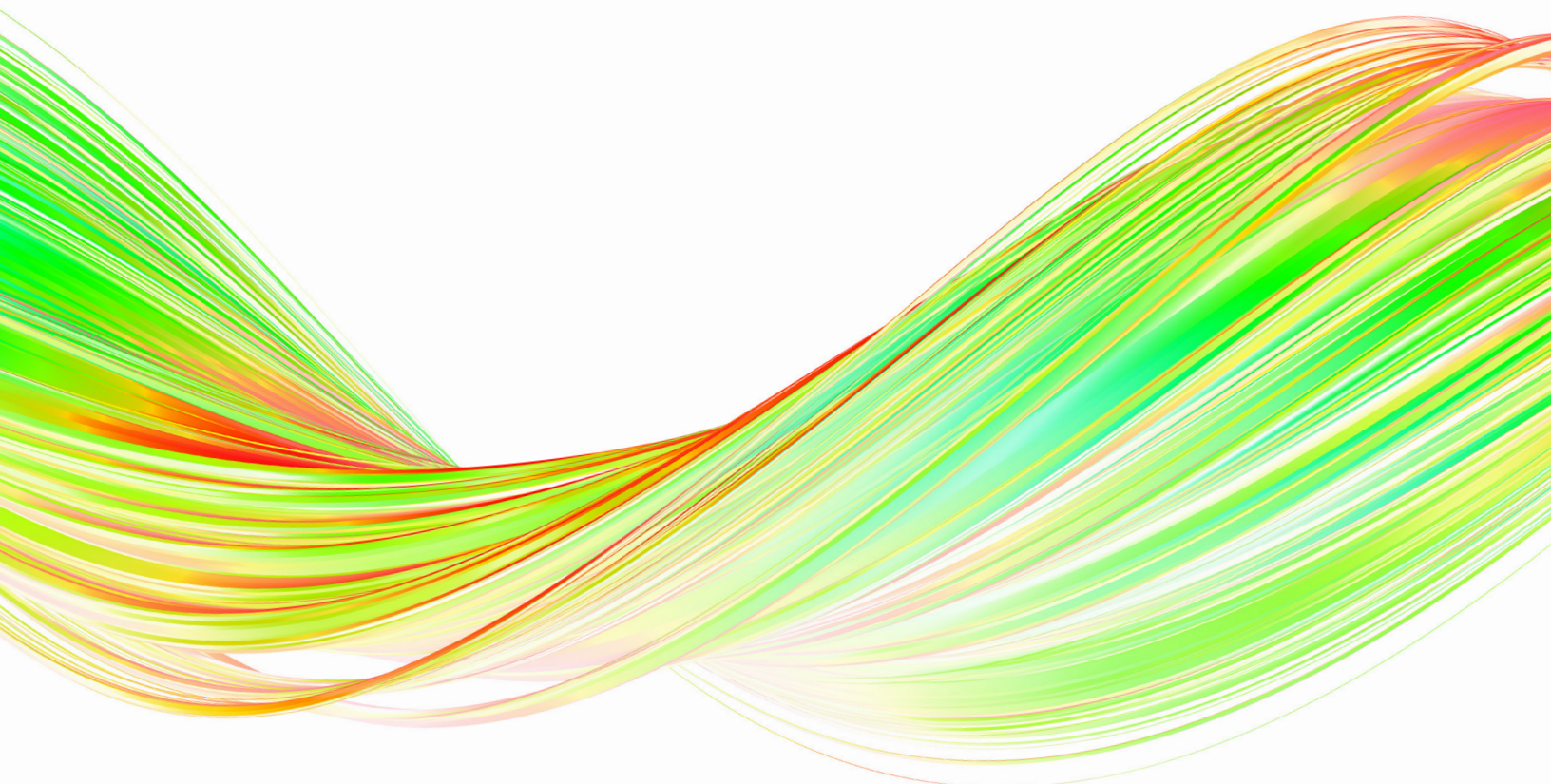


An Introduction to ANSYS® Fluent® 2019



John E. Matsson, Ph.D., P.E.



Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

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 inlet velocity for the 1 m long plate is 5 m/s, and we will be using air as the fluid for laminar calculations. 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.

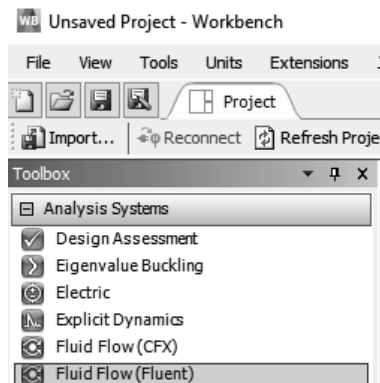


Figure 2.1 Selecting Fluid Flow

D. Launching ANSYS DesignModeler

2. 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.

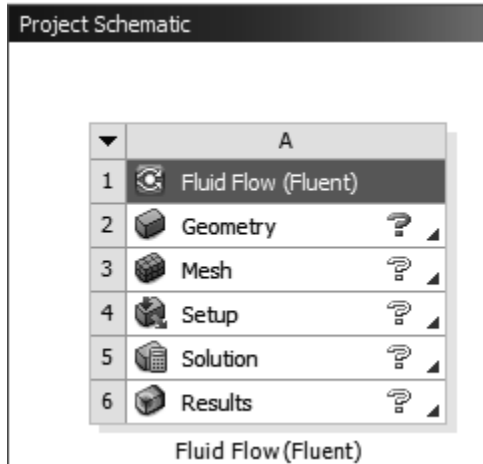


Figure 2.2a) Selecting Geometry

The screenshot shows the 'Properties of Schematic A2: Geometry' dialog box. It has two columns: 'A' (Property) and 'B' (Value). The 'Advanced Geometry Options' section is expanded, and the 'Analysis Type' property is set to '2D'.

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	FFF
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Basic Geometry Options	
10	Solid Bodies	<input checked="" type="checkbox"/>
11	Surface Bodies	<input checked="" type="checkbox"/>
12	Line Bodies	<input type="checkbox"/>
13	Parameters	Independent
14	Parameter Key	ANS;DS
15	Attributes	<input type="checkbox"/>
16	Named Selections	<input type="checkbox"/>
17	Material Properties	<input type="checkbox"/>
18	Advanced Geometry Options	
19	Analysis Type	2D
20	Use Associativity	3D
21	Import Coordinate Systems	2D

Figure 2.2b) Selecting 2D Analysis Type

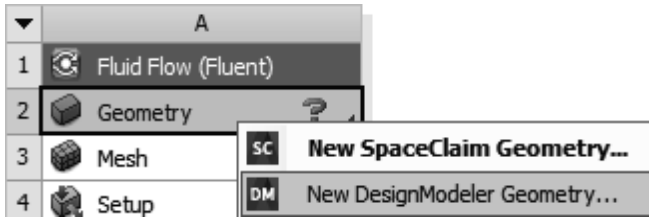


Figure 2.2c) Launching DesignModeler

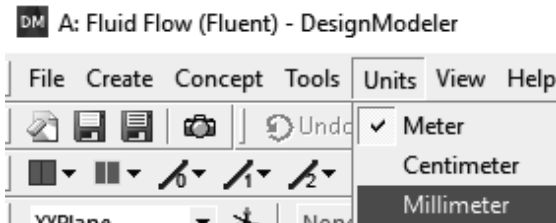


Figure 2.2d) Selecting the length unit

- 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 . Click on the Sketching tab 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.

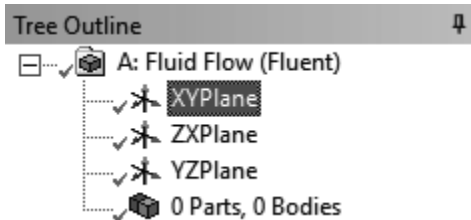


Figure 2.3a) Selection of XYPlane in Tree Outline

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

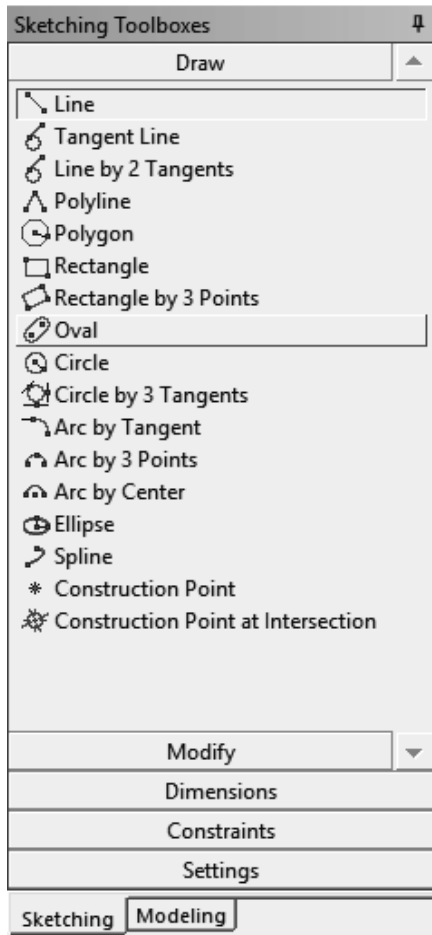


Figure 2.3b) Selection of Line tool

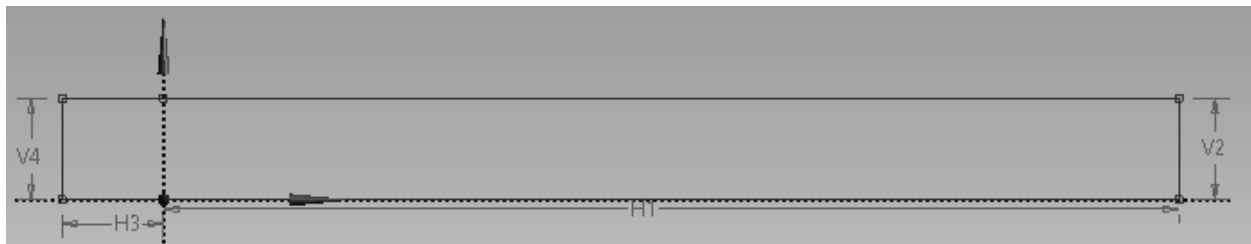


Figure 2.3c) Rectangle with dimensions

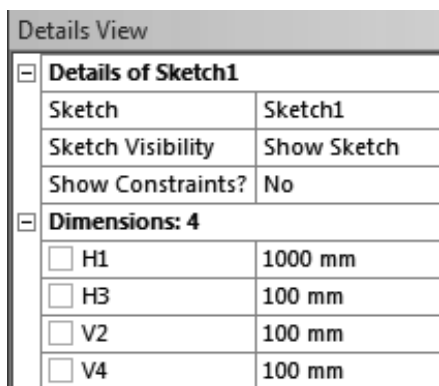
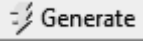


Figure 2.3d) Dimensions in Details View

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

- 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 . The rectangle turns gray. Right click in the graphics window and select Zoom to Fit and close DesignModeler.

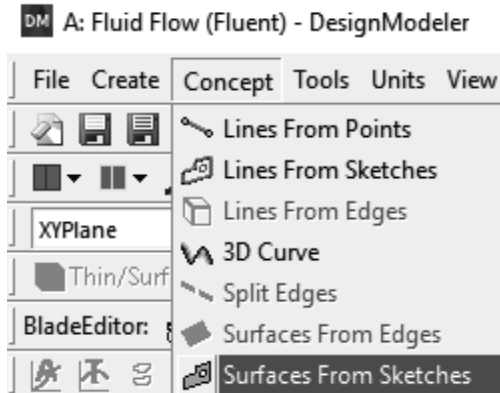


Figure 2.4a) Selecting Surfaces from Sketches

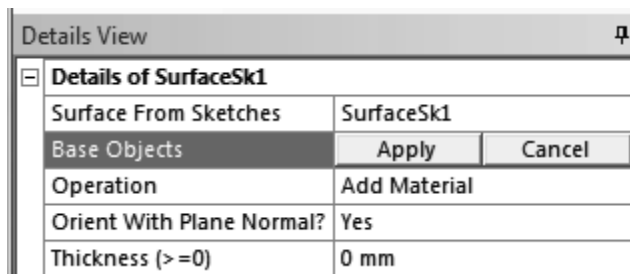


Figure 2.4b) Applying Base Objects

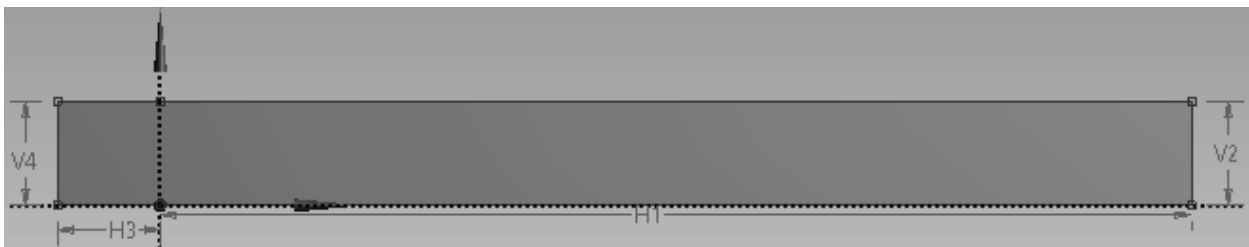


Figure 2.4c) Completed rectangle in DesignModeler

E. Launching ANSYS Meshing

- 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.

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

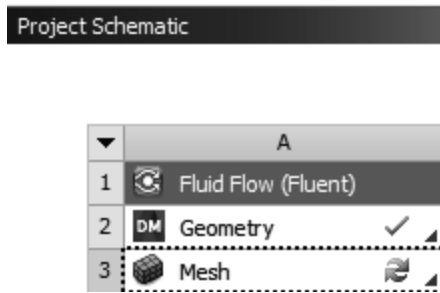


Figure 2.5a) Launching mesh

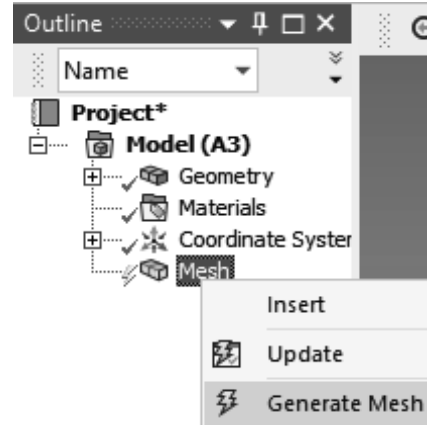


Figure 2.5b) Generating mesh



Figure 2.5c) Coarse mesh

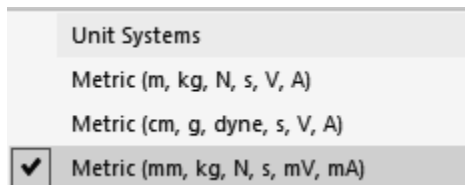



Figure 2.5d) Selection of units in meshing window

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 .

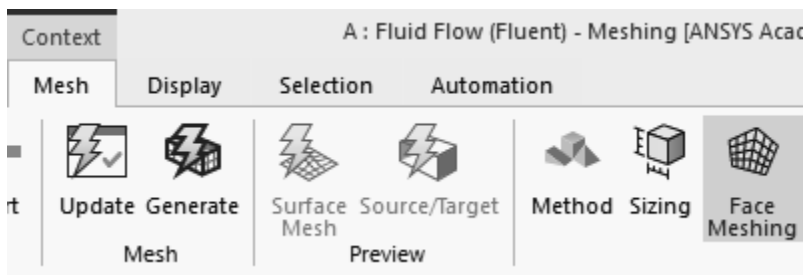


Figure 2.5e) Selection of face meshing



Figure 2.5f) Sizing

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 following Bias Type - - - - and enter 12.0 as the Bias Factor.

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

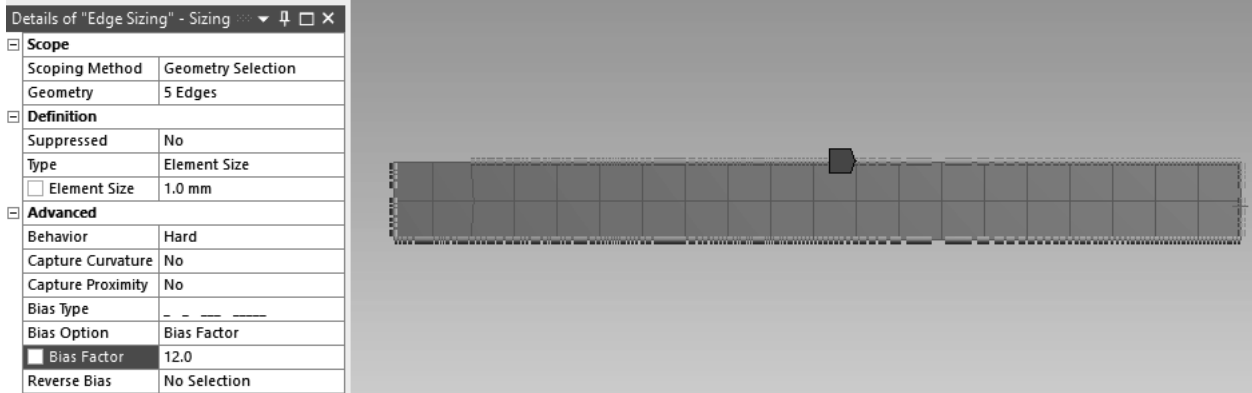


Figure 2.5g) Details of edge sizing for three of the horizontal edges and the vertical edges

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 Bias Type $\text{---} \text{---} \text{---}$. Click on Home>>Generate Mesh in the menu and select Mesh in the Outline. The finished mesh is shown in the graphics window.

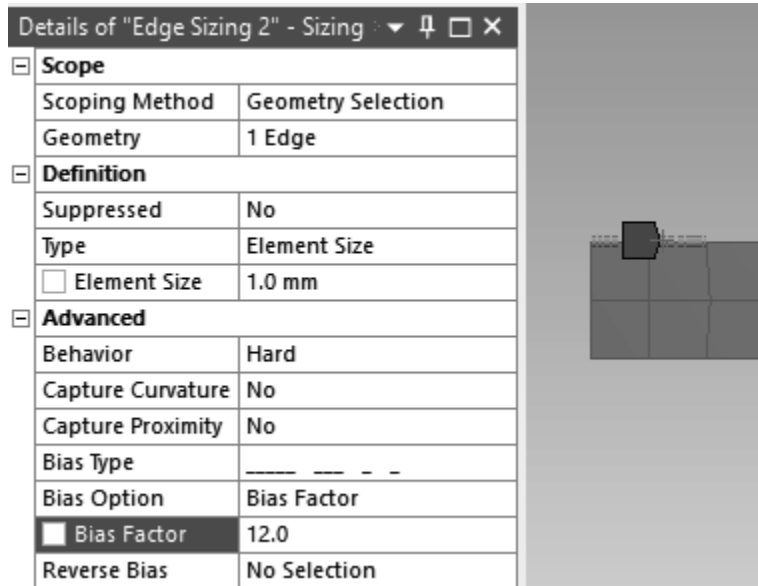


Figure 2.5h) Details of edge sizing for the remaining horizontal edge

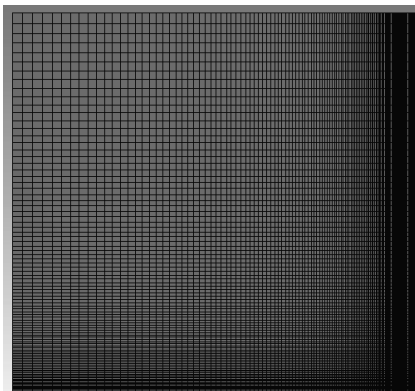


Figure 2.5i) Details of finished mesh

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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*.



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 2D Double Precision solver of Fluent. Check the Double Precision and Parallel as Processing Options. Set the number of Solver Processes equal to the number of cores (in this case there are 8 cores). Click on the plus sign next to Show More Options. Write down the location of the working directory as you will use this information later. Click on the OK button to launch ANSYS Fluent.

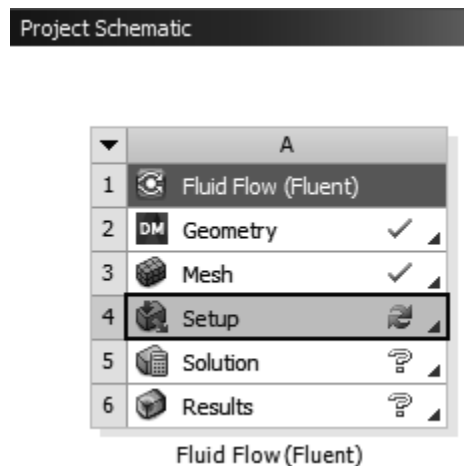


Figure 2.6a) Launching Setup

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

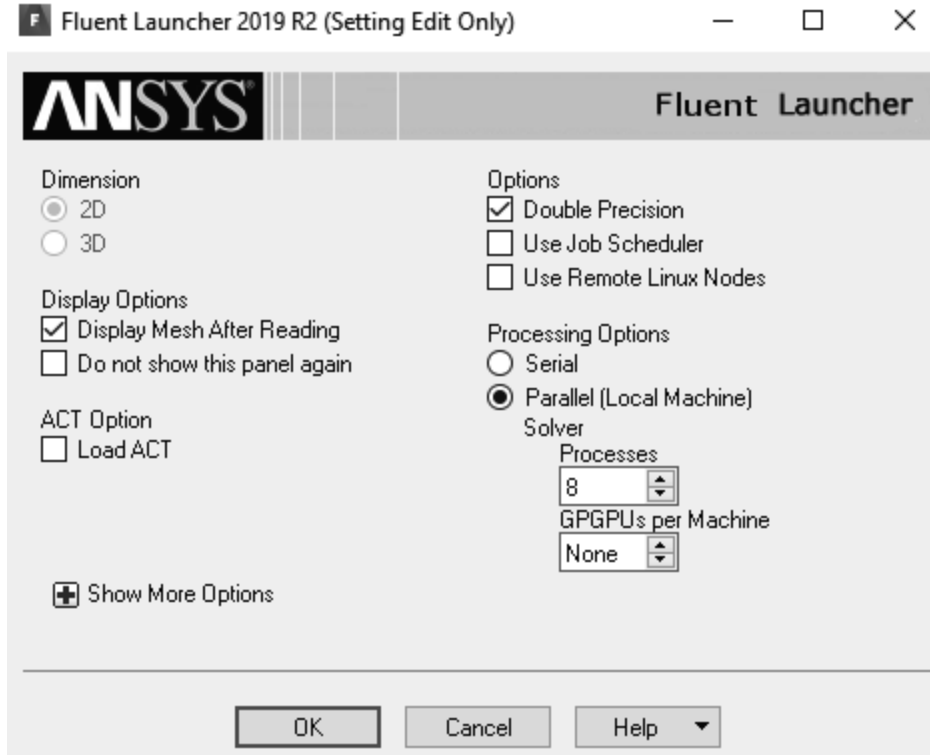


Figure 2.6b) ANSYS Fluent Launcher

Console							
ID	Hostname	Core	O.S.	PID	Vendor		
n7	DESKTOP-QQG8MCG	8/8	Windows-x64	13360	Intel (R)	Core (TM)	i7-7820HQ
n6	DESKTOP-QQG8MCG	7/8	Windows-x64	12752	Intel (R)	Core (TM)	i7-7820HQ
n5	DESKTOP-QQG8MCG	6/8	Windows-x64	4296	Intel (R)	Core (TM)	i7-7820HQ
n4	DESKTOP-QQG8MCG	5/8	Windows-x64	6872	Intel (R)	Core (TM)	i7-7820HQ
n3	DESKTOP-QQG8MCG	4/8	Windows-x64	11828	Intel (R)	Core (TM)	i7-7820HQ
n2	DESKTOP-QQG8MCG	3/8	Windows-x64	3988	Intel (R)	Core (TM)	i7-7820HQ
n1	DESKTOP-QQG8MCG	2/8	Windows-x64	11900	Intel (R)	Core (TM)	i7-7820HQ
n0*	DESKTOP-QQG8MCG	1/8	Windows-x64	12944	Intel (R)	Core (TM)	i7-7820HQ
host	DESKTOP-QQG8MCG		Windows-x64	8968	Intel (R)	Core (TM)	i7-7820HQ

Figure 2.6c) 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.

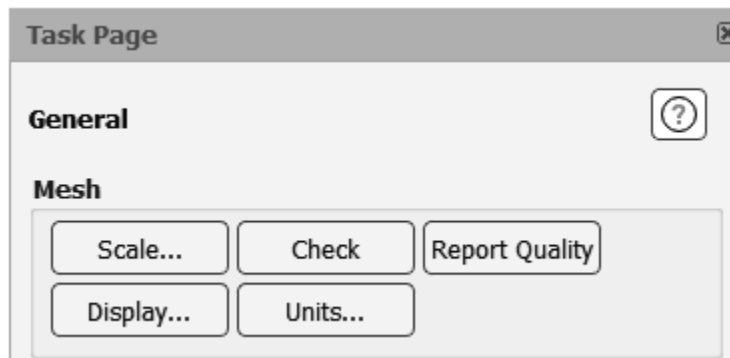


Figure 2.7a) Scale Check

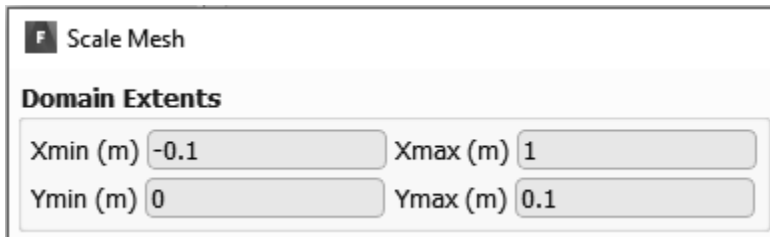


Figure 2.7b) Scale Mesh

8. 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 to 5 (m/s). 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. Click OK to close the window.

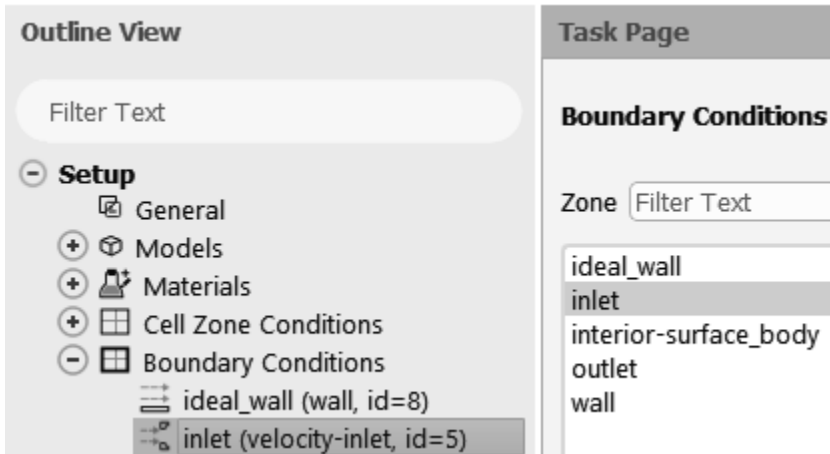


Figure 2.8a) Boundary Conditions for inlet

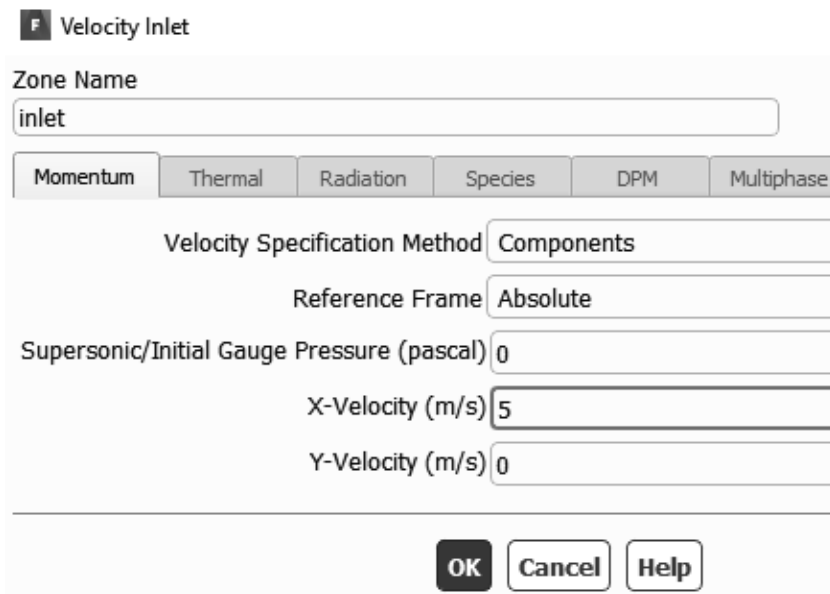


Figure 2.8b) Inlet velocity

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

F Wall

Zone Name
ideal_wall

Adjacent Cell Zone
surface_body

Momentum Thermal Radiation Species DPM Multiphase UDS

Wall Motion **Motion**

Stationary Wall Relative to Adjacent Cell Zone
 Moving Wall

Shear Condition **Shear Stress**

No Slip X-Component (pascal) 0
 Specified Shear Y-Component (pascal) 0
 Specularity Coefficient
 Marangoni Stress

