An Introduction to ANSYS Fluent 2020





Better Textbooks. Lower Prices. www.SDCpublications.com

Visit the following websites to learn more about this book:





Googlebooks



CHAPTER 2. FLAT PLATE BOUNDARY LAYER

A. Objectives

- Creating Geometry in ANSYS Workbench for ANSYS Fluent Flow Simulation
- Setting up ANSYS Fluent Simulation for Laminar Steady 2D Planar Flow
- Setting up Mesh
- Selecting Boundary Conditions
- Running Calculations
- Using Plots to Visualize Resulting Flow Field
- Compare with Theoretical Solution using Mathematica Code

B. Problem Description

In this chapter, we will use ANSYS Fluent to study the two-dimensional laminar flow on a horizontal flat plate. The size of the plate is considered as being infinite in the spanwise direction and therefore the flow is 2D instead of 3D. The inlet velocity for the 1 m long plate is 5 m/s and we will be using air as the fluid for laminar simulations. We will determine the velocity profiles and plot the profiles. We will start by creating the geometry needed for the simulation.



C. Launching ANSYS Workbench and Selecting Fluent

1. Start by launching ANSYS Workbench. Double click on Fluid Flow (Fluent) that is located under Analysis Systems in Toolbox.



Figure 2.1 Selecting Fluid Flow

D. Launching ANSYS DesignModeler

 Select Geometry under Project Schematic in ANSYS Workbench. Right-click on Geometry and select Properties. Select 2D Analysis Type under Advanced Geometry Options in Properties of Schematic A2: Geometry. Right-click on Geometry in Project Schematic and select to launch New DesignModeler Geometry. Select Units>>Millimeter as the length unit from the menu in DesignModeler.

Project Schematic 👻 🔻 🗙			×	Propertie	es of Schematic A2: Geometry		•	φ x		
							А	В		
					. 1	1	Property	Value		
	•		A	_		2	■ General			
	1	G	Fluid Flow (Fluent)		ų – I	3	Component ID	Geometry		
	2	9	Geometry	?	4	4	Directory Name	FFF		
	3	۲	Mesh	P ,		5	Notes			
	4		Setup	7		6	Notes			
	5	1	Solution	2		7	Used Licenses			
	6	۲	Results	2		8	Last Update Used Licenses			
L			Fluid Flow (Fluent)			9	Basic Geometry Options			
			riala rion (riaency			10	Solid Bodies	V		
						11	Surface Bodies	\checkmark		
						12	Line Bodies			
						13	Parameters	Independent		•
						14	Parameter Key	ANS;DS		
						15	Attributes			
						16	Named Selections			
						17	Material Properties			
						18	 Advanced Geometry Options 			
						19	Analysis Type	2D		-

Figure 2.2a) Selecting Geometry and 2D Analysis Type

▼	A		
1	🕃 Fluid Flow (Flue	ent)	
2	🥪 Geometry	la.	> ,
3	🍘 Mesh	sc	lew SpaceClaim Geometry
4	🎡 Setup	DM N	lew DesignModeler Geometry
D .	0.01) T 1	·	· NC 11

Figure 2.2b) Launching DesignModeler

A: Fluid Flow (Fluent) - DesignModeler

File Create Concept Tools	Units	View	Help
] 🖄 🔒 🛃 📾 🗍 🌖 Unda	🗸 М	eter	
	Ce	entimet	ter
VVDIane - St. None	М	illimete	≥r

Figure 2.2c) Selecting the length unit

I

3. Next, we will be creating the geometry in DesignModeler. Select XYPlane from the Tree Outline

on the left-hand side in DesignModeler. Select Look at Sketch in the Tree Outline and select the Line sketch tool.

Draw a horizontal line 1,000 mm long from the origin to the right. Make sure you have a P at the origin when you start drawing the line. Also, make sure you have an H along the line so that it is horizontal and a C at the end of the line.

Select Dimensions within the Sketching options. Click on the line and enter a length of 1,000 mm. Draw a vertical line upward 100 mm long starting at the end point of the first horizontal line. Make sure you have a P when starting the line and a V indicating a vertical line. Continue with a horizontal line 100 mm long to the left from the origin followed by another vertical line 100 mm long.

The next line will be horizontal with a length 100 mm starting at the endpoint of the former vertical line and directed to the right. Finally, close the rectangle with a 1,000 mm long horizontal line starting 100 mm above the origin and directed to the right.





Tree Outline





Figure 2.3c) Rectangle with dimensions

De	Details View				
Ξ	Details of Sketch1				
	Sketch	Sketch1			
	Sketch Visibility	Show Sketch			
	Show Constraints?	No			
Ξ	Dimensions: 4				
	🗌 H1	1000 mm			
	🗌 НЗ	100 mm			
	🗌 V2	100 mm			
	🗌 V4	100 mm			
. .	0.0.1) D'	·			

Figure 2.3d) Dimensions in Details View

4. Click on the Modeling tab under Sketching Toolboxes. Select Concept>>Surfaces from Sketches in the menu. Control select the six edges of the rectangle as Base Objects and select Apply in

Details View. Click on Generate in the toolbar Generate. The rectangle turns gray. Right click in the graphics window and select Zoom to Fit and close DesignModeler.

