An Introduction to COSMOSFloWorks 2008



John E. Matsson, Ph.D.



Schroff Development Corporation www.schroff.com

Better Textbooks. Lower Prices.

Chapter 2 Flat Plate Boundary Layer

Objectives

- Creating the SolidWorks part needed for the COSMOSFloWorks simulation
- Setting up FloWorks projects for internal flow
- Setting up a two-dimensional flow condition
- Initializing the mesh
- Selecting boundary conditions
- Inserting global goals, point goals and equation goals for the calculations
- Running the calculations
- Using Cut Plots to visualize the resulting flow field
- Use of XY Plots for velocity profiles, boundary layer thickness, displacement thickness, momentum thickness and friction coefficients
- Use of Excel templates for XY Plots
- Comparison of FloWorks results with theories and empirical data
- Cloning of the project

Problem Description

In this chapter, we will use COSMOSFloWorks to study the two-dimensional laminar and turbulent flow on a flat plate and compare with the theoretical Blasius boundary layer solution and empirical results. The inlet velocity for the 1 m long plate is 5 m/s and we will be using air as the fluid for laminar calculations and water to get a higher Reynolds number for turbulent boundary layer calculations. We will determine the velocity profiles and plot the profiles using the well known boundary layer similarity coordinate. The variation of boundary layer thickness, displacement thickness, momentum thickness and the local friction coefficient will also be determined. We will start by creating the part needed for this simulation, see figure 2.0.



Figure 2.0 SolidWorks model for flat plate boundary layer study

Creating the SolidWorks Part

 Start by creating a new part in SolidWorks: select File>>New and click on the OK button in the New SolidWorks Document window. Click on Front Plane in the FeatureManager design tree and select Front from the View Orientation drop down menu in the graphics window.



Figure 2.1a) Selection of front plane



Figure 2.1b) Selection of front view

2. Click on **Corner Rectangle** from the **Sketch** tools.



Figure 2.2 Selecting a sketch tool

3. Click to the left and below the origin in the graphics window and drag the rectangle to the right and upward. Fill in the parameters for the rectangle: 1000 mm wide and 100 mm high. Close the Rectangle dialog box by clicking on
Right click in the graphics window and select Zoom/Pan/Rotate>>
Zoom to Fit.

Para	meters 🔗
×	-100.00
•	0.00
×	-100.00
۲	100.00
×	1000.00
•	100.00
×	1000.00
•	0.00

Rectangle	?
v	
Close Dialog ype	~

Figure 2.3a) Parameter settings for the rectangle

Figure 2.3b) Closing the dialog box

	Zoom/Pan/Rotate	F	0	Zoom to Fit
Figu	re 2.3c) Zooming in the	g	raph	ics window

4. Repeat steps 2 and 3 but create a larger rectangle outside of the first rectangle. Dimensions are shown in Figure 2.4.



Figure 2.4 Dimensions of second larger rectangle

5. Click on the Features tab and select Extruded Boss/Base. Check the box for ☑ Direction 2 and click ✓ OK to exit the Extrude Property Manager.



Figure 2.5a) Selection of extrusion feature



Figure 2.5b) Closing the property manager

6. Select **Front** from the **View Orientation** drop down menu in the graphics window. Click on **Front Plane** in the **FeatureManager design tree.** Click on the **Sketch** tab and select the **Line** sketch tool.



Figure 2.6 Selection of the line sketch tool

7. Draw a vertical line in the Y-direction starting at the lower inner surface of the sketch. Set the
 Parameters and Additional Parameters to the values shown in the figure. Close the dialog



Figure 2.7 Parameters for vertical line

8. Repeat step 7 three more times and add three more vertical lines to the sketch, the second line at X = 400 mm with a length of 40 mm, the third line at X = 600 mm with a length of 60 mm and the fourth line at X = 800 mm with a length of 80 mm. These lines will be used to plot the boundary layer velocity profiles at different streamwise positions along the flat plate. Save the SolidWorks part with the following name: Flat Plate Boundary Layer Study. Rename the newly created sketch in the FeatureManager design tree, see figure 2.8b).



Figure 2.8a) Four vertical lines at different locations and with different lengths



Figure 2.8b) Renaming the sketch for boundary layer velocity profiles

9. Repeat step 6 and draw a horizontal line in the X-direction starting at the origin of the lower inner surface of the sketch. Set the **Parameters** and **Additional Parameters** to the values shown in the figure and close the dialog ✓. Rename the sketch in the **FeatureManager design tree** and call it x = 0 - 0.9 m.

Parameters 🕆	
900.00	
0.00°	
Additional Parameters	
🖌 0.00	
.00	
x 900.00	t t-
0.00	
Δx 900.00	
ΔY 0.00	

Figure 2.9 Adding a line in the X-direction

10. Next, we will create a split line. Repeat step 6 once again but this time select the **Top Plane** and draw a line in the Z-direction through the origin of the lower inner surface of the sketch. Set the **Parameters** and **Additional Parameters** to the values shown in the figure and close the dialog

Para	meters	*
1	20.00	;
Δ	90.00°	
Addit	ional Parameters	*
.∕x	0.00	}
4	-10.00	
/x	0.00	}
1	10.00	
Δx	0.00	
Δγ	20.00	×

Figure 2.10 Drawing a line in the Z-direction

Select Insert>>Curve>>Split Line... from the SolidWorks menu. For Faces to Split, select the surface where you have drawn your split line, see figure 2.11b). Close the dialog ✓. You have now finished the part for the flat plate boundary layer. Select File>>Save from the SolidWorks menu.



Figure 2.11a) Creating a split line

🖸 Split Line 🛛 📍 ?	
🗸 🗙	
Type of Split 🔅	
🚫 Silhouette	- 1
 Projection 	_
Intersection	
Selections	<u> </u>
Current Sketch.	
Face<1>	
Faces to Split	
	×
Single direction	
Reverse direction	

Figure 2.11b) Selection of surface for the split line



Figure 2.11c) Finished part for flat plate boundary layer

Setting Up the FloWorks Project

12. If FloWorks is not available in the menu, you have to add it from SolidWorks menu:

Tools>>Add Ins... and check the corresponding COSMOSFloWorks box. Select FloWorks>>Project>>Wizard to create a new FloWorks project. Create a new figuration named "Flat Plate Boundary Layer Study". Click on the Next > button. Select the default SI (m-kg-s) unit system and click on the Next> button once again.

FloW	Vorks Window Help	2	•	> -	8	ð •	9	- 🗟 -
	Project				New			
	Insert		•	, Ali	Wizard			

Figure 2.12a) Starting a new FloWorks project

Wizard - Project Configuration			? 🛛
Ele Edit View Insert Loois Flow	Configuration Create new Use current Configuration name: Current configuration: 	Flat Plate Boundary Layer Study Default)

Figure 2.12b) Creating a name for the project

13. Use the default Internal Analysis type.

Wizard - Analysis Type				?×
	Analysis type Internal External	Consider clo	sed cavities le cavities without flow conditions e internal space	>
	Physical Features		Value	
	Heat conduction in	solids		
	Radiation			
	Time-dependent			
	Gravity			
	Rotation			

Figure 2.13 Excluding cavities without flow conditions

14. Select Air from the Gases and add it as Project Fluid. Select Laminar Only from the Flow Type drop down menu. Click on the Next > button. Use the default Wall Conditions and click on the Next > button. Insert 5 m/s for Velocity in X-direction as Initial Condition and click on the Next > button. Slide the Result resolution to 8. Click on the Finish button.

Wizard - Default Fluid				? 2
	Fluids	Path	~	New
and the second	🖃 Gases			
	Acetone	Pre-Defined		
	Ammonia	Pre-Defined		
	Argon	Pre-Defined		
	Butane	Pre-Defined		
	Carbon dioxide	Pre-Defined		
	Chlorine	Pre-Defined		
	Ethane	Pre-Defined		
and the second	Ethanol	Pre-Defined		
	Ethylene	Pre-Defined		
	Fluorine	Pre-Defined	~	Add
and the second	Project Fluids	Default Fluid		Remove
	Air (Gases)			
The second second	Flow Characteristic	Value		
	Flow type	Laminar Only	~	
	Humidity			1
				0
	< Back	Next >	Cancel	Help

Figure 2.14 Selection of fluid for the project and flow type

15. Select FloWorks>>Computational Domain.... Click on the Boundary Condition tab and select XY-Plane Flow from the drop down menu. Click on the OK button to exit the Computational Domain window.

FloW	orks Window Help 🖉] - [
	Project	•
	Insert	•
1	General Settings	
	Units	
	Computational Domain	

Figure 2.15a) Modifying the computational domain

			Size	
Computational Domain				
Size Boundary Co	ndition Color Setting		× max:	
2D plane flow:	XY - Plane Flow	ОК	Y min:	
	None		Y max:	
At X min:	YZ - Plane Flow XZ - Plane Flow	Lancel	Z min:	
At X max:	XY - Plane Flow Default	Help	Z max:	

 Computational Domain

 Size
 Boundary Condition
 Color Setting

 X min:
 -0.12 m
 •

 X max:
 1.02 m
 •

 Y min:
 -0.02 m
 •

 Y max:
 0.12 m
 •

 Z min:
 -0.002 m
 •

 Z max:
 0.002 m
 •

Figure 2.15b) Selecting two dimensional flow condition

Figure 2.15c) Size of domain

16. Select FloWorks>>Initial Mesh.... Uncheck the Automatic setting box at the bottom of the window. Change both the Number of cells per X: to 300 and the Number of cells per Y: to 200. Click on the OK button to exit the Initial Mesh window.

FloW	orks Window Help 🖉 🗋	- (
	Project	۲
	Insert	•
	General Settings	
	Units	
	Computational Domain	
	Initial Mesh	

Figure 2.16a) Modifying the initial mesh

Initial Mesh			? 🗙
Basic Mesh Solid/Fluid Interface Refining Cells Narrow Chann	nels		
∩ Number of cells			ОК
Number of cells per X:	300	*	Cancel
Number of cells per Y:	200	•	Help
Number of cells per Z:	1	×	

Figure 2.16b) Changing the number of cells in both directions

Selecting Boundary Conditions

17. Select the COSMOSFloWorks analysis tree tab, open the Input Data folder by clicking on the plus sign next to it and right click on Boundary Conditions. Select Insert Boundary Condition.... Right click in the graphics window and select Zoom/Pan/Rotate>>Rotate View. Click and drag the mouse so that the inner surface of the left boundary is visible. Right click again and choose Select. Click on the left inflow boundary surface. Select Inlet Velocity in the Type portion of the Boundary Condition window and set the velocity to 5 m/s in the Flow Parameters window. Click OK ot exit the window. Right click in the graphics window and select Zoom/Pan/Rotate>> 2000 To Area and select an area around the left boundary.

🤏 👕 😫 🔶 🏧	
🍕 Flat Plate Boundary Layer	
🖕 🦉 Input Data	
Computational Domain	
🛁 🔂 Fluid Subdomains	
Sent a second	
Insert Boundary Cond	ition

Zoom/Pa	n/Rotate	Q	Zoom to Fit
		Q	Zoom to Area
		Q	Zoom In/Out
Cancer Class Cal	actions	3	Rotate View

Figure 2.17a) Inserting boundary condition

Figure 2.17b) Modifying the view

🛅 Boundary Condition ?	
🗸 🗙	
Selection 🕆	
Face<1>	
🚑 Face Coordinate System	
Reference axis: 🗙 💌	, v
Туре 🕆	Y
Inlet Mass Flow Inlet Volume Flow Inlet Velocity Inlet Mach Number	
Outlet Mass Flow Outlet Volume Flow Outlet Velocity	×
Flow Parameters	
V 5 m/s 5 fx Fully developed flow	3

Figure 2.17c) Including a velocity boundary condition on the inflow



Figure 2.17d) Inlet velocity boundary condition indicated by arrows

18. Red arrows pointing in the right direction appears indicating the inlet velocity boundary condition, see figure 2.17d). Right click in the graphics window and select Zoom/Pan/Rotate>> 200 Zoom to Fit. Right click again in the graphics window and select 200 Rotate View once again to rotate the part so that the inner right surface is visible in the graphics window. Right click and click on Select. Right click on Boundary Conditions in the COSMOSFloWorks analysis tree and select Insert Boundary Condition.... Click on the end

surface on the outlet boundary. Click on the **Pressure Openings** button in the **Type** portion of the **Boundary Condition** window and select **Static Pressure**. If you zoom in on the outlet boundary you will see blue arrows indicating the static pressure boundary condition, see figure 2.18b). Click OK 🖋 to exit the window.



Figure 2.18a) Selection of static pressure as boundary condition at the outlet of the flow region



Figure 2.18b) Outlet static pressure boundary condition

19. Enter the following boundary conditions: Ideal Wall for the upper wall and lower wall at the inflow region, see figures 2.19a) and 2.19b). These will be adiabatic and frictionless walls.

🗂 Boundary Condition ?	
Selection 🕆	
Face<1>	7
	∮ [∠]
Face Coordinate System	Y I
Reference axis: X	
Туре 🕆	
	×
Real Wall Ideal Wall	

Figure 2.19a) Ideal wall boundary condition for upper wall

📑 Boundary Condition ?	
Selection	
Face<1>	
	<u>X</u>
Ĵ ^v _z Face Coordinate System	
Reference axis: X V	• • · · · ·
Type 🔅	
	∮ •Z

Figure 2.19b) Ideal wall boundary condition for lower wall at the inflow region

20. The last boundary condition will be in the form of a real wall. We will study the development of the boundary layer on this wall.

📑 Boundary Condition 🤰		
✓ X		
Selection 🕆		
Face<1>		
F		Y
Face Coordinate System		<u>~</u>
Face Coordinate System Reference axis: X	Y Y	2 <u> </u>
↓ Face Coordinate System Reference axis: × Type ≈	Y X	2

Figure 2.20 Real wall boundary condition for the flat plate

Inserting Global Goals

21. Right click on Goals in the COSMOSFloWorks analysis tree and select Insert Global Goals.... Select X – Component of Shear Force as a global goal. Exit the Global Goals window. Right click on Goals in the COSMOSFloWorks analysis tree and select Insert Point

Goals.... Click on the **Point Coordinates** button. Enter **0.2 m** for X coordinate and **0.02 m**

for Y coordinate and click on the **Add Point** button \blacksquare . Add three more points with the coordinates shown in figure 2.21e). Check the **Value** box for **X-Component of Velocity**. Exit the **Point Goals** window. Rename the point goals as shown in figure 2.21g). Right click on **Goals** in the **COSMOSFloWorks analysis tree** and select **Insert Equation Goal...** Click on the **X-Component of Velocity at** x = 0.2 m goal in the **COSMOSFloWorks analysis tree**, multiply by 0.2 and divide by the kinematic viscosity of air at room temperature (1.516E-5) to get an expression for the Reynolds number in the **Equation Goal** window, see figure 2.21h). Select **No units** from the dimensionality drop down menu. Rename the equation goal to **Reynolds number at** x = 0.2 m. Insert three more equation goals corresponding to the Reynolds numbers at the three other *X* locations. For a definition of the Reynolds number, see page 2-19.







Figure 2.21c) Inserting point goals

🍍 Global Go	als	-	-	_	-		?		
🗸 🗙						_			
Parameter	_	_	_	_	_		~		
Parameter	Min	Av	Max	Bulk Av	Usi	~	1		
Heat Flux					~	_			
X - Component					~				
Y - Component					~				
Z - Component					~				
Heat Transfer					~				
X - Component					~				
Y - Component					~	_			
Z - Component					~				
Normal Force					~				
X - Component					~				
Y - Component					~				
Z - Component					~				
Force					~				
X - Component					~				
Y - Component					~				
Z - Component					~				
Shear Force									
X - Component			V		~				
Y - Component					4				
Z - Component				X - (Com	рог	hent	of Sh	ear



8	Point Goals
~	×
Sele	ction
	XYZ
*	Point Coordinates

Figure 2.21d) Selecting point coordinates

🍍 Point Goals					
✓ X					
Sele	ction				
		V F1	7 [1		
	x [m]	r[m]	2 [m]		
	0.2	0.02	0		
	0.4	0.02	0		
	0.6	0.02	0		
	0.8	0.02	0		

Figure 2 21e) Coordinates	for point goals
1 15010 2.210) Coordinates	for point gouis

1. 50	Carla
⊡ ~ ™	Goals
	GG X - Component of Shear Force 1
	$\mathbb{P}_{\mathbf{x}} = \mathbb{P}_{\mathbf{x}} + \mathbb{P}_{\mathbf{x}} = 0.2 \text{ m}$
	$\mathbb{P}_{\mathbf{x}} = \mathbb{P}_{\mathbf{x}} + \mathbb{P}_{\mathbf{x}} = 0.4 \text{ m}$
	$\mathbb{P}_{\mathbf{x}} = \mathbb{P}_{\mathbf{x}} + \mathbb{P}_{\mathbf{x}} = 0.6 \text{ m}$
	PG X - Component of Velocity at $x = 0.8$ m

Figure 2.21g) Renaming the point goals

Parameter		
Parameter	Value	Use for Conv.
Static Pressure		 Image: A set of the set of the
Total Pressure		 Image: A set of the set of the
Dynamic Pressure		 Image: A set of the set of the
Temperature of Fluid		 Image: A set of the set of the
Density		 Image: A set of the set of the
Velocity		 Image: A set of the set of the
X - Component of Velocity	~	Image: A start and a start

Figure 2.21f) Parameter for point goals

Equation Goal
Expression:
{PG X - Component of Velocity at x = 0.2 m}*0.2/1.516E-5
Figure 2.21h) Entering an equation goal

Running the Calculations

22. Select FloWorks>>Solve>>Run to start calculations. Click on the Run button in the Run window. Click on the goals *button* in the **Solver** window to see the **List of Goals**.

FloW	Yorks Window Help 🖉 🗋 🗸	P •	- 🖪 - 🎖	• 9	-
	Project •		65	\odot	P
	Insert •	Disp	elations	Quick Snaps	Ra
1	General Settings		*	Ŧ	
	Units				0
	Computational Domain				-
	Initial Mesh				
먭	Component Control				
	Radiation Transparent Bodies				
.	Calculation Control Options				
	Solve	⊳	Run		

Solver Figure 2.22a) Starting calculations

?× Run Startup Run 🗹 Create mesh Take previous results Close New calculation Help) Continue calculation

Figure 2.22b) Run window

📴 Solver: Flat Plate Boundary Layer(Flat Plate Boundary Layer Study,SLDPRT)							
File Calculation View Insert Wir	ndow Help						
	🛛 🕅 📈 😽	?					
E Log D Info							
Message	Iterations	Date	Parameter	Value			
Mesh generation started		15:59:12	Fluid cells	51002			
Mesh generation normally finished		15:59:15	Partial cells	4884			
Preparing data for calculation		15:59:18	Iterations	254			
Calculation started	0	15:59:22	Last iteration fini	16:10:34			
Refinement	157	16:05:58	CPU time per last	00:00:04			
Refinement	237	16:09:25	Travels	3.20796			
Calculation has converged since t	253	16:10:34	Iterations per 1 t	81			
Goals are converged	253		Cpu time	0:11:1			
Calculation finished	254	16:10:37	Calculation time left	0:0:0			
			Status	Solver is finished.			
🏁 List of Goals							
Goal Name	Cur	rent Value	Averaged Value	Minimum Value	Maximum Value	Progress	
GG X - Component of Shear Force 1	0.000	0147156 N	0.000154352 N	0.0001471 N	0.000364412 N	Achieved (IT = 81)	
PG X - Component of Velocity at x =	0.2 m 5.4	06865 m/s	5.06818 m/s	5.06794 m/s	5.06873 m/s	Achieved (IT = 87)	
PG X - Component of Velocity at x =	0.4 m 5.4	09419 m/s	5.09452 m/s	5.09414 m/s	5.09492 m/s	Achieved (IT = 96)	
PG X - Component of Velocity at x =	0.6 m 5.	11537 m/s	5.11524 m/s	5.11483 m/s	5.1159 m/s	Achieved (IT = 106)	
PG X - Component of Velocity at x =	0.8 m 5.	13267 m/s	5.13238 m/s	5.13218 m/s	5.13277 m/s	Achieved (IT = 115)	
Reynolds number at x = 0.8 m		270853	270838	270827	270859	Achieved (IT = 115)	
Reynolds number at $x = 0.6$ m		202455	202450	202434	202476	Achieved (IT = 106)	
Reynolds number at x = 0.4 m		134411	134420	134410	134431	Achieved (IT = 96)	
Reynolds number at x = 0.2 m		66868.7	66862.5	66859.4	66869.8	Achieved (IT = 87)	

Figure 2.22c) Solver window

Using Cut Plots to Visualize the Flow Field

23. Right click on Cut Plots in the COSMOSFloWorks analysis tree and select Insert....

Select the **Front Plane** from the **Select Pressure** Click on View Settings... Slide the **Number of colors:** slide bar to 101. Select **Pressure** from the **Parameter** drop down menu. Click on CK. Click OK V to exit the **Cut Plot** window. Figure 2.23a)

shows the high pressure region close to the leading edge of the flat plate. Rename the cut plot to **Pressure**. Repeat this step but instead choose **X-velocity** from the **Parameter** drop down menu. Rename the second cut plot to **X-velocity**. Figures 2.23b) and 2.23c) are showing the velocity boundary layer close to the wall.

101327 101326 101326		
_ 101326	V	
_ 101326		
_ 101326	Z X	
_ 101325		
101325 101325 Pressure [Pa]		

Figure 2.23a) Pressure distribution along the flat plate.



Figure 2.23b) X-velocity distribution on the flat plate.



Figure 2.23c) Close up view of the velocity boundary layer.

Using XY Plots with Templates

24. Place the file **"xy-plot figure 2.24c)**" into the **Local Disk**

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to

make it available in the **Template** list. Click on the 🧏 **FeatureManager design tree**. Click on

the sketch $\mathbf{x} = 0.2, 0.4, 0.6, 0.8 \text{ m}$. Click on the **COSMOSFloWorks analysis tree** tab. Right click **XY Plot** and select **Insert...** Check the **X-velocity** box. Open the **Resolution** portion of the **XY Plot** window and slide the **Geometry Resolution** as far as it goes to the right. Click on the **Evenly Distribute Output Points** button and increase the number of points to **500**. Open the **Options** portion and check the **Display boundary layer** box. Select the template "**xy-plot figure 2.24c**)" from the drop down menu. Click OK **v** to exit the **XY Plot** window. An Excel file will open with a graph of the velocity in the boundary layer at different streamwise positions, see figure 2.24c). Rename the inserted xy-plot in the COSMOSFloWorks analysis tree to **Laminar Velocity Boundary Layer**.





Figure 2.24a) Selecting the sketch for the XY Plot

Figure 2.24b) Settings for the XY Plot



Figure 2.24c) Boundary layer velocity profiles on a flat plate at different streamwise positions

Comparison of FloWorks Results with Theory and Empirical Data

25. We now want to compare this velocity profile with the theoretical Blasius velocity profile for laminar flow on a flat plate. First, we have to normalize the steamwise X velocity component with the free stream velocity. Secondly, we have to transform the wall normal coordinate into the similarity coordinate for comparison with the Blasius profile. The similarity coordinate is described by

$$\eta = y \sqrt{\frac{u}{vx}} \tag{1}$$

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

26. Place the file "xy-plot figure 2.25a)" into the Local Disk
(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the Template list. Repeat step 24 and select the new template for the XY-plot. Rename the xy-plot to Comparison with Blasius Profile.

We see in figure 2.25a) that all profiles at different streamwise positions approximately collapse on the same Blasius curve when we use the boundary layer similarity coordinate.



Figure 2.25a) Velocity profiles in comparison with the theoretical Blasius profile (full line)

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

$$Re_{\chi} = \frac{U\chi}{v}$$
(2)

The boundary layer thickness 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. The theoretical expression for the thickness of the laminar boundary layer is given by

$$\frac{\delta}{x} = \frac{4.91}{\sqrt{Re_x}} \tag{3}$$

and the thickness of the turbulent boundary layer

$$\frac{\delta}{x} = \frac{0.16}{Re_x^{1/7}}$$
(4)

From the data of figure 2.24c) we can see that the thickness of the laminar boundary layer is close to 3.88 mm at $Re_x = 66,900$ corresponding to x = 0.2 m. The free stream velocity at x = 0.2 m is

U = 5.069 m/s, see figure 2.22c) for list of goals in solver window, and 99% of this value is $U_{\delta} = 5.018$ m/s. The boundary layer thickness from FloWorks was found by finding the y position corresponding to the U_{δ} velocity. This value of the boundary layer thickness can be compared with a value of 3.80 mm from equation (3). In table 2.1 are comparisons shown between boundary layer thickness from FlowWorks and theory corresponding to the four different Reynolds numbers shown in figure 2.24c). The Reynolds number varies between $Re_x = 66,900$ at x = 0.2 m and $Re_x = 270,900$ at x = 0.8 m.

	δ (mm) FloWorks	δ (mm) Theory	Percent (%) Difference	U _δ (m/s)	U (m/s)	$\nu\left(\frac{m^2}{s}\right)$	Re _x
x = 0.2 m	3.88	3.80	2.1	5.018	5.069	0.00001516	66,900
x = 0.4 m	5.41	5.36	0.9	5.044	5.095	0.00001516	134,400
x = 0.6 m	6.53	6.55	0.3	5.065	5.116	0.00001516	202,500
x = 0.8 m	7.48	7.55	0.9	5.082	5.133	0.00001516	270,900

Table 2.1 Comparison between FloWorks and theory for laminar boundary layer thickness

27. Place the file "xy-plot figure 2.25b)" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and select the new template for the XY-plot as shown in figure 2.25b). Rename the xy-plot to **Boundary Layer Thickness**.



Figure 2.25b) Comparison between FloWorks and theory on boundary layer thickness

The displacement thickness is defined as the distance that a streamline outside of the boundary layer is deflected by the boundary layer and is given by the following integral

$$\delta^* = \int_0^\infty (1 - \frac{u}{v}) dy \tag{5}$$

The theoretical expression for the displacement thickness of the laminar boundary layer is given by

$$\frac{\delta^*}{x} = \frac{1.72}{\sqrt{Re_x}} \tag{6}$$

and the displacement thickness of the turbulent boundary layer

$$\frac{\delta^*}{x} = \frac{0.02}{Re_x^{1/7}}$$
(7)

	δ^* (mm)	δ^* (mm)	Percent (%)	Re_x
	FloWorks	Theory	Difference	
x = 0.2 m	1.3310	1.3303	0.05	66,900
x = 0.4 m	1.8418	1.8766	1.85	134,400
x = 0.6 m	2.2617	2.2936	1.39	202,500
x = 0.8 m	2.5858	2.6439	2.20	270,900

Table 2.2 Comparison between FloWorks and theory for laminar displacement thickness

Place the file "xy-plot figure 2.25c)" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and select the new template for the XY-plot as shown in figure 2.25c). Rename the xy-plot to **Displacement Thickness**.



Figure 2.25c) Comparison between FloWorks and theory on displacement thickness

The momentum thickness is related to the loss of momentum flux caused by the boundary layer. The momentum thickness is defined by an integral similar to the one for displacement thickness

$$\theta = \int_0^\infty \frac{u}{v} \left(1 - \frac{u}{v}\right) dy \tag{8}$$

The theoretical expression for the momentum thickness of the laminar boundary layer is given by

$$\frac{\theta}{x} = \frac{0.664}{\sqrt{Re_x}} \tag{9}$$

and the momentum thickness of the turbulent boundary layer

$$\frac{\theta}{x} = \frac{0.016}{Re_x^{1/7}}$$
(10)

	θ (mm)	θ (mm)	Percent (%)	Re_x
	FloWorks	Theory	Difference	
x = 0.2 m	0.5107	0.5136	0.56	66,900
x = 0.4 m	0.7132	0.7255	1.70	134,400
x = 0.6 m	0.8756	0.8854	1.11	202,500
x = 0.8 m	1.0016	1.0207	1.87	270,900

Table 2.3 Comparison between FloWorks and theory for laminar momentum thickness

Place the file "xy-plot figure 2.25d)" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and select the new template for the XY-plot as shown in figure 2.25d). Rename the xy-plot to **Momentum Thickness**.



Figure 2.25d) Comparison between FloWorks and theory on momentum thickness

Finally, we have the shape factor that is defined as the ratio of the displacement thickness and the momentum thickness.

$$H = \frac{\delta^*}{\theta} \tag{11}$$

	Н	Н	Percent (%)	Re_x
	FloWorks	Theory	Difference	
x = 0.2 m	2.6062	2.59	0.63	66,900
x = 0.4 m	2.5824	2.59	0.29	134,400
x = 0.6 m	2.5830	2.59	0.27	202,500
x = 0.8 m	2.5817	2.59	0.32	270,900

The theoretical value of the shape factor is H = 2.59 for the laminar boundary layer and H = 1.25 for the turbulent boundary layer.

Table 2.4 Comparison between FloWorks and theory for laminar shape factor

28. We now want to study how the local friction coefficient varies along the plate. It is defined as the local wall shear stress divided by the dynamic pressure:

$$C_{f,x} = \frac{\tau_w}{\frac{1}{2}\rho U^2} \tag{12}$$

The theoretical local friction coefficient for laminar flow is given by

$$C_{f,x} = \frac{0.664}{\sqrt{Re_x}}$$
 $Re_x < 5 \cdot 10^5$ (13)

and for turbulent flow

$$C_{f,x} = \frac{0.027}{Re_x^{1/7}} \qquad 5 \cdot 10^5 \le Re_x \le 10^7 \tag{14}$$

Place the file "xy-plot figure 2.26" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the Template list. Repeat step 24 but this time choose the sketch x = 0 - 0.9 m, uncheck the box for X-Velocity and check the box for Shear Stress. Rename the xy-plot to Local Friction Coefficient. An Excel file will open with a graph of the local friction coefficient versus the Reynolds number compared with theoretical values for laminar boundary layer flow.



Figure 2.26 Local friction coefficient as a function of the Reynolds number

The average friction coefficient over the whole plate C_f is not a function of the surface roughness for the laminar boundary layer but a function of the Reynolds number based on the length of the plate Re_L , see figure E3 in Exercise 8 at the end of this chapter. This friction coefficient can be determined in FloWorks by using the final value of the global goal, the X-component of the Shear Force F_f , see figure 2.22c) and dividing it by the dynamic pressure times the area A in the X-Z plane of the computational domain related to the flat plate.

$$C_f = \frac{F_f}{\frac{1}{2}\rho U^2 A} = \frac{0.000147156N}{\frac{1}{2} \cdot 1.204 kg/m^3 \cdot 5^2 m^2/s^2 \cdot 1m \cdot 0.004m} = 0.002444$$
(15)

$$Re_L = \frac{UL}{v} = \frac{5m/s \cdot 1m}{1.516 \cdot 10^{-5} m^2/s} = 3.3 \cdot 10^5$$
(16)

The average friction coefficient from FloWorks can be compared with the theoretical value for laminar boundary layers

$$C_f = \frac{1.328}{\sqrt{Re_L}} = 0.002312 \qquad Re_L < 5 \cdot 10^5 \tag{17}$$

This is a difference of 5.7 %. For turbulent boundary layers the corresponding expression is

$$C_f = \frac{0.0315}{Re_L^{1/7}} \qquad 5 \cdot 10^5 \le Re_L \le 10^7 \tag{18}$$

If the boundary layer is laminar on one part of the plate and turbulent on the remaining part the average friction coefficient is determined by

$$C_f = \frac{0.0315}{Re_L^{1/7}} - \frac{1}{Re_L} \quad (0.0315Re_{cr}^{\frac{6}{7}} - 1.328\sqrt{Re_{cr}}) \tag{19}$$

where Re_{cr} is the critical Reynolds number for laminar to turbulent transition.

Cloning of the Project

29. In the next step, we will clone the project. Select FloWorks>>Project>>Clone Project.... Create a new project with the name "Flat Plate Boundary Layer Study Using Water". Click on the OK button to exit the Clone Project window. Next, change the fluid to water in order to get higher Reynolds numbers. Start by selecting FloWorks>>General Settings... from the SolidWorks menu. Click on Fluids in the Navigator portion and click on the Remove button. Answer OK to the question if you want to continue. Select Water from the Liquids and Add it as the Project Fluid. Change the Flow type to Laminar and Turbulent, see figure 2.27d). Click on the OK button to close the General Setting window.



Figure 2.27a) Cloning the project

FloW	orks	Window	Help	9	•
	Proje	ct			•
	Inser	t			Þ
	Gene	ral Settings	5		



Figure 2.27b) Creating a new project

Figure 2.2/c) Selection of general settings	Figure 2.27c) Selection	of general	settings
---	--------------	-------------	------------	----------

Fluids	Path	~	New	Navigator
Gases				
Liquids				Analysis type
Acetone	Pre-Defined			
Ammonia	Pre-Defined			Fluids
Argon	Pre-Defined			
Ethane	Pre-Defined			Wall conditions
Ethanol	Pre-Defined			_
Ethylene	Pre-Defined			Initial conditions
····· Methane	Pre-Defined			
Methanol	Pre-Defined			
Nitroaen	Pre-Defined	~	Add	
Project Fluids	Default Fluid		Remove	
Water (Liquids)				
			Replace	
Flow Characteristic	Value			
Flow type	Laminar and Turbulent	~		

Figure 2.27d) Selection of fluid and flow type

30. Select FloWorks>>Computational Domain... Set the size of the computational domain to the values shown in figure 2.28a). Click on the OK button to exit. Select FloWorks>>Initial Mesh... from the SolidWorks menu and change the Number of cells per X: to 400 and the Number of Cells per Y: to 200. Also, in the Control Intervals portion of the window, change the Ratio for Y1 to -100 and the Ratio for X1 to -5. This is done to increase the number of cells close to the wall where the velocity gradient is high.). Click on the OK button to exit. Select FloWorks>>Calculation Control Options... from the SolidWorks menu. Change the Maximum travels value to 5 by first changing to Manual from the drop down menu. Travel is a unit characterizing the duration of the calculation.). Click on the OK button to exit.

Computational Domain						
Size	Boundary Condition Color S	etting				
× min:	-0.12 m	*				
×max	c 1.02 m	*				
Y min:	-1e-05 m	*				
Y max	c 0.10001 m	*				
Z min:	-0.002 m	*				
Z max	« 0.002 m	-				

Figure 2.28a) Setting the size of the computational domain

itial Me	sh					? 🛛
Basic Me	sh Solid/Fluid In	terface Refining C	Cells Narrow Channels			
Numbe	er of cells					ОК
Numb	er of cells per X:			400	•	Cancel
Number of cells per Y:			200	*	Help	
Numb	er of cells per Z:			1	•	
Contro	l intervals					
	Min	Max	Number of cells Rat	io	Add Plane	
X1	-0.12 m	1.02 m	400 -5			
Y1	-0.0001 m	0.1001 m	200 -10	0	Edit Plane	
Z1	-0.002 m	0.002 m	1 1			

Figure 2.28b) Increasing the number of cells and the distribution of cells

FloW	FloWorks Window Help & Calculation Control Options						
	Project		Finish				
	Insert		Finish	Refinement	Saving Adv	anced	
1	General Settings			Parameter		On/Off	Value
	Units		📮 Fin	ish Conditions			If one is satisfied
	Computational Domain			Minimum refiner	nent number	 Image: A set of the set of the	2
	Initial Mesh			Maximum iterati	ons		100
-12	Component Control			Maximum calcul	lation time		36000 s
	Radiation Transparent Bodies			Maximum travel	s	 Image: A set of the set of the	Manual 🔽 5
5	Calculation Control Options			Goals Converge	ence	 Image: A set of the set of the	

Figure 2.28c) Calculation control options

Figure 2.28d) Setting maximum travels

Select **FloWorks>>Project>>Show Basic Mesh** from the SolidWorks menu. We can see in figure 2.28f) that the density of the mesh is much higher close to the flat plate at the bottom wall in the figure as compared to the region further away from the wall.



Figure 2.28e) Showing the basic mesh



Figure 2.28f) Mesh distribution in the X-Y plane

31. Right click the Inlet Velocity Boundary Condition in the COSMOSFloWorks analysis tree and select Edit Definition.... Open the Boundary Layer section and select Laminar Boundary Layer. Click OK ✓ to exit the Boundary Condition window. Right click the Reynolds number at x = 0.2 m goal and select Edit Definition.... Change the viscosity value in the Expression to 1004F. 6. Click on the OK button to exit. Change the other three equation goals in

Expression to **1.004E-6**. Click on the OK button to exit. Change the other three equation goals in the same way. Select **FloWorks>>Solve>>Run** to start calculations. Check the **Create Mesh** box and select **New Calculation**. Click on the **Run** button in the **Run** window.



Figure 2.29a) Selecting a laminar boundary layer



Figure 2.29b) Modifying the equation goals

Run		? 🗙
Startup Create mesh New calculation Continue calculation Solver	Take previous results	Run Close Help

Figure 2.29c) Creation of mesh and starting a new calculation

💯 Solver: Flat Plate Boundary Layer Study Using Water(Flat Plate Boundary Layer Study.SLDPRT)								
File Calculation View Insert Window Help								
Log 🚺 Info								
Message	Iterations	Date	Parameter	Value				
Mesh generation started		09:10:55	Fluid cells	84960				
Mesh generation normally finished		09:10:59	Partial cells	3708				
Preparing data for calculation		09:11:02	Iterations	353				
Calculation started	0	09:11:06	Last iteration fini	09:57:53				
Refinement	173	09:39:22	CPU time per last	00:00:06				
Refinement	260	09:48:25	Travels	4.02354				
Calculation has converged since t	352	09:57:53	Iterations per 1 t	92				
Goals are converged	352		Cpu time	0:46:15				
Calculation finished	353	09:57:59	Calculation time left	0:0:0				
			Status	Solver is finished.				
🗭 List of Goals								
Goal Name	Cu	rrent Value	Averaged Value	Minimum Value	Maximum Value	Progress		
GG X - Component of Shear Force 1		0.22822 N	0.228643 N	0.22822 N	0.229164 N	Achieved (IT = 220)		
PG X - Component of Velocity at x = 0	D.2 m 5	.04166 m/s	5.04167 m/s	5.04166 m/s	5.04168 m/s	Achieved (IT = 331)		
PG X - Component of Velocity at x = 0.4 m 5.07044 m/s		5.07022 m/s	5.06967 m/s	5.07044 m/s	Achieved (IT = 349)			
PG X - Component of Velocity at x = 0.6 m 5.09548 m/s		.09548 m/s	5.09531 m/s	5.09495 m/s	5.09548 m/s	Achieved (IT = 346)		
PG X - Component of Velocity at x = (D.8 m 5	12023 m/s	5.12003 m/s	5.11909 m/s	5.12023 m/s	Achieved (IT = 353)		
Reynolds number at x = 0.8 m 4.07987e+006		4.07971e+006	4.07896e+006	4.07987e+006	Achieved (IT = 353)			
Reynolds number at x = 0.6 m	3.04	511e+006	3.04501e+006	3.04479e+006	3.04511e+006	Achieved (IT = 346)		
Reynolds number at x = 0.4 m	2.0	201e+006	2.02001e+006	2.01979e+006	2.0201e+006	Achieved (IT = 349)		
Reynolds number at x = 0.2 m	1.00)432e+006	1.00432e+006	1.00432e+006	1.00432e+006	Achieved (IT = 331)		

Figure 2.29d) Solver window and goals table for calculations of turbulent boundary layer

32. Place the file "xy-plot figure 2.30a)" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the Template list. Repeat step 24 and choose the sketch x = 0.2, 0.4, 0.6, 0.8 m and check the box for X-Velocity. Rename the xy-plot to Turbulent Velocity Boundary Layer. An Excel file will open with a graph of the streamwise velocity component versus the wall normal coordinate, see figure 2.30a). We see that the boundary layer thickness is much higher than the corresponding laminar flow case. This is related to higher Reynolds number at the same streamwise positions as in the laminar case. The higher Reynolds numbers are due to the selection of water as the fluid instead of air that has a much higher value of kinematic viscosity than water.



y (m)

Figure 2.30a) FloWorks comparison for turbulent boundary layers at $Re_x = 10^6 - 4.1 \cdot 10^6$

As an example, the turbulent boundary layer thickness from figure 2.30a) is 4.45 mm at x = 0.2 m which can be compared with a value of 4.44 mm from equation (4), see table 2.2.

	δ (mm)	δ (mm)	Percent (%)	U	m^2	Re_x
	FloWorks	Empirical	Difference	(m/s)	$\nu\left(\frac{1}{s}\right)$	
x = 0.2 m	4.45	4.44	0.1	5.042	0.000001004	1,004,320
x = 0.4 m	8.73	8.04	8.6	5.070	0.000001004	2,020,100
x = 0.6 m	12.5	11.4	9.9	5.095	0.000001004	3,045,110
x = 0.8 m	16.0	14.5	9.7	5.120	0.000001004	4,079,870

Table 2.5 Comparison for FloWorks and empirical results for turbulent boundary layer thickness

Place the file "xy-plot figure 2.30b)" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and choose the sketch x = 0.2, 0.4, 0.6, 0.8 m and check the box for X-Velocity. Rename the xy-plot to **Turbulent Boundary Layer Thickness**. An Excel file will open with a graph of the boundary layer thickness versus the Reynolds number, see figure 2.30b).





33. Place the file "xy-plot figure 2.31" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and choose the sketch x = 0.2, 0.4, 0.6, 0.8 m and check the box for X-Velocity. Rename the xy-plot to Comparison with One-Sixth Power Law. An Excel file will open with figure 2.31. In figure 2.31 we compare the results from FloWorks with the turbulent profile for n = 6. The power –law turbulent profiles suggested by Prandtl are given by

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{1/n} \tag{20}$$

It is seen that for the higher Reynolds numbers in figure 2.31, that the velocity profiles resembles the one-sixth power law.



Figure 2.31 Profiles in figure 2.30a) compared with one-sixth power law for turbulent profile.

34. Place the file "xy-plot figure 2.32" into the Local Disk

(C:)/SolidWorks/COSMOSFloWorks/FloWorks/lang/english/template/XY-Plots folder to make it available in the **Template** list. Repeat step 24 and choose the sketch $\mathbf{x} = \mathbf{0} - \mathbf{0.9}$ m, uncheck the box for **X-Velocity** and check the box for **Shear Stress**. Rename the xy-plot to **Local Friction Coefficient for Laminar and Turbulent Boundary Layer**. An Excel file will open with figure 2.32. Figure 2.32 is showing the FloWorks is able to capture the local friction coefficient in the laminar region in the Reynolds number range 10,000 – 150,000. At Re =

150,000 there is an abrupt increase in the friction coefficient caused by laminar to turbulent transition. In the turbulent region the friction coefficient is decreasing again but the local friction coefficient from FloWorks is significantly higher than empirical data.



Figure 2.32 Comparison between FloWorks (dashed line) and theoretical laminar and empirical turbulent friction coefficients

The average friction coefficient over the whole plate C_f is a function of the surface roughness for the turbulent boundary layer and also a function of the Reynolds number based on the length of the plate Re_L , see figure E3 in Exercise 8. This friction coefficient can be determined in FloWorks by using the final value of the global goal, the X-component of the Shear Force F_f and dividing it by the dynamic pressure times the area A in the X-Z plane of the computational domain related to the flat plate, see figure 2.28a) for the size of the computational domain.

$$C_f = \frac{F_f}{\frac{1}{2}\rho U^2 A} = \frac{0.22822N}{\frac{1}{2} \cdot 998 kg/m^3 \cdot 5^2 m^2/s^2 \cdot 1m \cdot 0.004m} = 0.00457$$
(21)

$$Re_L = \frac{UL}{\nu} = \frac{5m/s \cdot 1m}{1.004 \cdot 10^{-6} m^2/s} = 4.98 \cdot 10^6$$
(22)

The variation and final values of the goal can be found in the solver window during or after calculation by clicking on the associated flag, see figures 2.33 and 2.29d).



Figure 2.33 Obtaining the current value of the global goal

For comparison with FloWorks results we use equation (19) with $Re_{cr} = 150,000$

$$C_f = \frac{0.0315}{Re_L^{1/7}} - \frac{1}{Re_L} \left(0.0315 Re_{cr}^{\frac{6}{7}} - 1.328\sqrt{Re_{cr}} \right) = 0.0034$$
(23)

This is a difference of 34%.

References

[1] Çengel, Y. A., and Cimbala J.M., Fluid Mechanics Fundamentals and Applications, 1st Edition, McGraw-Hill, 2006.

[2] COSMOSFloWorks Fundamentals 2008.

[3] Fransson, J. H. M., Leading Edge Design Process Using a Commercial Flow Solver, *Exp.in Fluids* **37**, 929 – 932, 2004.

[4] Schlichting, H., and Gersten, K., Boundary Layer Theory, 8th Revised and Enlarged Edition, Springer, 2001.

[5] White, F. M., Fluid Mechanics, 4th Edition, McGraw-Hill, 1999.

Exercises

- 1. Change the number of cells per X and Y (see figure 2.16b)) for the laminar boundary layer and plot graphs of the boundary layer thickness, displacement thickness, momentum thickness and local friction coefficient versus Reynolds number for different combinations of cells per X and Y. Compare with theoretical results.
- 2. Choose one Reynolds number and one value of number of cells per X for the laminar boundary layer and plot the variation in boundary layer thickness, displacement thickness and momentum thickness versus number of cells per Y. Compare with theoretical results.

- 3. Choose one Reynolds number and one value of number of cells per Y for the laminar boundary layer and plot the variation in boundary layer thickness, displacement thickness and momentum thickness versus number of cells per X. Compare with theoretical results.
- 4. Import the file "Leading Edge of Flat Plate". Study the air flow around the leading edge at 5 m/s free stream velocity and determine the laminar velocity boundary layer at different locations on the upper side of the leading edge and compare with the Blasius solution. Also, compare the local friction coefficient with figure 2.26. Use different values of the initial mesh to see how it affects the results.



Figure E1 Leading edge of asymmetric flat plate, see Fransson (2004)

5. Modify the geometry of the flow region used in this chapter by changing the slope of the upper ideal wall so that it is not parallel with the lower flat plate. By doing this you get a streamwise pressure gradient in the flow. Use air at 5 m/s and compare your laminar boundary layer velocity profiles for both accelerating and decelerating free stream flow with profiles without a streamwise pressure gradient.



Figure E2 Example of geometry for a decelerating outer free stream flow

- 6. Determine the displacement thickness, momentum thickness and shape factor for the turbulent boundary layers in figure 2.30a) and determine the percent differences as compared with empirical data.
- 7. Change the distribution of cells using different values of the ratios in the X and Y directions, see figure 2.28b), for the turbulent boundary layer and plot graphs of the boundary layer

thickness, displacement thickness, momentum thickness and local friction coefficient versus Reynolds number for different combinations of ratios. Compare with theoretical results.

8. Use different fluids, surface roughness, free stream velocities and length of the computational domain to compare the average friction coefficient over the entire flat plate with figure E3.



Figure E3 Average friction coefficient for flow over smooth and rough plates, White (1999)