Old (4/1/2020) Flat Plate Boundary Layer - Numerical Results

Author: Rajesh Bhaskaran, Cornell University

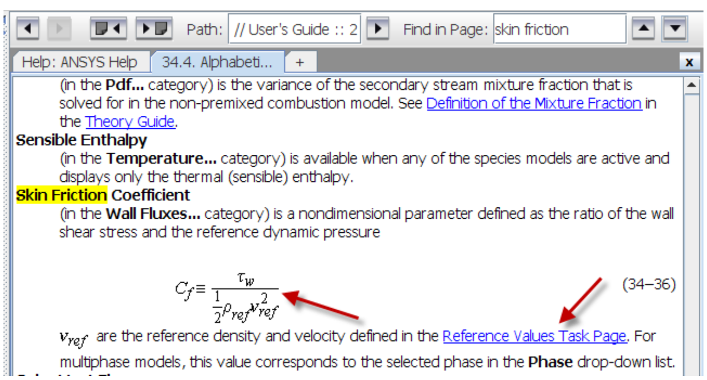
Numerical Results

Post Processing

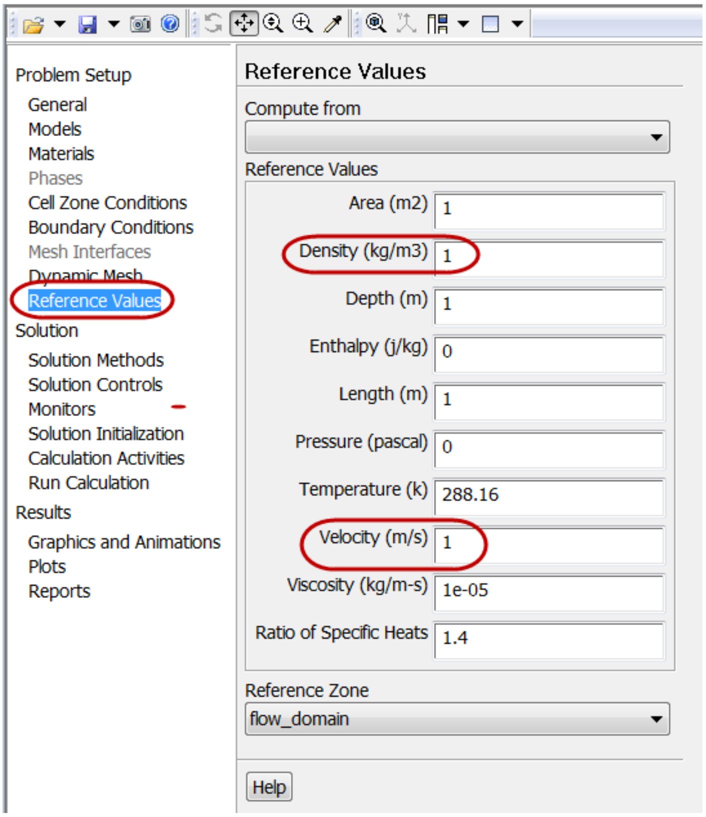
We will use CFD-Post as the primary post processing GUI. Steps for post processing in FLUENT can be found here.

Exporting Skin Friction from FLUENT into the Post-processor (CFD-Post)

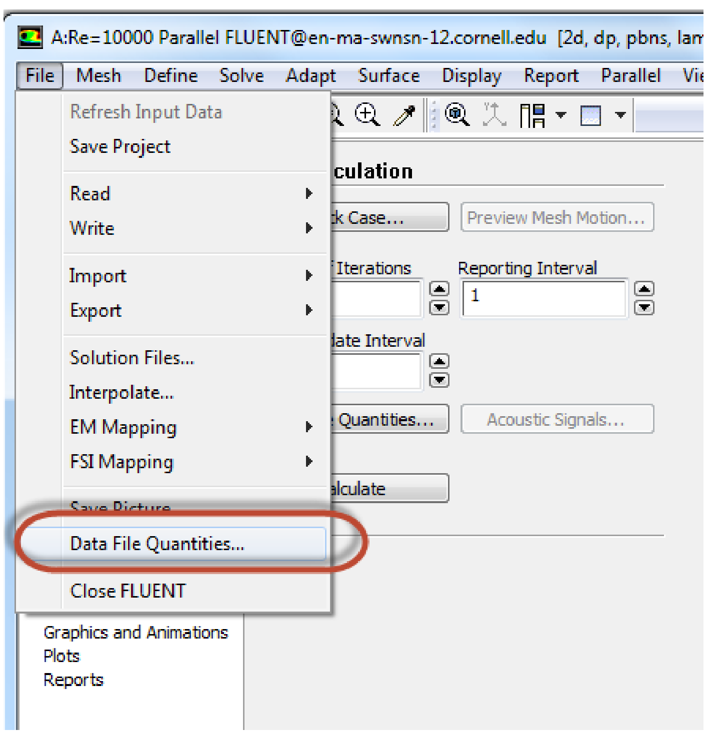
FLUENT calculates the skin friction coefficient as follows (see section 34.4 of the FLUENT User's Guide which is accessible from the "help" button on the FLUENT interface. In ANSYS 15.0, see section 33.4 Alphabetical Listing of Field Variables and Their Definitions).



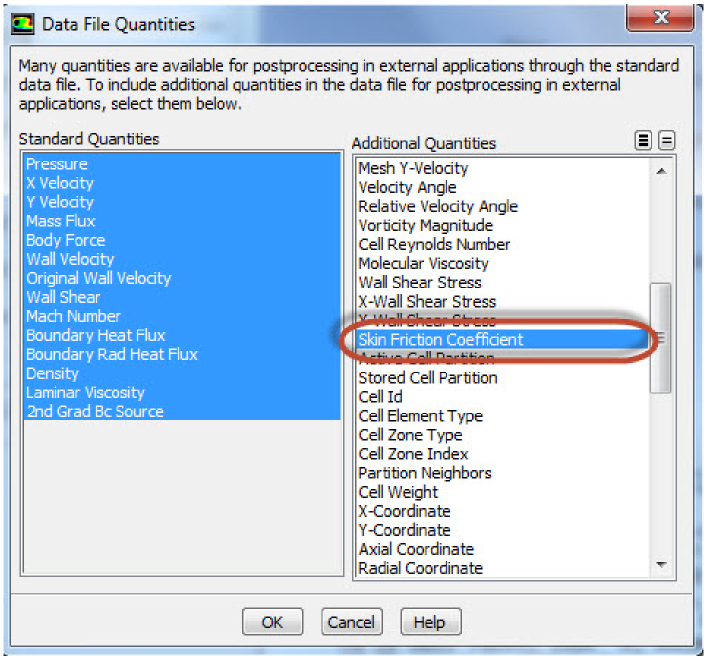
The wall shear in the numerator of this expression is calculated from the gradient of the velocity field. The "reference" density and velocity used in the denominator are specified through the reference values panel.



Basic entities such as velocity and pressure are automatically exported from FLUENT into CFD-Post. Others such as skin friction coefficient needs to be manually exported. To manually export the skin friction coefficient, in FLUENT, click on File > Data File Quantities:

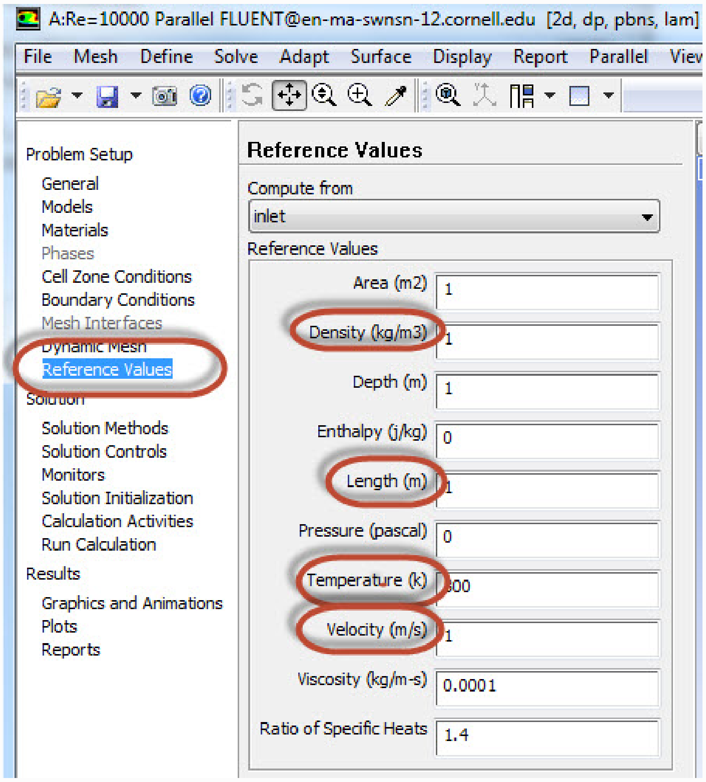


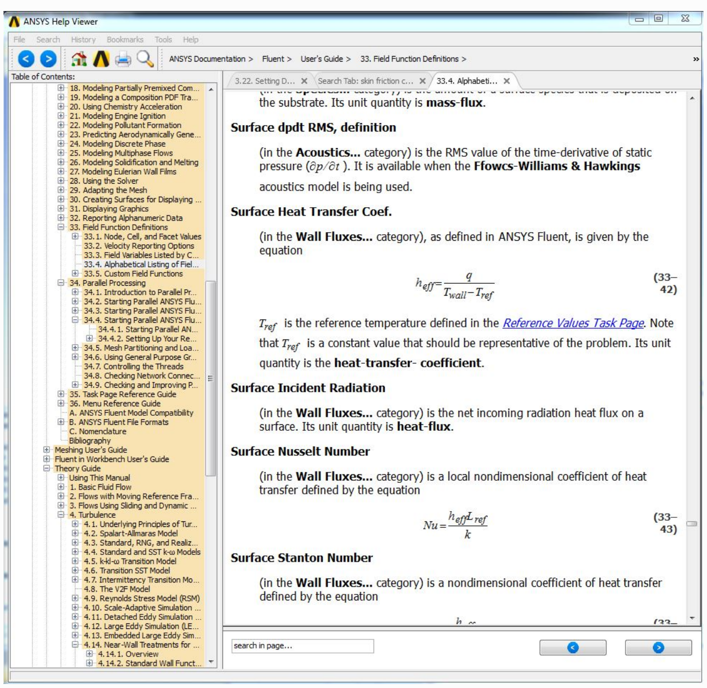
In Data File Quantities dialogue, select skin friction coefficient and click on OK.


Data File Quantities
Many quantities are available for postprocessing in external applications through the standard data file. To include additional quantities in the data file for postprocessing in external applications, select them below.
Standard Quantities
Pressure
X Velocity
Y Velocity
Mass Flux
Body Force
Wall Velocity
Original Wall Velocity
Wall Shear
Mach Number
Boundary Heat Flux
Boundary Rad Heat Flux
Density
Laminar Viscosity
2nd Grad Bc Source
Additional Quantities
Mesh Y-Velocity
Velocity Angle
Relative Velocity Angle
Vorticity Magnitude
Cell Reynolds Number
Molecular Viscosity
Wall Shear Stress
X-Wall Shear Stress
Y-Wall Shear Stress
Skin Friction Coefficient
Active Cell Partition
Stored Cell Partition
Cell Id
Cell Element Type
Cell Zone Type
Cell Zone Index
Partition Neighbors
Cell Weight
X-Coordinate
Y-Coordinate
Axial Coordinate
Radial Coordinate
OK
Cancel
Help

Note

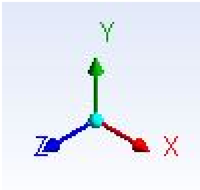
Students in MAE 6510 will need to select the Surface Heat Transfer Coefficient (NOT the Surface Nusselt Number) in addition to the Skin Friction Coefficient for your HW assignment, in order to calculate the local Nusselt number. You will need to specify the appropriate reference density, velocity, length, and temperature in FLUENT before you export the skin friction coefficient and heat transfer coefficient into the post-processor. One can use the heat transfer coefficient to then calculate the local Nusselt number in post-processing. If you select to export the Surface Nusselt Number directly from Data File Quantities, you will be exporting a Nusselt Number that is calculated at one x value only (the reference length of 1m in our case). It is always good practice to look up how ANSYS calculates quantities like these each time you choose to export.



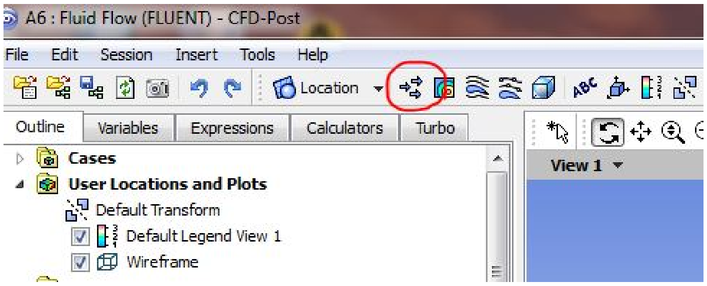


Double click on Results from the Workbench Window to launch CFD-Post.

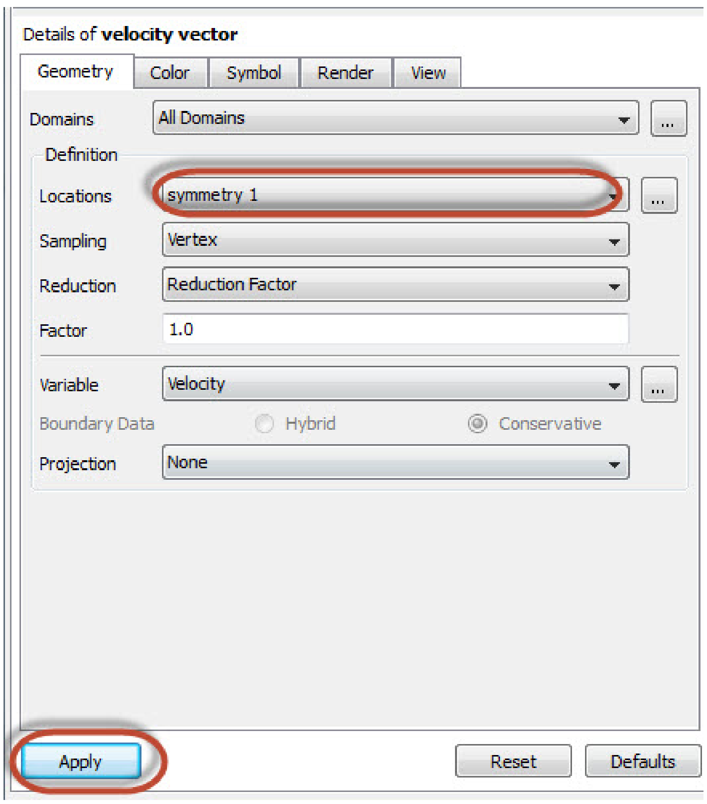
Velocity Vectors

Click on the z-axis, 

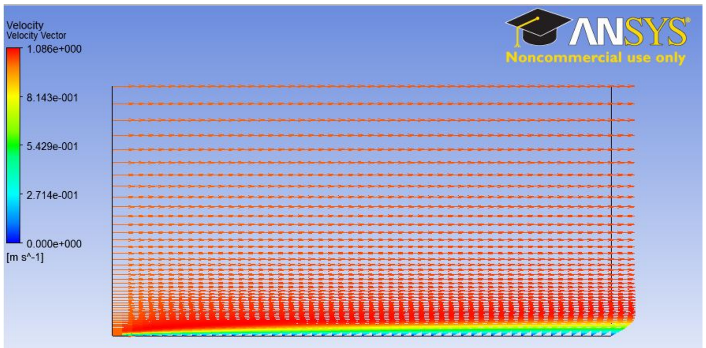
to view the XY plane. Click on the vector icon to insert a vector plot. Name it Velocity Vector.



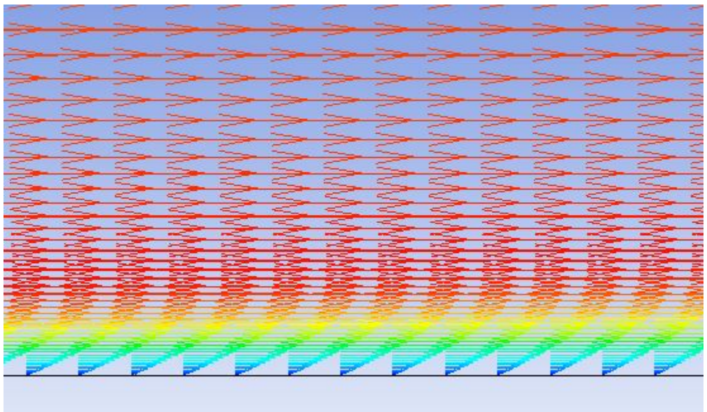
A panel named "Details of Velocity Vector" will appear right below the outline window. Set the ***Locations*** to ***symmetry 1***. Click on ***Apply*** to display the velocity vectors.



The velocity vectors will be displayed in the view window.

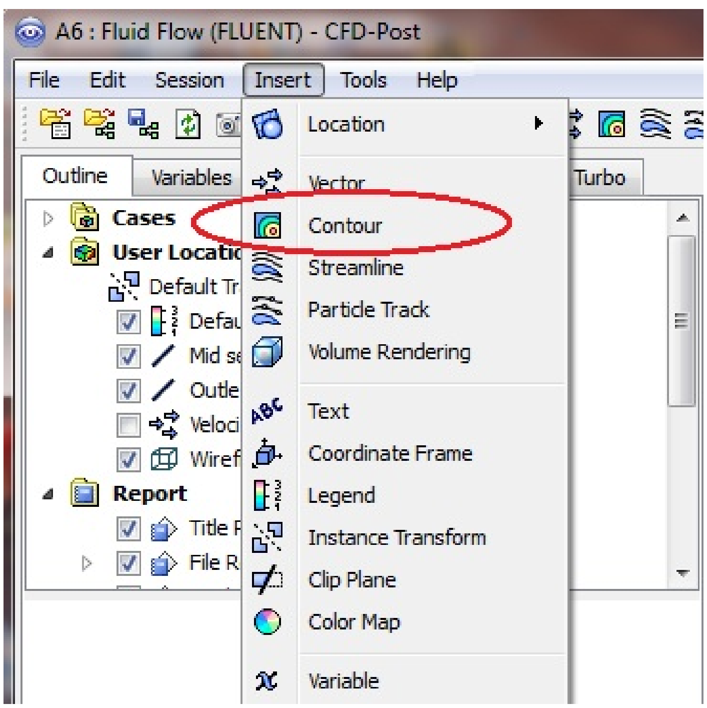


You can use the wheel button of the mouse to zoom into the region that closely surrounds the plate, to get a better view of the boundary layer velocities:

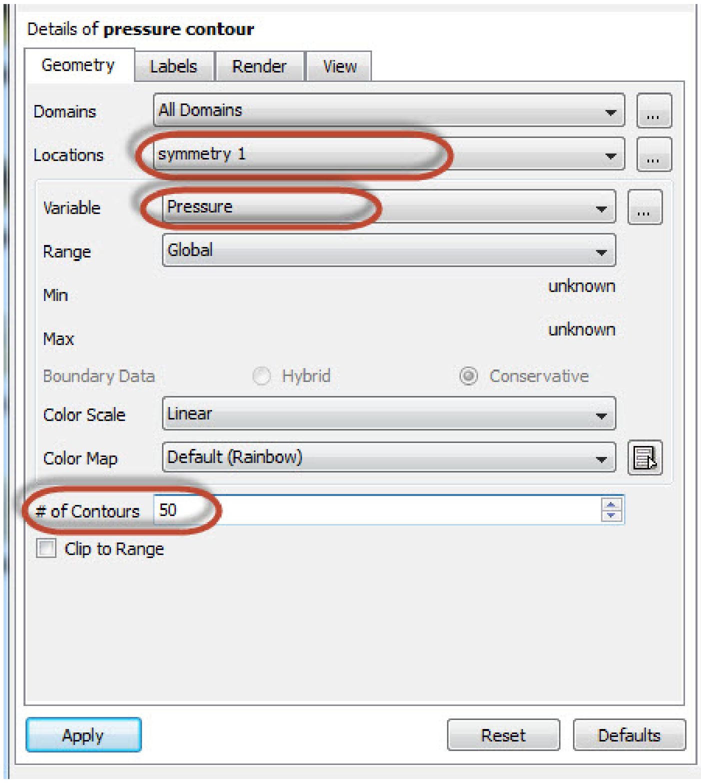


Pressure Contour

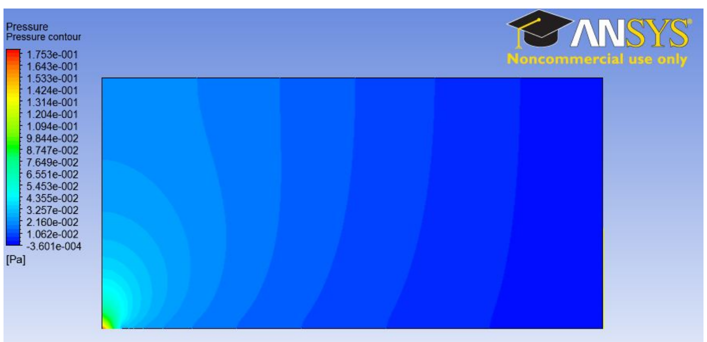
Insert > Contour. Name it Pressure contour.



In Details of Pressure contour, change the locations to ***symmetry 1***, change the variable to ***Pressure***, and change the number of contours to ***50***.

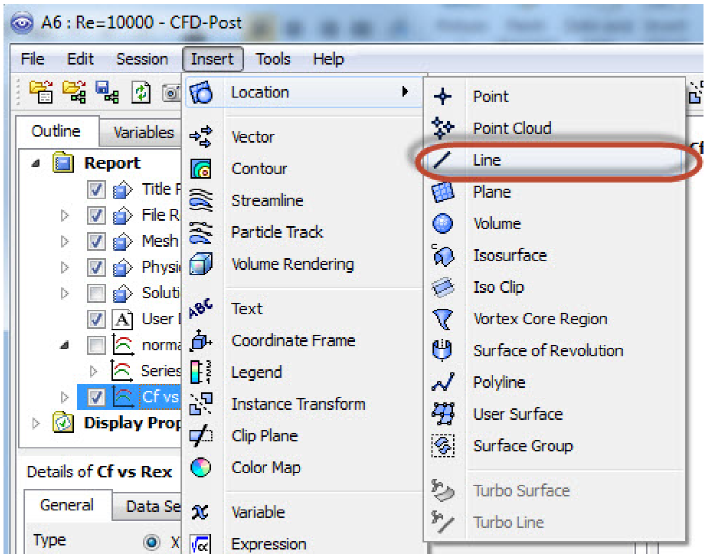


Click on Apply to view the contour.

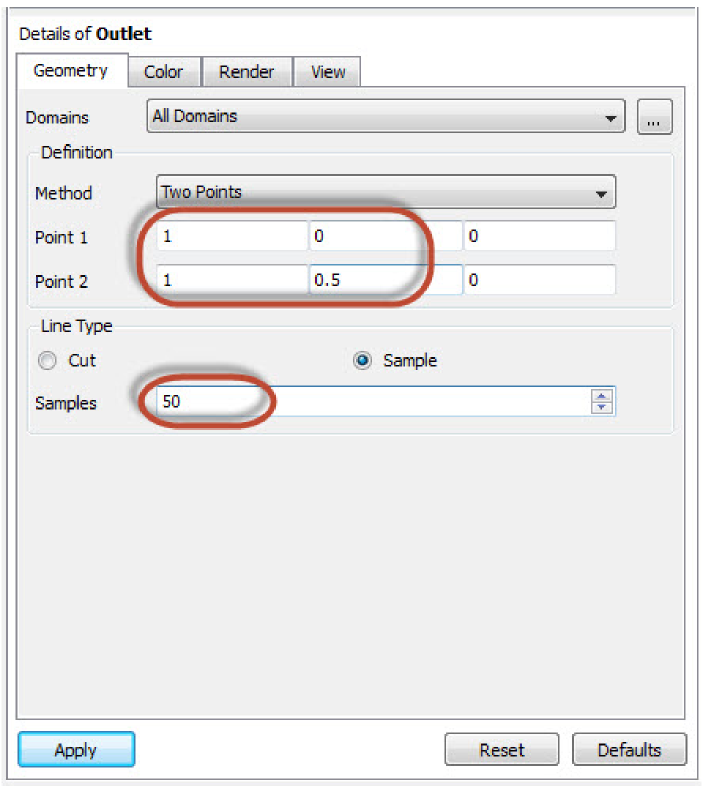


Outlet Velocity Profile

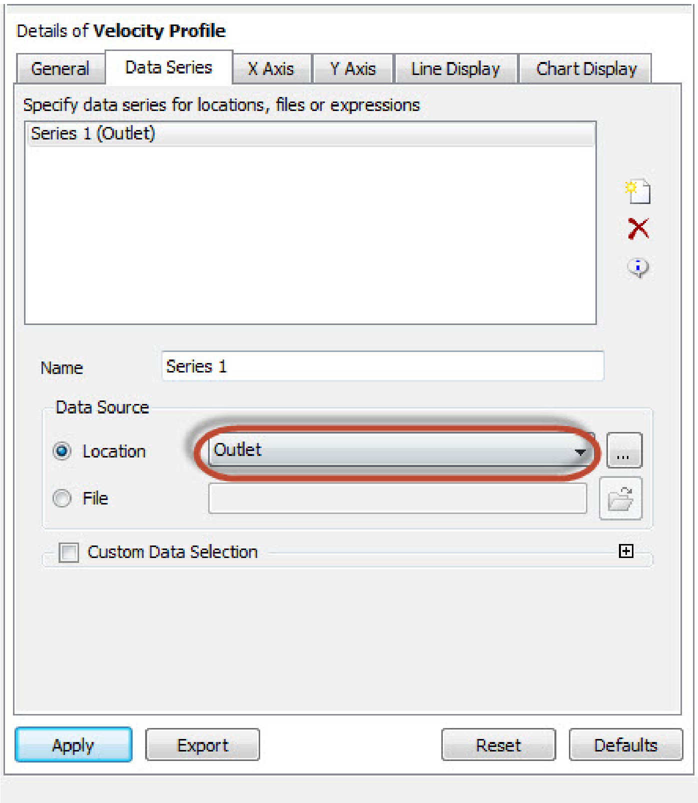
We will create a line that corresponds to the x=1 line (outlet). Then the velocity along this line can be plotted against the Y axis. From the toolbar, insert > location > line. Name it "Outlet"



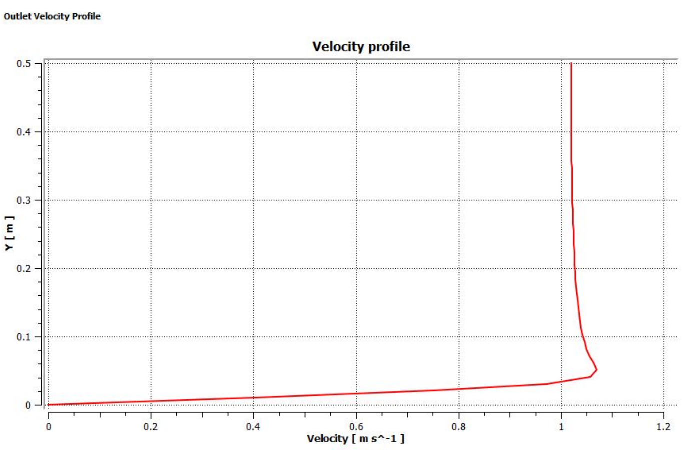
In the Details of Outlet panel, enter the following coordinates. Change the number of samples to ***50***. Click on ***Apply*** to create a line at the outlet.



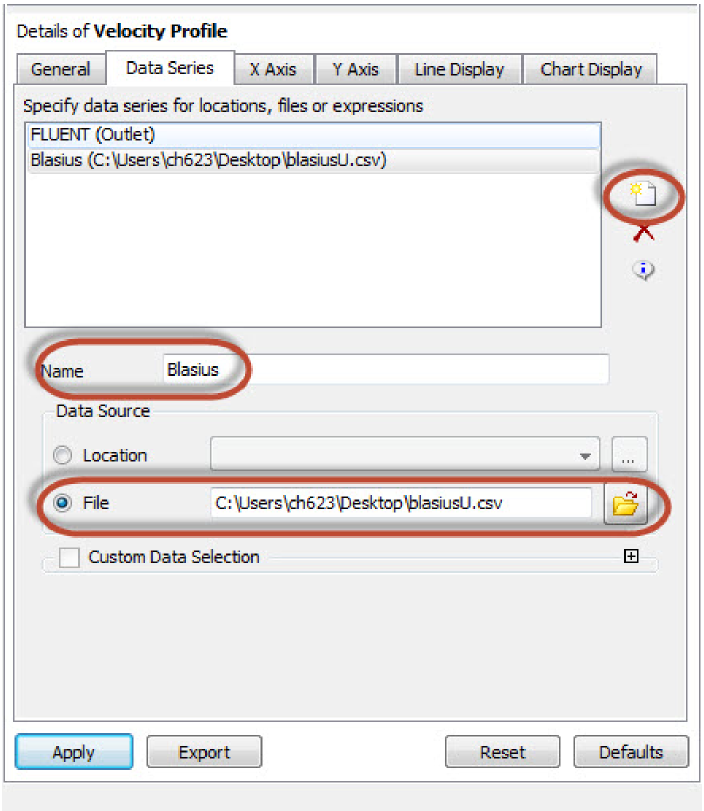
Insert a chart from the menu: insert > chart. Name the chart "Velocity Profile". Change the title to "Velocity Profile" in the ***General*** tab. In the ***Data Series*** tab, rename ***Series 1*** to ***FLUENT*** and select ***Outlet*** for location. Select ***Velocity*** as the variable in the ***X Axis*** tab and select ***Y*** as the variable in the ***Y Axis*** tab. Click on Apply to generate the chart.



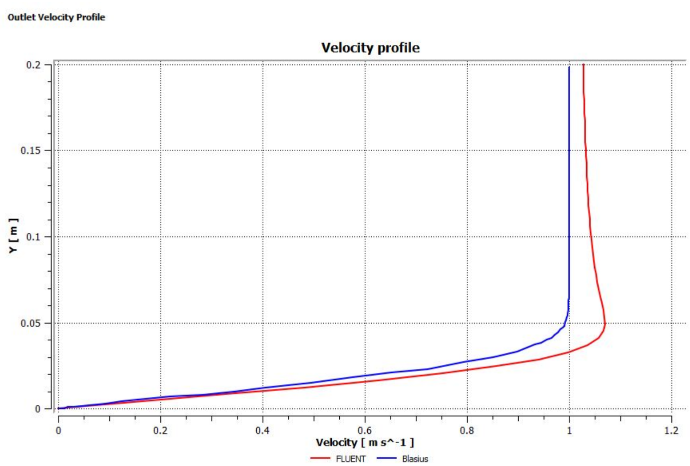
The velocity profile at the outlet is shown below:



We would like to compare the FLUENT result to the Blasius boundary layer solution. Download the Blasius solution here. Return to the ***Data Series*** tab and insert another data set. Rename it ***Blasius***. Instead of specifying the location of the data, select the Blasius solution file you have downloaded.