A: Fluid Flow (Fluent) - DesignModeler

File Create	Concept	Tools	Units	View
2 🛛 🖓	°∽ Lines	From P	oints	
.	🖾 Lines	From S	ketches	
	🔓 Lines	From E	dges	
	VA 3D Cu	irve		
] Thin/Surf	🛰 Split E	dges		
BladeEditor:	🗭 Surfa	ces Fror	n Edge	5
∦ K S	🔊 Surfa	ces Fror	n Sketc	hes

Figure 2.4a) Selecting Surfaces from Sketches

De	etails View			ņ
	Details of SurfaceSk1			
	Surface From Sketches	SurfaceSk1		
	Base Objects	Apply	Cancel	
	Operation	Add Material		
	Orient With Plane Normal?	Yes		
	Thickness (>=0)	0 mm		

Figure 2.4b) Applying Base Objects



Figure 2.4c) Completed rectangle in DesignModeler

E. Launching ANSYS Meshing

5. We are now going to double click on Mesh under Project Schematic in ANSYS Workbench to open the Meshing window. Select Mesh in the Outline of the Meshing window. Right click and select Generate Mesh. A coarse mesh is created. Select Unit Systems>>Metric (mm, kg, N ...) from the bottom of the graphics window.



Figure 2.5c) Coarse mesh



Figure 2.5d) Selection of units in graphics window

Select Mesh>> Controls>>Face Meshing from the menu. Click on the yellow region next to Geometry under Scope in Details of Face Meshing. Select the rectangle in the graphics window. Click on the Apply button for Geometry in Details of "Face Meshing". Select Mesh>>

Controls>>Sizing from the menu and select Edge above the graphics window.



Select the upper longer horizontal edge of the rectangle, control select the lower horizontal edges and the vertical edges for a total of 5 edges. Click on Apply for the Geometry in "Details of Edge Sizing". Under Definition in "Details of Edge Sizing", select Element Size as Type, 1.0 mm for Element Size, and Hard as Behavior. Select the second Bias Type - - --- and enter 12.0 as the Bias Factor.



Figure 2.5g) Details of edge sizing for three of the horizontal edges and the vertical edges

Repeat the selection of Mesh>>Controls>>Sizing from the menu once again but this time select the upper horizontal edge to the left of the origin. Enter the same Element Size, Behavior, and Bias Factor but the first Bias Type ---- - - - . Click on Home>>Generate Mesh in the menu and select Mesh in the Outline. The finished mesh is shown in the graphics window.

D	etails of "Edge Sizin	g 2" - Sizing 🕞 🖛 📮 🗖 🗙	
Ξ	Scope		
	Scoping Method	Geometry Selection	
	Geometry	1 Edge	
Ξ	Definition		
	Suppressed	No	_
	Туре	Element Size	
	Element Size	1.0 mm	
Ξ	Advanced	·	
	Behavior	Hard	
	Capture Curvature	No	
	Capture Proximity	No	
	Bias Type		
	Bias Option	Bias Factor	
	Bias Factor	12.0	
	Reverse Bias	No Selection	

Figure 2.5h) Details of edge sizing for the remaining horizontal edge



Figure 2.5i) Details of finished mesh

Why did we create a biased mesh? The reason for using a biased mesh is that we need a finer mesh close to the wall where we have velocity gradients in the flow. We also included a finer mesh where the boundary layer starts to develop on the flat plate.

We are now going to rename the edges for the rectangle. Select the left edge of the rectangle, right click and select Create Named Selection. Enter *inlet* as the name and click on the OK button.

Repeat this step for the right vertical edge of the rectangle and enter the name *outlet*. Create a named selection for the lower longer horizontal right edge and call it *wall*. Finally, control-select the remaining three horizontal edges and name them *ideal wall*. An ideal wall is an adiabatic and frictionless wall.



Figure 2.5j) Named selections

Select File>>Export...>>Mesh>>FLUENT Input File>>Export from the menu. Select Save as type: FLUENT Input Files (*.msh). Enter "boundary-layer-mesh" as file name and click on the Save button. Select File>>Save Project from the menu. Name the project "Flat Plate Boundary Layer". Close the ANSYS Meshing window. Right click on Mesh in Project Schematic and select Update.

F. Launching ANSYS Fluent

6. We are now going to double click on Setup under Project Schematic in ANSYS Workbench to open Fluent. Launch the Dimension 2D and Double Precision solver of Fluent. Check the Double Precision. Set the number of Solver Processes equal to the number of computer cores. To check the number of physical cores, press the Ctrl + Shift + Esc keys simultaneously to open the Task Manager. Go to the Performance tab and select CPU from the left column. You'll see the number of physical cores on the bottom-right side. Click on Show More Options. Write down the location of the working directory as you will use this information later. Click on the Start button to launch ANSYS Fluent.

Why do we use double precision? Double precision will give more accurate calculations than single precision.



Fluid Flow (Fluent) Figure 2.6a) Launching Setup

Performance App history Startup Use CPU 1% 1.41 GHz 0 Memory 10.3/31.8 GB (32%) 0 Disk 0 (B:) 0% 0% Disk 1 (C:) 0% 0% Bethernet Ethernet 60 Ethernet Startup 0% GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 1 NVIDIA Quadro P4000 1 0% Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Fluent Launcher solve, and post-pr Simulate a wide range of industri purpose setup, solve, and post-pr Simulate a wide range of industri	sers Details : CPU % Utilization % Utilization 50 seconds Utilization Sp 1% 1 Processes TP 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Speed 1.41 G Threads 4261 :05 y)	H(R) Xec	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	entropy of the second s	U @ 3.8(3.79 G 1 6 rs: 12 Enable 384 KE 1.5 Mi 12.0 N
CPU 1% 1.41 GHz Memory 10.3/31.8 GB (32%) Disk 0 (B:) 0% Disk 1 (C:) 0% Ethernet Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% GPU 1 NVIDIA Quadro P4000 0% Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting	CPU % Utilization 50 seconds Utilization Sp 1% 1 Processes Th 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Inte Speed 1.41 G Inreads 4261 :05	H(R) Xeo	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	e186G CPU	U @ 3.8(3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
1% 1.41 GHz Memory 10.3/31.8 GB (32%) Disk 0 (B:) 0% Disk 1 (C:) 0% Disk 1 (C:) 0% Bethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% Ogure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher setup, solve, and post-pr Fluent.	CPU % Utilization % Utilization 50 seconds Utilization Sp 1% 1 Processes Th 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Speed 1.41 G Ihreads 4261 :05 y)	H(R) Xeo	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	speed: tets:	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 N
Memory 10.3/31.8 GB (32%) Disk 0 (B:) 0% Disk 1 (C:) 0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting	60 seconds 60 seconds Utilization Si 1% 1 Processes Th 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	e speed: rets: ret	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 MI 12.0 M
Memory 10.3/31.8 GB (32%) Disk 0 (B:) 0% Disk 1 (C:) 0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	50 seconds Utilization SI 1% 1 Processes TP 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 cr L2 cr L3 cr	speed: tets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
Disk 0 (B:) 0% Disk 1 (C:) 0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	50 seconds Utilization Sp 1% 1 Processes Th 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 cr L2 cr L3 cr	speed: tets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
Disk 0 (B:) 0% Disk 1 (C:) 0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	50 seconds Utilization Sp 1% 1 Processes Th 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	e speed: tets: tets: taization: aache: aache: aache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 Mi 12.0 M
Disk 1 (C:) O% Disk 1 (C:) O% Ethernet Ethernet S: 0 R: 0 Kbps OF: 0 Kbps	50 seconds Utilization SI 1% 1 Processes TP 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	speed: tets:	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 Mi 12.0 N
Disk 1 (C:) 0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	60 seconds Utilization SI 1% 1 Processes Tr 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 cr L2 cr L3 cr	speed: tets: tets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
0% Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 0% GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher a wide range of industripurpose setup, solve, and post-pr Fluent.	50 seconds Utilization S 1% 1 Processes TH 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 cr L2 cr L3 cr	speed: tets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
Ethernet Ethernet S: 0 R: 0 Kbps GPU 0 Intel(R) UHD Graphics P630 O% GPU 1 NVIDIA Quadro P4000 O% 1 gure 2.6b) Taskmanager F Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	60 seconds Utilization S 1% 1 Processes TH 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virtu L1 ca L2 ca L3 ca	a speed: tets: tes: cal processor talization: aache: aache: aache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 Mi 12.0 M
	60 seconds Utilization Si 1% 1 Processes TP 217 4 Up time 12:02:23:0 g Edit Only	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Core Logi 29 Virta L1 ca L2 ca L3 ca	speed: tets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 Mi 12.0 M
GPU 0 Intel(R) UHD Graphics P630 O% GPU 1 NVIDIA Quadro P4000 O% GPU 1 NVIDIA Quadro P4000 O% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	Utilization S 1% 1 Processes TP 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Speed 1.41 G Threads 4261 :05 y)	Hz Handles 2401	Base Sock Logi 29 Virtu L1 cr L2 cr L3 cr	espeed: cets: cal processor ialization: ache: ache: ache: X	3.79 G 1 6 rs: 12 Enable 384 KE 1.5 ME 12.0 M
GPU 0 1 Intel(R) UHD Graphics P630 0% GPU 1 2 NVIDIA Quadro P4000 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	1% 1 Processes TI 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	1.41 G Threads 4261 :05 y)	Hz Handles 2401	29 Virtu L1 ca L2 ca L2 ca L3 ca	xets: ss: cal processor ialization: ache: ache: ache: X	1 6 5 Enabl 384 KE 1.5 ME 12.0 M
Intel(R) UHD Graphics P630 % GPU 1 NVIDIA Quadro P4000 % 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	Processes TH 217 4 Up time 12:02:23:0 g Edit Only rial applica processing	Threads 4261 :05 y)	Handles 2401	Core Logi 29 Virtu L1 ca L2 ca L3 ca	es: cal processor ialization: aache: aache: X	ь гs: 12 Епаbl 384 КЕ 1.5 МЕ 12.0 М
GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher avide range of industri purpose setup, solve, and post-pr Fluent.	217 4 Up time 12:02:23:0 g Edit Only rial applica processing	4261 :05 y)	2401	29 Virtu L1 ca L2 ca L3 ca	alization: ache: ache: ache: X	Enablı 384 Ki 1.5 Ml 12.0 M
GPU 1 NVIDIA Quadro P4000 0% 1 gure 2.6b) Taskmanager F Fluent Launcher 2020 R1 (Setting Fluent Launcher a wide range of industri purpose setup, solve, and post-pr Fluent.	up time 12:02:23:1 g Edit Only rial applica processing	:05 y)	_		ache: ache: ache: X	384 KE 1.5 MI 12.0 N
0% 1 gure 2.6b) Taskmanager 1 Fluent Launcher 2020 R1 (Setting Fluent Launcher 2020 R1 (Setting Simulate a wide range of industripurpose setup, solve, and post-pr Fluent.	12:02:23: g Edit Only rial applica processing	y)	_		x	12.0 N
gure 2.6b) Taskmanager Fluent Launcher 2020 R1 (Setting Fluent Launcher Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	g Edit Only rial applica processing	y)	_		×	
Fluent Launcher 2020 R1 (Setting Fluent Launcher Simulate a wide range of industri purpose setup, solve, and post-pr Fluent.	g Edit Only rial applica processing	y)	_		X	
Fluent.		g capa	using bilities	the ger of ANS	neral- SYS	
Dimension	-					
Dimensi	ion					
(•) 2D						
○ 3D						
Options	5					
Doul	uble Precis	ision				
		h After	Dandi			
Uisp	piay mesn	n Aiter	Redui	ng		
📋 Do n	not show	this p	anel a	gain		
🗌 Load	Id ACT					
Parallel	l (Local I	Machi	ne)			
Solver P	Processes	c		6		
Solver 1	10003303	3	l	•	•	
Solver G	00001			0	÷.	
	GPGPUs p	per Ma	chine	U		
	GPGPUs p	oer Ma	chine	U		
V Show More Ontions - V S	GPGPUs p	oer Ma	chine	0		
✓ Show More Options ✓ S	GPGPUs p	arning	chine Reso	urces		

Figure 2.6c) Fluent Launcher

Conse	ole						Ø
		_					
ID	Hostname	Core	0.5.	PID	Vendor		
n5	ljmatsson5	6/12	Windows-x64	165764	Intel(R)	Xeon(R)	E-2186G
n4	ljmatsson5	5/12	Windows-x64	172144	Intel(R)	Xeon(R)	E-2186G
n3	ljmatsson5	4/12	Windows-x64	169316	Intel(R)	Xeon(R)	E-2186G
n2	ljmatsson5	3/12	Windows-x64	169712	Intel(R)	Xeon(R)	E-2186G
nl	ljmatsson5	2/12	Windows-x64	170632	Intel(R)	Xeon(R)	E-2186G
n0*	ljmatsson5	1/12	Windows-x64	170060	Intel(R)	Xeon(R)	E-2186G
host	ljmatsson5		Windows-x64	172612	Intel(R)	Xeon(R)	E-2186G

Figure 2.6d) Example of console printout for four cores

7. Check the scale of the mesh by selecting the Scale button under Mesh in General on the Task Page. Make sure that the Domain Extent is correct and close the Scale Mesh window.

Task Page	
General	0
Mesh	
Scale Check Report Quality	
Display Units	

Figure 2.7a) Scale Check

Scale Mesh	
Domain Extents	
Xmin (m) -0.1	Xmax (m) 1
Ymin (m) 0	Ymax (m) 0.1

Figure 2.7b) Scale Mesh

8. Double click on Models and Viscous (SST k-omega). Select Laminar as Viscous Model. Click Ok to close the window. Double click on Boundary Conditions under Setup in the Outline View. Double click on *inlet* under Zone on the Task Page. Choose Components as Velocity Specification Method and set the X-Velocity (m/s) to 5. Click on the OK button to close the window. Double click on *ideal_wall* under Zones. Check Specified Shear as Shear Condition and keep zero values for specified shear stress since an ideal wall is frictionless. Click OK to close the window.

Why did we select Laminar as Viscous Model? For the chosen free stream velocity 5 m/s the Reynolds number is less than 500,000 along the plate and the flow is therefore laminar. Turbulent from along a flat plate occurs at Reynolds numbers above 500,000.

F Viscous Model ×
Model
○ Inviscid
 Laminar
O Spalart-Allmaras (1 eqn)
🔿 k-epsilon (2 eqn)
🔿 k-omega (2 eqn)
○ Transition k-kl-omega (3 eqn)
O Transition SST (4 eqn)
 Reynolds Stress (5 eqn)
Scale-Adaptive Simulation (SAS)
\bigcirc Detached Eddy Simulation (DES)
OK Cancel Help

Figure 2.8a) Viscous model

Outline View	Task Page
Filter Text	Boundary Conditions
 Setup General General Models Materials Cell Zone Conditions Cell Zone Conditions Boundary Conditions ideal_wall (wall, id=8) inlet (velocity-inlet, id=5) 	Zone Filter Text ideal_wall inlet interior-surface_body outlet wall

Figure 2.8b) Boundary Conditions for inlet

Velocity I	nlet						×
Zone Name							
inlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocity	Specificati	on Method	Component	ts			•
	Referen	nce Frame	Absolute				•
Supersonic/	Initial Gaug	e Pressure (pascal) 0				•
		X-Velocity (m/s) 5				•
		Y-Velocity (m/s) 0				•
		C	OK Can	cel) (Help	•		

Figure 2.8c) Inlet velocity

F Wall										×
Zone Name										
ideal_wall										
Adjacent Cell	Zone									
surface_bod	у									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	Strue	cture
Station	n ary Wall	Motion Relativ	ve to Adjace	ent Cell Z	Zone					
Moving	Wall	Sho	ar Stross							
No Slip Specifie Specula	ed Shear arity Coeffi	cient	Component Component	(pascal) (pascal)	0) •) •	
O Specula	arity Coeffi Joni Stress	cient			·					

Figure 2.8d) Specified shear for ideal wall

 Double click on Methods under Solution in the Outline View. Select *Standard* for Pressure and *First Order Upwind* for Momentum. Double click on Reference Values under Setup in the Outline View. Select *Compute from inlet* on the Task Page.

Outline View	<	Task Page	<
Filter Text		Solution Methods	(?)
 Setup ☑ General ④ ☑ Models ④ ☑ Materials ④ ☑ Cell Zone Conditions ④ ☑ Boundary Conditions ④ ☑ Boundary Conditions ☑ Mesh Interfaces ☑ Dynamic Mesh ☑ Reference Values ④ ☑ Reference Frames 		Pressure-Velocity Coupling Scheme Coupled Spatial Discretization Gradient Least Squares Cell Based Pressure Standard	
Solution Methods		Momentum First Order Upwind	•

Figure 2.9a) Solution Methods Task Page

Why do we use First Order Upwind method for Spatial Discretization of Momentum? The First Order Upwind method is generally less accurate but converges better than the Second Order Upwind method. It is common practice to start with the First Order Upwind method at the beginning of calculations and continue with the Second Order Upwind method.

Task Pag	e		
Referenc	æ Values		?
Compute f	from		
linlet			
	Reference Values		
	Area (m2)	1	
	Density (kg/m3)	1.225	
	Depth (m)	1	
	Enthalpy (j/kg)	0	
	Length (m)	1	
	Pressure (pascal)	0	
	Temperature (k)	288.16	
	Velocity (m/s)	5	
	Viscosity (kg/m-s)	1.7894e-05	
	Ratio of Specific Heats	1.4	
Reference	Zone		
			-

Figure 2.9b) Reference values

10. Double click on Initialization under Solution in the Outline View, select *Standard Initialization*, select *Compute from inlet*, and click on the *Initialize* button.

Outline View <	Task Page <
Filter Text	Solution Initialization
 General Models Materials Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Keference Frames 	Hybrid Initialization Standard Initialization Compute from inlet Reference Frame Image: Compute the compute of the co
 Named Expressions Solution Methods Controls Report Definitions Monitors Cell Registers Initialization Calculation Activities 	Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 5 Y Velocity (m/s) 0

Figure 2.10 Solution Initialization

11. Double click on Monitors under Solution in the Outline View. Double click on Residual under Monitors and enter 1e-9 as *Absolute Criteria* for all Residuals. Click on the OK button to close the window.



Options	Equations			
✓ Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria
✓ Plot	continuity	v	✓	1e-9
Window	x-velocity	v	-	1e-9
Curves Axes	y-velocity	v	\checkmark	1e-9
Iterations to Plot				
terations to Store				
1000	Convergence	Conditions		
	Show Advan	ced Options		

Figure 2.11b) Residual Monitors settings

Why did we set the Absolute Criteria to 1e-9? Generally, the lower the absolute criteria, the longer time the calculation will take and give a more exact solution. We see in Figure 2.12b) that the *x*-velocity and *y*-velocity equations have lower residuals than the continuity equation. The slopes of the residual curves for all three equations are about the same with a sharp downward trend.

12. Double click on Run Calculation under Solution and enter 5000 for *Number of Iterations*. Click on the *Calculate* button.

Outline View	< Task Page
Filter Text	Run Calculation
 Setup General ♥ Models ♥ Materials ♥ Cell Zone Conditions ♥ Boundary Conditions ♥ Delta Content Conditions ♥ Anterials ♥ Boundary Conditions ♥ Delta Content Conditions ♥ Anterials <li< td=""><td>Check Case Update Dynamic Mesh Pseudo Transient Settings Fluid Time Scale Time Step Method Time Scale Factor Automatic 1 Length Scale Method Verbosity Conservative 0 Parameters Number of Iterations Reporting Interval 1 5000 Profile Update Interval 1</td></li<>	Check Case Update Dynamic Mesh Pseudo Transient Settings Fluid Time Scale Time Step Method Time Scale Factor Automatic 1 Length Scale Method Verbosity Conservative 0 Parameters Number of Iterations Reporting Interval 1 5000 Profile Update Interval 1
E Report Files └── Report Plots	Solution Processing
Convergence Conditions	Data Sampling for Steady Statistics
 Initialization Calculation Activities 	Data File Quantities
Run Calculation Results	Solution Advancement
Surfaces	Calculate

Figure 2.12a) Running the calculations



Figure 2.12b) Scaled Residuals

G. Post-Processing

13. Select the Results tab in the menu and select Create>>Line/Rake under Surface. Enter 0.2 for x0 (m), 0.2 for x1 (m), 0 for y0 (m), and 0.02 m for y1 (m). Enter x=0.2m for the New Surface Name and click on Create. Repeat this step three more times and create vertical lines at x = 0.4m (length 0.04 m), x=0.6m (length 0.06 m), and x=0.8m (length 0.08 m). Close the window.

	<u>F</u> ile	Doma
	Surface	
	🕂 Create 🖕	\odot
	Zone	
	Partition	
	Imprint	
C	Point	
	Line/Rake	

Figure 2.13a) Selecting Line/Rake from the Post-processing menu

▶ Line/Rake Surface ×	■ Line/Rake Surface ×
New Surface Name	New Surface Name
x=0.2m	x=0.4m
Options Number of Points	Options Number of Points
Line Type 10	Line Type 10
Reset	Reset
End Points	End Points
x0 (m) 0.2 x1 (m) 0.2	x0 (m) 0.4 x1 (m) 0.4
y0 (m) 0 y1 (m) 0.02	y0 (m) 0 y1 (m) 0.04
z0 (m) 0 z1 (m) 0	z0 (m) 0 z1 (m) 0
Select Points with Mouse	Select Points with Mouse
Create Close Help	Create Close Help

Figure 2.13b) Line/Rake Surfaces at x=0.2 m and x=0.4m

14. Double click on Plots and XY Plot under Results in the Outline View. Uncheck Position on X Axis under Options and check Position on Y Axis. Set Plot Direction for X to 0 and 1 for Y. Select Velocity... and X Velocity as X Axis Function. Select the four surfaces x=0.2m, x=0.4m, x=0.6m, and x=0.8m.



ky-plot-1		
Arrow Process Options ✓ Node Values Position on X Axis ✓ Position on Y Axis Write to File Order Points	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Direction Vector X Axis Function Velocity X Velocity Surfaces Filter Text outlet wall x=0.2m x=0.4m x=0.6m x=0.8m
		New Surface 💂

Figure 2.14b) Settings for Solution XY Plot

15. Click on the *Axes* button in the Solution XY Plot window. Select the X Axis, uncheck Auto Range under Options, enter 6 for Maximum Range, select general Type under Number Format and set Precision to 0. Click on the Apply button. Select the Y Axis, uncheck the Auto Range, enter 0.01 for Maximum Range, select general Type under Number Format, and click on the Apply button. Close the Axes window.

