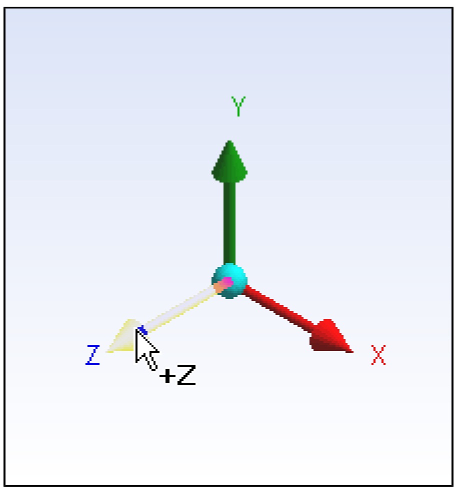
2D Steady Convection - Numerical Results

Author: Benjamin Mullen, Cornell University

Numerical Results

Open the Post Processor

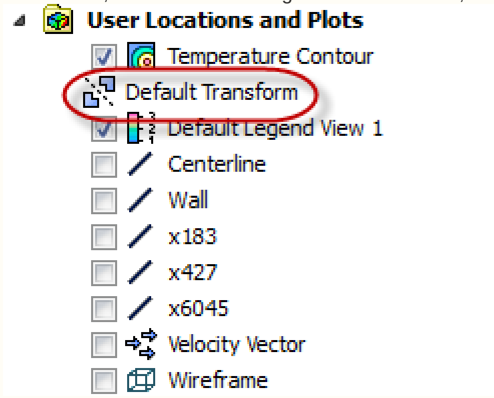
In the *Project Schematic* double click ***Results*** to open the post processor. When the *A6: Fluid Flow (FLUENT) - CFD - Post* Window opens, look at the geometry by clicking the +Z axis on the compass

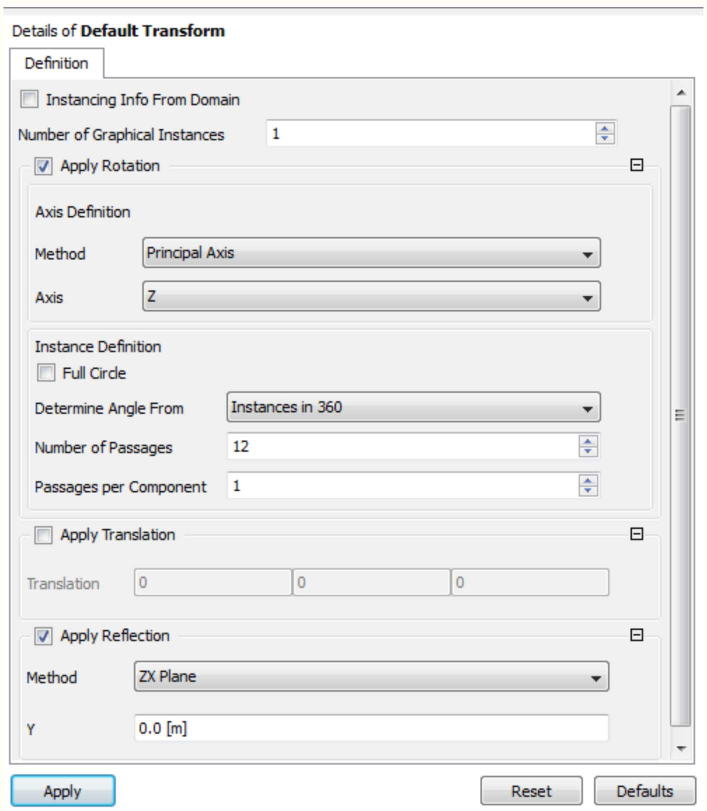


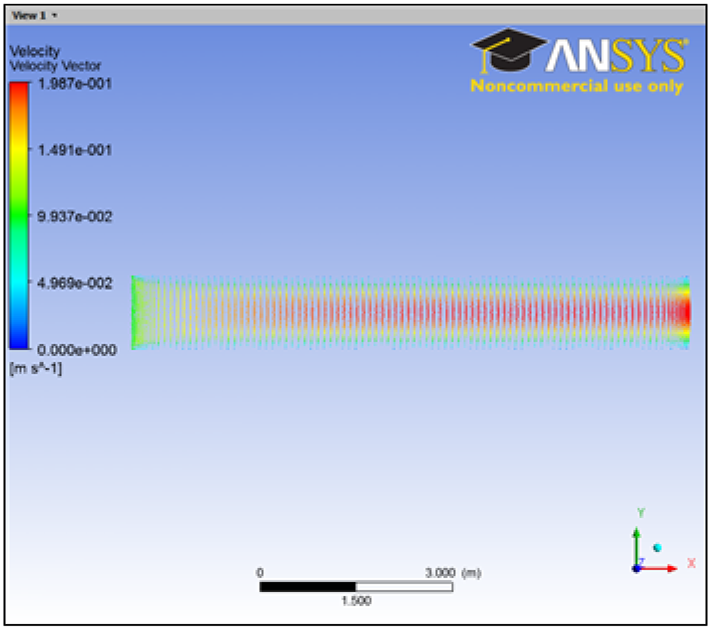
Velocity Vectors

In the Post Processing window, click the Vector icon  to create a vector result. When prompted, name the result `Velocity Vector`. In the *Details of Velocity Vector* window, begin on the ***Geometry*** tab. Under ***Locations***, select ***Periodic 1***. This will show the velocity along the entire geometry surface periodically. Next, click on the ***Symbol*** tab. Change the ***Symbol Size*** to `0.1`. Finally, move to the ***View*** tab. We want to see the entire geometry of the pipe: not just half of it like we currently see. To see the whole pipe, check the box next to ***Apply Reflection/Mirroring***, and change the ***Method*** to ***ZX Plane***. Because the pipe is long and skinny, it will be difficult to see the results. This post processor allows us to stretch the results to make the results easier to see. To apply a scaling, check the box next to ***Apply Scale***, and change the ***Scale*** to `1,10,1` (this will scale the y-direction by 10). When finished, press ***Apply*** to see the result. If you wish to see the result without the wireframe of the pipe, uncheck the box next to ***Wireframe*** under *User Location and Plots*.

In ANSYS version 14.5 and later, only one half of the pipe cross-section is displayed after using the mirroring option. You can work around this by applying the mirroring condition in the "Default transform" setting and not in the "View" Tab. To do this select "Default Transform" in the left-hand menu, uncheck "Instancing Info from Domain", check "Apply Reflection" and select to mirror about the ZX Plane.

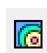


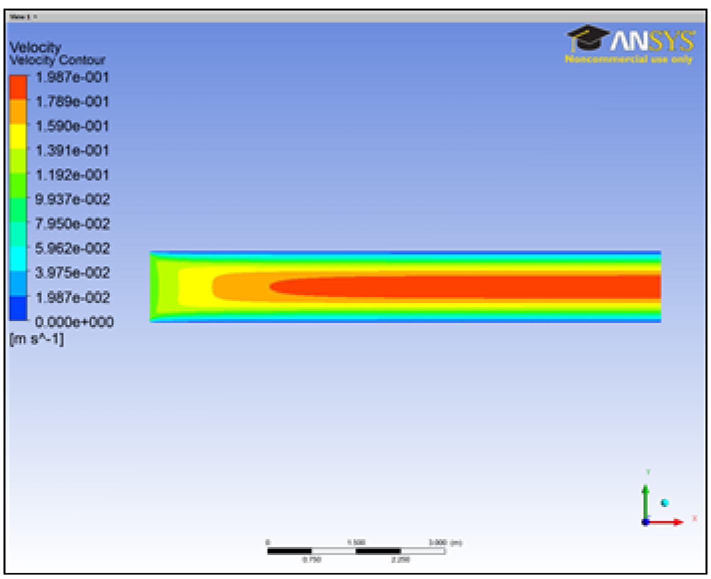




click here to enlarge

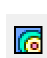
Velocity Contour

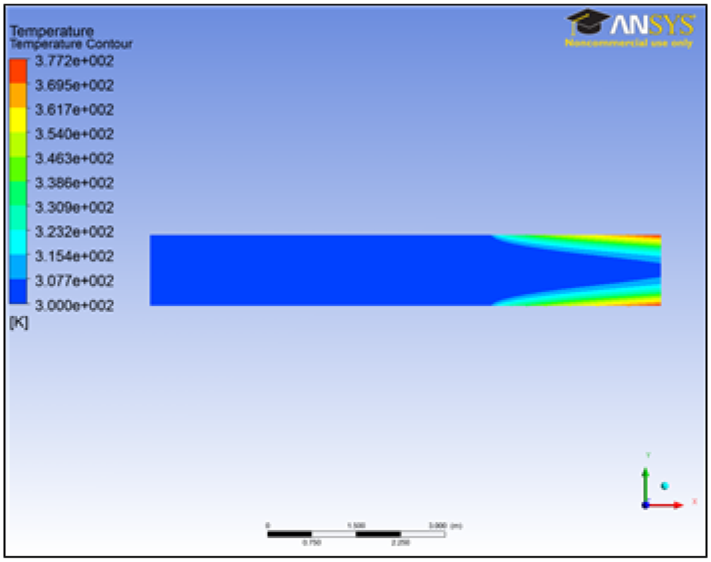
In the Post Processing window, click the Contour icon  to create a Contour result. When prompted, name the result `Velocity Contour`. In the *Details of Velocity Contour* window, begin on the ***Geometry*** tab. Under ***Locations***, again select ***Periodic 1***. Also, change the ***Variable*** to ***Velocity***. Next, move to the ***View*** tab. Check the box next to ***Apply Reflection/Mirroring***, and change the ***Method*** to ***ZX Plane*** and again,check the box next to ***Apply Scale***, and change the ***Scale*** to `1,10,1`. When finished, press ***Apply*** to see the result. Finally, we need to remove the Velocity Vectors from the Graphic Window. Do this by unchecking the box next to ***Velocity Vector*** in the *Outline* window under *User Location and Plots*.



Click here to enlarge

Temperature Contour

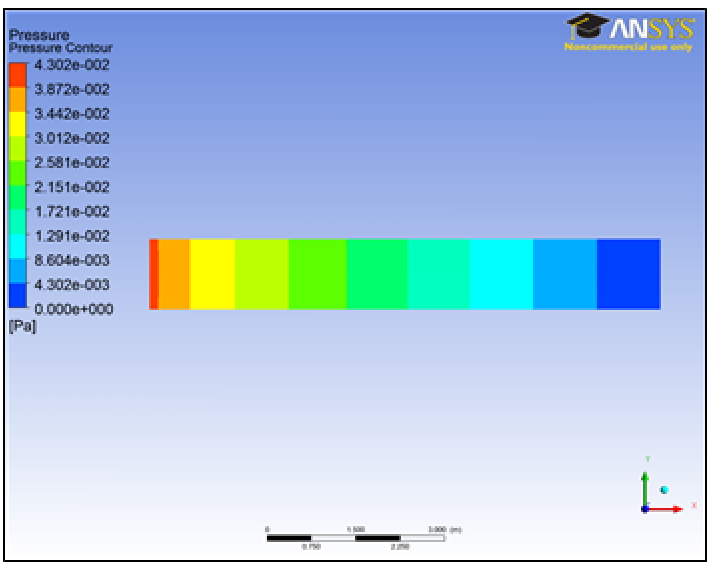
In the Post Processing window, click the Contour icon  to create another Contour result. When prompted, name the result `Temperature Contour`. In the *Details of Temperature Contour* window, begin on the ***Geometry*** tab. Under ***Locations***, select ***Periodic 1***. This time, change the ***Variable*** to ***Temperature***. Next, move to the ***View*** tab. Apply the same mirroring and scaling as we did for the Velocity Contours. When finished, press ***Apply***. Uncheck the box next to ***Velocity Contour*** to only see the Temperature Contours.



Click here to enlarge

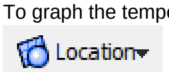
Pressure Contour

Create another contour result, and name `Pressure Contour`. Use all of the same settings as the previous results but this time choosing ***Variable > Pressure*** in the ***Geometry*** tab.

