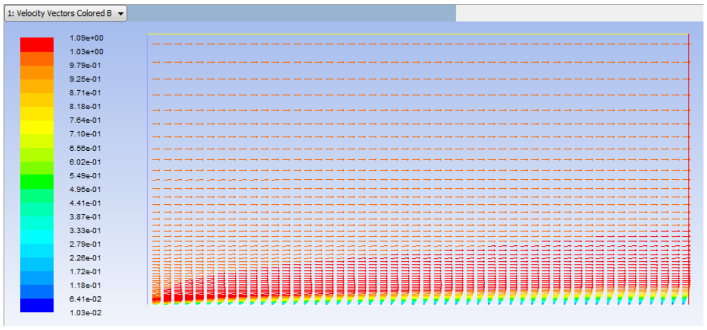
FLUENT - Flat Plate Boundary Layer - FLUENT Post Processing

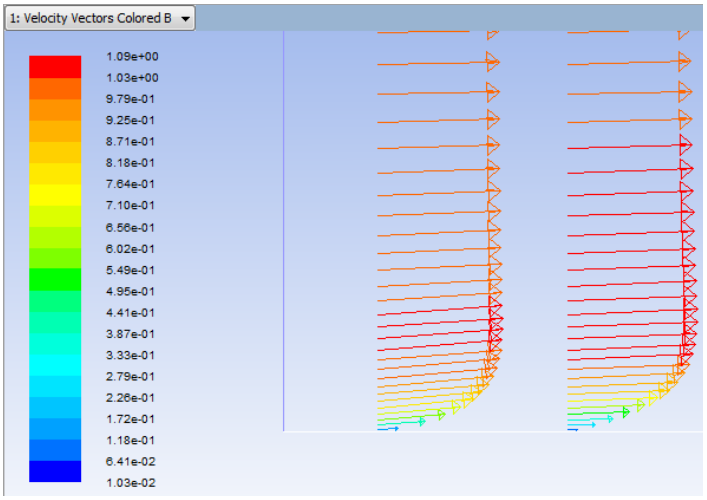
Step 6: Results

Velocity Vectors

One can plot vectors in the entire domain, or on selected surfaces. Here, the vectors will be plotted for the entire domain. First, click on ***Graphics & Animations*** . Next, double click on ***Vectors*** which is located under ***Graphics***. Then, click on ***Display*** in the ***Vectors*** menu. You should obtain, the following output.

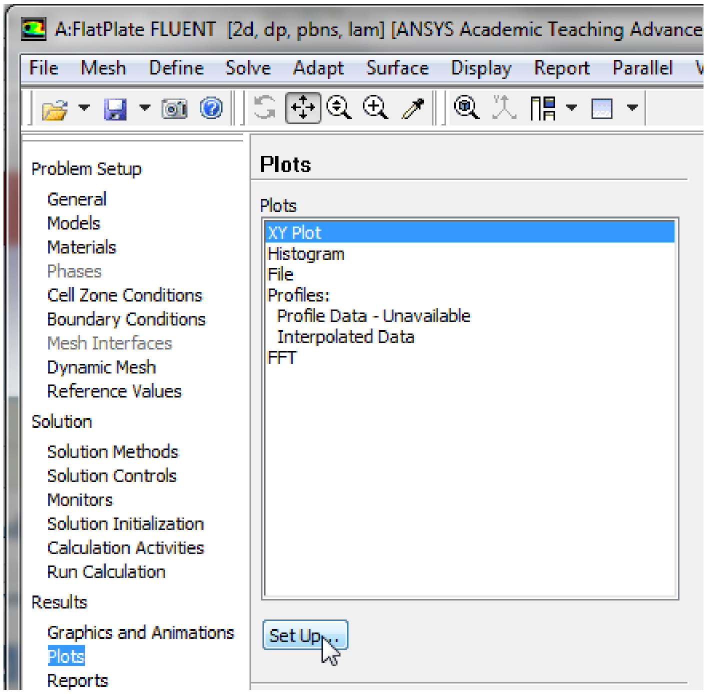


You can use the wheel button of the mouse to zoom into the region that closely surrounds the plate, to get a better view of the boundary layer velocities.

