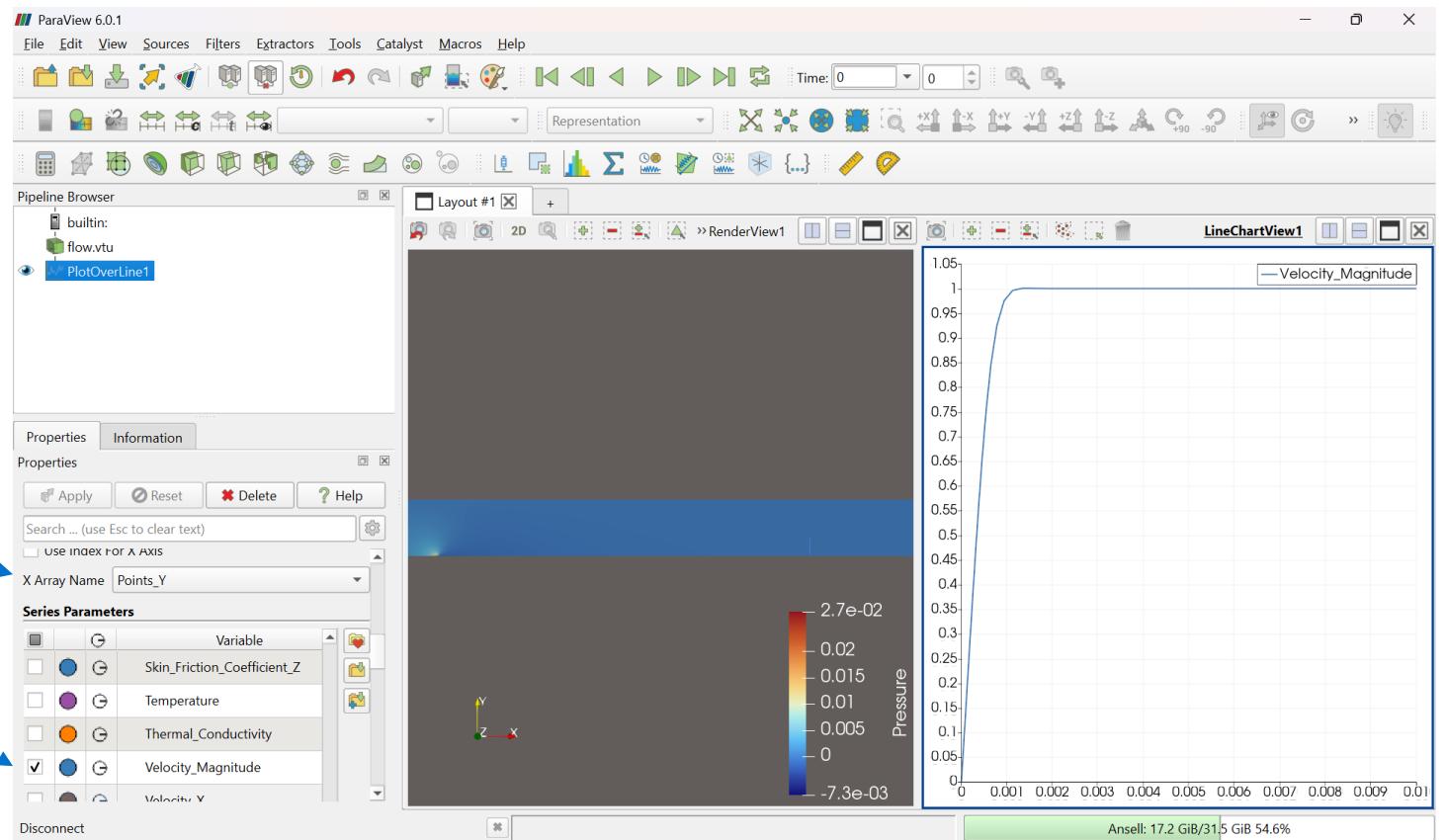
- Click “Apply”





Extracting Data

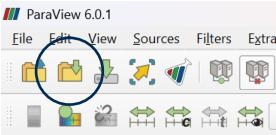
- A line chart will appear on the right side of the screen
 - Scroll down in the “Properties” tab on the left side of the screen
 - To plot *just* the velocity magnitude
 - Change “X Array Name” to “Points_Y”
 - Uncheck all variables except for “Velocity_Magnitude”
 - You can now see (a sideways plot 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



A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	
1	Density	Heat_Cap	Heat_Flux	Laminar_V	Pressure	(Skin_Fricti	Skin_Fricti	Skin_Fricti	Temperatu	Thermal_C	Velocity:0	Velocity:1	Velocity:2	Y_Plus	vtkValidPo	arc_length	Points:0	Points:1	
2	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0.000714	0	0	1	2.03E-05	0	0	0	1	0	0.2	0	
3	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.015237	1.90E-07	0	0	1	1.00E-05	0.2	1.00E-05
4	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.030474	7.53E-07	0	0	1	2.00E-05	0.2	2.00E-05
5	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.045711	1.70E-06	0	0	1	3.00E-05	0.2	3.00E-05
6	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.060947	3.02E-06	0	0	1	4.00E-05	0.2	4.00E-05
7	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.076181	4.70E-06	0	0	1	5.00E-05	0.2	5.00E-05
8	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.091412	6.77E-06	0	0	1	6.00E-05	0.2	6.00E-05
9	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.10664	9.25E-06	0	0	1	7.00E-05	0.2	7.00E-05
10	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.12186	1.21E-05	0	0	1	8.00E-05	0.2	8.00E-05
11	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.13707	1.53E-05	0	0	1	9.00E-05	0.2	9.00E-05
12	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.15228	1.89E-05	0	0	1	0.0001	0.2	0.0001
13	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.16747	2.29E-05	0	0	1	0.00011	0.2	0.00011
14	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.18265	2.72E-05	0	0	1	0.00012	0.2	0.00012
15	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.19781	3.19E-05	0	0	1	0.00013	0.2	0.00013
16	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.21295	3.70E-05	0	0	1	0.00014	0.2	0.00014
17	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.22808	4.22E-05	0	0	1	0.00015	0.2	0.00015
18	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.24315	4.83E-05	0	0	1	0.00016	0.2	0.00016
19	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.25821	5.45E-05	0	0	1	0.00017	0.2	0.00017
20	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.27328	6.06E-05	0	0	1	0.00018	0.2	0.00018
21	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.28823	6.78E-05	0	0	1	0.00019	0.2	0.00019
22	1	62.5	0	2.34E-07	-7.50E-05	-0.00015	0	0	0	1	2.03E-05	0.30316	7.52E-05	0	0	1	0.0002	0.2	0.0002