

## AE 462

### Lab 1

Lab Session: Monday, February 2, 2026

Due: Monday, February 9, 2026, 9am CST

This laboratory assignment will closely follow the laminar flat-plate boundary layer example provided in Tutorial 1. However, we will extend the length of the plate, extract several boundary-layer velocity profiles, and compare these outcomes to the known Blasius boundary layer solution. The items required for your assignment submission are given here, alongside helpful instructions to completing this lab.

### Report Instructions:

Using CFD-generated data from this exercise, provide a response to the following:

- 1) Co-plot the velocity profile of the laminar boundary layer corresponding to  $x = 0.1, 0.2, 0.3, 0.4,$  and  $0.5$  m. Provide your  $u/V_\infty$  values on the  $x$  axis and the corresponding  $y$  position (in mm) on the  $y$  axis of the plot. Be sure to label your axes, use a reasonably distinct color/symbol scheme for the velocity profiles, and provide a legend indicating which plots correspond to each  $x$  position.
- 2) At these same  $x$  positions, determine the boundary-layer thickness ( $\delta$ ), assuming that the edge of the boundary layer is produced when  $u = 0.99V_\infty$ . Report the thickness in mm for these five locations.
- 3) Calculate the local Reynolds number ( $Re_x$ ) corresponding to these five  $x$  locations. Recall that the freestream conditions defined in the input file correspond to:  $V_\infty = 30$  m/s,  $\rho = 1.2886$  kg/m<sup>3</sup>, and  $\mu = 1.716 \times 10^{-5}$  N-s/m<sup>2</sup>.
- 4) The Blasius solution for a zero pressure gradient, laminar boundary layer indicates that:

$$\delta = \frac{5.0x}{\sqrt{Re_x}}$$

Report the values of  $\delta$  calculated using this method and provide a plot comparing the theoretical boundary-layer thickness, on the  $y$  axis, with respect to Reynolds number, on the  $x$  axis. Include the five boundary-layer thickness values determined in part 2) (provide as filled symbols). As with 1), label your axes, indicate units, and provide a legend.

- 5) For each  $x$  position, calculate and report the proportional (percent) difference between the CFD-determined boundary layer thickness and the theoretical result from Blasius. In 2-3 sentences, indicate if you notice any trends in increases/decreases in discrepancies with streamwise distance.

In your submission, provide the previous items in a single PDF file. Upload this file, along with your .geo and .cfg files. Upload any scripts, spreadsheets, or other work used during data analysis.

On the EWS machines in the Talbot Aero Lab and Engineering Hall, you can find the SU2\_CFD module in the **C:\SU2\v8.4.0-mpi\bin\** directory. However, the C:\ drive on these computers is *not* shared across the UIUC network, and any files placed on this drive will *only* remain this specific computer. For this lab, and future use of the EWS machines, I encourage you to create folders for your work on the U:\ drive. For this lab, you might want to create a “**U:\AE 462**” folder, and a sub-folder specific to this lab, “**U:\AE 462\Lab 1**”.

## Lab Instructions:

### *Create a mesh*

Either build your own structured mesh following the instructions from Tutorial 1 or use the “flatplate.geo” Gmsh file from the tutorial as a starting point to represent the flat-plate flow domain. Change the streamwise length of the plate from 0.3 m to 0.5 m and adjust the mesh definition (see “Transfinite Curve” input parameters below). If you create your own mesh, you can define this 0.5-m long plate when entering the downstream points of the domain, as well as the modified node distribution when entering the number and progression of the transfinite curve definitions. Alternatively, if you are starting with the provided “flatplate.geo” file, you can open the Gmsh script file and implement modifications directly in the .geo file. Note that you can use any standard text editor, such as Notepad, to make changes to flatplate.geo:

```
Point(4) = {0.5, 0.03, 0, 1.0};
```

```
//+
```

```
Point(5) = {0.5, 0, 0, 1.0};
```

```
Transfinite Curve {1, 3, 2} = 60 Using Progression 1.2;
```

```
//+
```

```
Transfinite Curve {5, 4} = 20 Using Progression 1.1;
```

```
//+
```

```
Transfinite Curve {6, 7} = 60 Using Progression 1.05;
```

If you created your own mesh and defined the points in a different order than the tutorial, note that Point(4) corresponds to the upper-right corner of the flow domain, and Point(5) corresponds to the lower-right corner of the flow domain. Your specific point numbering might be slightly different, as the point numbers correspond to the order in which they are entered. Similarly, the mesh density and progression changes to Transfinite Curve {6, 7} correspond to the distribution of nodes across the horizontal, downstream borders of the domain (upper far-field and plate surface), and your transfinite curves for this same position in the domain might be a different number if defined in a different order from that of the tutorial.

Following these modifications, mesh your 2D flow domain. If you have the .geo file open in Gmsh and make any changes via a text editor, be sure to save changes in the .geo file and click “Reload script” in the Gmsh interface. If you generated a new .geo file, save it for later submission. Then, click on Modules → Mesh → 2D to generate the mesh. Export the mesh as “**flatplate.su2**” within your U:\ directory (e.g., U:\AE 462\Lab 1) following the export instructions from Tutorial 1. Note that the “**.su2**” extension must specifically be included in the file name. Simply changing the “Save as type” to the .su2 option in the export menu will not automatically apply this file type extension. Download the “**flatplate.cfg**” file on the Canvas page and place it within your working folder (U:\AE 462\Lab 1).

### *Run the Solver*

Next, we will run SU2 based on this input file. Begin by opening the “flatplate.cfg” file in a text editor (e.g., Notepad). Change the inlet velocity from 70 m/s to 30 m/s, by changing Line 31 to:

```
INC_VELOCITY_INIT= ( 30, 0.0, 0.0 )
```

and change Line 107 to:

```
MARKER_INLET= ( inlet, 288.15, 30, 1.0, 0.0, 0.0 )
```

Save the file and close the text editor. On an EWS machine, start by opening a command window. You can do this by going to the start search bar and enter “**cmd**” then hit enter. On the command line, you will first want to navigate to your working directory. To do this, type (including quotations):

```
cd “U:\AE 462\Lab 1”
```

and press enter. You will now see “U:\AE 462\Lab 1>” on the bottom line of the terminal window. To run SU2, type (including quotations):

```
“C:\SU2\v8.4.0-mpi\bin\SU2_CFD” flatplate.cfg
```

The code will now begin to run, and you will see the convergence progress with each iteration. Note that, since we are including a greater number of mesh elements than the tutorial, it will take a bit longer to converge. However, you should reach a converged solution within < 400 iterations.

### *Post-Processing the Data*

Your flow solution is now saved in the “U:\AE 462\Lab 1\flow.vtu” file. Open Paraview and load this file into the pipeline browser. Using the method demonstrated in Tutorial 1, extract boundary-layer velocity profiles across five vertical lines corresponding to  $x = 0.1, 0.2, 0.3, 0.4,$  and  $0.5$  m, with a height of 5 mm. For example, the  $x = 0.1$  m line, designation can be performed by entering the following points and selecting “Apply”:

Point 1:  $x = 0.1, y = 0, z = 0$

Point 2:  $x = 0.1, y = 0.005, z = 0$

If saving as a .csv file like was done in Tutorial 1, the  $u/V_\infty$  velocity is stored under the “Velocity:0” variable and the corresponding  $y$  position (in m) for each point is stored under the “Points:1” variable.

You now have the CFD-generated data needed to respond to the items required for the submission.