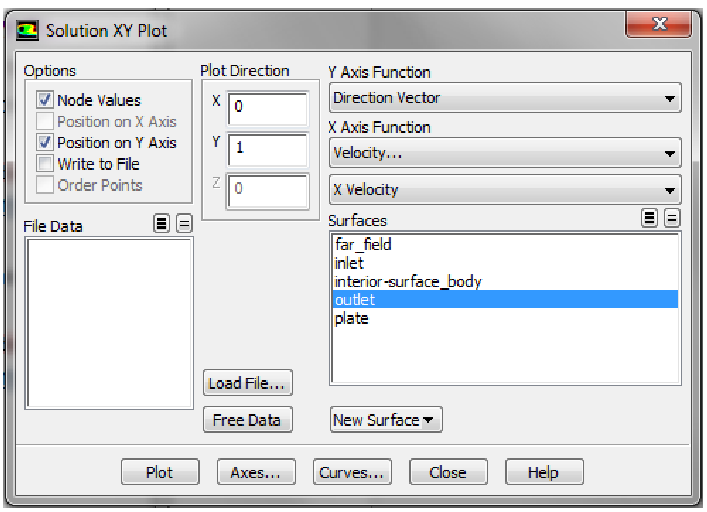


Outlet Velocity Profile

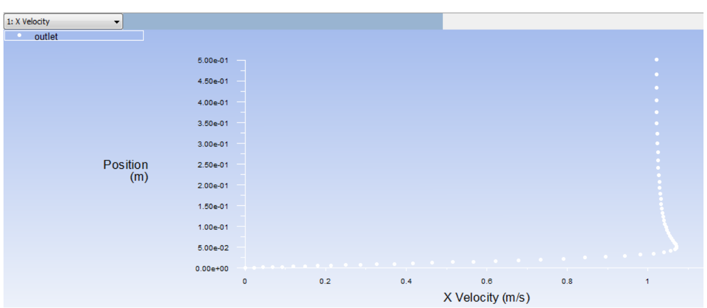
In this section we will first plot the variation of the x component of the velocity along the outlet. Then we will plot the Blasius solution to see how the numerical solution compares. In order to start the process ***(Click) Results > Plots > XY Plot... > Set Up..*** as shown below.



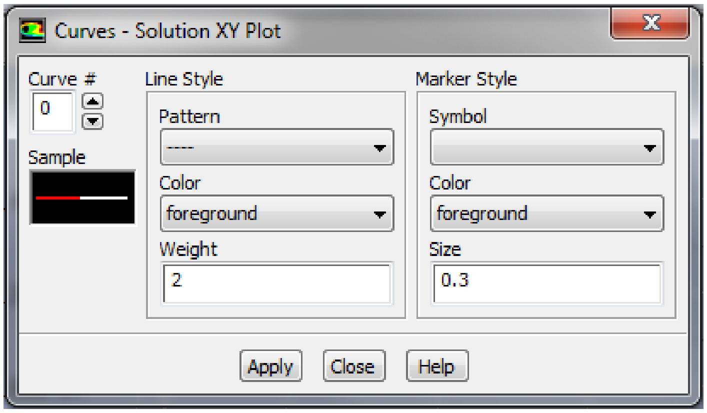
In the ***Solution XY Plot*** menu make sure that ***Position on Y Axis*** is selected , and ***X*** is set to `0` and ***Y*** is set to `1`. This tells FLUENT to plot the y-coordinate value on the ordinate of the graph. Next, select ***Velocity...*** for the first box underneath ***X Axis Function*** and select ***X Velocity*** for the second box. Please note that ***X Axis Function*** and ***Y Axis Function*** describe the *x* and *y* axes of the *graph*, which should not be confused with the *x* and *y* directions of the geometry. Finally, select ***outlet*** under ***Surfaces*** since we are plotting the x component of the velocity along the *outlet*. This finishes setting up the plotting parameters. Your ***Solution XY Plot*** menu should look exactly the same as the following image.



Now, click ***Plot***. The plot of the x component of the velocity as a function of distance along the *outlet* now appears.

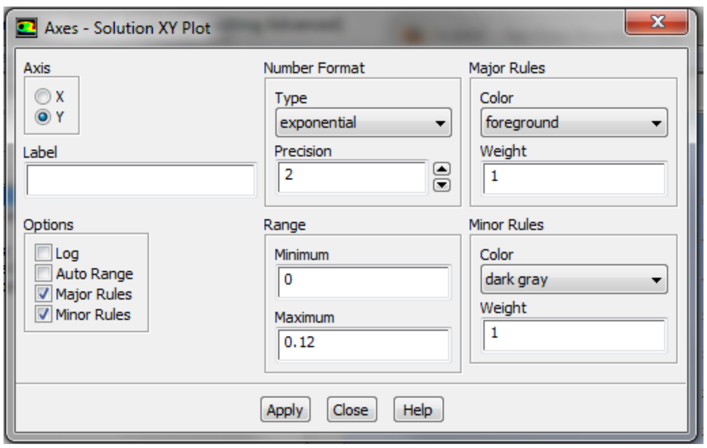


In order to increase the legibility of the graph, we will plot the data as a line rather than points. To turn on the line feature, click on ***Curves...*** in the ***Solution XY Plot*** menu. Then, set ***Pattern*** to ***----***, set the ***Weight*** to 2 and select nothing for ***Symbol***, as shown below.

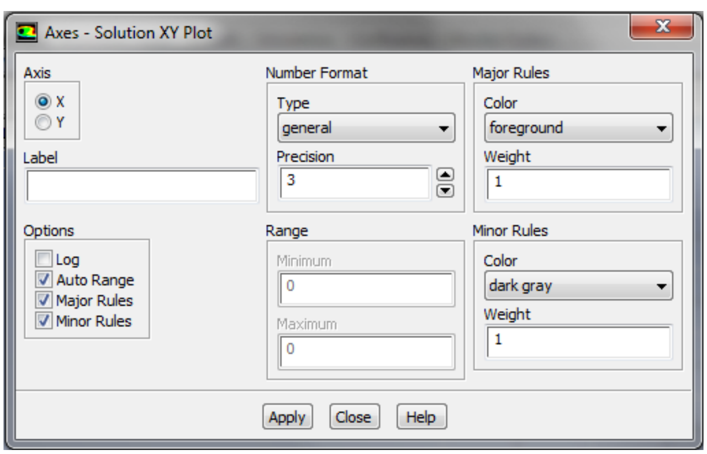


Next, click ***Apply*** in the ***Curves - Solution XY Plot*** menu. Next, close the ***Curves - Solution XY Plot*** menu.

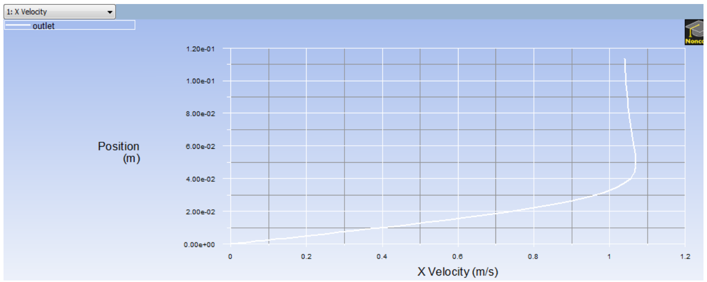
Now, the range of the y axis will be truncated, as we are not interested in far field velocity. Furthermore, the grid lines will be turned on. In order to implement these two changes. First click ***Axes*** in the ***Solution XY Plot*** menu. Next, select ***Y*** for ***Axis***, deselect ***Auto Range***, select ***Major Rules***, select ***Minor Rules***. Then, set ***Minimum*** to 0 and set ***Maximum*** to 0.12. Your ***Axes - Solution XY Plot*** menu, should look exactly like the image below.



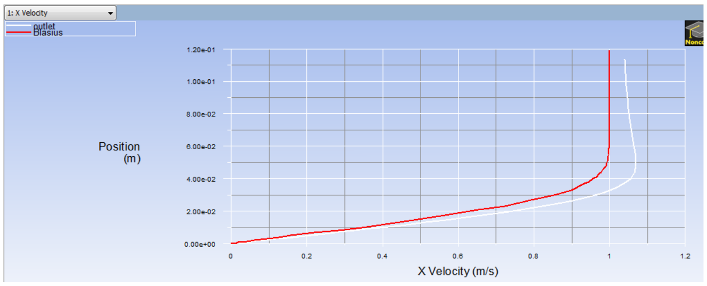
Then, click ***Apply*** in the ***Axes - Solution XY Plot*** menu. Now, select ***X*** for ***Axis*** and select ***Major Rules*** and ***Minor Rules***, as shown below.



Next, click ***Apply*** in the ***Axes - Solution XY Plot*** menu. Close the ***Axes - Solution XY Plot*** menu. Now, click ***Plot*** in the ***Solution XY Plot*** menu. You should obtain the following output.



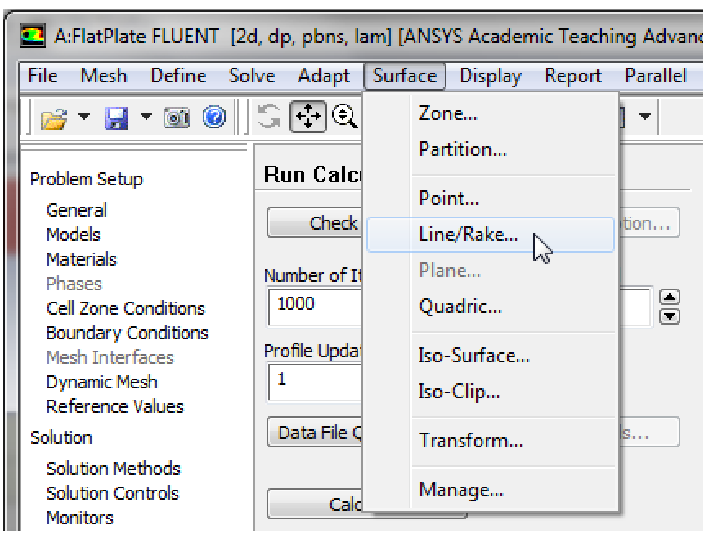
It is of interest to compare the numerical velocity profile to the velocity profile obtained from the Blasius solution. In order to plot the theoretical results, first click here to download the necessary file. Save the file to your working directory. Next, go to the ***Solution XY Plot*** menu and click ***Load File...*** and select the file that you just downloaded, *BlasiusU.xy*. Lastly, click ***Plot*** in the ***Solution XY Plot*** menu. You should then obtain the following figure.



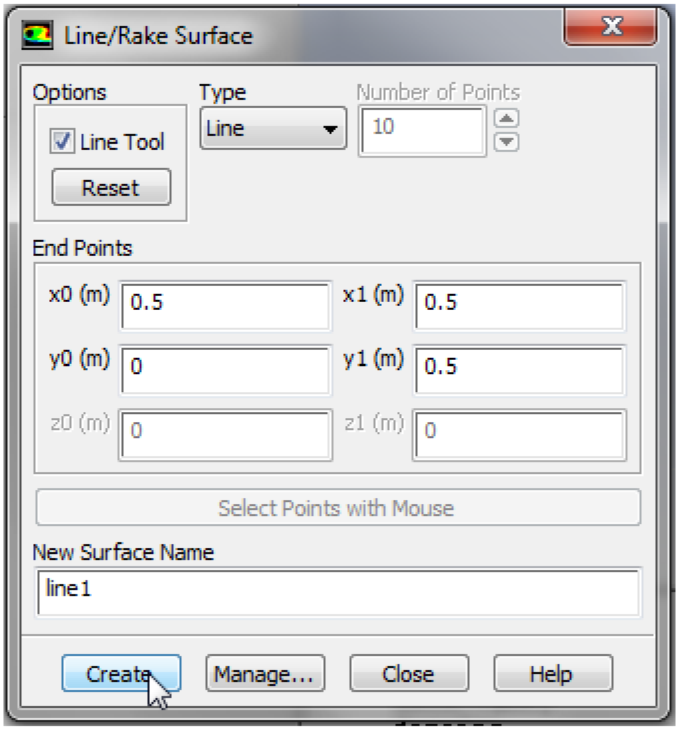
Lastly, select ***Write to File*** located under ***Options*** in the ***Solution XY Plot*** menu. Then, click ***Write...***. When prompted for a filename, enter *XVelOutlet.xy* and save the file in your working directory.

Mid-Section Velocity Profile

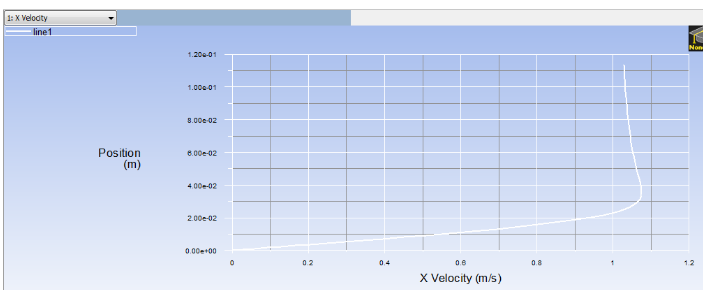
Here, we will plot the variation of the x component of the velocity along a vertical line in the middle of the geometry. In order to create the profile, we must first create a vertical line at x=0.5m, using the ***Line/Rake*** tool. First, ***(Click) Surface < Line/Rake*** as shown in the following image.



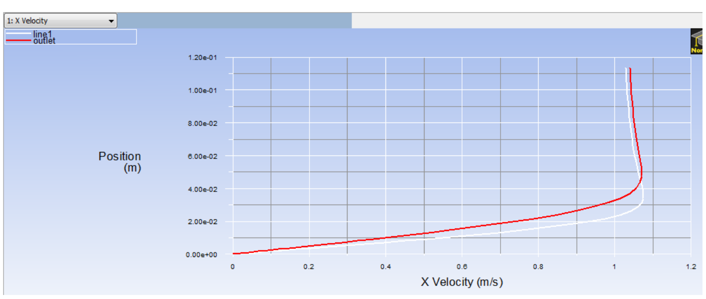
We'll create a straight vertical line from (x0,y0)=(0.5,0) to (x1,y1)=(0.5,0.5). Select ***Line Tool*** under ***Options***. Enter ***x0***=`0.5`, ***y0***=`0`,***x1***=`0.5`, ***y1***=`0.5`. Enter `line1` under ***New Surface Name***. Your ***Line/Rake Surface*** menu should look exactly like the following image.



Next, click ***Create***. Now, that the vertical line has been created we can proceed to the plotting. Click on ***Plots***, then double click ***XY Plot*** to open the ***Solution XY Plot*** menu. In the ***Solution XY Plot*** menu, use the settings that were used from the section above, except select *line1* under ***Surfaces*** and deselect any other geometry sections. Make sure that ***Write to File*** is not selected, then click ***Plot***. You should obtain the following output.



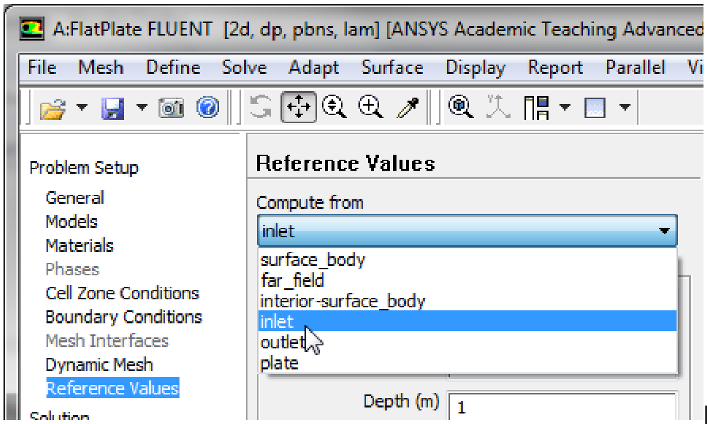
Then, return to the ***Solution XY Plot*** menu and select both *line1* and *outlet* under ***Surfaces***. Next, click ***Plot*** and you should obtain the following figure.



Once again, return to the ***Solution XY Plot*** menu, select ***Write to File***, then click ***Write...***. When prompted for a filename, enter *XVelProfs.xy* and save the file in your working directory.

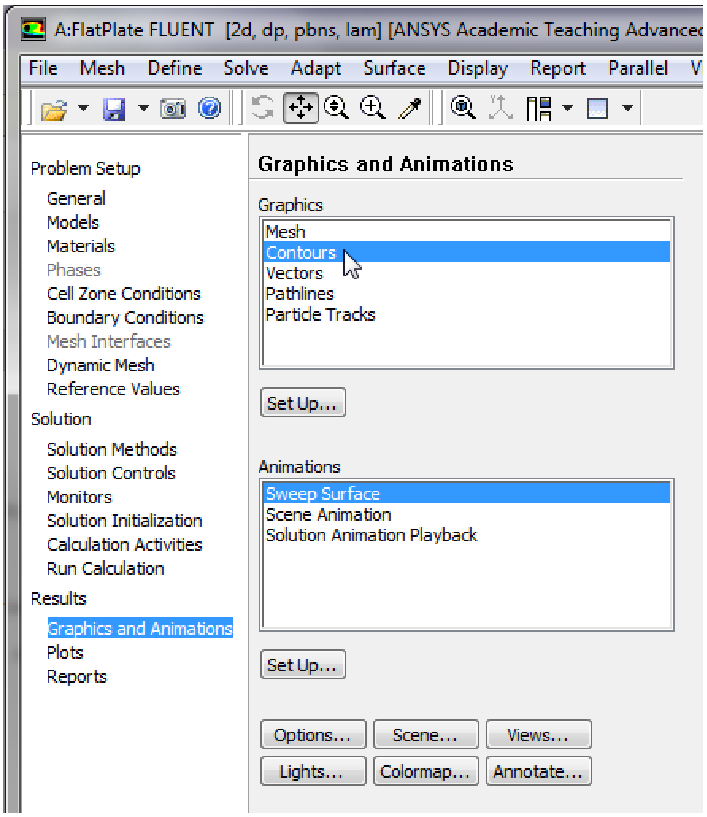
Pressure Coefficients

In this section we will create contour plots for the pressure coefficients. Before we begin, we must first set the reference values for velocity. In order to do so, first click on ***Reference Values*** then set ***Compute from*** to *inlet*, as shown below.

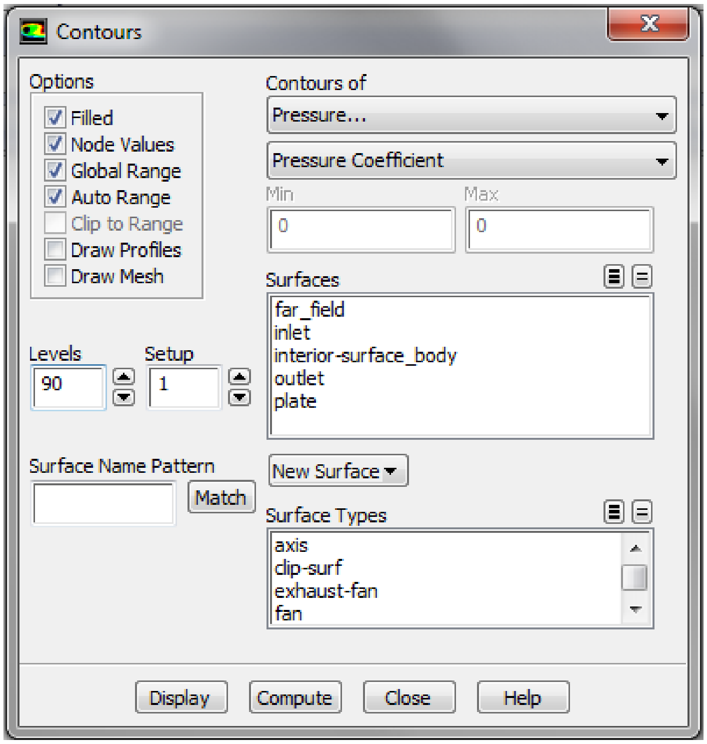


|^CompInlet_Full.png]

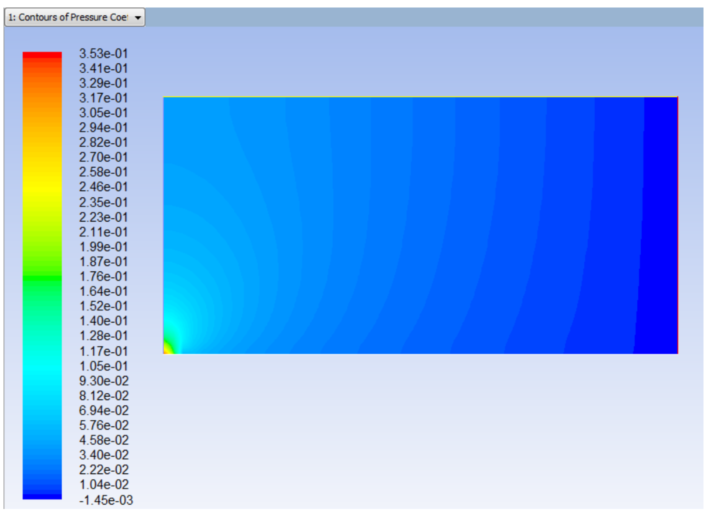
Next, click on ***Graphics and Animations***, then double click on ***Contours***, as shown below.



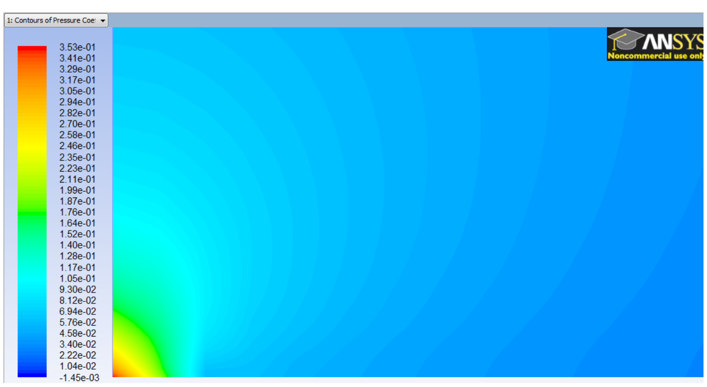
In the ***Contours*** menu, set ***Contours of*** to ***Pressure...*** and set the box below to ***Pressure Coefficient***. Next, select ***Filled*** and set ***Levels*** to 90. Your ***Con tours*** menu should look exactly like the following image.



Lastly, click ***Display*** in the ***Contours*** menu to generate the contour plot. You should obtain the following output.

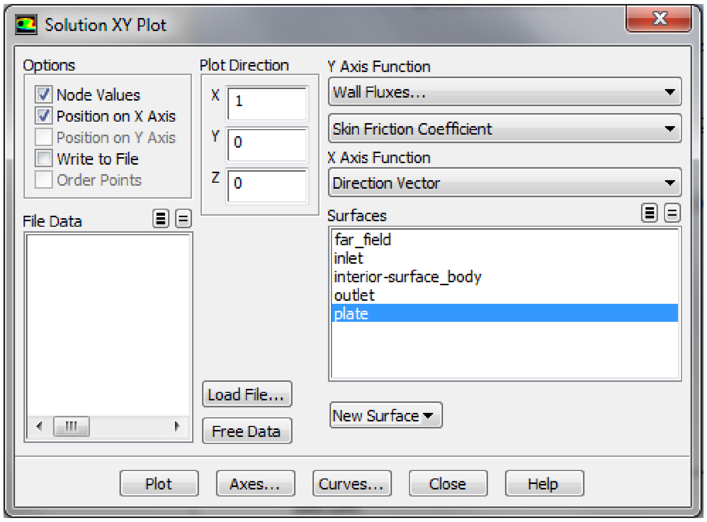


You can then zoom in to the leading edge of the plate with the wheel mouse button as shown below.

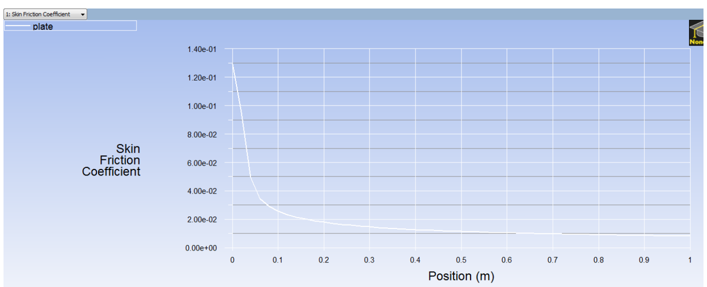


Skin Friction Coefficient

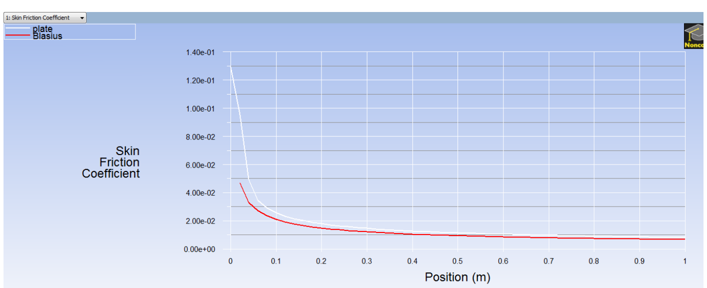
Here, the skin friction coefficient will be plotted as a function of distance along the plate. First, click on ***Plots***, then double click on ***XY Plot***. In the ***Solution XY Plot*** menu deselect ***Write to File***, select ***Position on X Axis***, set ***X*** to `1` and set ***Y*** to `0`. Then, set the box located underneath ***Y Axis Function*** to ***Wall Fluxes*** and set the box below to ***Skin Friction Coefficient***. Next, select ***plate*** under ***Surfaces*** and deselect any other geometry features. At this point your ***Solution XY Plot*** menu should look the same as the following image.



Make sure that for both the x and y axes, that ***Auto Range*** is selected. Remember, that you must click ***Apply*** to implement the changes you make. Then, click ***Plot*** in the ***Solution XY Plot*** menu and you should obtain the following output.



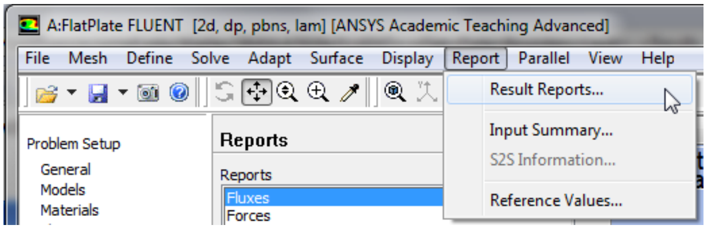
It is of interest to compare the numerical skin friction coefficient profile to the skin friction coefficient profile obtained from the Blasius solution. In order to plot the theoretical results, first click here to download the necessary file. Save the file to your working directory. Next, go to the ***Solution XY Plot*** menu and click ***Load File...*** and select the file that you just downloaded, *BlasiusSkin.xy*. Lastly, click ***Plot*** in the ***Solution XY Plot*** menu. You should then obtain the following figure.



Lastly, select ***Write to File*** located under ***Options*** in the ***Solution XY Plot*** menu. Then, click ***Write...***. When prompted for a filename, enter *SkinFriction.xy* and save the file in your working directory.

Drag

Now, we will obtain the drag on the plate. First, click on ***Report*** then click on ***Result Reports...***, as shown in the following image.



Next, double click on ***Forces*** and click ***Print*** in the ***Force Reports*** menu. You should then obtain the following output in the command pane.

```
Forces
                         Forces (n)
Zone                     Pressure                                   Viscous                                    Total                                      Coefficients
plate                    (0 -0.01319054 0)                          (0.008376893 0.00018661246 0)              (0.008376893 -0.013003927 0)               Pressure
                                                                                                                                                          (0 -0.02638108 0)
------------------------ ------------------------------------------ ------------------------------------------ ------------------------------------------ -----------------
Net                      (0 -0.01319054 0)                          (0.008376893 0.00018661246 0)              (0.008376893 -0.013003927 0)               (0 -0.02638108 0)

Forces - Direction Vector (1 0 0)
                         Forces (n)                                  Coefficients
Zone                     Pressure      Viscous       Total          Pressure      Viscous       Total
plate                    0             0.008376893   0.008376893    0             0.016753786   0.016753786
------------------------ ------------- ------------- -------------- ------------- ------------- -------------
Net                      0             0.008376893   0.008376893    0             0.016753786   0.016753786
```

As one can see from the data above, the plate experiences a drag of approximately 0.008377 Newtons. Furthermore, the data states that the drag coefficient is approximately 0.01675. The drag coefficient is defined by the following equation.

$$C_d = \frac{F_d}{0.5\,\rho\,U^2 A}$$

In the case here, the density, velocity and area all have values of 1. Thus, the equation above reduces to the following equation.

$$\boxed{C_d = 2F_d}$$

The results from ANSYS FLUENT agree with the theory here since the drag coefficient is approximately twice the value of the drag.

Now, save your work in the FLUENT window, then close the FLUENT window.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules