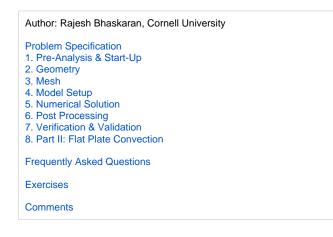
# Old (4/1/2020) Flat Plate Boundary Layer - Numerical Results



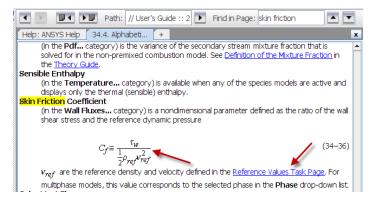
## Numerical Results

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.

i 💕 🕶 🛃 🕶 🞯 🖉 🕻 🕻	⊉Q € ∥∥® 밨 ⊪ ▾ □ ▾
Problem Setup	Reference Values
General Models	Compute from
Materials Phases	Reference Values
Cell Zone Conditions Boundary Conditions	Area (m2) 1
Mesh Interfaces	Density (kg/m3)
Reference Values	Depth (m) 1
Solution Methods	Enthalpy (j/kg)
Solution Controls Monitors -	Length (m) 1
Solution Initialization Calculation Activities	Pressure (pascal)
Run Calculation Results	Temperature (k) 288.16
Graphics and Animations Plots	Velocity (m/s) 1
Reports	Viscosity (kg/m-s) 1e-05
	Ratio of Specific Heats 1.4
	Reference Zone
	flow_domain
	Help

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:

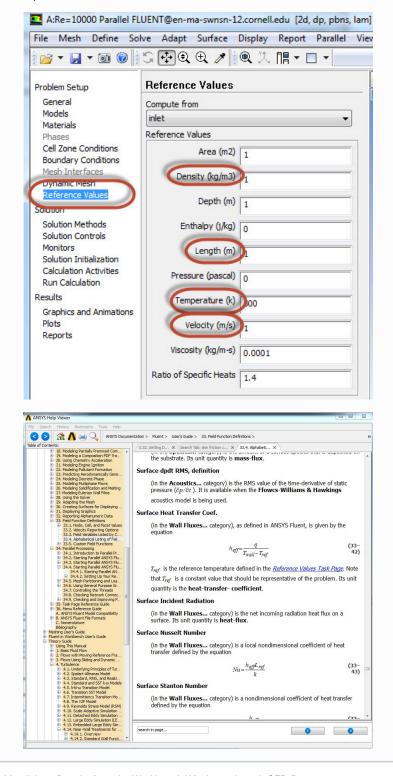
A:Re=10000 Parallel FLUE	NT@en-ma-swnsn-12.cornell.edu [2d, dp, pbns, la Adapt Surface Display Report Parallel \
Refresh Input Data	<b>↓ ① 〃 ◎ ↓ 開 - □ -</b>
Save Project	
Read	culation
Write	K Case     Preview Mesh Motion
Import	Iterations Reporting Interval
Export	
Solution Files	late Interval
Interpolate	
EM Mapping	Quantities     Acoustic Signals
FSI Mapping	•
Save Dicture	alculate
Data File Quantities	
Close FLUENT	
Graphics and Animations	
Plots Reports	

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

Data File Quantities		×
	oostprocessing in external applications through ntities in the data file for postprocessing in ex	
Standard Quantities	Additional Quantities	
Pressure X Velocity Mass Flux Body Force Wall Velocity Original Wall Velocity Wall Shear Mach Number Boundary Heat Flux	Mesh Y-Velocity Velocity Angle Relative Velocity Angle Vorticity Magnitude Cell Reynolds Number Molecular Viscosity Wall Shear Stress X-Wall Shear Stress Ste Erician Coefficient	
Boundary Rad Heat Flux	Skin Friction Coefficient	
Density Laminar Viscosity 2nd Grad Bc Source	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,

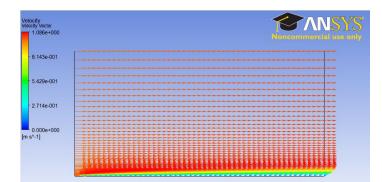
to view the XY plane. Click on the vector icon to insert a vector plot. Name it Velocity Vector.

A6 : Fluid F	low (FLUI	ENT) - CFD-Pos	st		
File Edit S	ession 1	Insert Tools	Help		
😤 🕰 🔩	1	DOF	Location 🚽	+*	S 🕸 🗛 📢 🕼
Outline V	ariables	Expressions	Calculators	Turbo	*\ 5 ↔ Q (
D	Location efault Tran	Legend View 1			View 1 🔻
~					

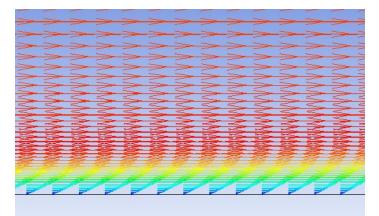
A panel named "Details of Velocity Vector" will appear right below the outline window. Set the *Locations* to symmetry 1. Click on *Apply* to display the velocity vectors.

Geometry	Color	Symbol	Render	View		
Domains	All Dom	ains				<b>•</b> ] [.
Definition	-					
Locations	symm	etry 1				)
Sampling	Verte	x				•
Reduction	Redu	ction Factor				•
Factor	1.0					
Variable	Veloci	ty				•
Boundary Data	а	0 н	lybrid		Conservat	tive
Projection	None					-
Projection	None					•

The velocity vectors will be displayed in the view window.

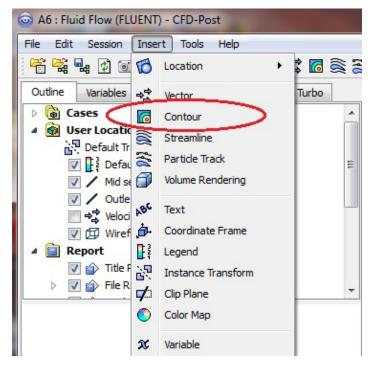


You can use the wheel button of the mouse to zoom into the region that closely surrounds the plate, to get a better view of the boundary layer velocities:



#### **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.

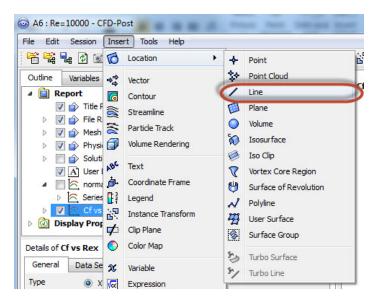
Geometry	Labels Render View	
omains	All Domains	•
ocations	symmetry 1	-
Variable	Pressure	•
Range	Global	-
Min		unknown
Max		unknown
Boundary Da	ata 🔘 Hybrid	Onservative
Color Scale	Linear	•
Color Map	Default (Rainbow)	•
f of Contours	and an and a second	×

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"



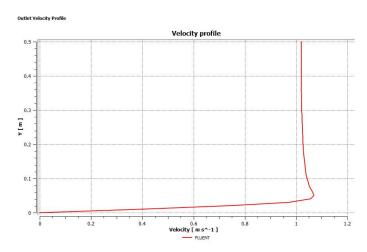
In the Details of Outlet panel, enter the following coordinates. Change the number of samples to 50. Click on Apply to create a line at the outlet.

Geometry	Color	Render	View		
omains	All Dor	nains			•
Definition					
Method	Two F	Points			•
Point 1	1		0	0	
Point 2	1		0.5	0	
Line Type					
Cut	-	_	و ھ	ample	
Samples	50				* *
	_				

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.

General	Data Series	X Axis	Y Axis	Line Display	Chart Display
pecify dat	ta series for loca	tions, files	or express	ions	
Series 1 (	Outlet)				
					*
					×
					- D
					~
Name	Series 1				
Name	Jenes 1				
Data S	Source				_
Data S	1	utlet			
O Loo	cation	utlet			
	cation	utlet			
Loc File	cation				
Loc File	cation				
Loc File	cation				
Loc File	cation				
Loc File	cation				
Loc File	cation				

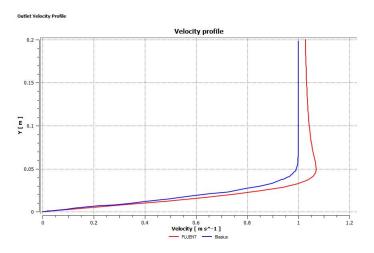
The velocity profile at the outlet is shown below:



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.

General	Data Series	X Axis	Y Axis	Line Display	Chart Display
ecify data	a series for loo	ations, files	or expressi	ions	
LUENT (O	utlet)				
asius (C:	Users\ch623	/Desktop/bla	siusU.csv)		0
					CE
					7
					~
6	Dission				
Name	Blasius	)			
Name Data So		)			
-	ource	)			
Data So	ource ation (				
Data So	ource ation (		523\Deskto	p\blasiusU.csv	•
Data So O Loca O File	ation	C: \Users\ché	523\Desktoj	p\blasiusU.csv	ß
Data So O Loca O File	ource ation (	C: \Users\ché	523\Desktoj	p\þlasiusU.csv	•
Data So O Loca O File	ation	C: \Users\ché	523\Desktoj	p\blasiusU.csv	ß
Data So O Loca O File	ation	C: \Users\ché	523\Deskto	p\blasiusU.csv	ß
Data So O Loca O File	ation	C: \Users\ché	523\Deskto	p\blasiusU.csv	ß
Data So O Loca O File	ation	C: \Users\ché	523\Desktoj	p\blasiusU.csv	ß

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  $U_{infinity}$ .

File Edit	Session Ir	isert Tools Help	-		
r i i i i i i i i i i i i i i i i i i i	e 🖸 🖻 🕻	Location	(+	Point	Ì
4 🚱 Use 19 19 19 19 19 19 19 19 19 19 19 19 19	Re 1000 er Locatic Default Tr Default Tr Default Tr Defau Mid se Defau Mid se Defau File Fil	Contour Streamline Particle Track Volume Rendering Text Coordinate Frame Legend Instance Transform Clip Plane Color Map	なく むん きゅう	Point Cloud Line Plane Volume Isosurface Iso Clip Vortex Core Region Surface of Revolution Polyline User Surface Surface Group Turbo Surface	
Type Title Report Caption	Data Se 👷 © X 🖟 © H E Cfal	Expression Table Chart Comment	\$7	Turbo Line	

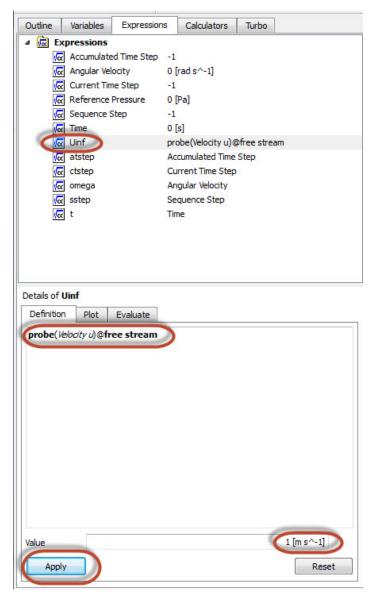
Enter the following coordinates and click on Apply.

Geometry	Color	Symbol	Render	View		
Domains	All Dor	mains				•
Definition						
Method	XYZ					•
Point	0		0.5		0	
Apply	1				Reset	Defaults

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

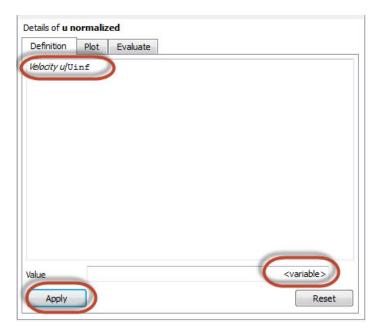
	ables Expressions Calculators Turbo	. % 🖸 🗘 Q Q Q 🖉 🗆 - %
	scattions and Plots ault Transform E Default Legend Vew 1 Mid section —	
🖌 🛄 Report		
etails of veloc	ity vector Color Symbol Render View	
	Al Domeins 👻	
Locations	symmetry 1 🔹	
Sampling	Vertex	
Reduction	Reduction Factor	
Factor	1.0	
Variable	Velocity	
Boundary Date		
Projection	None +	
		ч. така така така така така така така так
		t.
		0 0.150 0.300 (m)
		0.075 0.225
Apply	Reset Defaults	30 Venet Table Venet Chart Venet Comment Venet Report Venet

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.

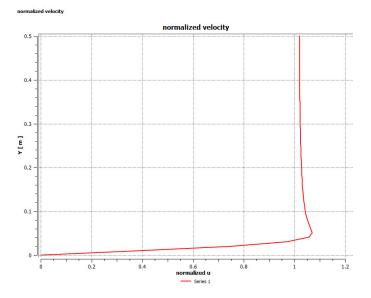
Insert another expression and name it "u normalized". Enter the following command and notice its value is a variable. This is because the u-velocity varies in the y direction.



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.

			12 U.S.		~ ~~		-
Outline	Variables	Expressions	Calculators	Turbo			
De 🕱 De	rived						
🖻 🕱 Ge	ometric						
a 🕱 So	lution						
x	Density						
x	Heat Flux						
x	Mass Flow						
X	Pressure						
x	Skin Friction	Coefficient					
x	Total Pressu	ire					
x	Total Pressu	ure in Stn Frame					
x	Wall Heat F	lux					
X	Wall Radiati	ive Heat Flux					
> 🄉	Force						
> 🏹	Velocity						
> 🏞	Velocity in S	tn Frame					
Þ 🏹	Wall Shear						
Details of <b>n</b>	ormalized	u (scalar)				-	
Method	Ever	ession					
Method	Lyn	ession					
S	calar		O Vector			_	
Express	C	ormalized					
Express	aion	ormalizeu			_	-	
Calcu	ulate Global F	lange					
_							
Apply							

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**

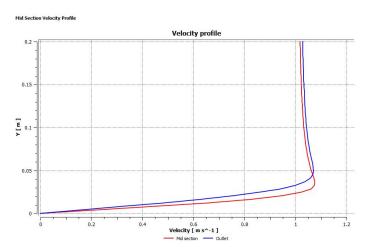
Here, we will plot the variation of the x component of the velocity along a vertical line in the middle of the geometry. In order to create the profile, we must first create a vertical line at x=0.5m. Insert another line, same as the previous step, and name it Mid section. Enter the following numbers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

Geometry	Color Ren	der View		
Domains	All Domains			▼
Definition				
Method	Two Points			•
Point 1	0.5	0	0	
Point 2	0.5	0.5	0	
Line Type				
Cut	$\sim$	Sa	mple	
Samples	50			*
	-			(
Apply			Reset	Default

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.

General	Data Serie	s X Axis	Y Axis	Line Di	splay	4	) Þ
Specify dat	ta series for lo	ocations, files	or expressi	ions			•
Mid section	n (Mid section	1)					
Outlet (Ou	10et)				×		
Name	Mid s	ection					I
Name Data S		ection					
	ource	ection Mid section					
Data S	ource ation				: 3		
Data S Loc File	ource ation	Mid section					-

The velocity profile comparison is shown below:



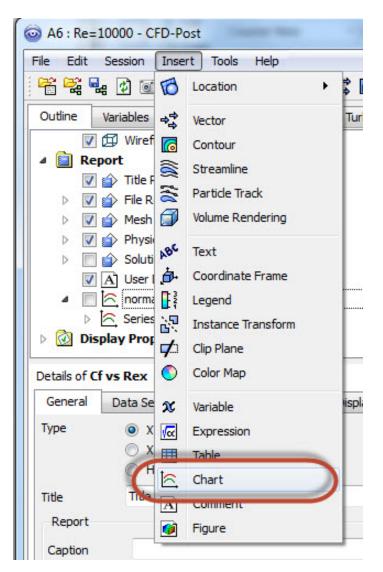
### **Skin Friction Coefficient**

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:

🚡 🛃 💂 ⊅ 🗃 🔞 Dutline Variables 🚓	Location Vector	+	Point Point Cloud
Report     Image: Constraint of the second sec	Contour Streamline Particle Track Volume Rendering Text Coordinate Frame Legend Instance Transform Clip Plane Color Map	(1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	Line Plane Volume Isosurface Iso Clip Vortex Core Region Surface of Revolution Polyline User Surface Surface Group
General Data Se	Variable Expression	50 51	Turbo Surface Turbo Line

Geometry	Color	Render	View		
Domains	All Dor	nains			•
Definition					
Method	Two	Points			-
Point 1	0		0	0	
Point 2	1		0	0	
Line Type	-				
Cut			Sa	ample	
Samples	100				*

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

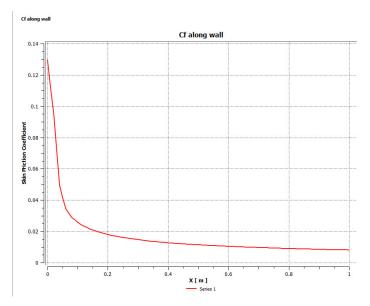


From the Data Series tab, Select "plate wall" for the location.

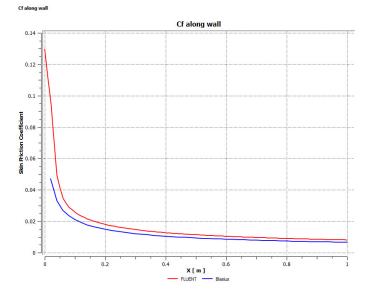
General	Data Series	X Axis	Y Axis	Line Display	Chart Display
pecify dat	ta series for loca	tions, files	or express	ions	
	plate wall)			196652	-
					*
					×
					~
		0			
Name	Series 1				
Data S	ource	_			-
Loc	ation p	late wall			•
		_			
File					
- 🗖 Q	ustom Data Sele	tion			Ŧ

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.

General	Data Series	X Axis	Y Axis	Line Display	Chart Display
Type	<ul> <li>XY</li> <li>XY - Tra</li> <li>Histogram</li> </ul>	ansient or S am	Sequence		
ītle	Cf along w	all			
Report					
Caption					
- East	Fourier Transfor				Ŧ

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules