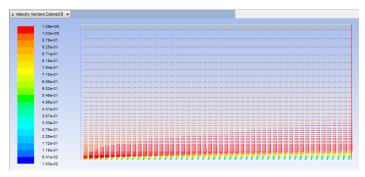
# FLUENT - Flat Plate Boundary Layer - FLUENT Post Processing

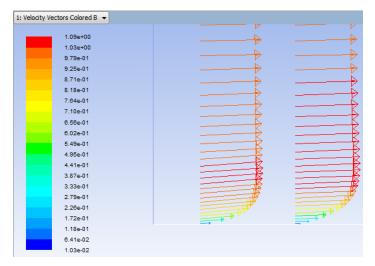
# Step 6: Results

## **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.

1		
ſ	A:FlatPlate FLUENT [20	l, dp, pbns, lam] [ANSYS Academic Teaching Advance
	File Mesh Define Sol	ve Adapt Surface Display Report Parallel
	) 🚰 🕶 🛃 👻 🚳 🔞	\$ 全€ ♥ ∥® 洗 ⊪ - □ -
	Problem Setup	Plots
	General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Plots           YY Plot           Histogram           File           Profiles:           Profile Data - Unavailable           Interpolated Data           FFT
	Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	
	Results Graphics and Animations Plots Reports	Set Up

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 ycoordinate 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.

Solution XY Plot		
Options V Node Values Position on X Axis V Position on Y Axis V Vite to File Order Points File Data	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Direction Vector   X Axis Function Velocity  X Velocity  Surfaces  [] =  [far_field inlet interior-surface_body outlet plate New Surface ▼
Plot	Axes	Curves Close Help

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

1: X Velocity 👻						
outlet						
	5.00e-01					
	4.50e-01					
	4.00e-01					
	3.50e-01					
	3.00e-01					
Position	2.50e-01					
(m)	2.00e-01					
	1.50e-01					
	1.00e-01					
	5.00e-02					
	0.00e+00					
	0	0.2	0.4	<sub>0.6</sub> K Velocity (m/s)	0.8	1

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.

Curves - Solution	XY Plot	×						
Curve # Line Sty 0 • Patter Sample Color foreg Weigh 2	n v round v	Marker Style Symbol Color foreground Size 0.3						
Apply Close Help								

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

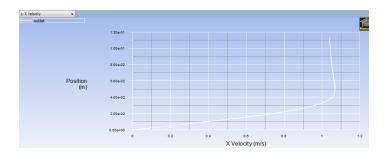
Now, the range of the y axis will be truncated, as we are not interested in far field velocity. Furthermore, the grid lines will be turned on. In order to implement these two changes. First click *Axes* in the *Solution XY Plot* menu. Next, select Y for *Axis*, deselect *Auto Range*, select *Major Rules*, select *Mi nor 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.

Axes - Solution XY Plot		<b></b>		
Axis X Y Label	Number Format Type exponential Precision 2	Major Rules Color foreground Weight 1		
Options Log Auto Range Ø Major Rules Ø Minor Rules	Range Minimum 0 Maximum 0.12 Apply Close Help	Minor Rules Color dark gray Weight 1		

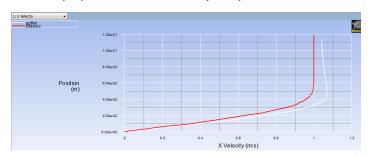
Then, click Apply in the Axes - Solution XY Plot menu. Now, select X for Axis and select Major Rules and Minor Rules, as shown below.

Axes - Solution XY Plot		<b>X</b>		
Axis © X © Y Label	Number Format Type general Predsion 3	Major Rules Color foreground  Weight 1		
Options Log V Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray - Weight 1		
	Apply Close Help			

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.



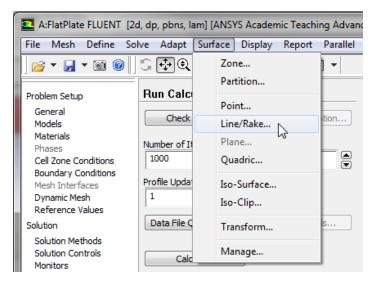
It is of interest to compare the numerical velocity profile to the velocity profile obtained from the Blasius solution. In order to plot the theoretical results, first click here to download the necessary file. Save the file to your working directory. Next, go to the *Solution XY Plot* menu and click *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**

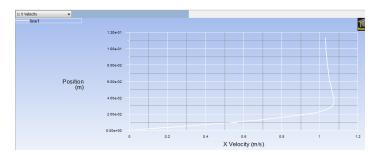
Here, we will plot the variation of the x component of the velocity along a vertical line in the middle of the geometry. In order to create the profile, we must first create a vertical line at x=0.5m, using the *Line/Rake* tool. First, *(Click) Surface < Line/Rake* as shown in the following image.



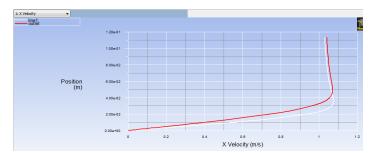
We'll create a straight vertical line from (x0,y0)=(0.5,0) to (x1,y1)=(0.5,0.5). Select *Line Tool* under *Options*. Enter x0=0.5, y0=0,x1=0.5, y1=0.5. Enter line1 under *New Surface Name*. Your *Line/Rake Surface* menu should look exactly like the following image.

Line/Rake Surfac	e X
Options Type Line Tool Reset	
End Points	
x0 (m) 0.5	x1 (m) 0.5
y0 (m) 0	y1 (m) 0.5
z0 (m) 0	z1 (m) 0
Se	elect Points with Mouse
New Surface Name	
line1	
Create Ma	anage Close Help

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 *Soluti* on *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.



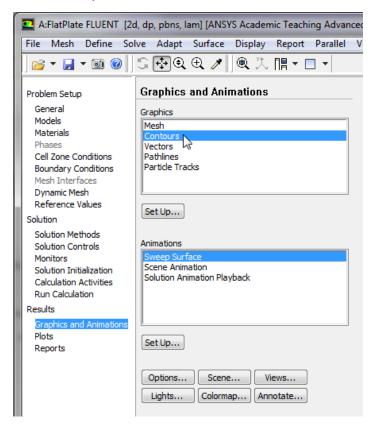
Once again, return to the Solution XY Plot menu, select Write to File, then click Write.... When prompted for a filename, enter XVelProfs.xy and save the file in your working directory.

#### **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.

A:FlatPlate FLUENT [2	d, dp, pbns, lam] [ANSYS Academic Teaching Advanced
File Mesh Define So	lve Adapt Surface Display Report Parallel Vi
🞯 🕶 🛃 🕶 🎯 🔘	\$ ❹€ € ↗   € Հ 唱 - □ -
Problem Setup	Reference Values
General	Compute from
Models	inlet 🗸
Materials	surface body
Phases Cell Zone Conditions	far_field
Boundary Conditions	interior-surface_body inlet
Mesh Interfaces	outlet
Dynamic Mesh	plate
Reference Values	Depth (m) 1
Solution	Complniet_F

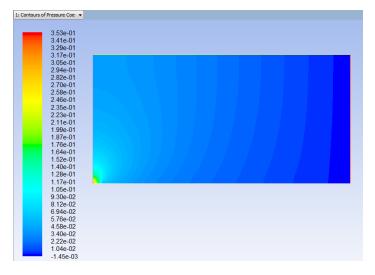
Next, click on *Graphics and Animations*, then double click on *Contours*, as shown below.



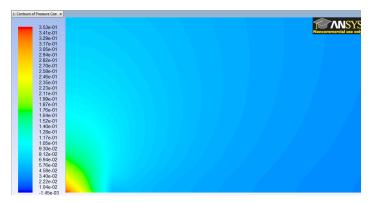
In the Contours menu, set Contours of to Pressure... and set the box below to Pressure Coefficient. Next, select Filled and set Levels to 90. Your Con tours menu should look exactly like the following image.

Contours		×
Options	Contours of	
Filled	Pressure	
Vode Values	Pressure Coefficient	•
Auto Range	Min Max	
Clip to Range	0	
Draw Profiles		
Draw Mesh	Surfaces	
	far_field inlet	
Levels Setup	interior-surface_body	
90 🚔 1	outlet     plate	
	plate	
Surface Name Patter	n New Surface 🔻	
	Aatch	
	Surface Types	
	axis	
	dip-surf exhaust-fan	
	fan	-
Display	y Compute Close Help	

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.

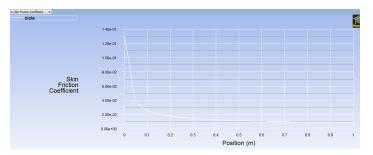


**Skin Friction Coefficient** 

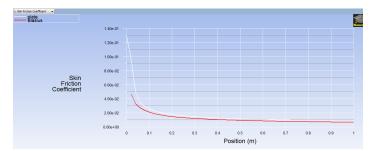
Here, the skin friction coefficient will be plotted as a function of distance along the plate. First, click on *Plots*, then double click on *XY Plot*. In the *Solution XY Plot* menu deselect *Write to File*, select *Position on X Axis*, set X to 1 and set Y to 0. Then, set the box located underneath *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.

Solution XY Plot		<b>X</b>
Options V Node Values V Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Wall Fluxes Skin Friction Coefficient X Axis Function Direction Vector Surfaces far_field inlet interior-surface_body outlet plate New Surface New Surface
Plot	Axes	Curves Close Help

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.

A:FlatPlate FLUENT [20	d, dp, pbns, lam] [ANS)	/S Academ	nic Teach	ing Advan	ced]	_	
File Mesh Define So	lve Adapt Surface	Display	Report	Parallel	View	Help	
📴 🕶 🛃 🕶 🞯 🔘	] 📂 ▾ 🛃 ▾ 🚳 @ 🛛 🕾 🔂 🤁 🗶 🥒 🔍 🔍				Result Reports		
Problem Setup General	Reports Reports Fuxes Forces			p <b>ut Summ</b> S Informat	1 - C		
Models Materials			Re	ference Va	lues		

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

Zone	Forces (m) Pressure (8 -8.81319854 )	1)		iscous 0.000376893 0.00	818661246 8)		Total (8.008376893	-8.013883927		Coefficients Pressure (0 -0.02630100 0)
Net	(8 -8.01319054	1)	(	0.008376893 0.00	@18661246 B)		(0.008376893	-0.013083927	8)	(0 -0.02638108 0)
Forces - Direction Vector	(1 0 0) Forces (n)			Coefficients						
		Viscous	Tetal	Pressure	Viscous	Total				
plate	8	8.008376893	8.888376893	8	8.816753786	8.016753786				
Het	9	8,898376893	9,008376893		8.816753786	8, @16753786				

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.

$$C_d = 2F_d$$

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