

Laminar Boundary Layer

1 Problem Description

[Files and video.](#)

- The flow Reynolds number is around 3.5×10^4 at the end of the plate, thus the entire boundary layer is laminar. The Laminar Model is hence used.
- Boundary conditions for the model:
 - Inlet: velocity inlet $u = 1 \text{ m/s}$
 - Top: velocity inlet $u = 1 \text{ m/s}$
 - Symmetry: symmetry
 - Outlet: pressure outlet $p = 0 \text{ Pa}$ (relative to atmospheric pressure)
 - Flat Plate: no-slip wall
- A steady solution is obtained using the pressure-based Coupled Pseudo Transient solver in Ansys Fluent.

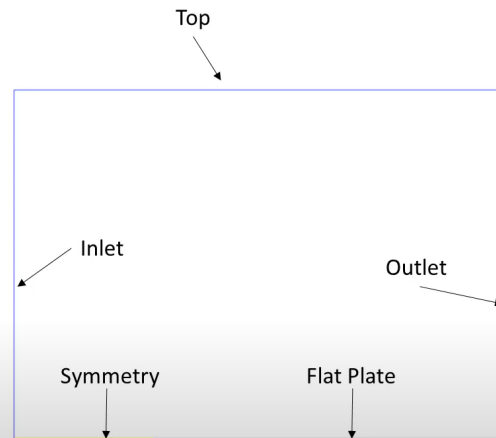


Figure 1: Problem Description.

2 Import Mesh

2.1 Get Mesh File

Open Fluent Launcher, then click **Solution**. Click **Mesh** on the right hand side, then choose **flat-plate-05m.msh**, click **Start With Selected Options**, shown in Fig. 2.

2.2 Check Mesh

Click **Check, Perform Mesh Check**

2.3 Evaluate Mesh

Click **Quality, Evaluate Mesh Quality**

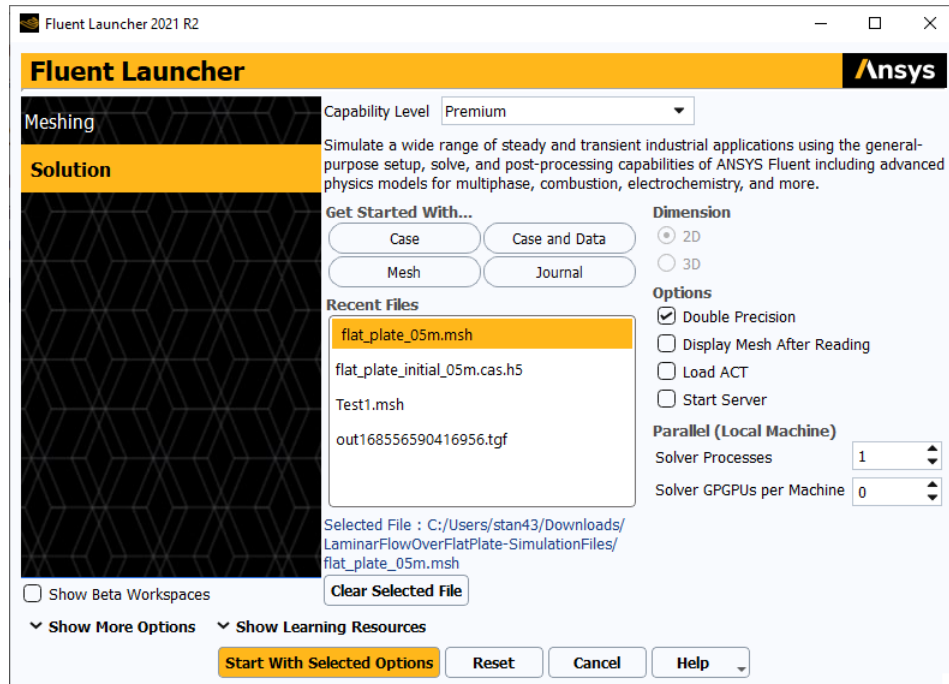


Figure 2: Fluent Launcher.

2.4 Display Mesh

Click Display, Edges, Colors, Display, Close, as shown in Fig. 3

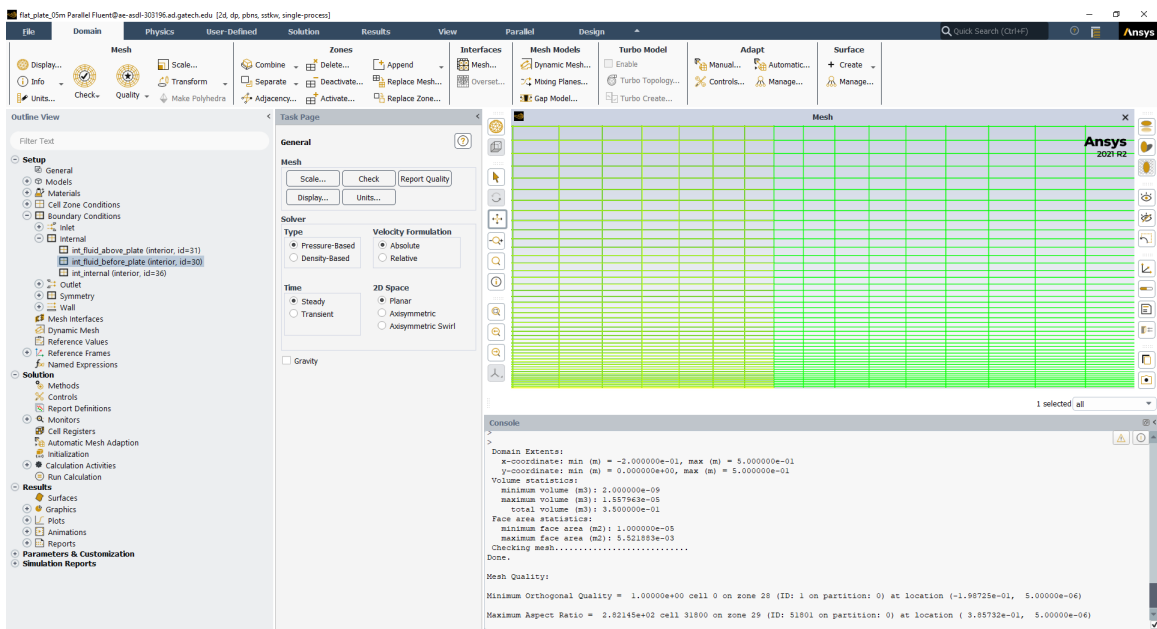


Figure 3: Display Mesh.

3 Physics Setup

3.1 General Setting

Click **Physics**, **General**, keep all the default values.

3.2 Viscous Model

Click **Viscous**, **Laminar**, **OK**, as shown in Fig. 4

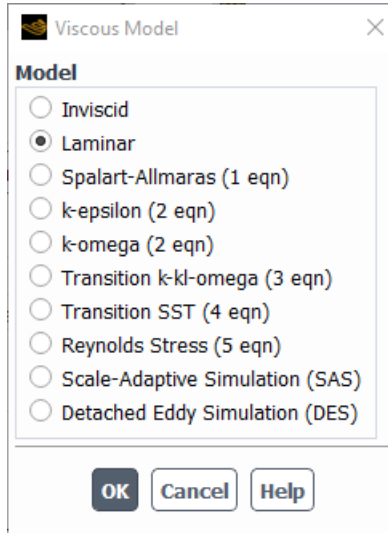


Figure 4: Viscous Model.

3.3 Flow Properties

Click **Create/Edit Materials**, keep all the default values, as shown in Fig. 5.

3.4 Cell Zone

Click **Cell Zones**, make sure every zone's type is fluid, as shown in Fig. 6.

3.5 Boundary Conditions

1. Click **Boundaries**, **Group by Zone Type**
2. Click **inlet**, choose the type as **velocity-inlet**, then click **Edit**. Change the **Velocity Specification Method** to **Components**, then set **X-Velocity** as 1, as shown in Fig. 7
3. Click **Outlet**, choose the type as **pressure-outlet**, keep all the default values.
4. Click **top-f**, choose the type as **velocity-outlet**, change the **Velocity Specification Method** to **Components**, then set **X-Velocity** as 1.
5. Click **top-r**, same set up as **top-f**.

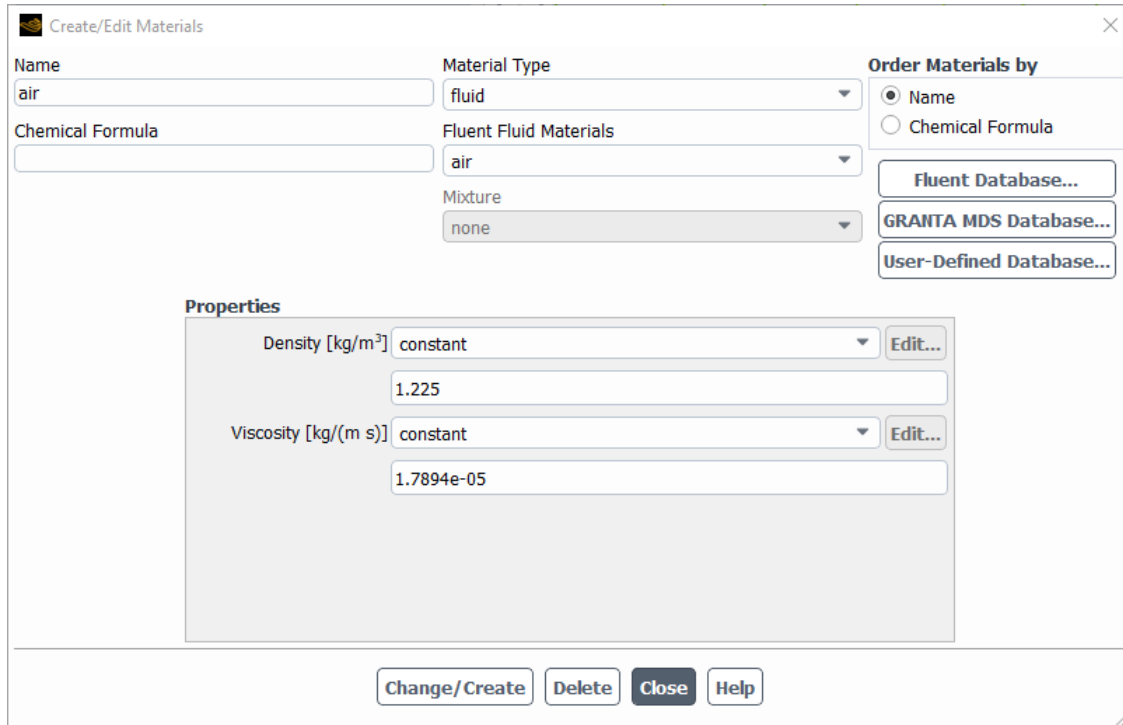


Figure 5: Flow Properties.

3.6 Reference Values

Click **Reference Values**, keep all the default values, as shown in Fig. 8

4 Numerical Solution

4.1 Choose CFD Method

Click **Solution**, **Solution Methods**, keep all the default values, as shown in Fig. 9

4.2 Choose Control Factors

Click **Controls**, keep all the default values, as shown in Fig. 10

4.3 Set Residuals

Click **Residuals**, set all the residuals as $1e-6$, as shown in Fig. 11

4.4 Definition Report

Click **Definitions**, **New**, **Flux Report**, **Mass Flow Rate**, select **inlet**, **top-f**, **top-r**, **outlet**, **symmetry**, click **Report Plot**, **Print to Console**, as shown in Fig. 12.

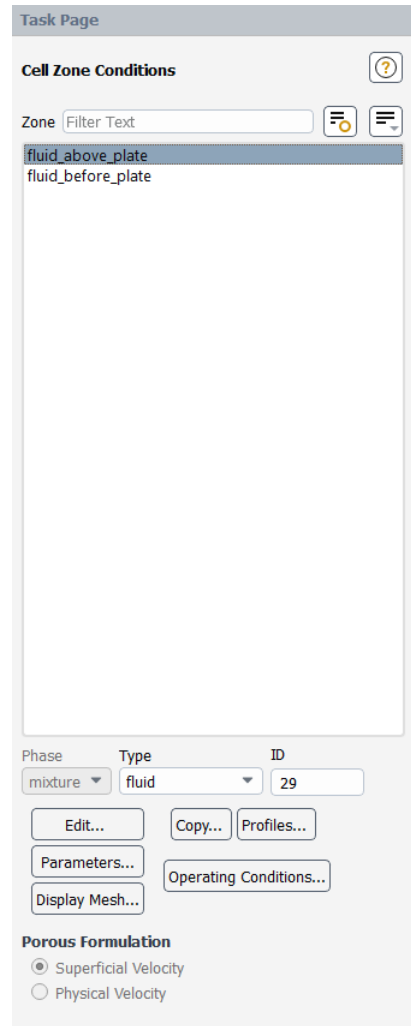


Figure 6: Cell Zone.

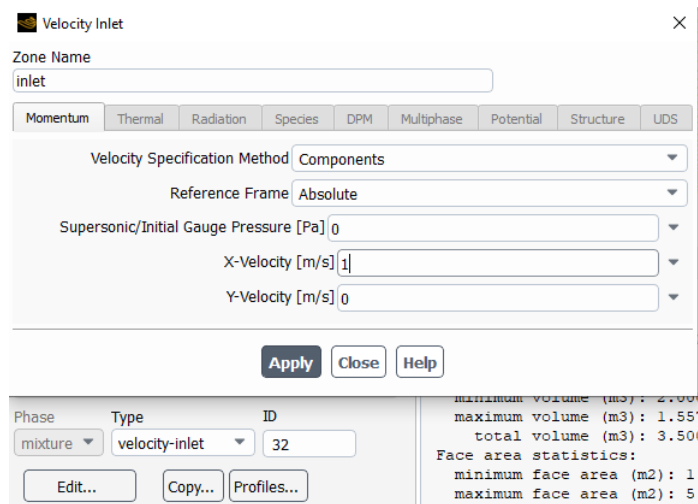


Figure 7: Inlet Boundary Conditions.

Task Page

Reference Values

Compute from

Reference Values

Area [m ²]	1
Density [kg/m ³]	1.225
Depth [m]	1
Enthalpy [J/kg]	0
Length [m]	1
Pressure [Pa]	0
Temperature [K]	288.16
Velocity [m/s]	1
Viscosity [kg/(m s)]	1.7894e-05
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300

Reference Zone

Figure 8: Reference Values.

Solution Methods

Pressure-Velocity Coupling

Scheme: Coupled

Flux Type: Rhie-Chow: distance based ☐ Auto Select

Spatial Discretization

Gradient: Least Squares Cell Based

Pressure: Second Order

Momentum: Second Order Upwind

Transient Formulation

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☒ Pseudo Transient

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation

Structure Transient Formulation

Default

Figure 9: Solution Method.

4.5 Initialize Solution

Click **Initialize**. Change **No. of Iterations** to 500. Then click **Check Case**.

4.6 Save Data

Click **File, Write, Case/Data**, save this into a .cas.h5 file, label as initial.

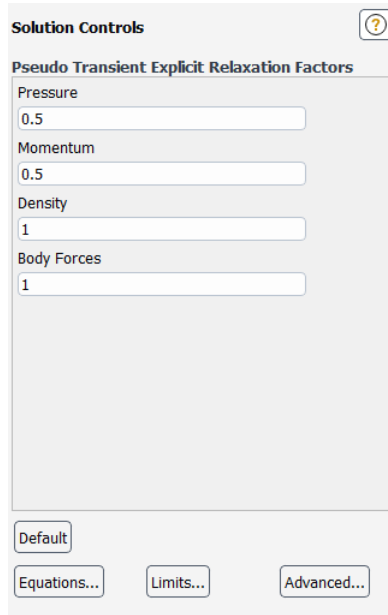


Figure 10: Solution Control.

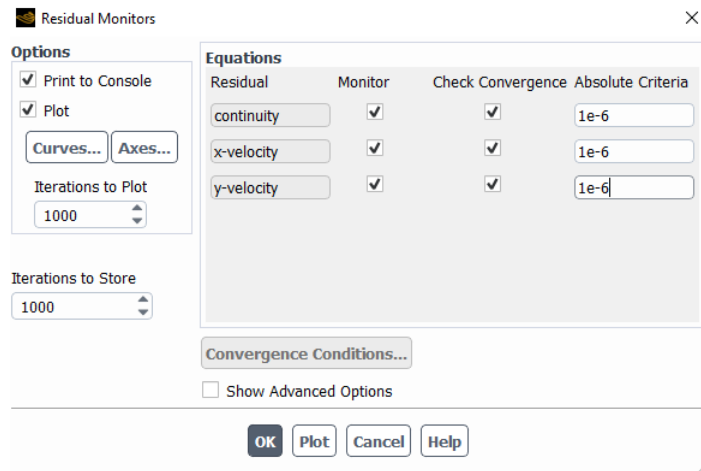


Figure 11: Solution Residual.

4.7 Calculate

Click **Calculate**

5 Results

5.1 Check Residuals

Click **Scales Residuals** plot, as shown in Fig. 13

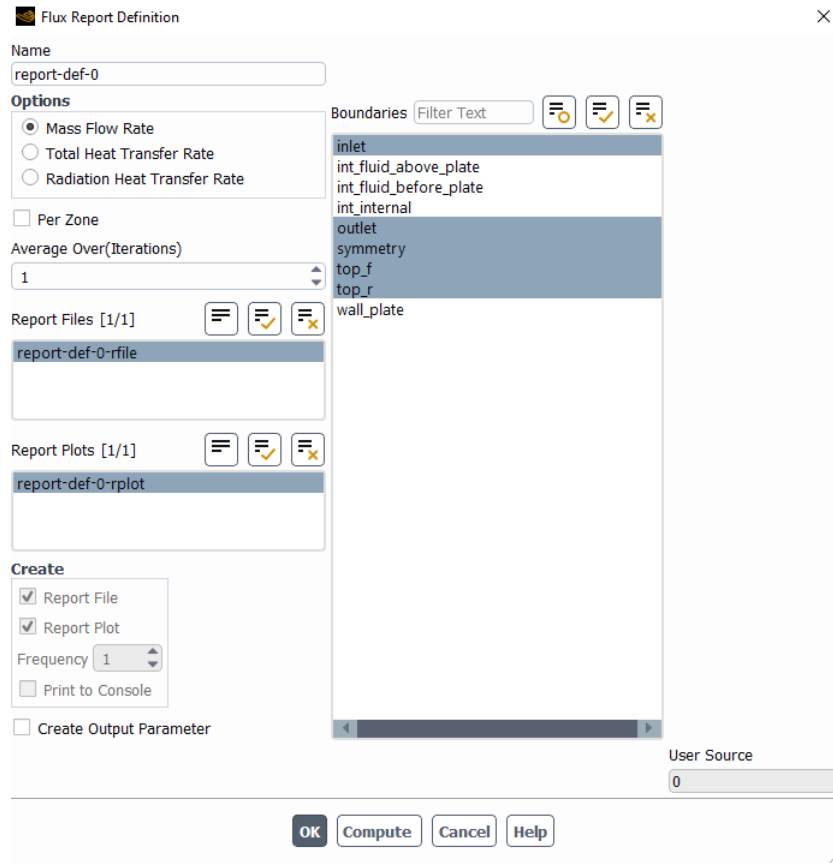


Figure 12: Definition Report.

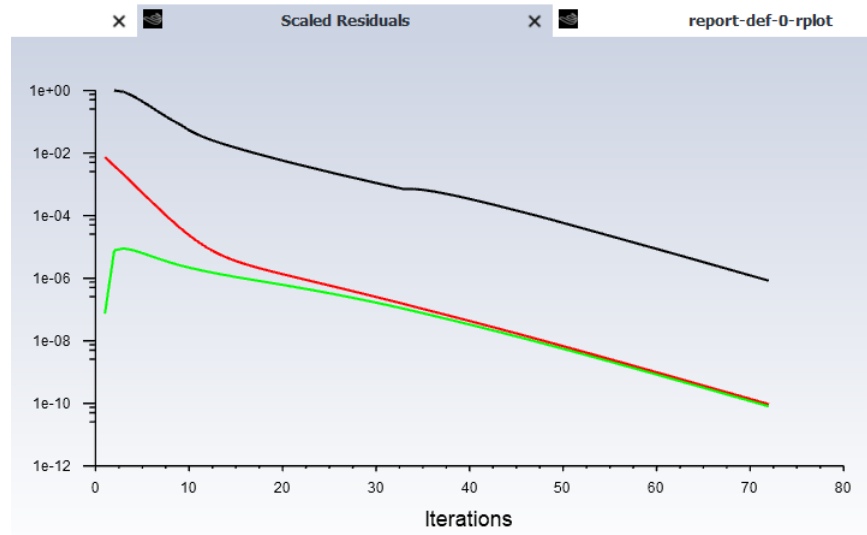


Figure 13: Check Residual.