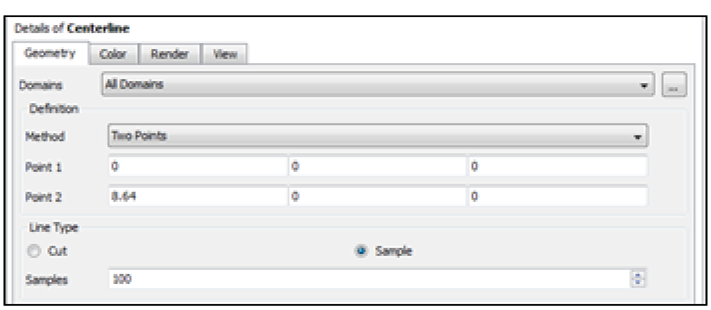


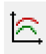
Click here to enlarge

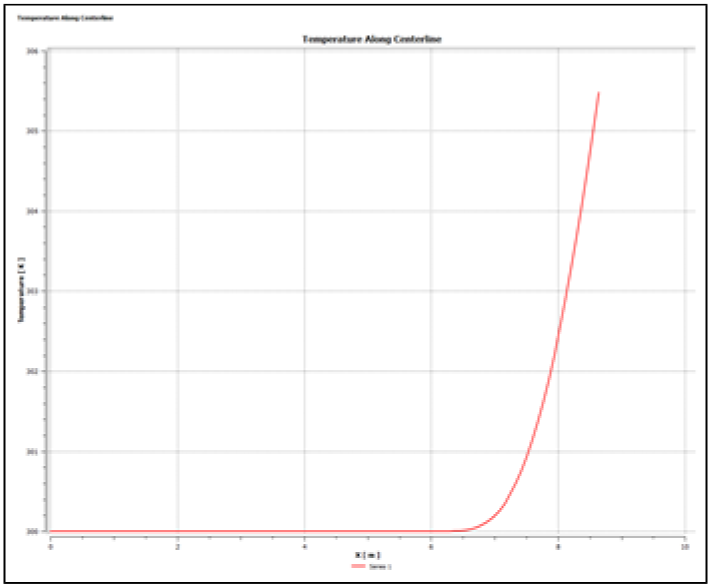
Graph of Temperature along Centerline

To graph the temperature along the centerline, we first need to create the centerline as a path. To accomplish this, click on the Location icon 

, select ***Line***, and name the line `Centerline`. In the *Details of Centerline* window, set the ***Method*** to two points. Point 1 is (0,0,0), and Point 2 is (8.64,0,0). Enter these values into the *details* window. Next, change the number of ***Samples*** to 100. Press ***Apply*** once finished.

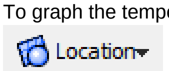


To create a chart, press the chart icon . When prompted, name the page `Temperature Along Centerline`. In the *Details of Temperature Along Centerline* window, begin on the ***General*** tab. In the ***Title***, enter `Temperature Along Centerline`. Next, click on the ***Data Series*** tab. Under *Data Source*, in the drop down menu next to ***Location***, select ***Centerline***. Now move to the ***X Axis*** tab. In the drop down menu next to ***Variable***, scroll all the way down and select ***X***. In the ***Y Axis*** tab, change the ***Variable*** to ***Temperature***. when finished, press ***Apply*** to see the chart.

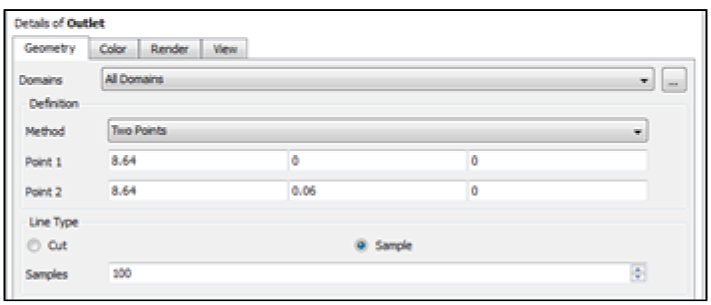


Click here to enlarge

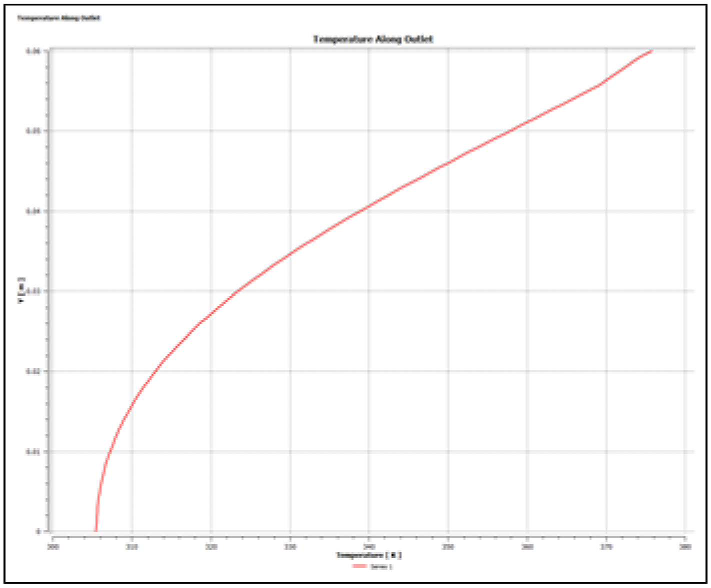
Graph of Temperature along Outlet

To graph the temperature along the outlet, we need to create the outlet as a path much like we did with the centerline. Click on the Location icon 

, select ***Line***, and name the line `Outlet`. In the *Details of Centerline* window, set the ***Method*** to two points. Point 1 is (8.64,0,0), and Point 2 is (8.64,0.06,0). Enter these values into the *details* window. Next, change the number of ***Samples*** to 100. Press ***Apply*** once finished.



Next, press the chart icon . When prompted, name the page `Temperature Along Outlet`. In the *Details of Temperature Along Outlet* window, begin on the ***General*** tab. In the ***Title***, enter `Temperature Along Outlet`. Next, click on the ***Data Series*** tab. Under *Data Source*, in the drop down menu next to ***Location***, select ***Outlet***. Now move to the ***X Axis*** tab. In the drop down menu next to ***Variable***, and select ***Temperature***. In the ***Y Axis*** tab, change the ***Variable*** to ***Y***. when finished, press ***Apply*** to see the chart.



Click here to enlarge

Graph of Nusselt Number along the heated section of the pipe

The ***Nusselt number*** is a non-dimensional parameter that provides a measure of the convection heat transfer at a surface. It is the ratio of convection to pure conduction heat transfer. We will now derive the Nusselt number as a function of the given parameters and temperature.
The convection heat transfer at the pipe wall is:

$$q''_w = h(T_w - T_m)$$

We can rearrange terms to find an expression for h, the ***convection coefficient***:

$$h = \frac{q''_w}{(T_w - T_m)}$$

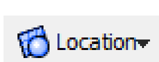
Substitute the ***convection coefficient*** expression into the ***Nusselt Number*** expression:

$$Nu = \frac{hL}{k} = \frac{q''_w(2R)}{k(T_w - T_m)}$$

where
h is the convection coefficient.
k is the thermal conductivity.
L is the length scale. Similar to the Reynold's Number, the length scale is the diameter of the pipe for an internal pipe flow.
q''_w is the heat flux at the heated surface, 37.5 W/m^2.
Tw is the pipe wall temperature at a given location along the pipe.
Tm is the mean temperature in the pipe at the location where Tw is defined.

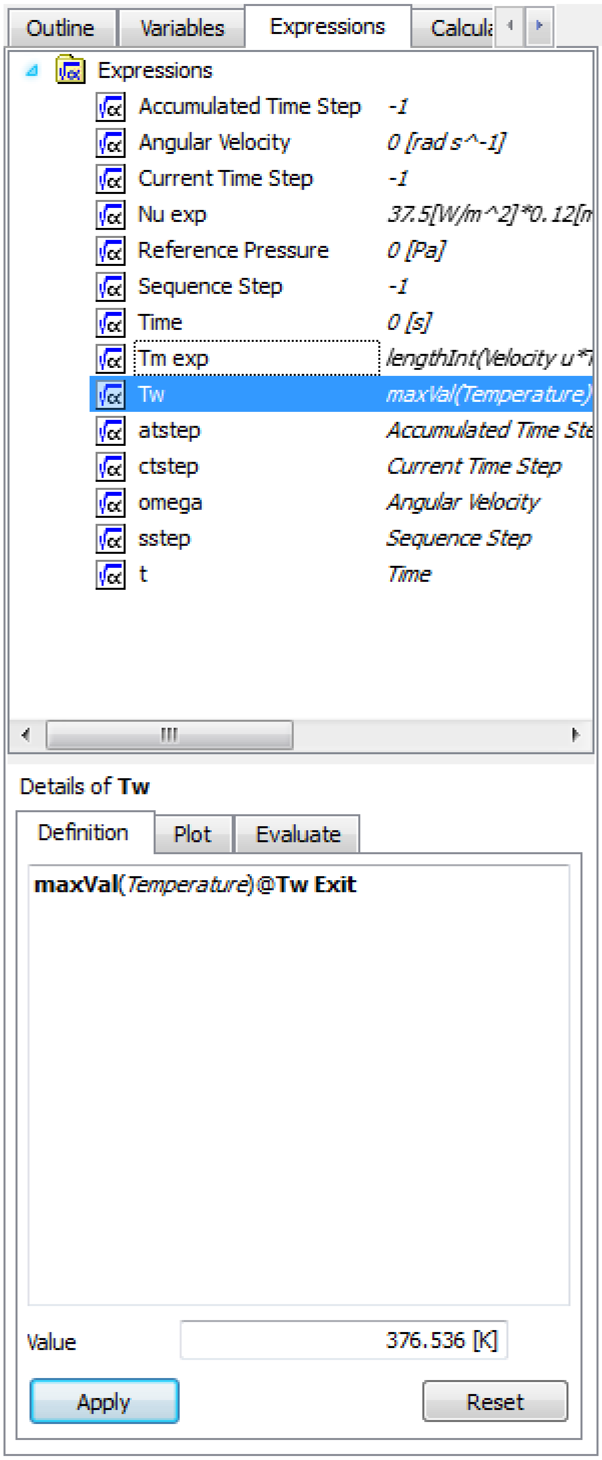
Wall Temperature

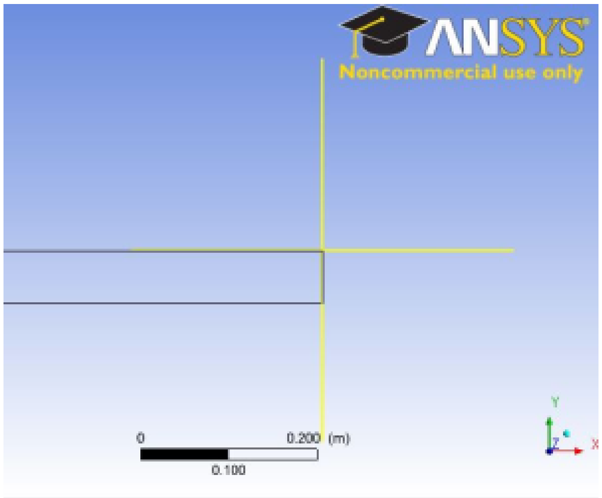
To find the temperature at the wall, click on ***insert*** >> ***location*** >> ***point***, and name it *Tw exit*. In the Details of *Tw exit* window, set Method to XYZ and enter (8.64, 0.06, 0) in Point.Click Apply to create a point at the upper right corner of the pipe.

Click on Expression right below 

and right click in the window to create a new expression named *Tw*. Under ***Details of Tw*** panel, enter **maxVal(Temperature)@Tw exit** in the ***Definition*** tab.

Tw now gives the temperature at the location (8.64, 0.06, 0), which is on the exit pipe wall.


Outline
Variables
Expressions
Calcul
Expressions
Accumulated Time Step -1
Angular Velocity 0 [rad s^-1]
Current Time Step -1
Nu exp 37.5[W/m^2]*0.12[m
Reference Pressure 0 [Pa]
Sequence Step -1
Time 0 [s]
Tm exp lengthInt(Velocity u*T
Tw maxVal(Temperature)
atstep Accumulated Time Ste
ctstep Current Time Step
omega Angular Velocity
sstep Sequence Step
t Time
Details of Tw
Definition
Plot
Evaluate
maxVal(Temperature)@Tw Exit
Value
376.536 [K]
Apply
Reset



Mixed Mean Temperature

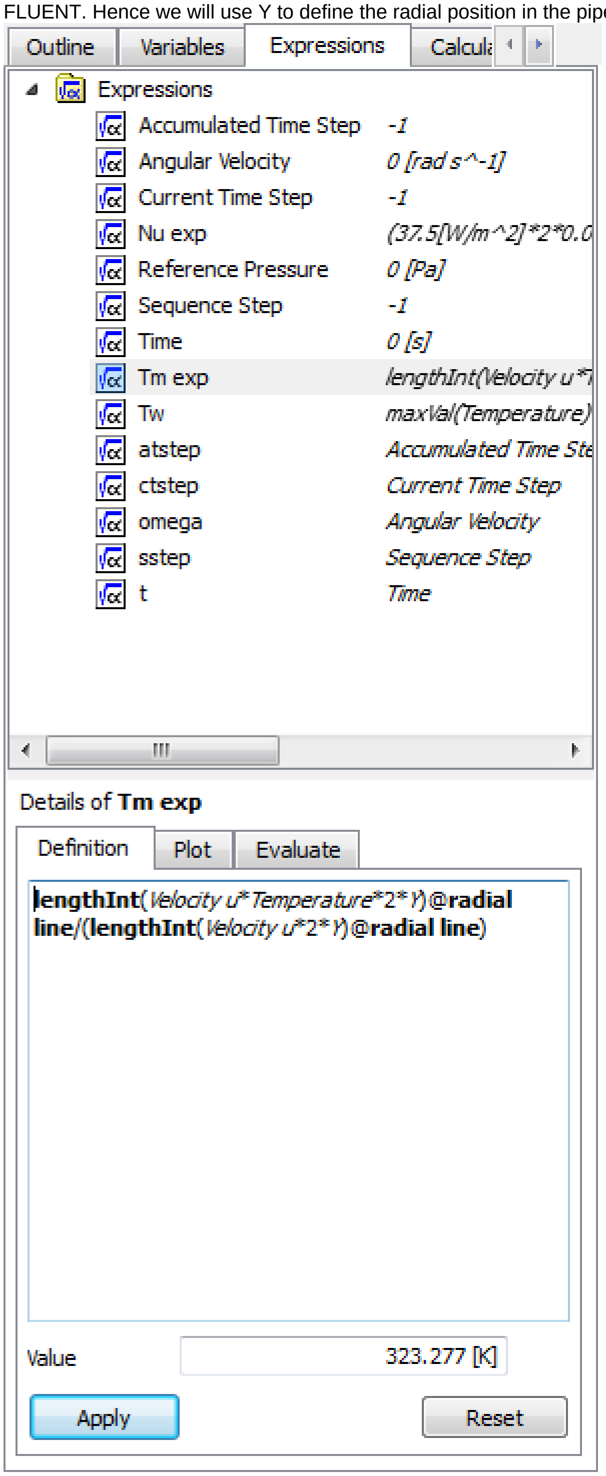
To find the mean temperature at a given location in the pipe, click on ***insert*** >> ***location*** >> ***line***, and name it exit. In the Details of exit window, set Method to Two Points and enter (8.64, 0, 0) for Point 1 and (8.64, 0.06, 0) for Point 2. Click Apply to create a line at the exit of the pipe. The mean temperature is the area weighted average temperature and we can use integral to find the appropriate mean Temperature:

$$T_{m\ exit} = \frac{\int_0^R uT(2\pi r)\,dr}{\int_0^R u(2\pi r)\,dr} = \frac{\int_0^R urT\,dr}{\int_0^R ur\,dr}$$

Click on the ***Calculators*** tab and double click on ***Function Calculator***. Select ***lengthInt*** for the Function, ***exit*** for the Location, and ***Velocity u*** for the Variable. Check ***show equivalent expression*** and click Calculate. The expression "lengthInt(Velocity u)@exit is essentially the integral of u*dr and can be conveniently used to calculate the mean temperature.
Under ***Expressions***, right click in the window to create a new expression and name it Tm. In the Details of Tm window, enter the following:

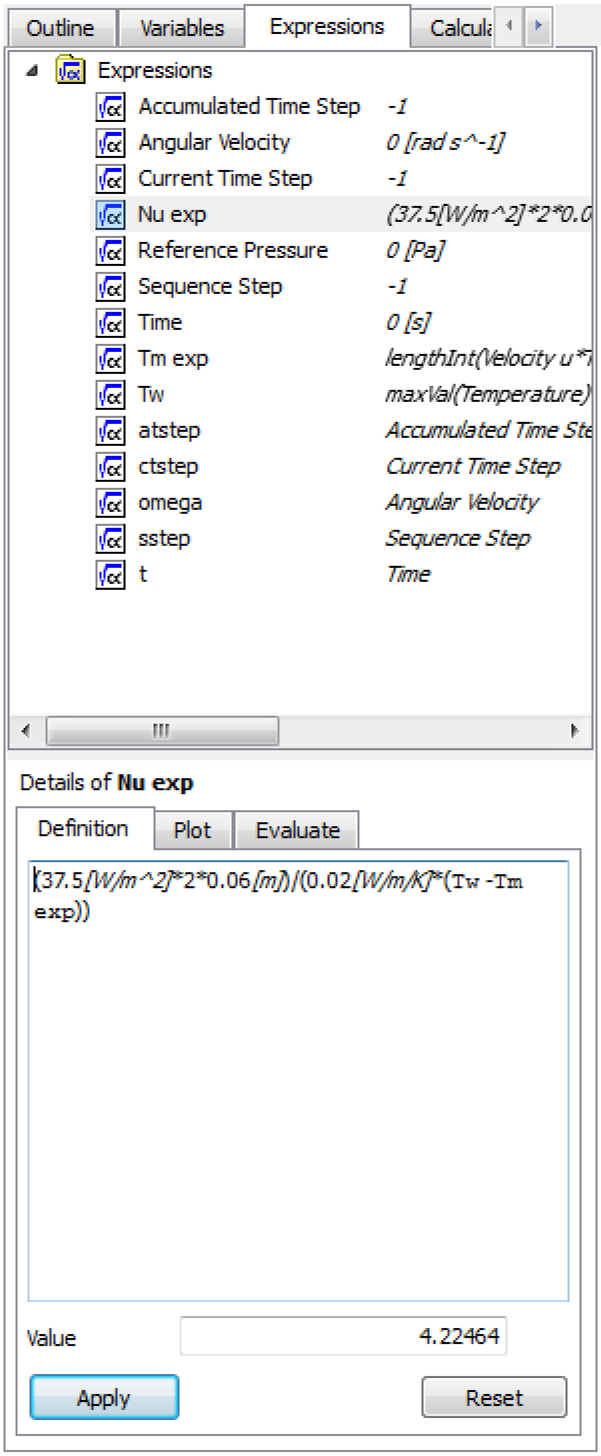
lengthInt(Velocity u*Y*Temperature)@exit/lengthInt(Velocity u*Y)@exit

This expression will now give the mean temperature at the location in which we called "exit". Recall the pipe radius r is defined in the Y direction in FLUENT. Hence we will use Y to define the radial position in the pipe, as shown in the expression above.



Nusselt Number

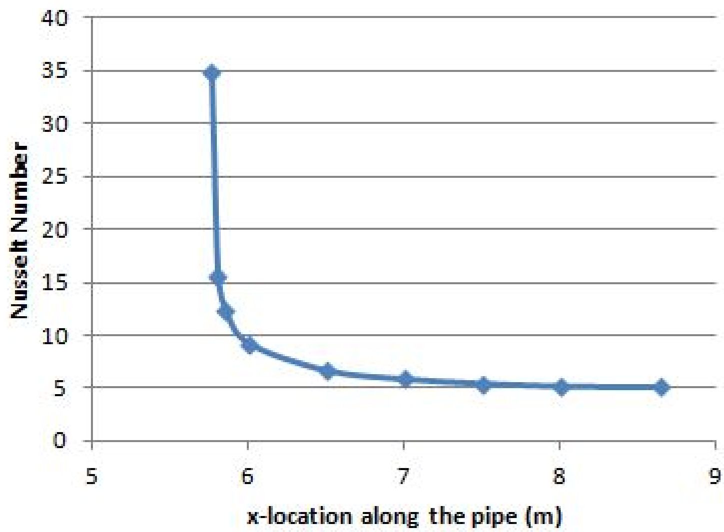
We are now ready to find the Nusselt Number. Create another expression and name it ***Nu exp***. Under the Definition tab, enter the Nusselt Number expression shown in the equation above. The units are entered in square brackets, this is done to ensure the expression for the Nusselt Number is dimensionless.



You may get a slightly higher or lower value for the Nusselt Number here.

We would like to compare the Nusselt Number along the heated section of the pipe. We can generate the Nusselt Number at a different location by simply changing the x-coordinate of ***exit*** and ***Tw exit***, which we defined earlier. Once the new coordinates defined in ***exit*** and ***Tw exit*** are updated, the associated expression ***Tw***, ***Tm***, and ***Nu exp*** will be updated automatically.

We can expect a maximum and dominant convection heat transfer at the entrance of the heated section of the pipe. The convection heat transfer raises the temperature inside the pipe, as well as mean temperature, along the downstream direction. The mean temperature near the exit is higher relative to the entrance and therefore a lower convection heat transfer is expected at the exit. Again, the Nusselt Number is a measure of convection heat transfer relative to conduction heat transfer. Thus we should expect the Nusselt Number to decrease along the length of the pipe.



To export the data, click on the "export" button. Comma Seperated Value (.csv) is able to be read by matlab and Excel, so it should be fine.

We are now ready to validate and verify our results.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules