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
Chapter 2

Creating the SolidWorks Part

1. 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.

![Selection of front plane](image1.png)

Figure 2.1a) Selection of front plane

![Selection of front view](image2.png)

Figure 2.1b) Selection of front view

2. Click on Corner Rectangle from the Sketch tools.

![Selecting a sketch tool](image3.png)

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.

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

![Figure 2.5b) Closing the property manager](image)

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](image)
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](image)

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](image)

![Figure 2.8b) Renaming the sketch for boundary layer velocity profiles](image)
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.

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.

Figure 2.10 Drawing a line in the Z-direction
11. 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

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.

Figure 2.12a) Starting a new FloWorks project

Figure 2.12b) Creating a name for the project

13. Use the default Internal Analysis type.

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.
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.

Figure 2.14 Selection of fluid for the project and flow type

Figure 2.15a) Modifying the computational domain

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.

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 to exit the window. Right click in the graphics window and select Zoom/Pan/Rotate>>Zoom to Area and select an area around the left boundary.
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. Right click again in the graphics window and select 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.

![Figure 2.19a)](image1) Ideal wall boundary condition for upper wall

![Figure 2.19b)](image2) 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.

![Figure 2.20](image3) 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 XYZ Point Coordinates button. Enter 0.2 m for X coordinate and 0.02 m for Y coordinate and click on the Add Point button. 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.
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.
23. Right click on Cut Plots in the COSMOSFloWorks analysis tree and select Insert…

Select the Front Plane from the FeatureManager design tree. Click on Slide the Number of colors: slide bar to 101. Select Pressure from the Parameter drop down menu. Click on Click OK 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.
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 x = 0.2, 0.4, 0.6, 0.8 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 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 2.24c) 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

\[ \eta = \frac{y}{ \sqrt{vx} } \]  

(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.

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

\[ Re_x = \frac{U_x}{v} \]  

(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

\[ \frac{\delta}{x} = \frac{4.91}{\sqrt{Re_x}} \]  

(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
$U = 5.069 \text{ m/s}$, see figure 2.22c) for list of goals in solver window, and 99% of this value is $U_\delta = 5.018 \text{ m/s}$. The boundary layer thickness from FloWorks was found by finding the $y$ position corresponding to the $U_\delta$ 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 $x = 0.2 \text{ m}$ and $Re_x = 270,900$ at $x = 0.8 \text{ m}$.

<table>
<thead>
<tr>
<th>$x$ (m)</th>
<th>$\delta$ (mm) FloWorks</th>
<th>$\delta$ (mm) Theory</th>
<th>Percent (%) Difference</th>
<th>$U_\delta$ (m/s)</th>
<th>$U$ (m/s)</th>
<th>$\nu \left( \frac{m}{s} \right)$</th>
<th>$Re_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>3.88</td>
<td>3.80</td>
<td>2.1</td>
<td>5.018</td>
<td>5.069</td>
<td>0.00001516</td>
<td>66,900</td>
</tr>
<tr>
<td>0.4</td>
<td>5.41</td>
<td>5.36</td>
<td>0.9</td>
<td>5.044</td>
<td>5.095</td>
<td>0.00001516</td>
<td>134,400</td>
</tr>
<tr>
<td>0.6</td>
<td>6.53</td>
<td>6.55</td>
<td>0.3</td>
<td>5.065</td>
<td>5.116</td>
<td>0.00001516</td>
<td>202,500</td>
</tr>
<tr>
<td>0.8</td>
<td>7.48</td>
<td>7.55</td>
<td>0.9</td>
<td>5.082</td>
<td>5.133</td>
<td>0.00001516</td>
<td>270,900</td>
</tr>
</tbody>
</table>

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.

![Graph](image)

**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

$$
\delta^* = \int_0^\infty (1 - \frac{y}{U})dy
$$

(5)
The theoretical expression for the displacement thickness of the laminar boundary layer is given by

\[
\frac{\delta^*}{x} = \frac{1.72}{\sqrt{Re_x}}
\]  

and the displacement thickness of the turbulent boundary layer

\[
\frac{\delta^*}{x} = \frac{0.02}{Re_x^{1/7}}
\]  

<table>
<thead>
<tr>
<th>x (m)</th>
<th>( \delta^* ) (mm) FloWorks</th>
<th>( \delta^* ) (mm) Theory</th>
<th>Percent (%) Difference</th>
<th>( Re_x )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>1.3310</td>
<td>1.3303</td>
<td>0.05</td>
<td>66,900</td>
</tr>
<tr>
<td>0.4</td>
<td>1.8418</td>
<td>1.8766</td>
<td>1.85</td>
<td>134,400</td>
</tr>
<tr>
<td>0.6</td>
<td>2.2617</td>
<td>2.2936</td>
<td>1.39</td>
<td>202,500</td>
</tr>
<tr>
<td>0.8</td>
<td>2.5858</td>
<td>2.6439</td>
<td>2.20</td>
<td>270,900</td>
</tr>
</tbody>
</table>

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.

![Graph showing \( \delta^* \) vs. \( Re_x \)]

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

\[
\theta = \int_0^x u \left(1 - \frac{u}{U}\right)dy
\]  

Chapter 2 - 21 -
The theoretical expression for the momentum thickness of the laminar boundary layer is given by

\[ \frac{\theta}{x} = \frac{0.664}{\sqrt{Re_x}} \]  

(9)

and the momentum thickness of the turbulent boundary layer

\[ \frac{\theta}{x} = \frac{0.016}{Re_x^{1/7}} \]  

(10)

<table>
<thead>
<tr>
<th></th>
<th>FloWorks</th>
<th>Theory</th>
<th>Percent (%) Difference</th>
<th>$Re_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x = 0.2$ m</td>
<td>0.5107</td>
<td>0.5136</td>
<td>0.56</td>
<td>66,900</td>
</tr>
<tr>
<td>$x = 0.4$ m</td>
<td>0.7132</td>
<td>0.7255</td>
<td>1.70</td>
<td>134,400</td>
</tr>
<tr>
<td>$x = 0.6$ m</td>
<td>0.8756</td>
<td>0.8854</td>
<td>1.11</td>
<td>202,500</td>
</tr>
<tr>
<td>$x = 0.8$ m</td>
<td>1.0016</td>
<td>1.0207</td>
<td>1.87</td>
<td>270,900</td>
</tr>
</tbody>
</table>

Table 2.3 Comparison between FloWorks and theory for laminar momentum thickness.

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

\[ H = \frac{\delta^*}{\theta} \]  

(11)

Figure 2.25d) Comparison between FloWorks and theory on momentum thickness.
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>
<thead>
<tr>
<th></th>
<th>$H_{\text{FloWorks}}$</th>
<th>$H_{\text{Theory}}$</th>
<th>Percent (%) Difference</th>
<th>$Re_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x = 0.2 \text{ m}$</td>
<td>2.6062</td>
<td>2.59</td>
<td>0.63</td>
<td>66,900</td>
</tr>
<tr>
<td>$x = 0.4 \text{ m}$</td>
<td>2.5824</td>
<td>2.59</td>
<td>0.29</td>
<td>134,400</td>
</tr>
<tr>
<td>$x = 0.6 \text{ m}$</td>
<td>2.5830</td>
<td>2.59</td>
<td>0.27</td>
<td>202,500</td>
</tr>
<tr>
<td>$x = 0.8 \text{ m}$</td>
<td>2.5817</td>
<td>2.59</td>
<td>0.32</td>
<td>270,900</td>
</tr>
</tbody>
</table>

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:

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

The theoretical local friction coefficient for laminar flow is given by

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

and for turbulent flow

$$C_{f,x} = \frac{0.027}{Re_x^{3/7}} \quad 5 \cdot 10^5 \leq Re_x \leq 10^7$$

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 $x = 0 – 0.9 \text{ m}$, 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](image-url)
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.

\[
C_f = \frac{F_f}{\frac{1}{2} \rho U^2 A} = \frac{0.000147156N}{\frac{1}{2} \cdot 1.204kg/m^3 \cdot 5^2 m^2 / s^2 \cdot 1m - 0.004m} = 0.002444
\]  

(15)

\[
Re_L = \frac{UL}{\nu} = \frac{5m/s \cdot 1m}{1.516 \cdot 10^{-5} m^2 / s} = 3.3 \cdot 10^5
\]  

(16)

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

\[
C_f = \frac{1.328}{\sqrt{Re_L}} = 0.002312 \quad \text{when} \quad Re_L < 5 \cdot 10^5
\]  

(17)

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

\[
C_f = \frac{0.0315}{Re_L^{1/7}} \quad \text{when} \quad 5 \cdot 10^5 \leq Re_L \leq 10^7
\]  

(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

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

(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.
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.
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.
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 **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 2.28e)** Showing the basic mesh

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

**Figure 2.29a)** Selecting a laminar boundary layer

**Figure 2.29b)** Modifying the equation goals

---

**Equation Goal**

| Expression | (FG X - Component of Velocity at x = 0.2 m)^0.2/1.004E-6 |

---

Chapter 2 - 27 -
Figure 2.29c) Creation of mesh and starting a new calculation

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.
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.

<table>
<thead>
<tr>
<th></th>
<th>$\delta$ (mm) FloWorks</th>
<th>$\delta$ (mm) Empirical</th>
<th>Percent (%) Difference</th>
<th>$U$ (m/s)</th>
<th>$\nu \left( \frac{m^2}{s} \right)$</th>
<th>$Re_x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x = 0.2$ m</td>
<td>4.45</td>
<td>4.44</td>
<td>0.1</td>
<td>5.042</td>
<td>0.0000001004</td>
<td>1,004,320</td>
</tr>
<tr>
<td>$x = 0.4$ m</td>
<td>8.73</td>
<td>8.04</td>
<td>8.6</td>
<td>5.070</td>
<td>0.0000001004</td>
<td>2,020,100</td>
</tr>
<tr>
<td>$x = 0.6$ m</td>
<td>12.5</td>
<td>11.4</td>
<td>9.9</td>
<td>5.095</td>
<td>0.0000001004</td>
<td>3,045,110</td>
</tr>
<tr>
<td>$x = 0.8$ m</td>
<td>16.0</td>
<td>14.5</td>
<td>9.7</td>
<td>5.120</td>
<td>0.0000001004</td>
<td>4,079,870</td>
</tr>
</tbody>
</table>

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).

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

Figure 2.30b) Boundary layer thickness for turbulent boundary layers at $Re_x = 10^6 - 4.1 \cdot 10^6$
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

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

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 $x = 0 – 0.9$ m, uncheck the box for X-Velocity and check the box for Shear Stress. Rename the xy-plot to Local Friction Coefficient 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.

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

$$Re_L = \frac{UL}{\nu} = \frac{5 \text{ m/s} \times 1 \text{ m}}{1.004 \times 10^{-6} \text{ m}^2 / \text{s}} = 4.98 \times 10^6$$  \hspace{1cm} (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).

![Image](image.png)

Figure 2.33 Obtaining the current value of the global goal

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

$$C_f = \frac{0.0315}{Re_L^{3/7}} \left( \frac{1}{Re_L} \left( 0.0315 Re_{cr}^{8/7} - 1.328 \sqrt{Re_{cr}} \right) \right) = 0.0034$$

(23)

This is a difference of 34%.

References


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.

![Leading Edge of Asymmetric Flat Plate](image1)

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.

![Example of Geometry for a Decelerating Outer Free Stream Flow](image2)

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)](image)