Wall Roughness

Roughness Height (m) 0
Roughness Constant 0.5

OK Cancel Help

Figure 2.8c) Specified shear for ideal wall

9. 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

Filter Text

- Setup
 - General
 - + Models
 - + Materials
 - + Cell Zone Conditions
 - + Boundary Conditions
 - Dynamic Mesh
 - Reference Values
 - + Reference Frames
 - Named Expressions
- Solution
 - Methods
 - Controls

Task Page

Solution Methods

Pressure-Velocity Coupling

Scheme
Coupled

Spatial Discretization

Gradient
Least Squares Cell Based

Pressure
Standard

Momentum
First Order Upwind

Figure 2.9a) Solution Methods Task Page

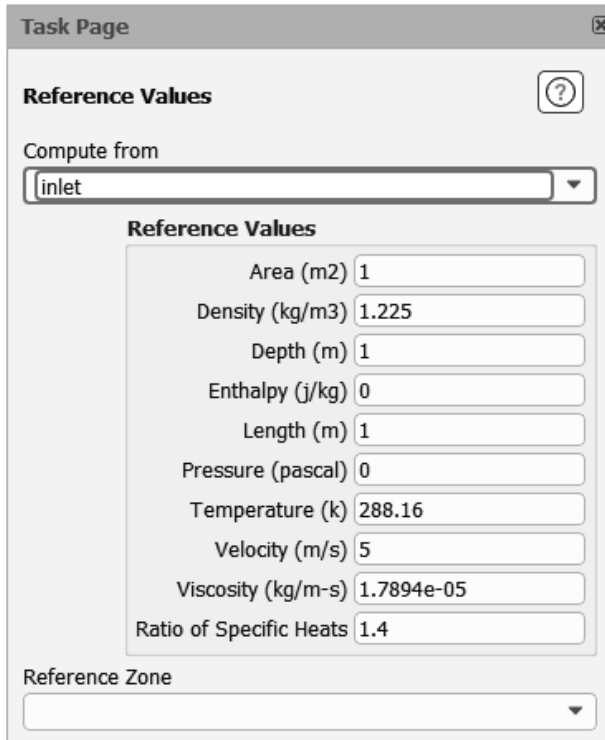


Figure 2.9b) Reference values

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

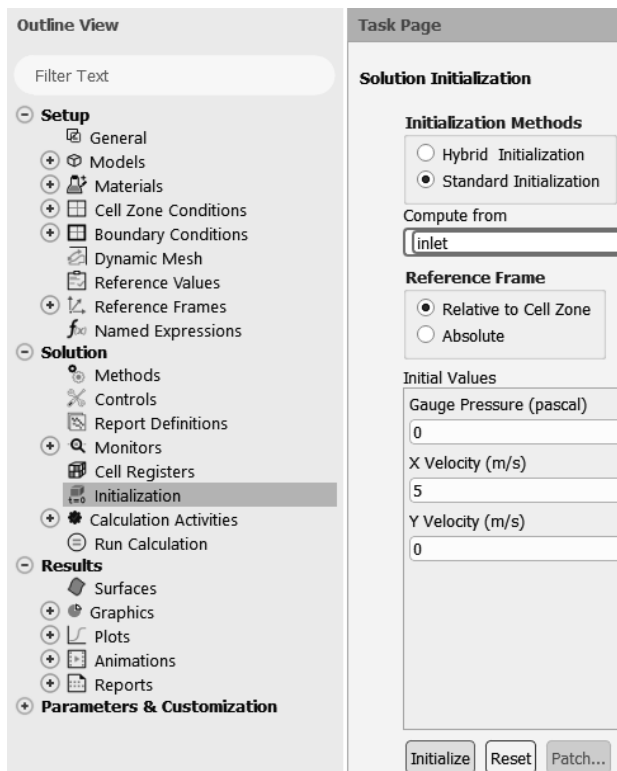


Figure 2.10 Solution Initialization

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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.

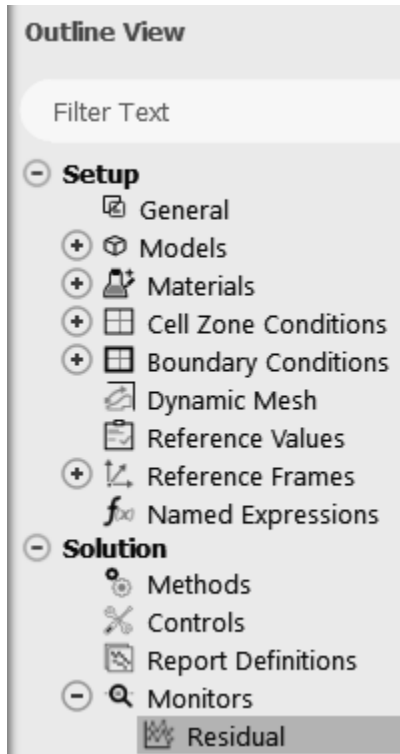


Figure 2.11a) Residual monitors

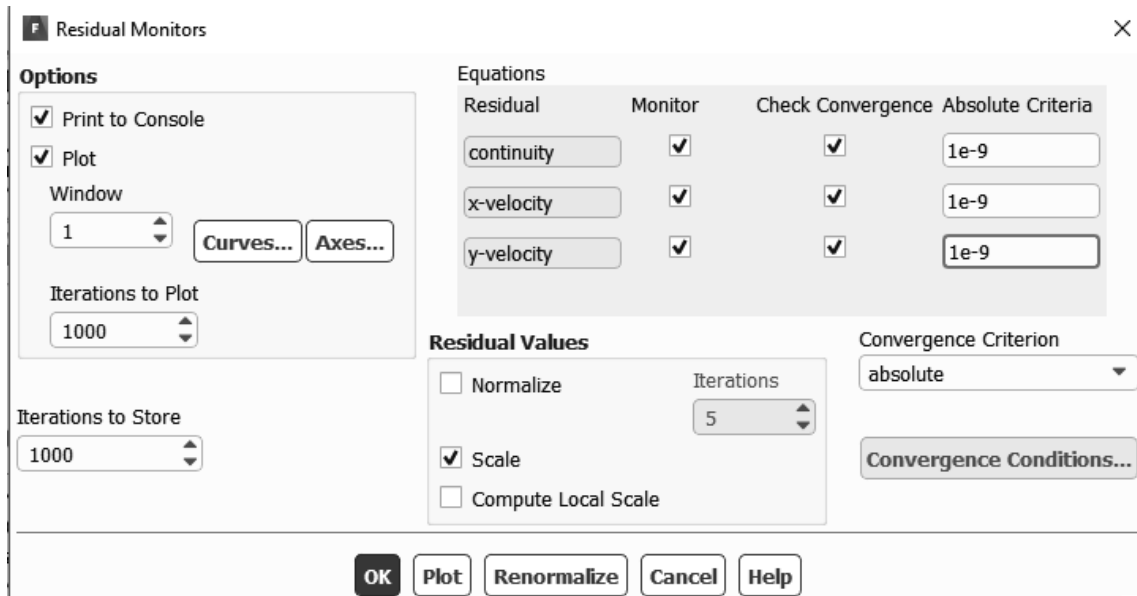


Figure 2.11b) Residual Monitors settings

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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



Figure 2.12a) Running the calculations

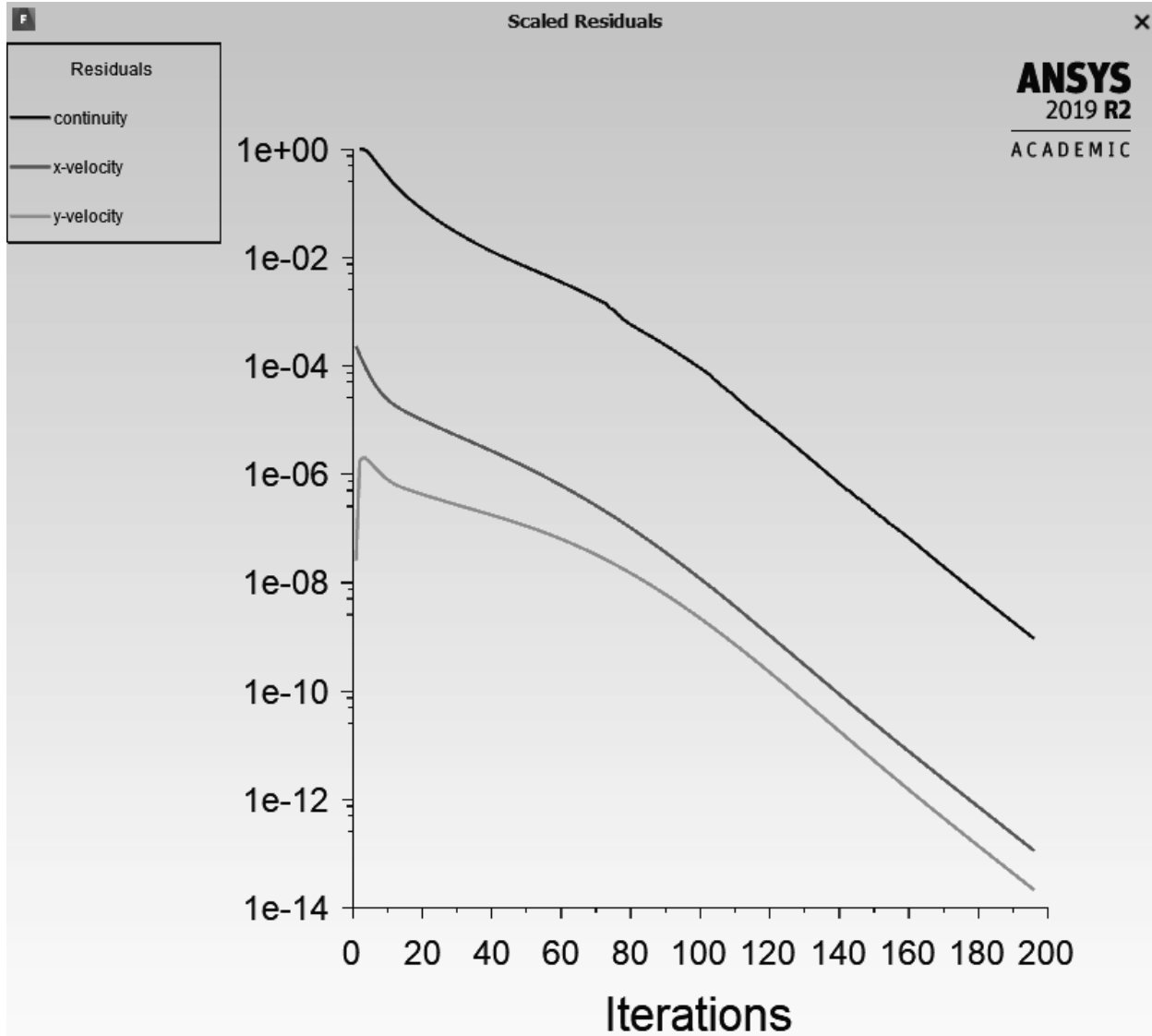


Figure 2.12b) Scaled Residual

G. Post-Processing

13. Select the Results tab in the menu and select Create>>Line/Rake under Surface. Enter 0.2 for x_0 (m), 0.2 for x_1 (m), 0 for y_0 (m), and 0.02 m for y_1 (m). Enter $x=0.2$ m for the New Surface Name and click on Create. Repeat this step three more times and create vertical lines at $x = 0.4$ m (length 0.04 m), $x=0.6$ m (length 0.06 m), and $x=0.8$ m (length 0.08 m). Close the window.

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

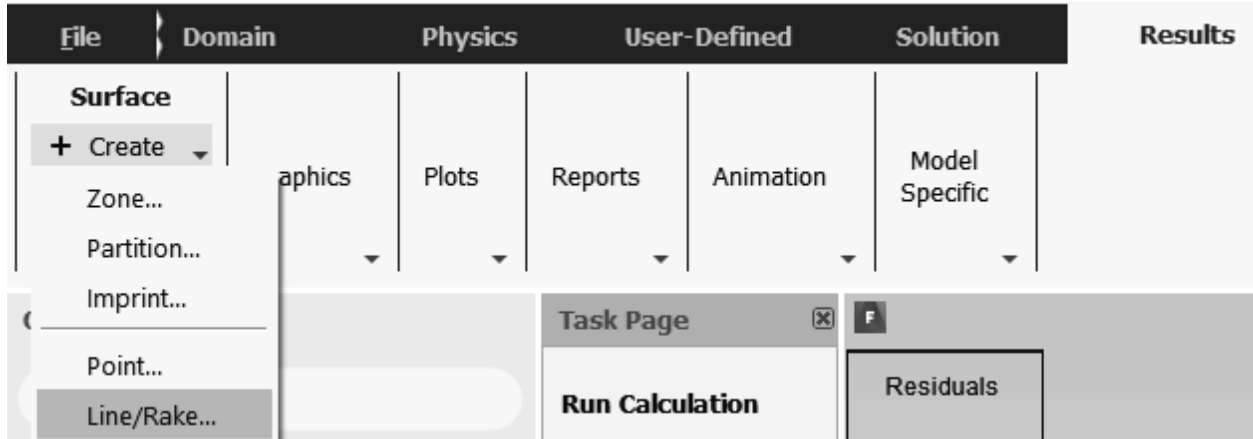


Figure 2.13a) Selecting Line/Rake from the Post-processing menu

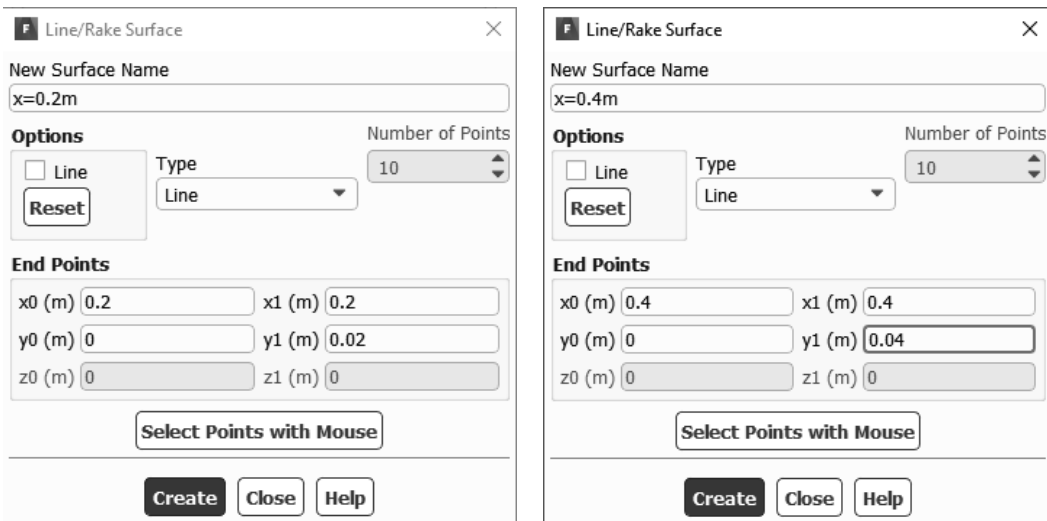


Figure 2.13b) Line/Rake Surfaces at $x=0.2$ m and $x=0.4$ m

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 $x=0.2$ m, $x=0.4$ m, $x=0.6$ m, and $x=0.8$ m. Click on the *Axes* button.

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

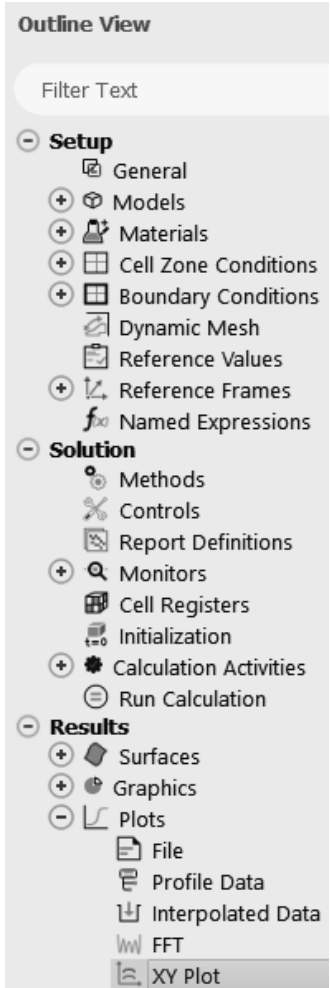


Figure 2.14a) XY Plot setup

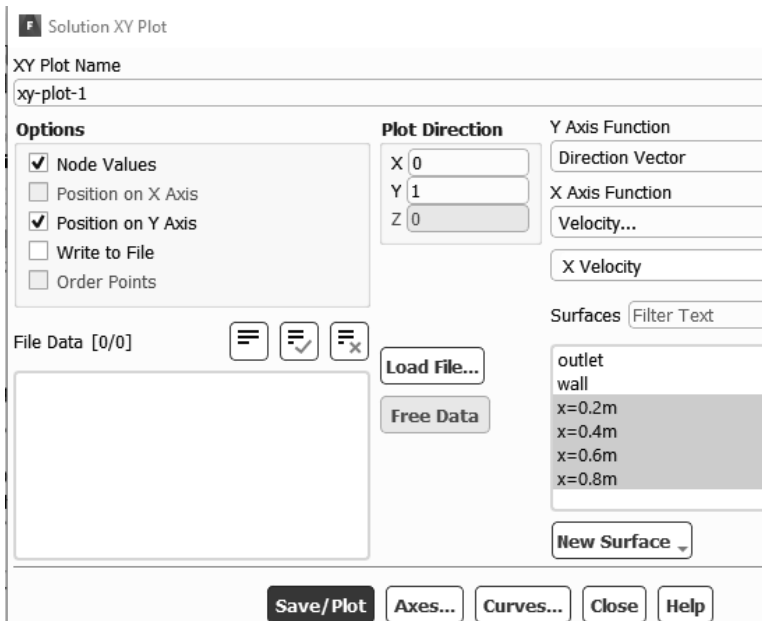


Figure 2.14b) Settings for Solution XY Plot

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

15. 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.

The screenshot shows the 'Axes - Solution XY Plot' dialog box. The 'Axis' section has the X-axis selected. The 'Number Format' section has 'Type' set to 'general' and 'Precision' set to 0. The 'Range' section has 'Maximum' set to 6. The 'Major Rules' section has 'Color' set to 'foreground' and 'Weight' set to 1. The 'Options' section has 'Auto Range' unchecked. The 'Minor Rules' section has 'Color' set to 'dark gray' and 'Weight' set to 1. The 'Apply' button is highlighted.

Figure 2.15a) Settings for X Axes

The screenshot shows the 'Axes - Solution XY Plot' dialog box. The 'Axis' section has the Y-axis selected. The 'Number Format' section has 'Type' set to 'general' and 'Precision' set to 2. The 'Range' section has 'Maximum' set to 0.01. The 'Major Rules' section has 'Color' set to 'light gray' and 'Weight' set to 1. The 'Options' section has 'Auto Range' unchecked. The 'Minor Rules' section has 'Color' set to 'light gray' and 'Weight' set to 1. The 'Apply' button is highlighted.

Figure 2.15b) Settings for Y Axes

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

- 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.

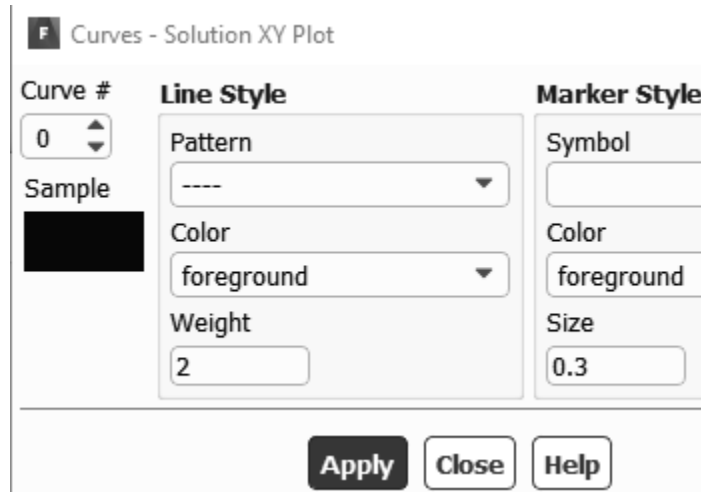


Figure 2.16a) Settings for curve # 0

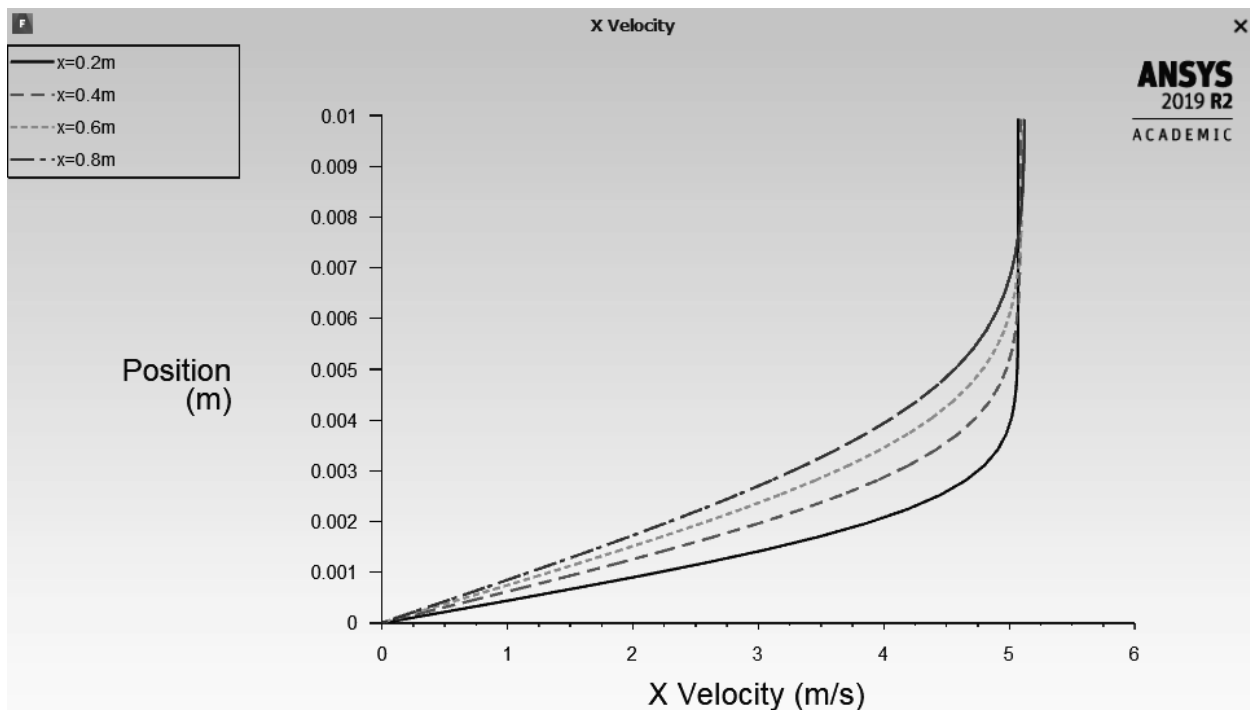


Figure 2.16b) Velocity profiles for a laminar boundary layer on a flat plate

Select the *User Defined* tab in the menu and *Custom*. Select *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

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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.

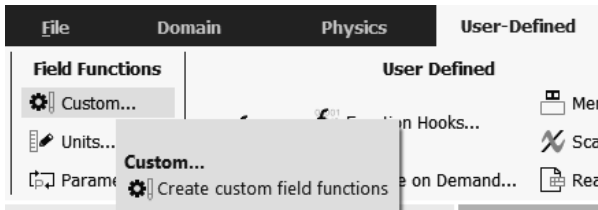


Figure 2.16c) Custom Field Functions

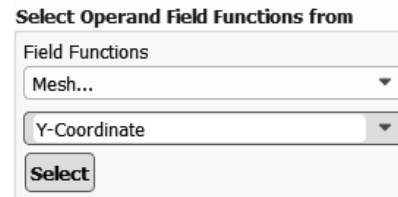


Figure 2.16d) Operand Field Function

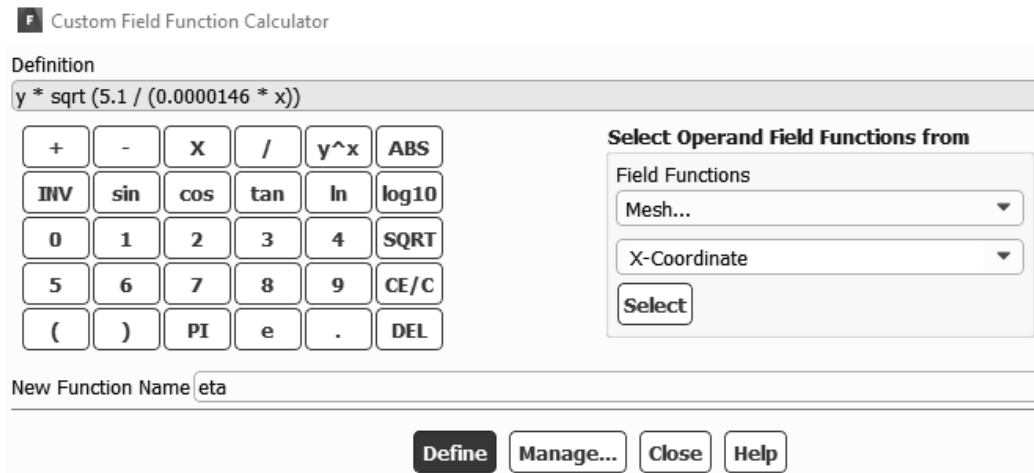


Figure 2.16e) Custom field function for self-similar coordinate

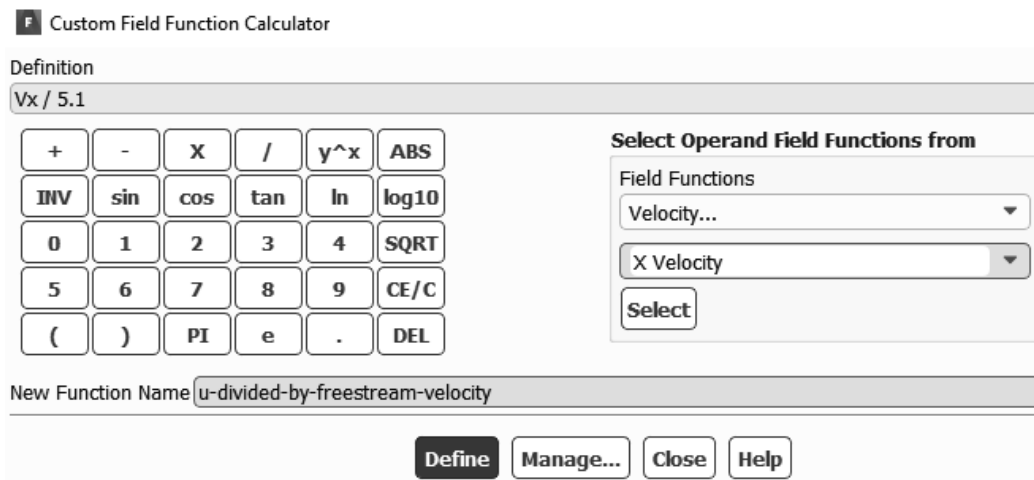


Figure 2.16f) Custom field function for non-dimensional velocity

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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*.

Place the file “*blasius.dat*” in your working directory. This file can be downloaded from *sdcpublications.com*. See Figure 1.19 for the Mathematica code used to generate the theoretical Blasius velocity profile for laminar boundary layer flow over a flat plate. In this case the working directory is *C:\Users\John Matsson*. Click on *Load File*. Select Files of type: All Files (*) and select the file “*blasius.dat*”. Select the four surfaces *x=0.2m*, *x=0.4m*, *x=0.6m*, *x=0.8m* and the loaded file *Theory*. Click on the Axes button.

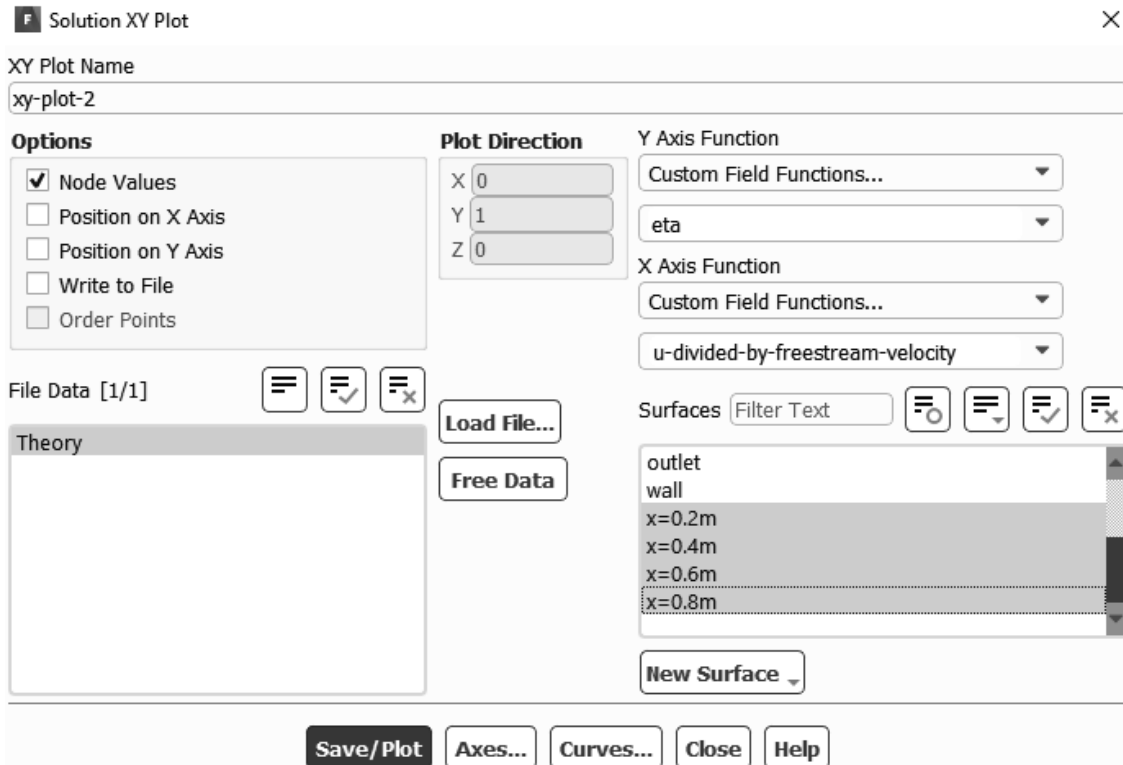


Figure 2.17a) Solution XY Plot for self-similar velocity profiles

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 Precision to 0 under Number Format. Set the Label to *eta* and click on *Apply*.

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

F Axes - Solution XY Plot

Axis <input type="radio"/> X <input checked="" type="radio"/> Y Label <input type="text" value="eta"/>	Number Format Type <input type="text" value="float"/> Precision <input type="text" value="0"/>	Major Rules Color <input type="text" value="foreground"/> Weight <input type="text" value="1"/>
Options <input type="checkbox"/> Log <input type="checkbox"/> Auto Range <input type="checkbox"/> Major Rules <input type="checkbox"/> Minor Rules	Range Minimum <input type="text" value="0"/> Maximum <input type="text" value="10"/>	Minor Rules Color <input type="text" value="dark gray"/> Weight <input type="text" value="1"/>

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

F Axes - Solution XY Plot

Axis <input checked="" type="radio"/> X <input type="radio"/> Y Label <input type="text" value="u/U"/>	Number Format Type <input type="text" value="float"/> Precision <input type="text" value="1"/>	Major Rules Color <input type="text" value="foreground"/> Weight <input type="text" value="1"/>
Options <input type="checkbox"/> Log <input checked="" type="checkbox"/> Auto Range <input type="checkbox"/> Major Rules <input type="checkbox"/> Minor Rules	Range Minimum <input type="text" value="0"/> Maximum <input type="text" value="0"/>	Minor Rules Color <input type="text" value="dark gray"/> Weight <input type="text" value="1"/>

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

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

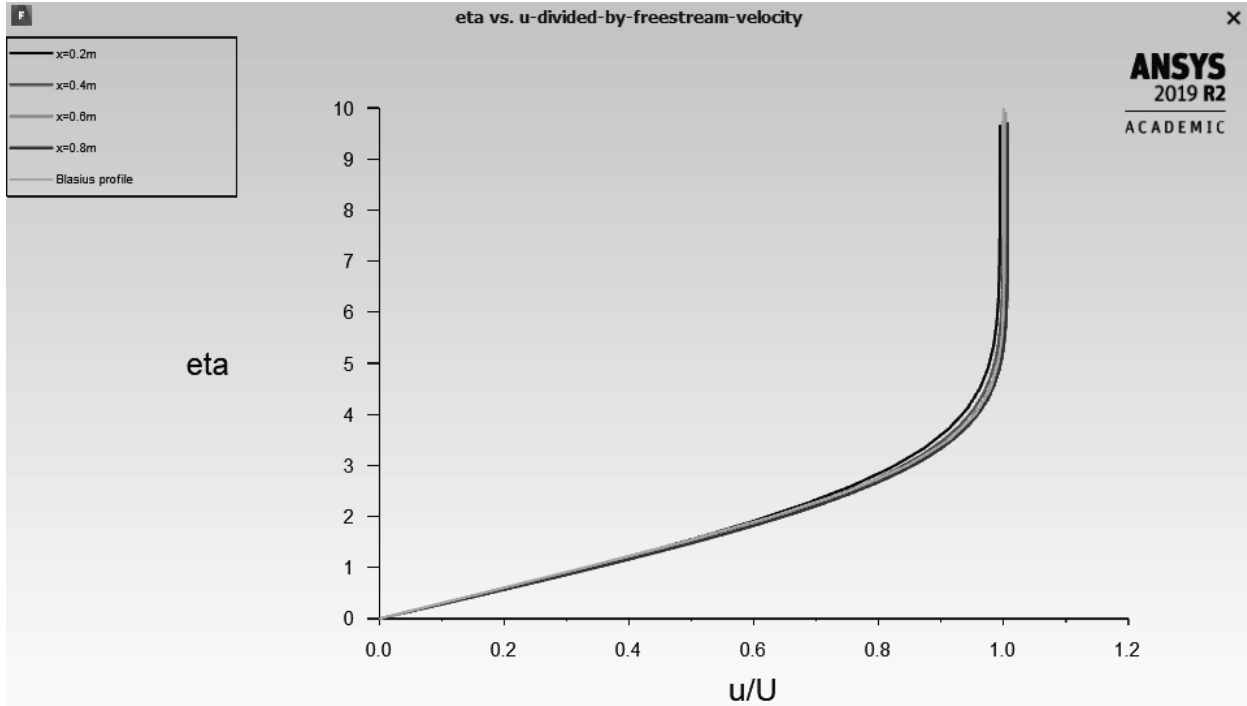


Figure 2.17d) Self-similar velocity profiles for a laminar boundary layer on a flat plate

Select the User Defined tab in the menu and Custom. Select 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.

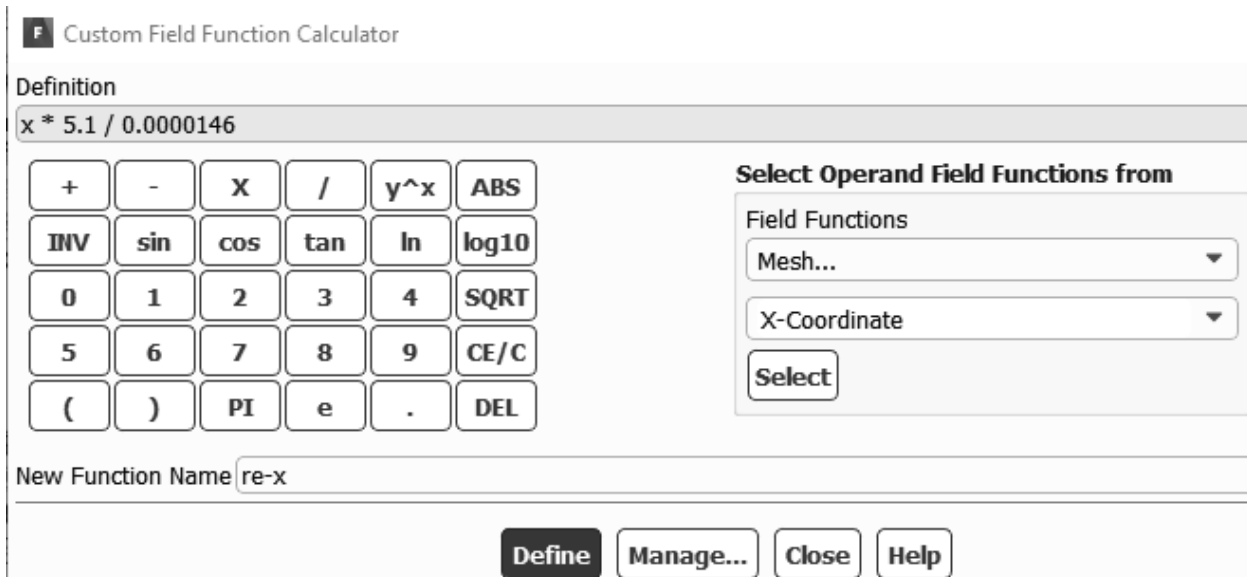


Figure 2.17e) Custom field function for Reynolds number

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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.

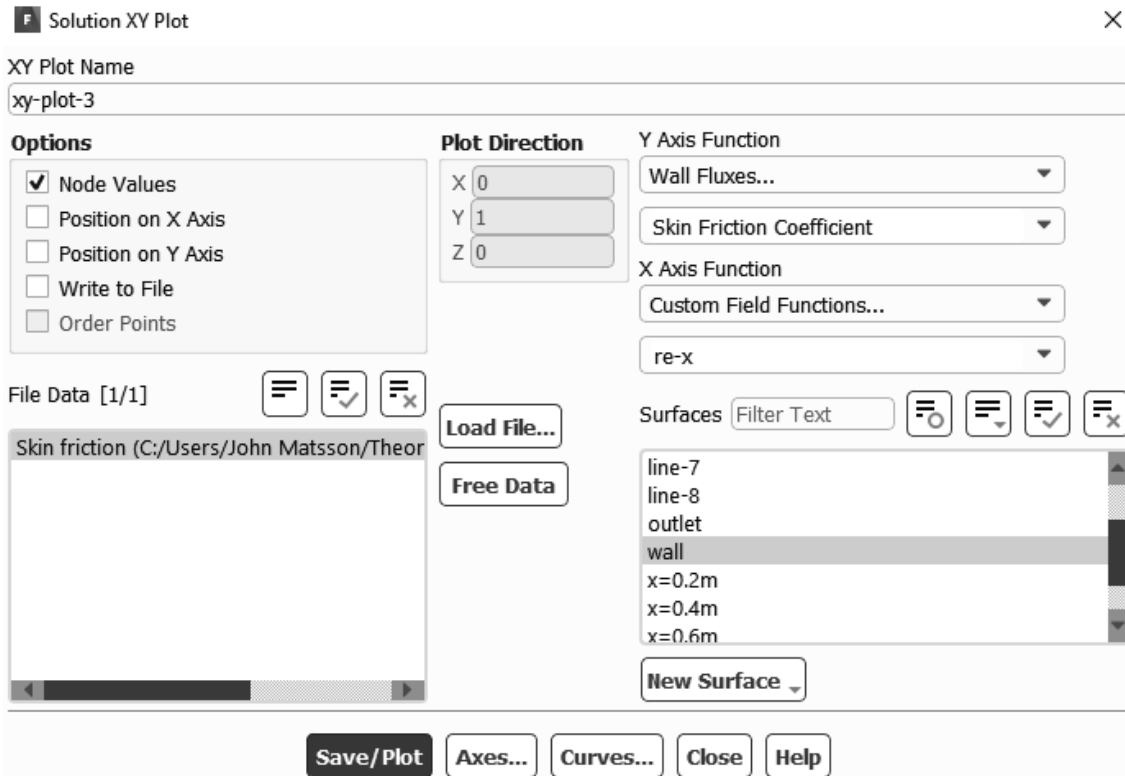


Figure 2.18a) Solution XY Plot for skin friction coefficient

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 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 Precision to 3 and click on Apply. Close the window. Click on Save/Plot in the Solution XY Plot window.

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

F Axes - Solution XY Plot

Axis	Number Format	Major Rules
<input checked="" type="radio"/> X <input type="radio"/> Y	Type float	Color foreground
Label Re-x	Precision 0	Weight 1
Options	Range	Minor Rules
<input checked="" type="checkbox"/> Log <input type="checkbox"/> Auto Range <input type="checkbox"/> Major Rules <input type="checkbox"/> Minor Rules	Minimum 100 Maximum 1000000	Color dark gray Weight 1

Apply **Close** **Help**

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

F Axes - Solution XY Plot

Axis	Number Format	Major Rules
<input type="radio"/> X <input checked="" type="radio"/> Y	Type float	Color foreground
Label Cf-x	Precision 3	Weight 1
Options	Range	Minor Rules
<input checked="" type="checkbox"/> Log <input type="checkbox"/> Auto Range <input type="checkbox"/> Major Rules <input type="checkbox"/> Minor Rules	Minimum 0.001 Maximum 0.1	Color dark gray Weight 1

Apply **Close** **Help**

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

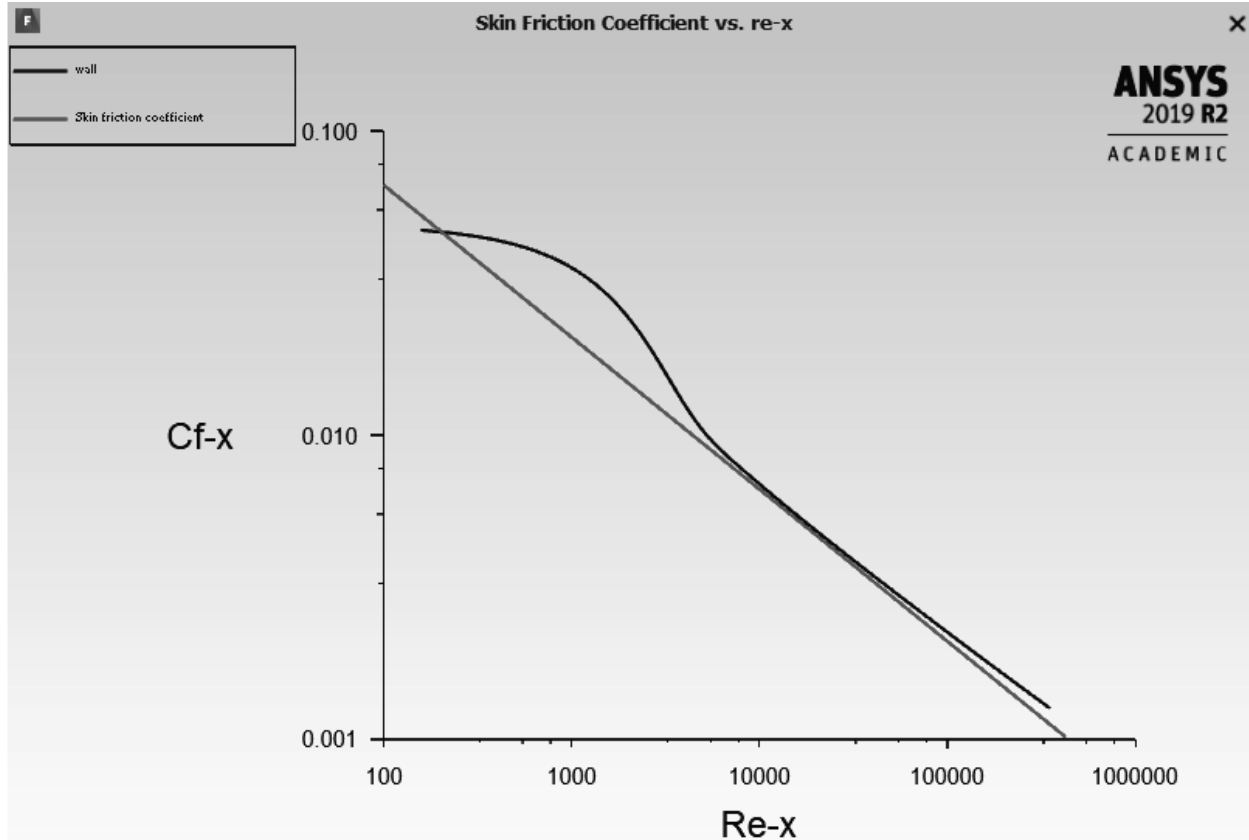


Figure 2.18d) Comparison between ANSYS Fluent and theoretical skin friction coefficient for laminar boundary layer flow on a flat plate

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. The similarity coordinate is defined by

$$\eta = y \sqrt{\frac{U}{\nu x}} \quad (2.1)$$

where y (m) is the wall normal coordinate, U (m/s) is the free stream velocity, x (m) is the distance from the start of the wall and ν (m^2/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

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

with the following boundary conditions

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

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

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

$$Re_x = \frac{Ux}{\nu} \quad (2.4)$$

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 with streamwise distance x and Reynolds number Re_x .

$$\delta = \frac{4.91x}{\sqrt{Re_x}} \quad (2.5)$$

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

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

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

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

The theoretical local friction coefficient for laminar flow is determined by

$$C_{f,x} = \frac{0.664}{\sqrt{Re_x}} \quad Re_x < 5 \cdot 10^5 \quad (2.8)$$

and for turbulent flow we have the following relation

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

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

```

sol = NDSolve[{{f'''[η] + 0.5 f[η] f''[η] == 0, f[0] = f'[0] = 0, f'[10] == 1}, f, η];
myplot = Plot[f'[η] /. First[sol], {η, 0, 10}, AxesLabel → {"u/U", "η"}];
axisFlip # /. {x_Line | x_GraphicsComplex ⇒ MapAt[#, Reverse~2 &, x, 1],
  x : (PlotRange → _) ⇒ x~Reverse~2} &;
myplot // axisFlip

SetDirectory["C:\\Users\\jmatsson"];
mylist = Table[{f'[η] /. First[sol], η}, {η, 0, 10, 0.5}];
TableOfValues1 = Prepend[mylist, {""}];
TableOfValues1 = Prepend[TableOfValues1, {"(xy/key/label \"Blasius profile\")"}];
TableOfValues1 = Prepend[TableOfValues1, {""}];
TableOfValues1 = Prepend[TableOfValues1, {"(labels \"X Velocity\" \"η-Coordinate\")"}];
TableOfValues1 = Prepend[TableOfValues1, {"(title \"Theory\")"}];
TableOfValues1 = Append[TableOfValues1, {""}];
Grid[TableOfValues1]
Export["blasius.dat", TableOfValues1]

```

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.

x (m)	δ (mm) Fluent	δ (mm) Theory	Percent Difference	U_δ (m/s)	U (m/s)	ν (m ² /s)	Re_x
0.2						0000146	
0.4						0000146	
0.6						0000146	
0.8						0000146	

Table 2.1 Comparison between Fluent and theory for boundary layer thickness

CHAPTER 2. FLAT PLATE BOUNDARY LAYER

- 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 finally create an XY Plot for the skin friction coefficient versus Reynolds number.
- 2.4 Use the results from the ANSYS Fluent simulation in Exercise 1.3 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.

x (m)	δ (mm) Fluent	δ (mm) Theory	Percent Difference	U_δ (m/s)	U (m/s)	ν (m ² /s)	Re_x
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