Click on Apply. The comparison should look like the following plot:



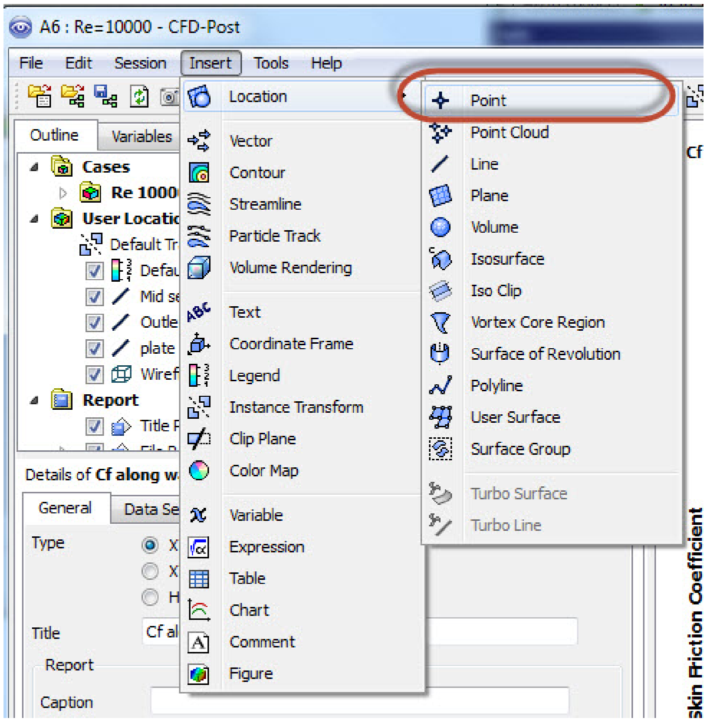
Overshoot in velocity profile: Explanation

We get an overshoot in the velocity profile in the laminar FLUENT solution that is not predicted by classical boundary layer theory. It turns out this is a real effect that is missed by boundary layer theory due to one of the assumptions it makes, namely, that the outer flow is the inviscid flow past a flat plate. This is true if the boundary layer is infinitely thin which is valid only in the limit as Reynolds no. tends to infinity. At a finite Reynolds number, the boundary layer has a thickness which displaces the outer flow and causes the overshoot seen in the FLUENT solution. This is explained in more detail in the following video.

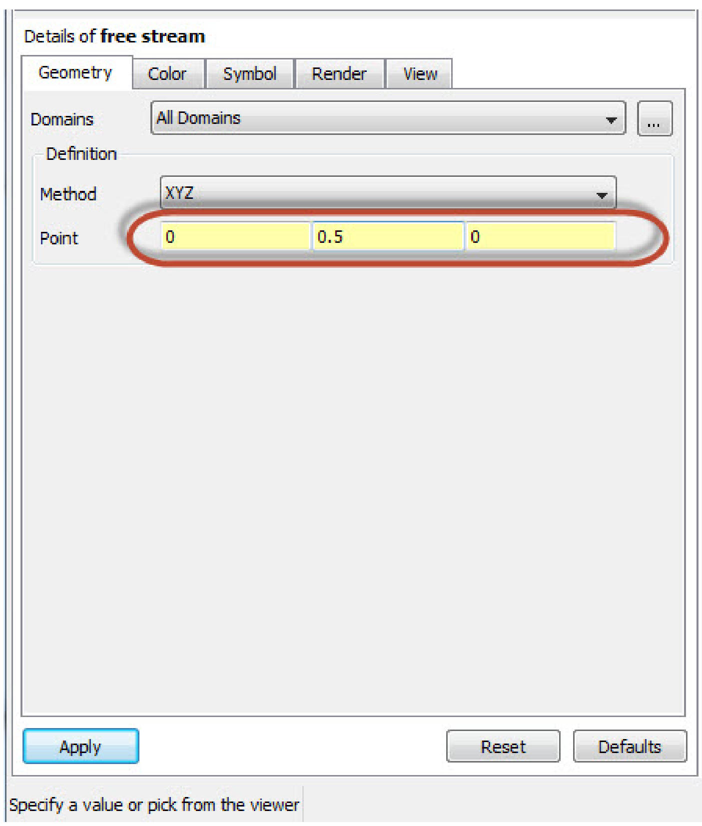
For a closer look at this, you can go to the exercises page here.

Normalized velocity profile

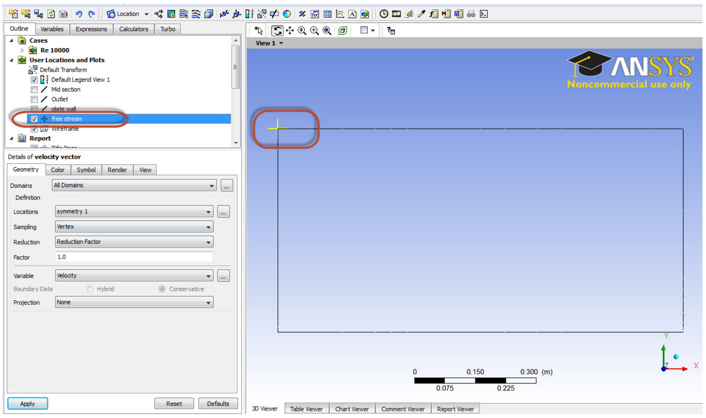
We will observe the normalized velocity (u/U_infinity) at the outlet. Insert a point and call it free stream. The velocity at this point will be extracted and set to the free stream velocity (U_infinity). The velocity profile found in the previous step will be divided by U_infinity.



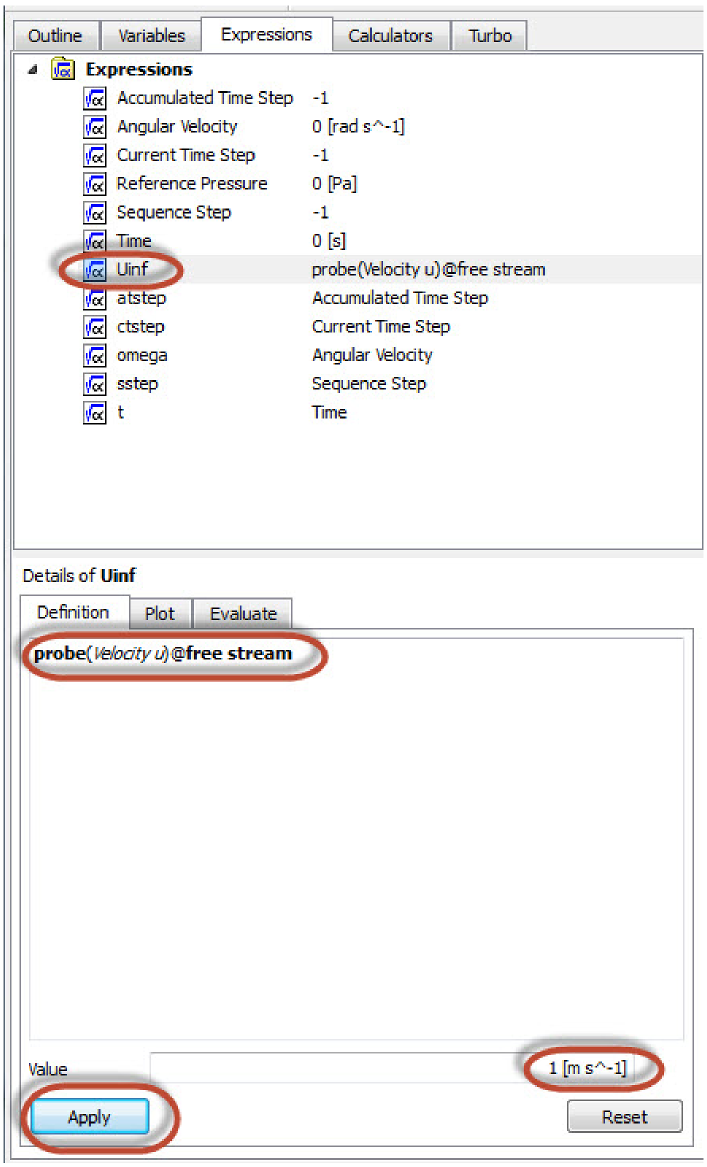
Enter the following coordinates and click on Apply.



The location of this point can be visualized in the 3D viewer:

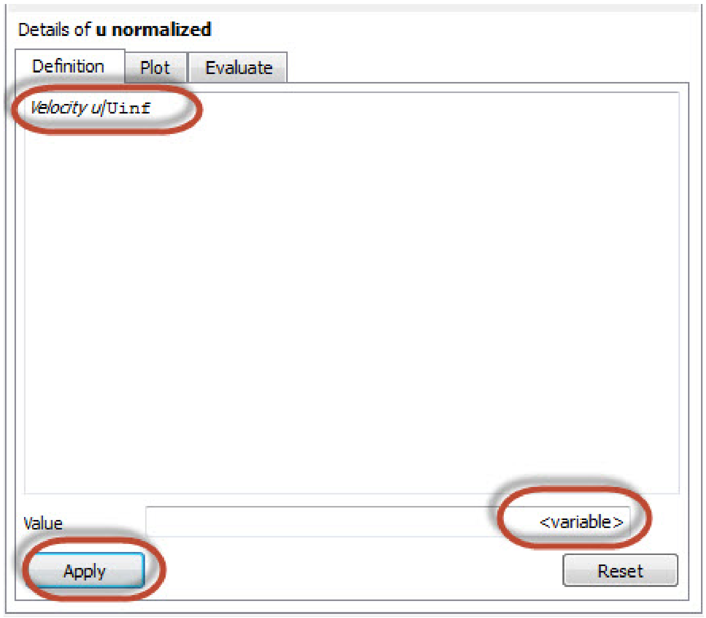


In the expressions tab, create a new expression and name it "Uinf". In Details of Uinf, enter the following command and click on Apply:

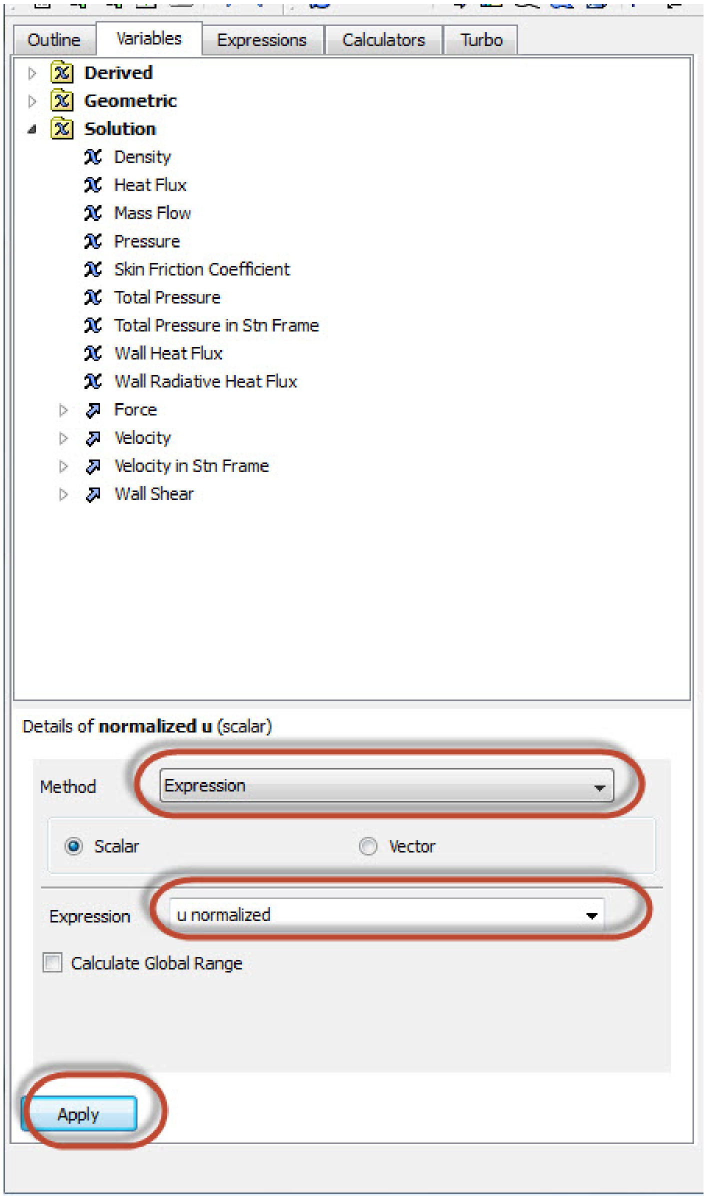


Notice Uinf returns 1 m/s as the velocity at the point where we defined as free stream. This is the same free stream velocity that we have set up in FLUENT.

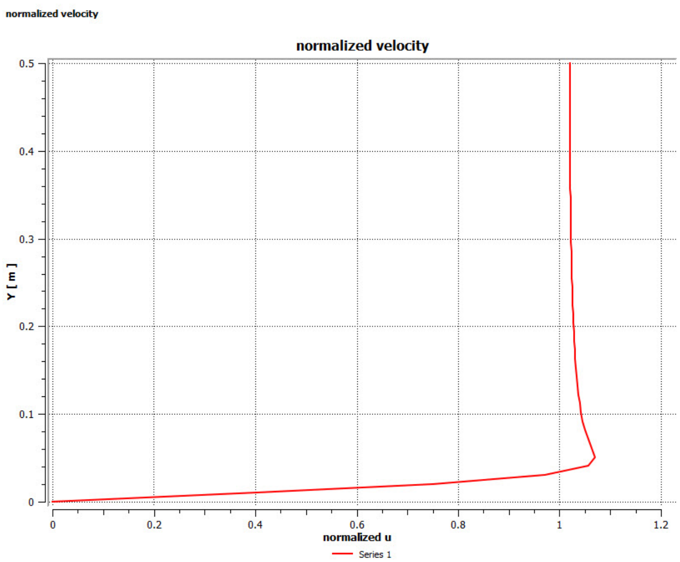
Insert another expression and name it "u normalized". Enter the following command and notice its value is a variable. This is because the u-velocity varies in the y direction.



In the Variables tab, create a new variable and name it "normalized u". Retain Expression for the Method and change the expression to "u normalized" from the drop down list. Click on Apply. normalized u now appears as an variable.



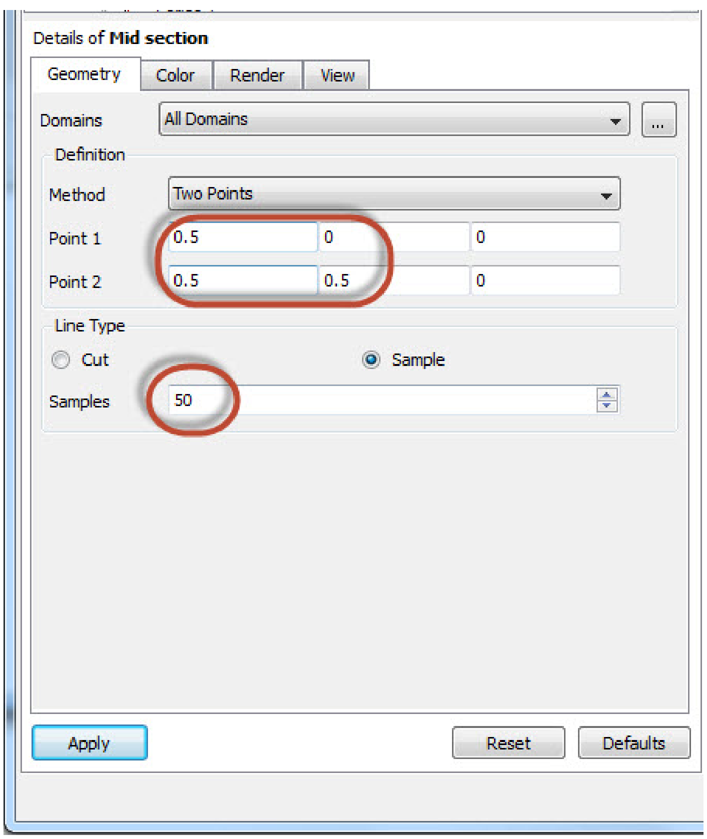
Insert a chart and name it "normalized velocity". Select ***Outlet*** for the location in Data Series. Select ***normalized u*** for the ***X variable*** and ***Y*** for the ***Y variable***. Click on Apply to view the chart.



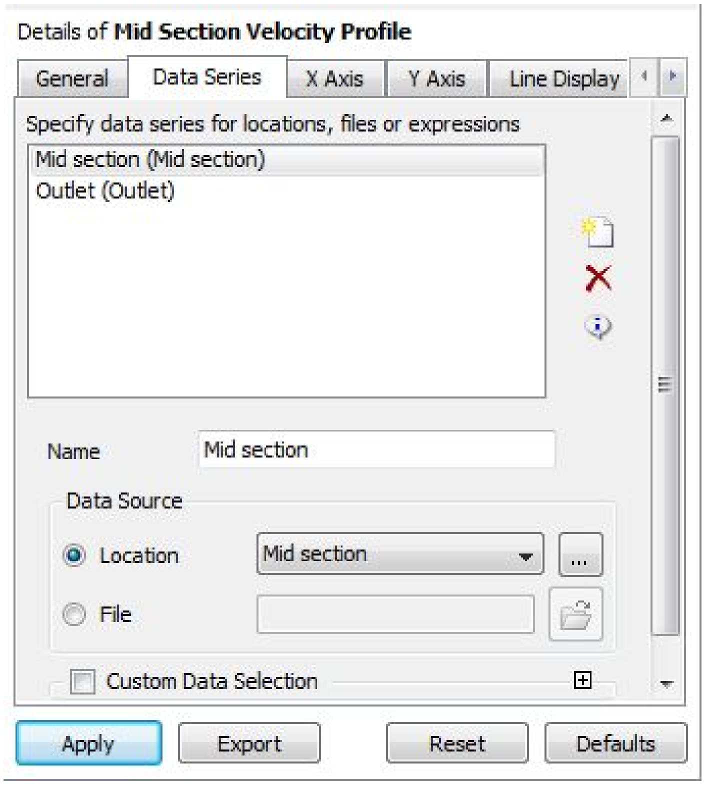
Notice the scale of this profile is exactly the same as that of the outlet velocity profile. This is because the free stream velocity, Uinf, is 1 m/s.

Mid-Section Velocity Profile

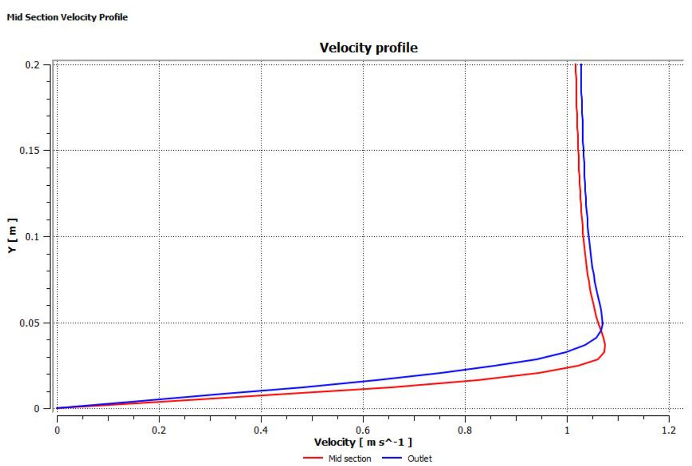
Here, we will plot the variation of the x component of the velocity along a vertical line in the middle of the geometry. In order to create the profile, we must first create a vertical line at x=0.5m. Insert another line, same as the previous step, and name it Mid section. Enter the following numbers to create a vertical line at x=0.5m. Set the number of samples to 50. Remember to click on ***Apply*** to finish.



Insert another chart and name it Mid Section Velocity Profile. In the General tab, change the title to "Velocity profile". Select ***Mid Section*** as the location and rename ***Series 1*** to ***Mid section***. We will compare the velocity profiles at the mid section and at the outlet. Repeat the procedure in the previous step to insert the velocity profile at the outlet. Change the variable to ***Velocity*** in the ***X Axis*** tab and change the variable to ***Y*** in the ***Y Axis*** tab. Click on ***Apply*** to generate the chart.

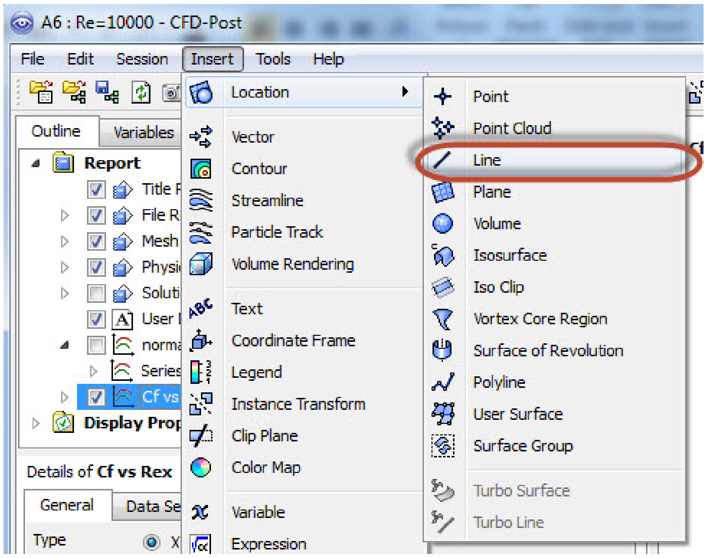


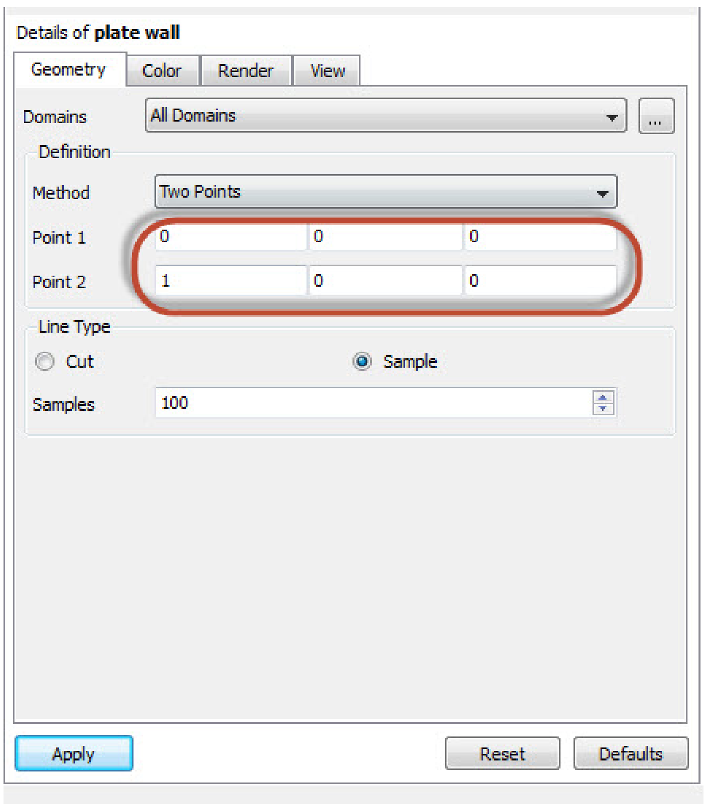
The velocity profile comparison is shown below:



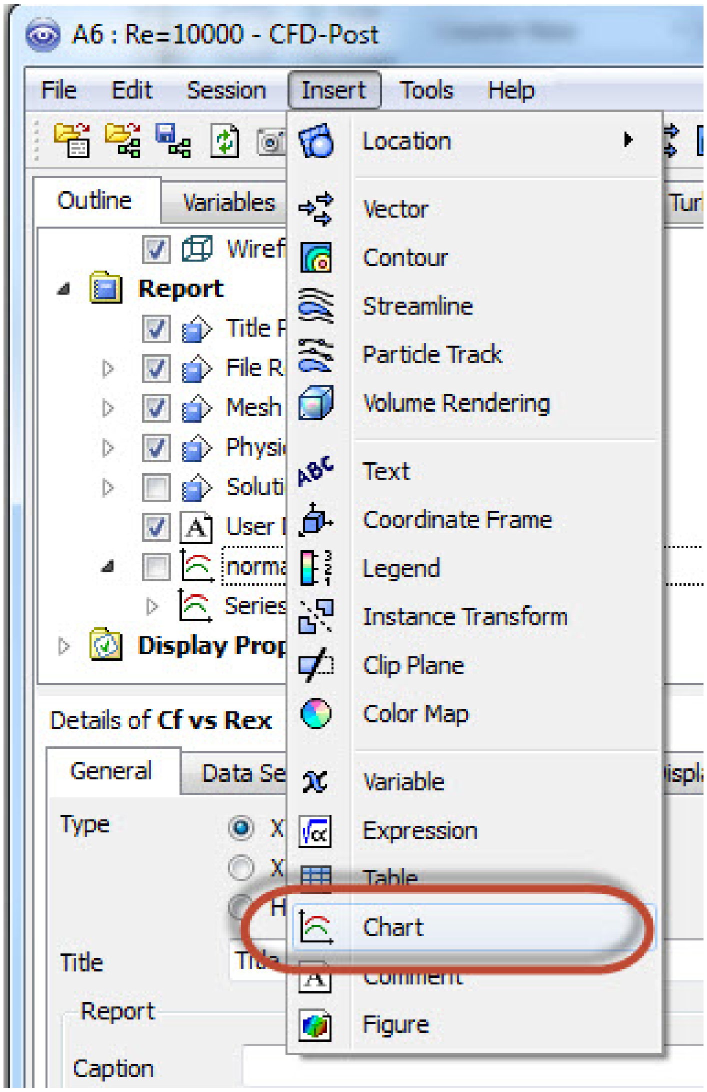
Skin Friction Coefficient

We can plot the skin friction coefficient imported from FLUENT as a function of distance along the plate. Insert a line and name it "plate wall". Enter the end points of the line and the number of samples as the following:

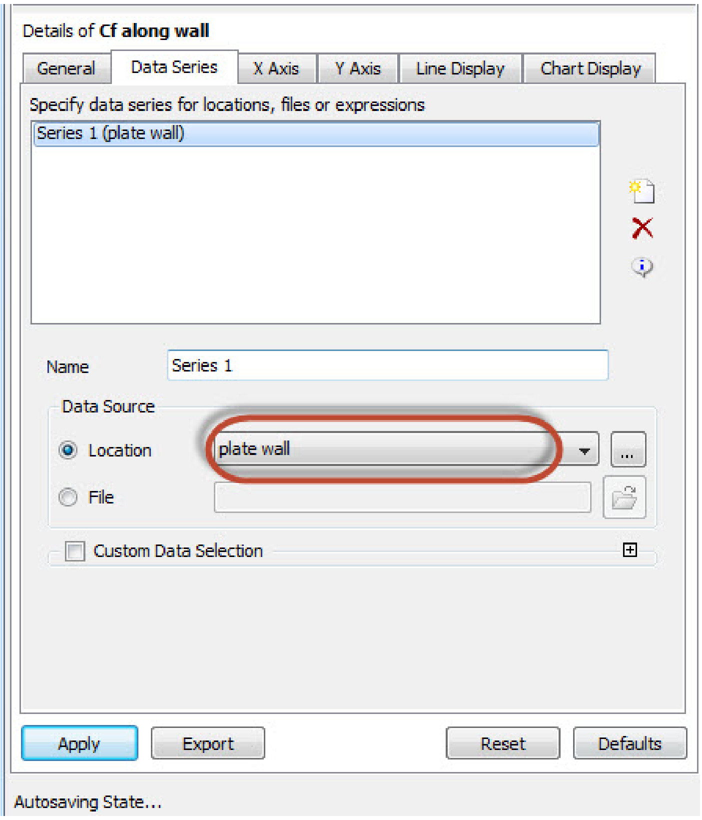




Insert a chart and name it "Cf along wall". Use the same name for the chart title.

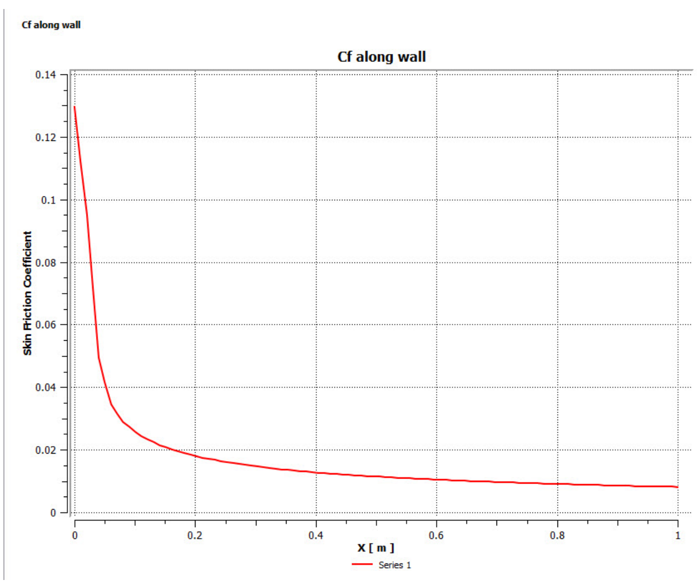


From the Data Series tab, Select "plate wall" for the location.

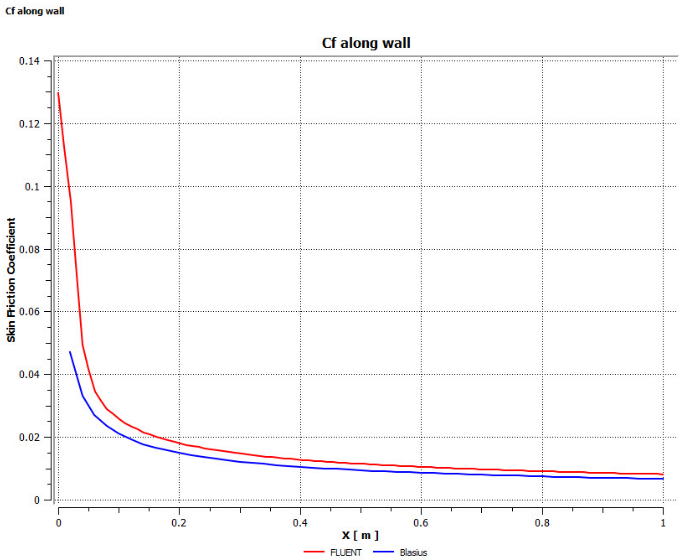


In the X Axis tab, select X as the Variable. In the Y Axis tab, select Skin Friction Coefficient as the Variable.

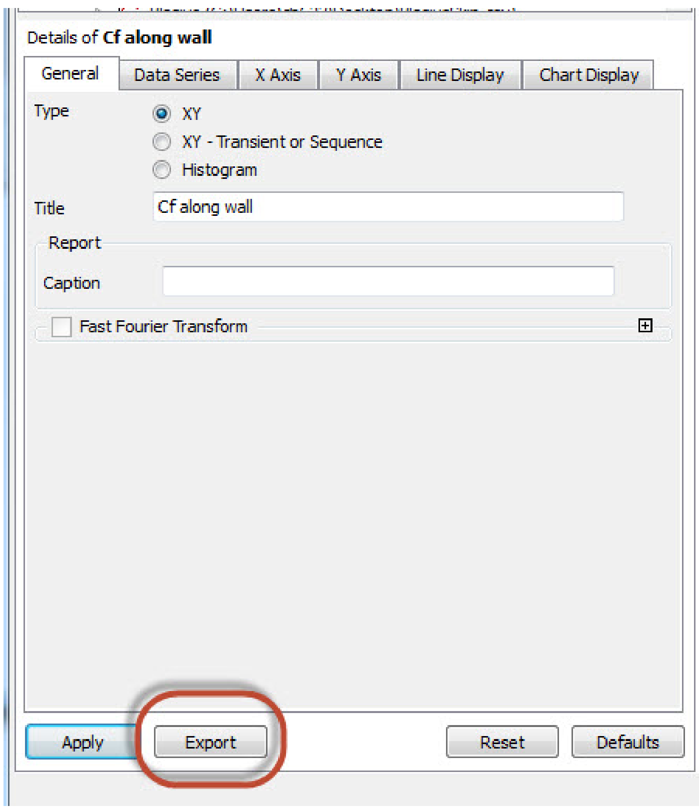
The skin friction coefficient along the plate is shown below:



It is of interest to compare the numerical skin friction coefficient profile to the skin friction coefficient profile obtained from the Blasius solution. We will compare the FLUENT result to the Blasius solution. Download the Blasius solution here. The comparison is shown below:



You can export the skin friction coefficient for data manipulation.



Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules