FLUENT - Forced Convection over a Flat Plate step 6

Step 6: Analyze Results

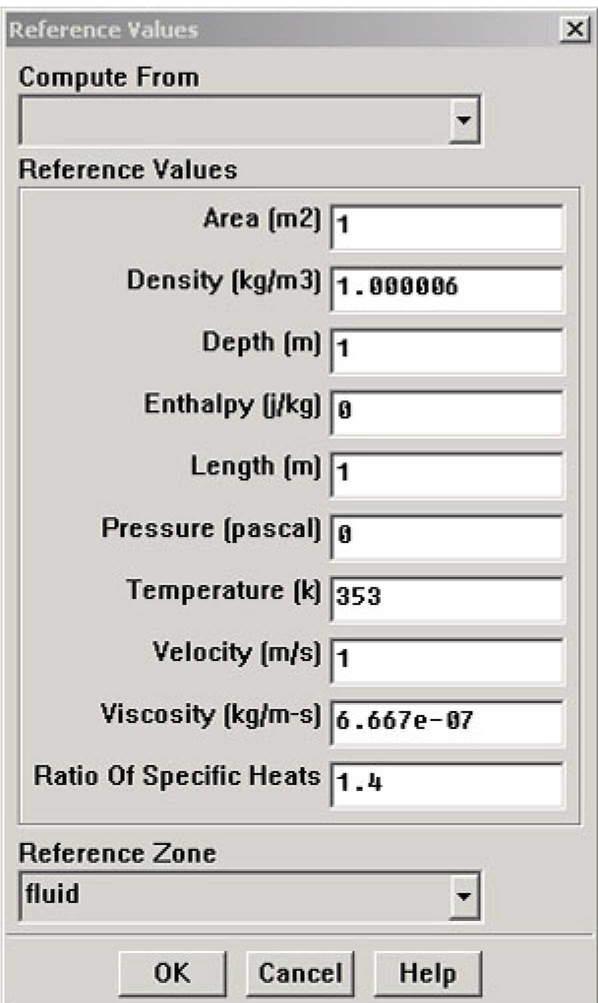
y+

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...

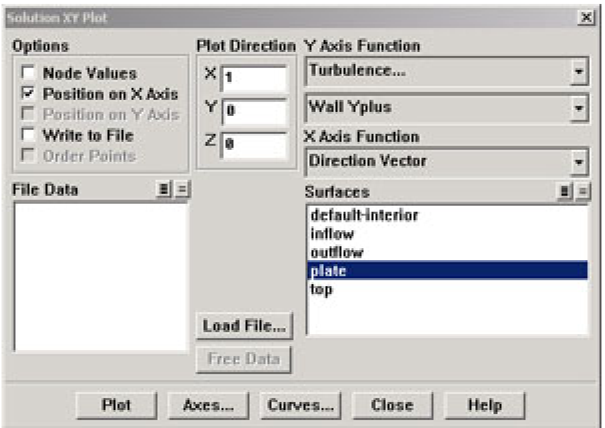
Select ***inflow*** under ***Compute From*** to tell FLUENT to use values at the inflow for the reference values. Check that the reference value for velocity is *1 m/s,* temperature is *353 K*, and coefficient of viscosity is *6.667e-7 kg/m-s* as given in the Problem Specification. These reference values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



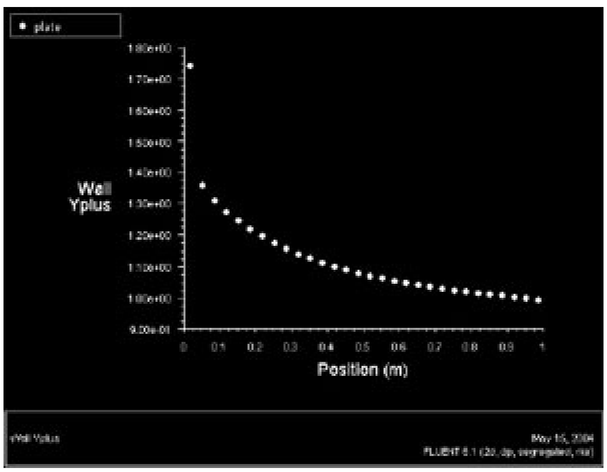
By using the following method, plot *y+* values for wall-adjacent cells to check how they compare with the recommendation mentioned above.

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 anamolous 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

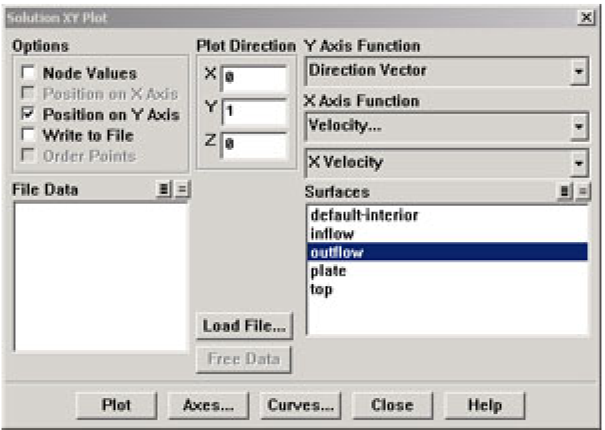
In the *Solution XY Plot Window*, check the ***Write to File*** box under ***Options***. The ***Plot*** button should have changed to the ***Write...*** button. Click on ***Write....*** Enter `yplus.xy` as the filename and click ***OK***. Check that this file has been created in your FLUENT working directory.

Velocity at x = 1m

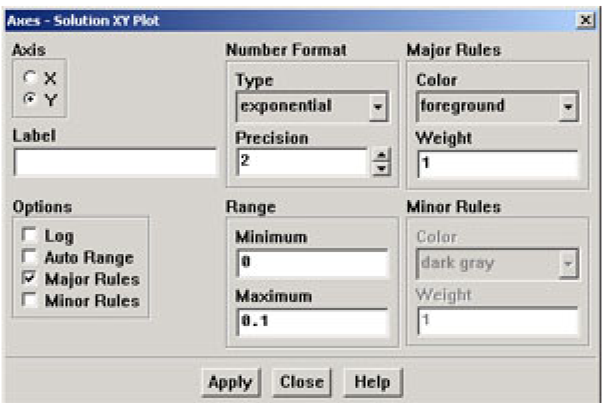
Main Menu > Plot > XY Plot...

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.

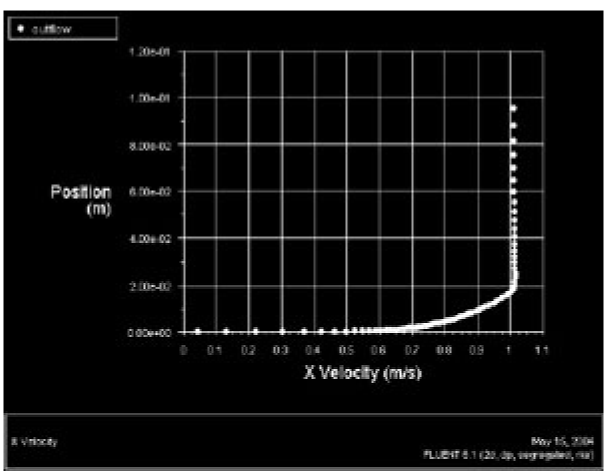
Under ***X Axis Function***, pick ***Velocity...*** and then in the box under that, pick ***X Velocity***. Finally, select ***outflow*** under ***Surfaces*** since we are plotting the velocity profile at the outflow. De-select ***plate*** under ***Surfaces***.



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 ***A pply***. 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***.



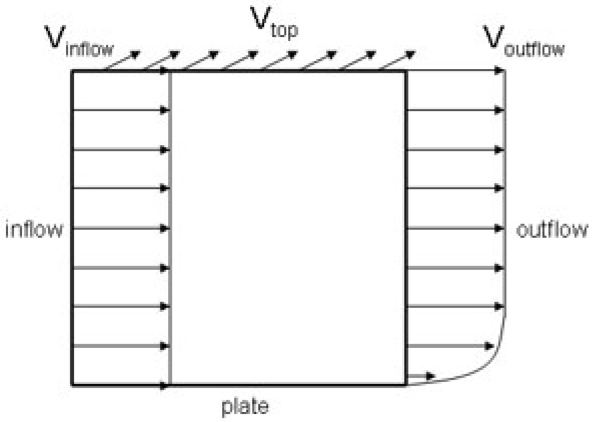
Uncheck ***Write to File***. Click ***Plot***.



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 = 1m, 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

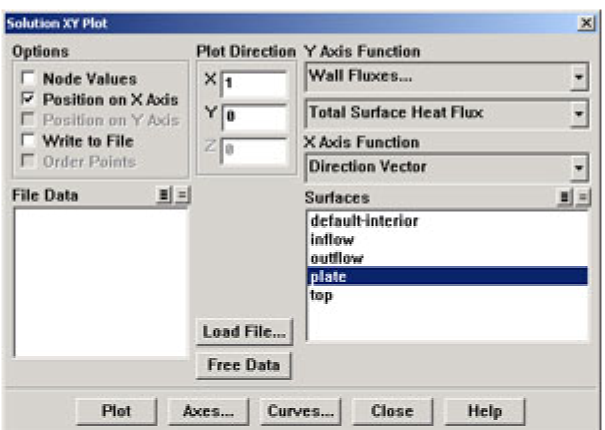
Recall that the Nusselt Number is a non-dimensional heat transfer coefficient that relates convective and conductive heat transfer.

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

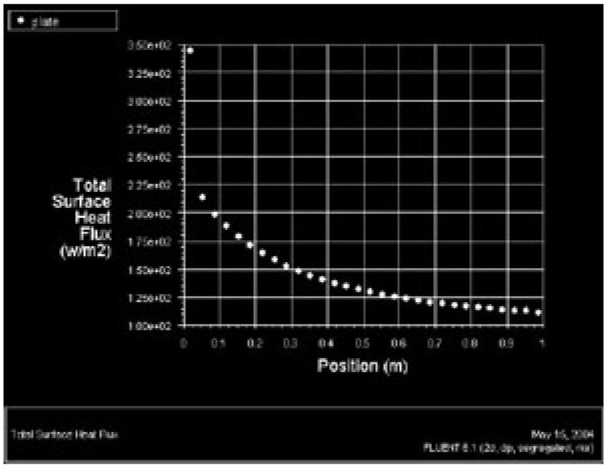
In order to obtain the Nusselt Number from FLUENT, we will begin by plotting Total Surface Heat Flux.

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...***.



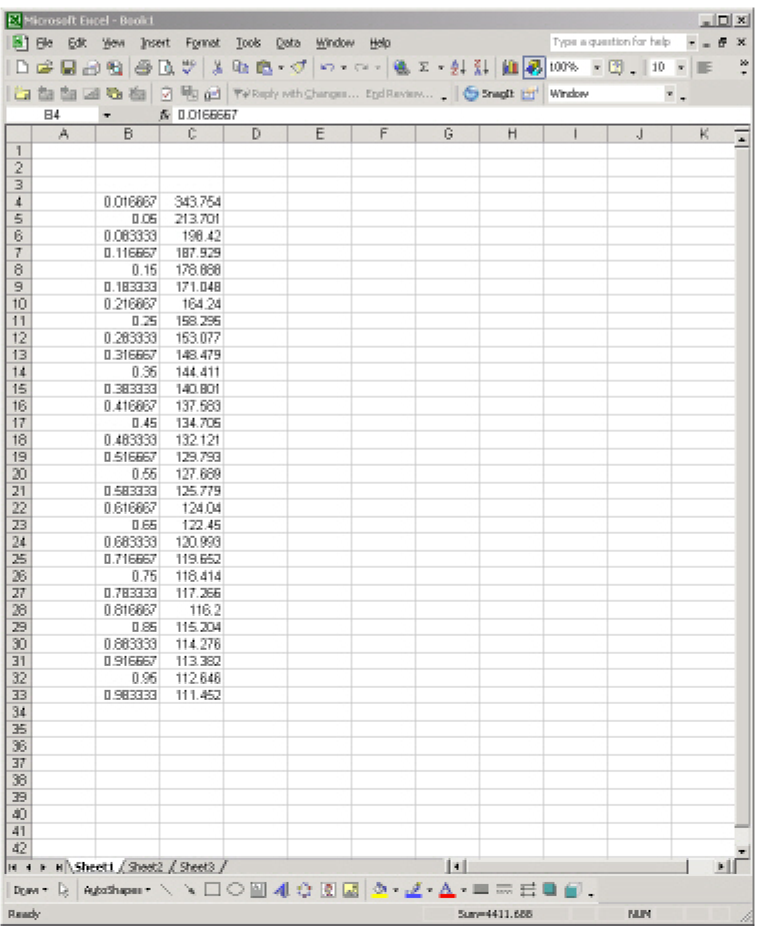
Click ***Plot***.



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
4		0.016667	343.754
5		0.05	213.701
6		0.083333	198.42
7		0.116667	187.929
8		0.15	178.868
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	137.583
17		0.45	134.705
18		0.483333	132.121
19		0.516667	129.793
20		0.55	127.689
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	113.382
32		0.95	112.646
33		0.983333	111.452

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.

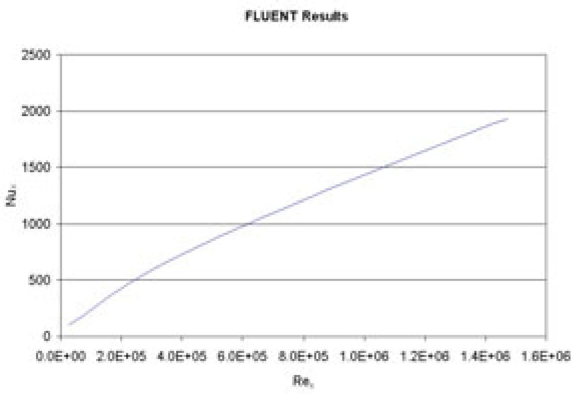
$$q''_x = h_x \left(T_{fluid}(x, y=0) - T_{plate} \right)$$

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

Reynolds Number can be defined at each x location by

$$\mathrm{Re}_x = (\mathrm{Re}_L)(x) = 1{,}500{,}000x$$

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



Higher Resolution Image

Compare Results with Correlation & Experiment

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

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

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. $10^5 < \mathrm{Re} < 10^7$

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):

$$C_f = 0.0296 \mathrm{Re}_x^{-0.2}$$

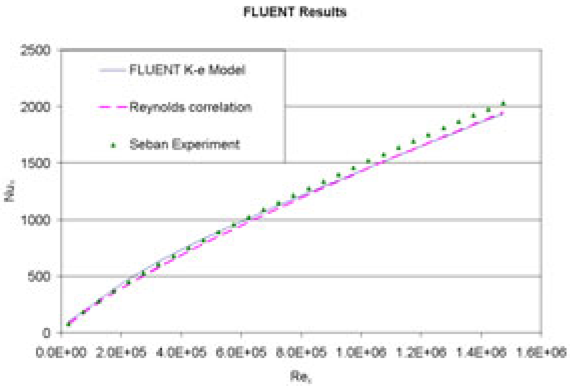
Add the Reynolds correlation for Nusselt Number to your Excel spreadsheet.

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