5.2 Velocity Vector Plot

Click **Results, Vectors, New**, change style to **filled-arrow** and scale to 0.03 as shown in Fig. 14. The plot is shown in Fig. 15

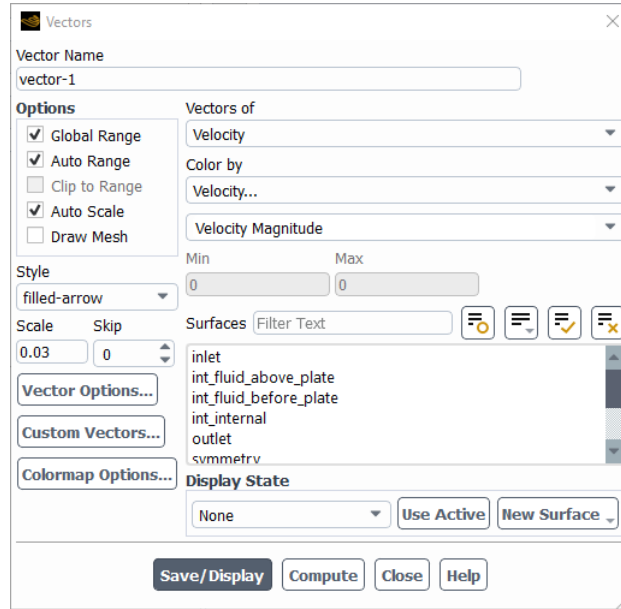


Figure 14: Vector Plot Settings.

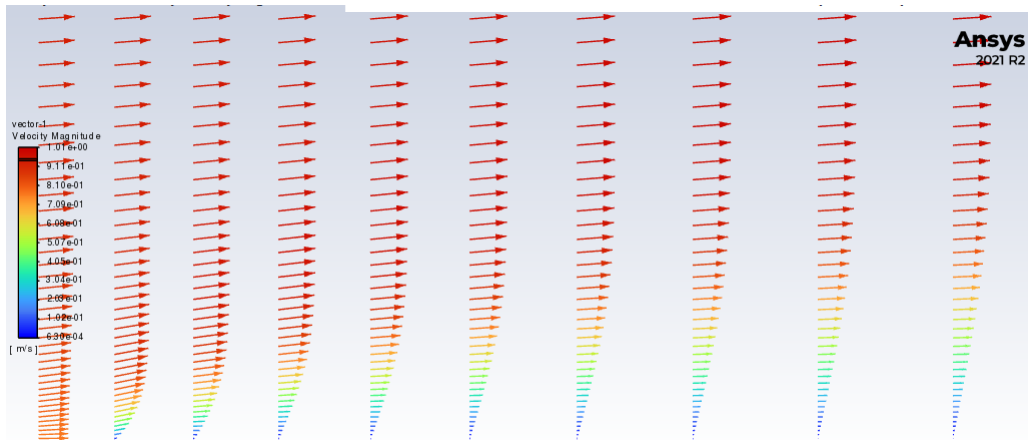


Figure 15: Vector Plot.

5.3 Velocity Contour Plot

Click **Contours, Velocity, Display**. The plot is shown in Fig. 16

5.4 Create Line Data

Click **Create, Line/Rake Surface**, change $x_0 = 0.1$, $y_0 = 0$, $x_1 = 0.1$, $y_1 = 0.1$, then save the line, as shown in Fig. 17. Repeat this for $x_0 = 0.2, 0.3, 0.4, 0.5$.

5.5 Create XY Plot

Click **XY Plot, New**. Set **Plot Direction** $X = 0$, $Y = 1$. Set **Y Axis Function** as **Velocity, X Velocity**, select the lines saved before then display, as shown in Fig. 18. Then click **Write to File** to save the data into .xy file.

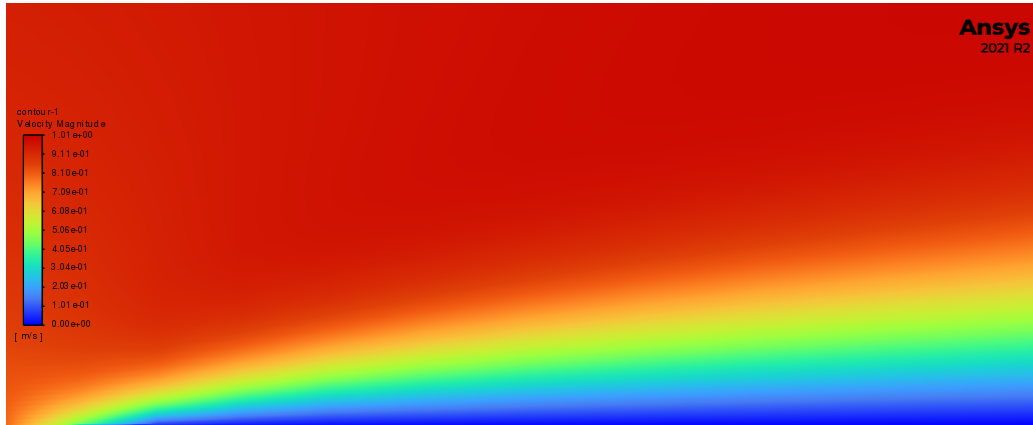


Figure 16: Contour Plot.

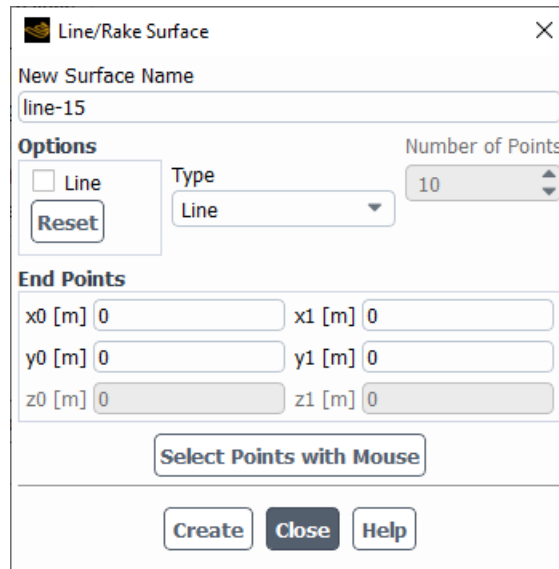


Figure 17: Create Lines.

5.6 Skin Coefficient Plot

Set **Plot Direction** $X = 1$, $Y = 0$. Set **Y Axis Function** as **Wall Fluxes, Skin Friction Coefficient**, then save and display, as shown in Fig. 19

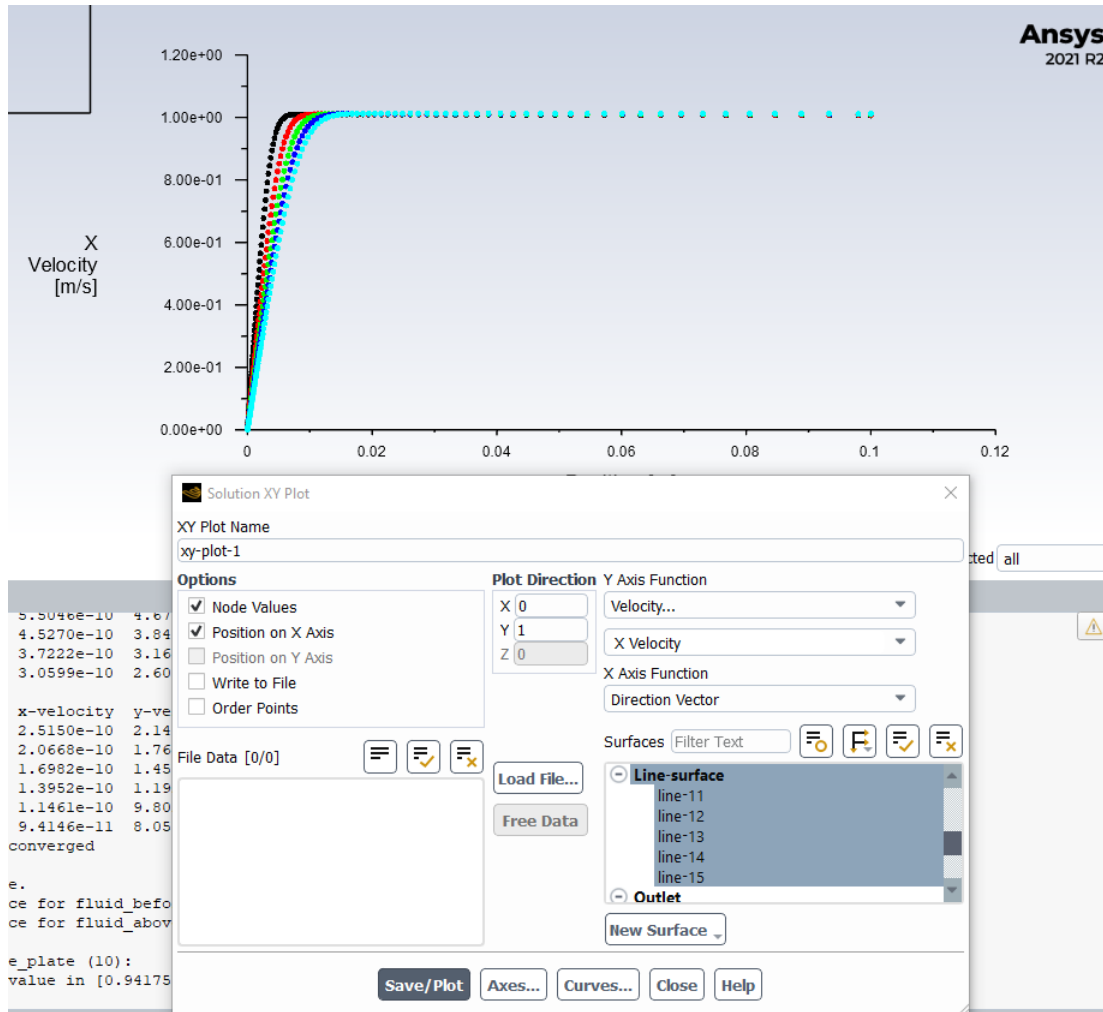


Figure 18: XY Plot.

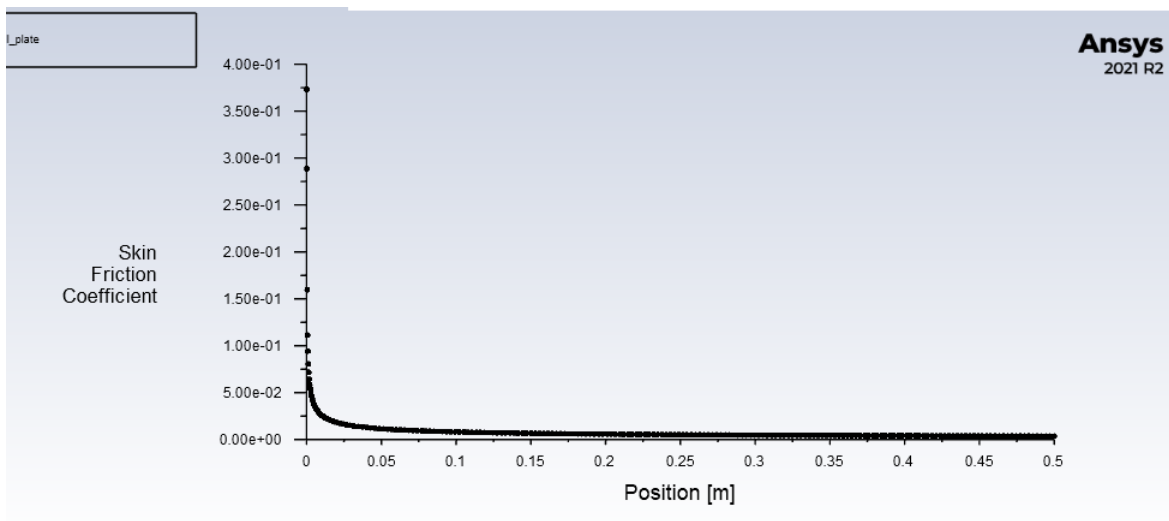


Figure 19: Skin Friction Plot.