

FLUENT - Forced Convection over a Flat Plate step 6

Problem Specification

1. Create Geometry in GAMBIT
2. Mesh Geometry in GAMBIT
3. Specify Boundary Types in GAMBIT
4. Set Up Problem in FLUENT
5. Solve
6. **Analyze Results**
7. Refine Mesh

Step 6: Analyze Results

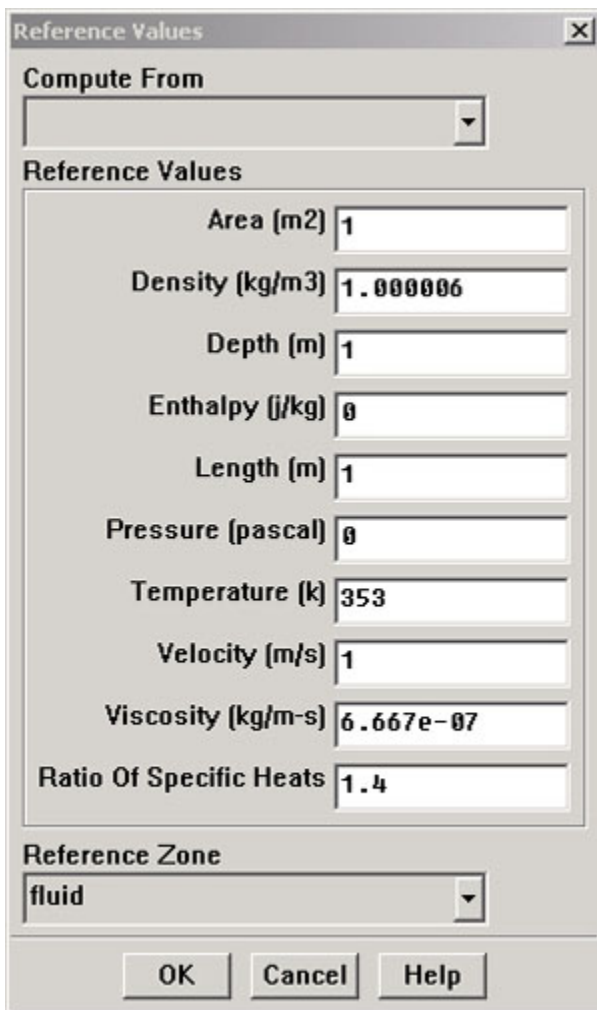
y+

Turbulent flows are significantly affected by the presence of walls. The *k-epsilon* turbulence model's validity is grid-independent away from walls but requires verification to make sure it is valid when used near walls. The near-wall model is sensitive to the grid resolution, which is assessed in the wall unit y^+ , as discussed in [Step 4](#).

First, we need to set the reference values needed to calculate y^+ .

Main Menu > Report > Reference Values...

Select *inflow* under *Compute From* to tell FLUENT to use values at the inflow for the reference values. Check that the reference value for velocity is 1 m/s , temperature is 353 K , and coefficient of viscosity is $6.667\text{e-}7\text{ kg/m-s}$ as given in the [Problem Specification](#). These reference values will be used to non-dimensionalize the distance of the cell center from the wall to obtain the corresponding y^+ values. Click **OK**.



The image shows the 'Reference Values' dialog box in FLUENT. It has a title bar with a close button. Inside, there is a 'Compute From' dropdown menu. Below it is a section titled 'Reference Values' containing several input fields with their respective units and values: Area (m2) is 1, Density (kg/m3) is 1.0000006, Depth (m) is 1, Enthalpy (J/kg) is 0, Length (m) is 1, Pressure (pascal) is 0, Temperature (K) is 353, Velocity (m/s) is 1, Viscosity (kg/m-s) is 6.667e-07, and Ratio Of Specific Heats is 1.4. At the bottom, there is a 'Reference Zone' dropdown menu set to 'fluid'. At the very bottom are three buttons: 'OK', 'Cancel', and 'Help'.

Parameter	Value
Area (m2)	1
Density (kg/m3)	1.0000006
Depth (m)	1
Enthalpy (J/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (K)	353
Velocity (m/s)	1
Viscosity (kg/m-s)	6.667e-07
Ratio Of Specific Heats	1.4

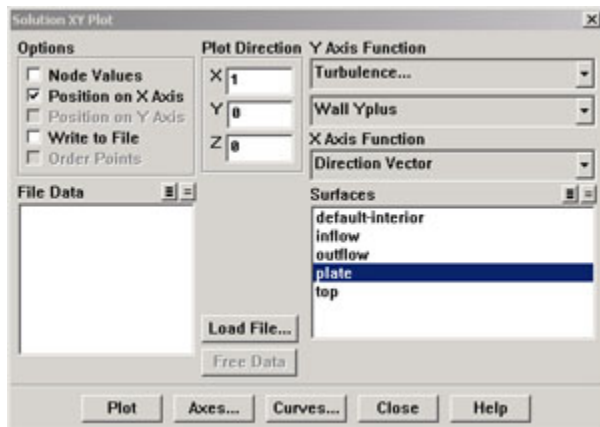
Reference Zone: fluid

Buttons: OK, Cancel, Help

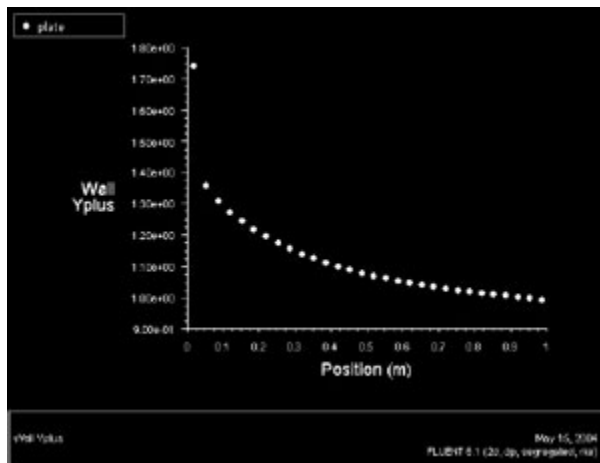
By using the following method, plot y^+ values for wall-adjacent cells to check how they compare with the recommendation mentioned above.

Main Menu > Plot > XY Plot...

Make sure that **Position on X Axis** is set under **Options**, that 1 is the value next to **X**, and 0 is the value next to **Y** under **Plot Direction**. Recall that this tells FLUENT to plot the x-coordinate value on the abscissa of the graph. Select **Turbulence...** under **Y Axis Function** and select **Wall Yplus** from the drop down list under that. Since we want the y^+ value for cells adjacent to the wall of the pipe, choose **plate** under **Surfaces**.



Click **Plot**.



[Higher Resolution Image](#)

As we can see, the wall y^+ value is between 1.0 and 1.4 (ignoring the anomalous at the inflow). Because these values are less than 5, the near-wall mesh resolution is in the laminar sublayer, which is the most accurate region to which we can resolve the boundary layer.

Save Plot

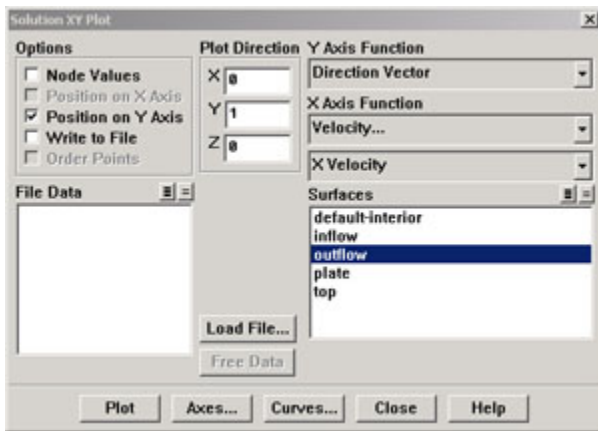
In the *Solution XY Plot Window*, check the **Write to File** box under **Options**. The **Plot** button should have changed to the **Write...** button. Click on **Write...**. Enter `yp1us.xy` as the filename and click **OK**. Check that this file has been created in your FLUENT working directory.

Velocity at $x = 1\text{m}$

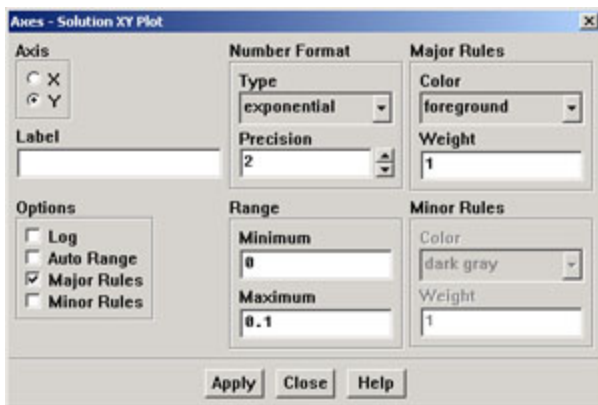
Main Menu > Plot > XY Plot...

Under **Options**, unselect **Position on X Axis** and select **Position on Y Axis**. Under **Plot Direction**, enter 0 in the **X** box and 1 in the **Y** box. This tells FLUENT to plot a vertical rather than horizontal profile.

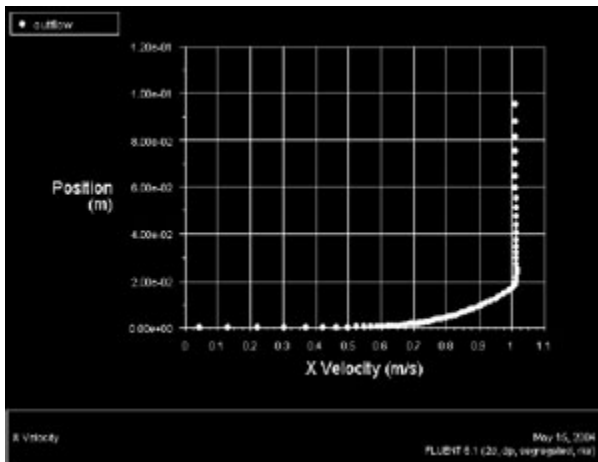
Under **X Axis Function**, pick **Velocity...** and then in the box under that, pick **X Velocity**. Finally, select **outflow** under **Surfaces** since we are plotting the velocity profile at the outflow. De-select **plate** under **Surfaces**.



Click on **Axes...** in the *Solution XY Plot* window. Select **X** in the Axis box. In the *Options* box select **Major Rules** to turn on the grid lines in the plot. Click **Apply**. Then select the **Y** in the Axis box, select **Major Rules** again, and turn off **Auto Range**. In the *Range* box enter 0.1 for the *Maximum* so that we may view the velocity profile in the boundary layer region more closely. Click **Apply** and **Close**.



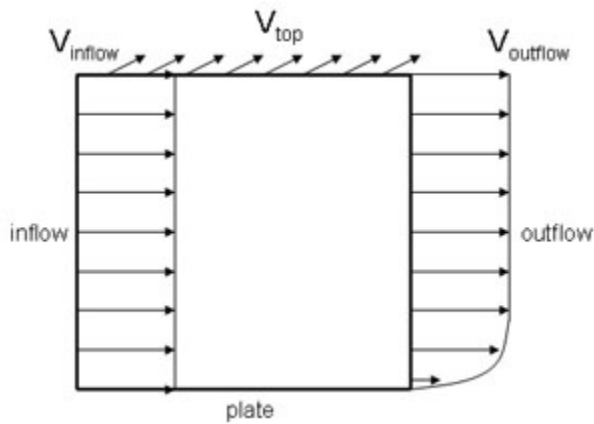
Uncheck **Write to File**. Click **Plot**.



Higher Resolution Image

We notice here that the x velocity reaches 1 m/s at approximately $y = 0.02$ m. This shows the relative thinness of the boundary layer compared to the length scale of the plate. We also notice that the velocity profile is slightly greater than 1 m/s above the boundary layer. We know this would not happen in real flow, rather it is a result of the boundary condition we have chosen for our model. We chose the **Symmetry** boundary condition at the top of our flow field, which is essentially a wall without the no-slip condition. Thus, no flow is permitted to escape through this boundary.

In a real external flow, there is no such boundary at the top and flow is permitted to pass through freely. When we consider the inflow and outflow velocity profiles in terms of conservation of mass, the uniform velocity profile of 1 m/s at $x = 0$ has more mass entering the flow field than the non-uniform velocity profile at $x = 1$ m, in which the velocity is lower near the plate. In addition, the fluid is expanding near the plate because its temperature is increasing, further increasing the y-velocity of the fluid above it. These factors require that some mass must escape through the top of our flow field in order to satisfy conservation of mass.



Choosing a **Pressure Outlet** for the top boundary condition would represent real external flow more accurately. Unfortunately, this cannot be used in our flow field without encountering convergence problems, so selecting the **Symmetry** boundary condition was the next best option. Because we are not allowing flow to escape through the top boundary, we observe an outflow velocity profile in which outflow velocity is greater than 1 above the boundary layer in order to satisfy conservation of mass. Fortunately, the inaccuracies resulting from the model we chose have no significant effect on the heat transfer coefficients at the plate.

Select **Write to File** and save the data for this plot as `outflow_profile.xy`.

Plot Nusselt Number vs. Reynolds Number

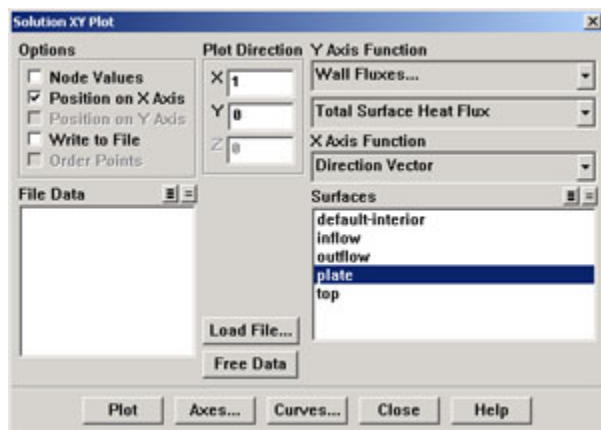
Recall that the Nusselt Number is a non-dimensional heat transfer coefficient that relates convective and conductive heat transfer.

$$Nu_x = \frac{h_x x}{k}$$

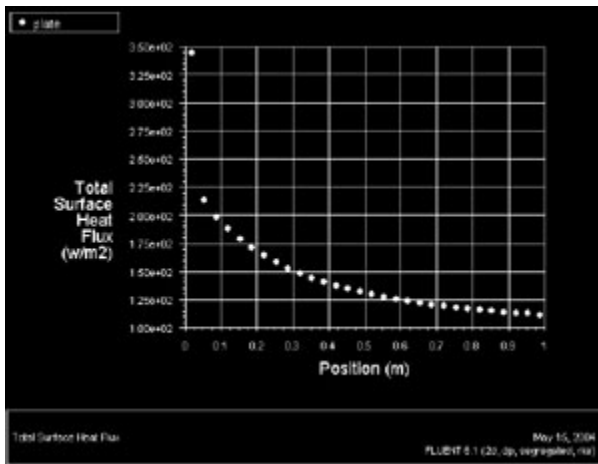
In order to obtain the Nusselt Number from FLUENT, we will begin by plotting Total Surface Heat Flux.

Main Menu > Plot > XY Plot...

In the **Options** box, change back to **Position on X Axis**. In the **Plot Direction** box, enter the default values of 1 in the **X** box and 0 in the **Y** box. Under **Y-Axis Function** choose **Wall Fluxes**. In the box below, chose **Total Surface Heat Flux**. Select **Plate** under **Surfaces**. Before plotting, be sure to turn on **Auto Range** for the Y axis under **Axes...**



Click **Plot**.



[Higher Resolution Image](#)

Now Select **Write to File**. Save the data for this plot as `heatFlux.xy`. Click **Write....**

Open the file `heatFlux.xy` using Wordpad or a similar application. You can simply copy and paste the data into Excel.

	A	B	C	D	E	F	G	H	I	J	K
1											
2											
3											
4		0.016667	343.754								
5		0.05	213.701								
6		0.083333	198.42								
7		0.116667	187.929								
8		0.15	178.898								
9		0.183333	171.048								
10		0.216667	164.24								
11		0.25	158.295								
12		0.283333	153.077								
13		0.316667	148.479								
14		0.35	144.411								
15		0.383333	140.801								
16		0.416667	137.553								
17		0.45	134.705								
18		0.483333	132.121								
19		0.516667	129.793								
20		0.55	127.668								
21		0.583333	125.779								
22		0.616667	124.04								
23		0.65	122.45								
24		0.683333	120.993								
25		0.716667	119.652								
26		0.75	118.414								
27		0.783333	117.266								
28		0.816667	116.2								
29		0.85	115.204								
30		0.883333	114.276								
31		0.916667	113.362								
32		0.95	112.446								
33		0.983333	111.452								
34											
35											
36											
37											
38											
39											
40											
41											
42											

If Excel does not automatically separate the data into columns, separate it by selecting the column of data and then using the Text to Columns function:

Main Menu > Data > Text to Columns

The first column is the x location on the plate and the second column is the total surface heat flux (q'') at the corresponding x location. We now need to determine the Nusselt number from these values at each x location. We will define positive q'' as heat transfer into the fluid. Use the following expression to convert q'' to Nusselt Number in your Excel spreadsheet.

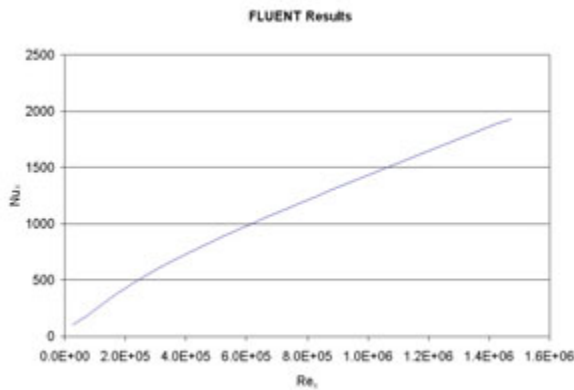
$$q'_z = h_z (T_{fluid}(x, y=0) - T_{plate})$$

$$Nu_x = \frac{h_z x}{k} = \left(\frac{q'_z}{T_{plate} - T_{\infty}} \right) \left(\frac{x}{k} \right) = \left(\frac{q'_z}{60} \right) \left(\frac{x}{9.4505e-4 \text{ W/m K}} \right)$$

Reynolds Number can be defined at each x location by

$$Re_x = (Re_L)(x) = 1,500,000x$$

Now plot Re vs. Nu in Excel. Your plot should look like this:



[Higher Resolution Image](#)

Compare Results with Correlation & Experiment

Validate your results from FLUENT by comparing to a correlation and experimental results. The correlation we will use is derived by Reynolds [1]:

$$Nu_x = 0.0296 (Re_x^{0.8}) \left(Pr^{0.6} \left(\frac{T_{plate}}{T_{\infty}} \right)^{-0.4} \right)$$

All properties in this correlation are evaluated at the free-stream static temperature of 300K. This correlation assumes the following:

1. $Pr = 0.7$
2. $10^5 < Re < 10^7$
3. Fluid properties evaluated at free-stream conditions
4. Turbulent compressible boundary layer
5. Flat plate
6. Friction factor calculated from the following relation (implicit in Nu equation above, does not need to be calculated in your analysis):

$$C_f = 0.0296 Re_x^{-0.2}$$

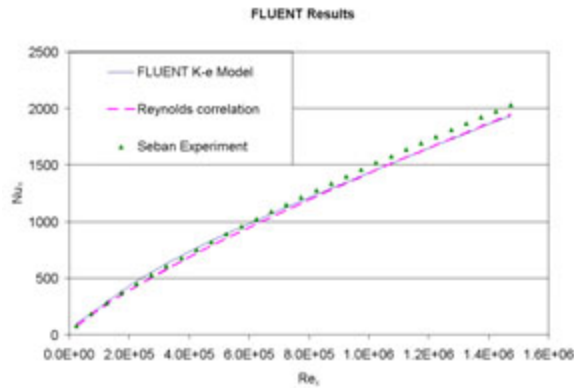
Add the Reynolds correlation for Nusselt Number to your Excel spreadsheet.

Seban & Doughty [2] performed a heated flat plate experiment for which they derived the following expression for Nusselt Number:

$$Nu_x = 0.0236 \left(\frac{\rho u x}{\mu} \right)^{4/5}$$

The Seban & Doughty experiment was performed with air as the fluid ($Pr = 0.7$) and at various Reynolds Numbers in the range $1e5 < Re < 4e6$. Add the this experimental relation for Nusselt Number to your Excel spreadsheet.

Now plot and compare Re vs. Nu from FLUENT, the Reynolds Correlation, and Seban's experiment.



[Higher Resolution Image](#)

As we can see, there is very little variation between these 3 results. The largest % error between the FLUENT results and the Reynolds correlation is only 7.5%. In turbulent flow as we have here, similar results between FLUENT and correlation are more difficult to come by than in laminar flow because a turbulent model must be used in FLUENT, which does not solve the Navier-Stokes Equations exactly. Experimental error (in experiments from which correlations are derived) also accounts for some of this 7.5% error. Each of the turbulence models that FLUENT offers produces results similar to these, although the k-epsilon model is the most appropriate model to use in this case.

Go to [Step 7: Refine Mesh](#)

[1] Reynolds, W.C., Kays, W.M., Kline, S.J. "Heat Transfer in the Turbulent Incompressible Boundary Layer." NASA Memo 12-1-58W. December 1958.

[2] Seban, R.A. and Doughty, D.L. "Heat Transfer to Turbulent Boundary Layers with Variable Freestream Velocity." *Journal of Heat Transfer* **78**:217 (1956).

[See and rate the complete Learning Module](#)

Go to [all FLUENT Learning Modules](#)