Axis	Number Format	Major Rules
• x	Туре	Color
О Ү	general 🔹	foreground
ahol	Precision	Weight
	0	1
Options	Range	Minor Rules
Log	Minimum	Color
Auto Range	0	dark gray
Major Rules	Maximum	Weight
Minen Dules	6	
Figure 2.15a) Settings for	Apply Close He	lp
Figure 2.15a) Settings for Axes - Solution XY Plot	Apply Close He X Axes	lp Major Rules
 igure 2.15a) Settings for Axes - Solution XY Plot Axis X 	Apply Close He X Axes Number Format Type	I Ip Major Rules Color
 Minor Rules Figure 2.15a) Settings for Axes - Solution XY Plot Axis X Y 	Apply Close He X Axes Number Format Type general	Major Rules Color foreground
 igure 2.15a) Settings for Axes - Solution XY Plot Axis X Y 	Apply Close He X Axes Number Format Type general • Precision	I Major Rules Color foreground Weight
igure 2.15a) Settings for Axes - Solution XY Plot Xis X Y	Apply Close He X Axes Number Format Type general • Precision 2 •	Major Rules Color foreground Weight 1
igure 2.15a) Settings for Axes - Solution XY Plot Axis X Y abel	Apply Close He X Axes Number Format Type general • Precision 2 • Range	Major Rules Color foreground Weight 1 Minor Rules
 Minor Rules Figure 2.15a) Settings for 1 Axes - Solution XY Plot Axis X Y Dptions Log 	Apply Close He X Axes Number Format Type general Precision 2 2 Range Minimum	I Major Rules Color foreground Weight 1 Minor Rules Color
 igure 2.15a) Settings for Axes - Solution XY Plot Axis X Y Dptions Log Auto Range 	Apply Close He X Axes Number Format Type general • Precision 2 • Range Minimum 0	Major Rules Color foreground Weight 1 Minor Rules Color dark gray
Figure 2.15a) Settings for Axes - Solution XY Plot Axis X Y Label Dptions Log Auto Range Major Rules	Apply Close He X Axes Number Format Type general • Precision 2 • Range Minimum 0 Maximum	I Major Rules Color foreground Weight 1 Minor Rules Color dark gray Weight

Figure 2.15b) Settings for Y Axes

16. Click on the Curves... button in the Solution XY Plot window. Select the first pattern under Line Style for Curve # 0. Select no Symbol for Marker Style and click on the Apply button.

Next select Curve # 1, select the next available Pattern for Line Style, no Symbol for Marker Style, and click on the Apply button. Continue this pattern of selection with the next two curves # 2 and # 3. Close the Curves – Solution XY Plot window. Click on the Save/Plot button in the Solution XY Plot window and close this window.

Curve #	Line Style	Marker Style
0	Pattern	Symbol
Sample	v	
	Color	Color
	foreground 🔻	foreground
	Weight	Size
	2	0.3

Figure 2.16a) Settings for curve # 0



Figure 2.16b) Velocity profiles for a laminar boundary layer on a flat plate

Select the User Defined tab in the menu and Custom. Select a specific Operand Field Functions from the drop-down menu by selecting Mesh... and Y-Coordinate. Click on Select and enter the definition as shown in Figure 2.16e). You need to select Mesh... and X-Coordinate to complete the definition of the field function. Enter eta as New Function Name, click on Define and close the window.

Repeat this step to create another custom field function. This time, we select *Velocity*... and *X Velocity* as *Field Functions* and click on *Select*. Complete the *Definition* as shown in Figure 2.16f) and enter *u-divided-by-freestream-velocity* as *New Function Name*, click on *Define* and close the window.





E Custom Field Function Calculator

Definition

Select Operand Field Functions fro	m
Field Functions	
Mesh	•
Y-Coordinate	-
Select	

Figure 2.16d) Operand Field Function

+	-	x	\bigcap	y^x	ABS	Select Operand Field Functions from
INV	sin	cos	tan		log10	Field Functions
0		2	3	4	SQRT	Mesn
5	6	7	8	9	CE/C	X-Coordinate
($\overline{)}$	PI	e		DEL	Select

New Function Name eta

Define Manage... Close

se Help

Figure 2.16e) Custom field function for self-similar coordinate

Why did we create a self-similar coordinate? It turns out that by using a self-similar coordinate, the velocity profiles at different streamwise positions will collapse on one self-similar velocity profile that is independent of the streamwise location.

x / 5.1						
+ INV 0 5 (- sin 1 6	X Cos 2 7 PI	/ tan 3 8 e	y^x In 9	ABS log10 SQRT CE/C DEL	Select Operand Field Functions from Field Functions Velocity • X Velocity Select
_				<u> </u>		

Figure 2.16f) Custom field function for non-dimensional velocity

17. Double click on Plots and XY Plot under Results in the Outline View. Set X to 0 and Y to 1 as Plot Direction. Uncheck *Position on X Axis* and *Position on Y Axis* under *Options*. Select *Custom Field Functions* and *eta* for *Y Axis Function* and select *Custom Field Functions* and *u-divided-by-freestream-velocity* for *X Axis Function*.

Define Manage... Close Help

Place the file "blasius.dat" in your working directory. This file can be downloaded from *sdcpublications.com*. See Figure 2.19 for the Mathematica code that can be used to generate the theoretical Blasius velocity profile for laminar boundary layer flow over a flat plate. As an example, in this case the working directory is C:\Users\jmatsson. Click on Load File. Select Files of type: All Files (*) and select the file "blasius.dat" from your working directory. Select the four surfaces x=0.2m, x=0.4m, x=0.6m, x=0.8m and the loaded file Theory.

Options	Plot Direction	Y Axis Function
Node Values Position on X Axis Position on Y Axis Write to File Order Points	X 0 Y 1 Z 0	Custom Field Functions eta X Axis Function Custom Field Functions
File Data [1/1] Theory	Load File	u-divided-by-freestream-velocity
		x=0.2m x=0.4m x=0.6m x=0.8m

Figure 2.17a) Solution XY Plot for self-similar velocity profiles

Click on the Axes button. Select *Y Axis* in Axes-Solution XY Plot window and uncheck *Auto Range*. Set the *Minimum Range* to 0 and *Maximum Range* to 10. Set the Type to float and Precision to 0 under Number Format. Enter the Label as *eta* and click on *Apply*.

Select *X* Axis in Axes-Solution XY Plot window and enter the Label to u/U. Check the box for *Auto Range*. Set the Type to float and Precision to 1 under Number Format. Click on Apply and close the window.

Click on the Curves... button in the Solution XY Plot window. Select the first pattern under Line Style for Curve # 0. Select no Symbol for Marker Style and click on the Apply button.

Next select Curve # 1, select the next available Pattern for Line Style, no Symbol for Marker Style, and click on the Apply button. Continue this pattern of selection with the next two curves # 2 and # 3. Close the Curves – Solution XY Plot window. Click on the Save/Plot button in the Solution XY Plot window and close this window.

Axis	Number Format	Major Rules
Οx	Туре	Color
• Y	float 💌	foreground
I ahel	Precision	Weight
eta	0	1
Options	Range	Minor Rules
□.	Minimum	Color
Log		
Log Auto Range	0	dark gray
Log Auto Range Major Rules	0 Maximum	dark gray Weight

Figure 2.17b) Axes - Solution XY Plot window settings for Y Axis

Axis	Number Format	Major Rules
• x	Туре	Color
⊖ Y	float 💌	foreground
Label u/U	Precision	Weight 1
Ontions	Range	Minor Rules
options		Finiter Hunter
	Minimum	Color
Log Auto Range	Minimum	Color dark gray
 Log ✓ Auto Range Major Rules 	Minimum 0 Maximum	Color dark gray Weight

Figure 2.17c) Axes – Solution XY Plot window settings for X Axis

Axes - Solution XY Plot



Figure 2.17d) Self-similar velocity profiles for a laminar boundary layer on a flat plate

Select the User Defined tab in the menu and Custom. Select a specific Operand Field Functions from the drop-down menu by selecting Mesh... and X-Coordinate. Click on Select and enter the definition as shown in Figure 2.17e). Enter *re-x* as New Function Name, click on Define and Close the window.

Cust Definitio	tom Field n / 0.0000	Function	n Calculi	ator		
+ INV 0 5 (New Fur	- sin 1 6) 1	X Cos 2 7 PI me re->	/ tan 3 8 e	y^x In 4 9	ABS log10 SQRT CE/C DEL	Select Operand Field Functions from Field Functions Mesh X-Coordinate Select
					Define	Manage Close Help

Figure 2.17e) Custom field function for Reynolds number

18. Double click on Plots and XY Plot under Results in the Outline View. Set X to 0 and Y to 1 under Plot Direction. Uncheck Position on X Axis and Position on Y Axis under Options. Select *Wall Fluxes* and *Skin Friction Coefficient* for Y Axis Function and select Custom Field Functions and *re-x* for X Axis Function.

Place the file "*Theoretical Skin Friction Coefficient*" in your working directory. Click on Load File. Select Files of type: All Files (*) and select the file "*Theoretical Skin Friction Coefficient*". Select *wall* under Surfaces and the loaded file Skin Friction under File Data. Click on the Axes button.

KY Plot Name		
xy-plot-3		
Options	Plot Direction Y Axis Function	
✓ Node Values	X 0 Wall Fluxes	•]
Position on X Axis	Y 1	ant T
Position on Y Axis	Z 0	enc •
Write to File	X Axis Function	
Order Points	Custom Field Functio	ns 🔻
	re-x	•
File Data [1/1]		
Skin friction	Surfaces Filter Text	
	Free Data outlet	A
	wall	
	x=0.2m	
	x=0.4m	_
	x=0.8m	
		¥
	New Surface 🚽	

Figure 2.18a) Solution XY Plot for skin friction coefficient

Check the X Axis, check the box for Log under Options, enter Re-x as Label, uncheck Auto Range under Options, set Minimum to 100 and Maximum to 1000000. Set Type to float and Precision to 0 under Number Format and click on Apply.

Check the Y Axis, check the box for Log under Options, enter Cf-x as Label, uncheck Auto Range, set Minimum to 0.001 and Maximum to 0.1, set Type to float, Precision to 3 and click on Apply. Close the window. Click on Save/Plot in the Solution XY Plot window.

Click on the Curves... button in the Solution XY Plot window. Select the first pattern under Line Style for Curve # 0. Select no Symbol for Marker Style and click on the Apply button.

Next select Curve # 1, select the next available Pattern for Line Style, no Symbol for Marker Style, and click on the Apply button. Close the Curves – Solution XY Plot window. Click on the Save/Plot button in the Solution XY Plot window and close this window.

Axis	Number Format	Major Rules
• x	Туре	Color
<u>О</u> Ү	float 🔹	foreground
Label	Precision	Weight
Re-x		
Options	Range	Minor Rules
✓ Log	Minimum	Color
Auto Range	100	dark gray
Major Rules	Maximum	Weight
Minor Rules	100000	1



Figure 2.18b) Axes - Solution XY Plot window settings for X Axis

Axis	Number Format	Major Rules
○x	Туре	Color
• Y	float 💌	foreground
Label	Precision	Weight
Cf-x	3	1
Options	Range	Minor Rules
	Minimum	Color
✓ Log		
✓ Log Auto Range	0.001	dark gray
 Log Auto Range Major Rules 	0.001 Maximum	dark gray Weight

Figure 2.18c) Axes – Solution XY Plot window settings for Y Axis



Figure 2.18d) Comparison between ANSYS Fluent and theoretical skin friction coefficient (dashed line) for laminar boundary layer flow on a flat plate

H. Theory

19. In this chapter we have compared ANSYS Fluent velocity profiles with the theoretical Blasius velocity profile for laminar flow on a flat plate. We transformed the wall normal coordinate to a similarity coordinate for comparison of profiles at different streamwise locations. The similarity coordinate is defined by

$$\eta = y \sqrt{\frac{U}{vx}} \tag{2.1}$$

where y (m) is the wall normal coordinate, U (m/s) is the free stream velocity, x (m) is the distance from the streamwise origin of the wall and v (m²/s) is the kinematic viscosity of the fluid.

We also used the non-dimensional streamwise velocity u/U where u is the dimensional velocity profile. u/U was plotted versus η for ANSYS Fluent velocity profiles in comparison with the Blasius theoretical profile and they all collapsed on the same curve as per definition of self-similarity. The Blasius boundary layer equation is given by

$$f'''(\eta) + \frac{1}{2}f(\eta)f''(\eta) = 0$$
(2.2)

with the following boundary conditions

$$f(0) = f'(0) = 0, f'(\infty) = 0$$
(2.3)

The Reynolds number for the flow on a flat plat is defined as

$$Re_x = \frac{Ux}{v} \tag{2.4}$$

The boundary layer thickness δ is defined as the distance from the wall to the location where the velocity in the boundary layer has reached 99% of the free stream value. For a laminar boundary-layer we have the following theoretical expression for the variation of the boundary layer thickness with streamwise distance x and Reynolds number Re_x .

$$\delta = \frac{4.91x}{\sqrt{Re_x}} \tag{2.5}$$

The corresponding expression for the boundary layer thickness in a turbulent boundary layer is given by

$$\delta = \frac{0.16x}{Re_x^{1/7}}$$
(2.6)

The local skin friction coefficient is defined as the local wall shear stress divided by dynamic pressure.

$$C_{f,x} = \frac{\tau_w}{\frac{1}{2}\rho U^2}$$
(2.7)

The theoretical local friction coefficient for laminar flow is determined by

$$C_{f,x} = \frac{0.664}{\sqrt{Re_x}} \qquad Re_x < 5 \cdot 10^5 \tag{2.8}$$

and for turbulent flow we have the following relation

$$C_{f,x} = \frac{0.027}{Re_x^{1/7}} \qquad 5 \cdot 10^5 < Re_x < 10^7 \tag{2.9}$$

Figure 2.19 Mathematica code for theoretical Blasius laminar boundary layer

I. References

- 1. Çengel, Y. A., and Cimbala J.M., Fluid Mechanics Fundamentals and Applications, 1st Edition, McGraw-Hill, 2006.
- 2. Richards, S., Cimbala, J.M., Martin, K., ANSYS Workbench Tutorial Boundary Layer on a Flat Plate, Penn State University, 18 May 2010 Revision.
- 3. Schlichting, H., and Gersten, K., Boundary Layer Theory, 8th Revised and Enlarged Edition, Springer, 2001.
- 4. White, F. M., Fluid Mechanics, 4th Edition, McGraw-Hill, 1999.

J. Exercises

2.1 Use the results from the ANSYS Fluent simulation in this chapter to determine the boundary layer thickness at the streamwise positions as shown in the table below. Fill in the missing information in the table. U_{δ} is the velocity of the boundary layer at the distance from the wall equal to the boundary layer thickness and U is the free stream velocity.

<i>x</i> (m)	δ (mm)	δ (mm)	Percent	U_{δ}	U	ν	Re_{x}
	Fluent	Theory	Difference		(m/s)	(m^{2}/s)	
				(m/s)			
0.2						.0000146	
0.4						.0000146	
0.6						.0000146	
0.8						.0000146	

Table 2.1 Comparison between Fluent and theory for boundary layer thickness

- 2.2 Change the element size to 2 mm for the mesh and compare the results in XY Plots of the skin friction coefficient versus Reynolds number with the element size 1 mm that was used in this chapter. Compare your results with theory.
- 2.3 Change the free stream velocity to 3 m/s and create an XY Plot including velocity profiles at x = 0.1, 0.3, 0.5, 0.7 and 0.9 m. Create another XY Plot with self-similar velocity profiles for this lower free stream velocity and create an XY Plot for the skin friction coefficient versus Reynolds number.
- 2.4 Use the results from the ANSYS Fluent simulation in Exercise 2.3 to determine the boundary layer thickness at the streamwise positions as shown in the table below. Fill in the missing information in the table. U_{δ} is the velocity of the boundary layer at the distance from the wall equal to the boundary layer thickness and U is the free stream velocity.

<i>x</i> (m)	δ (mm)	δ (mm)	Percent	U_{δ}	U	ν	Re _x
	Fluent	Theory	Difference		(m/s)	(m^{2}/s)	
				(m/s)			
0.1						.0000146	
0.2						.0000146	
0.5						.0000146	
0.7						.0000146	
0.9						.0000146	

Table 2.2 Comparison between Fluent and theory for boundary layer thickness

K. Notes