Turbulent Pipe Flow - Numerical Solution

Author: Rajesh Bhaskaran, Cornell University

Useful Information

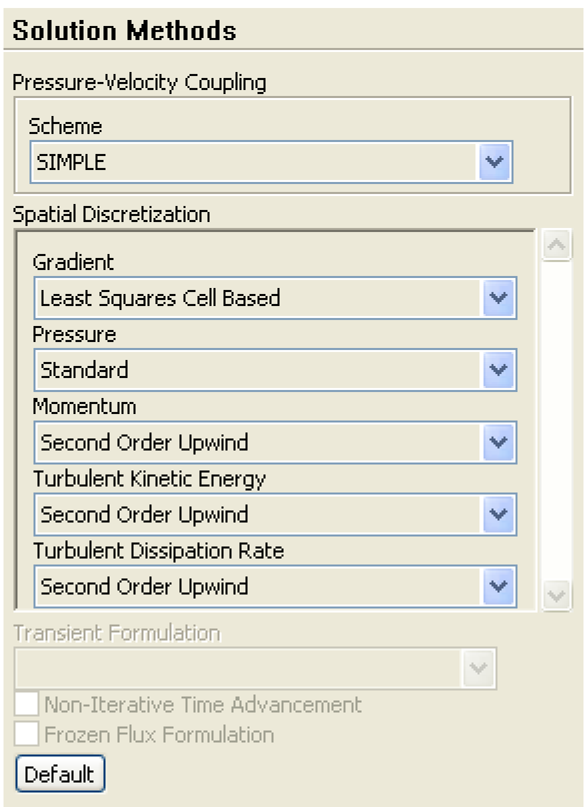
Click here for the FLUENT 6.3.26 version.

Numerical Solution

We'll use second-order discretization for the momentum equation, as in the laminar pipe flow tutorial, and also for the turbulence kinetic energy equation which is part of the *k-epsilon* turbulence model.

Solution > Solution Methods

Change the Discretization for ***Momentum***, ***Turbulence Kinetic Energy*** and ***Turbulence Dissipation Rate*** equations to ***Second Order Upwind*** (if you do not see all of the equations scroll down to see them).



The order of discretization that we just set refers to the convective terms in the equations; the discretization of the viscous terms is always second-order accurate in FLUENT. Second-order discretization generally yields better accuracy while first-order discretization yields more robust convergence. If the second-order scheme doesn't converge, you can try starting the iterations with the first-order scheme and switching to the second-order scheme after some iterations.

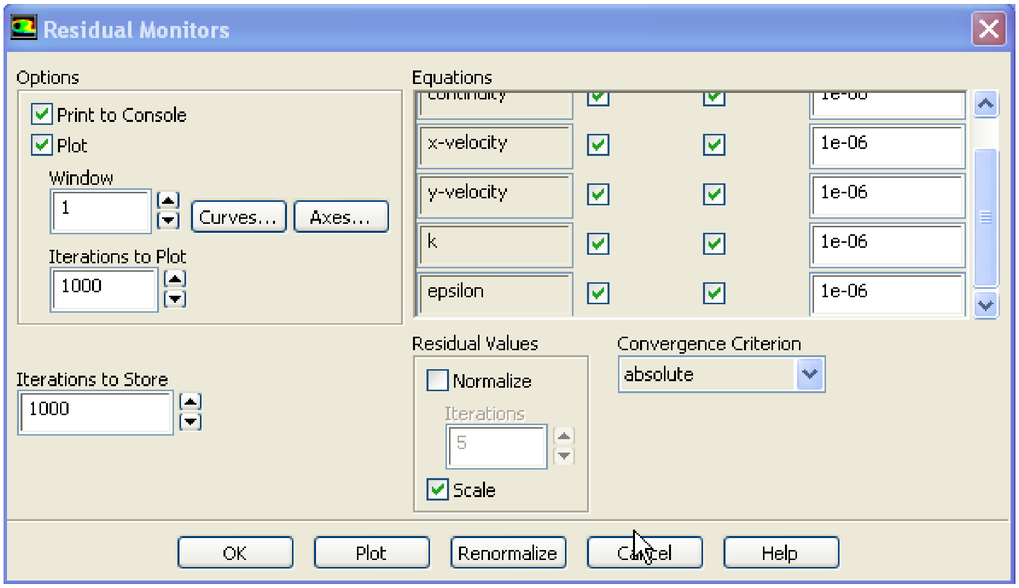
Set Convergence Criteria

Recall that FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-6.

Solution > Monitors > Residuals, Statistic and Force Monitors

Double click on ***Residuals.***Notice that ***Convergence Criterion*** has to be set for the *k* and *epsilon* equations in addition to the three equations in the last tutorial. Set the ***Convergence Criterion*** to be 1e-06 for all five equations being solved.

Select ***Print to Console*** and ***Plot*** under ***Options*** (these are the defaults). This will print as well plot the residuals as they are calculated which you will use to monitor convergence.

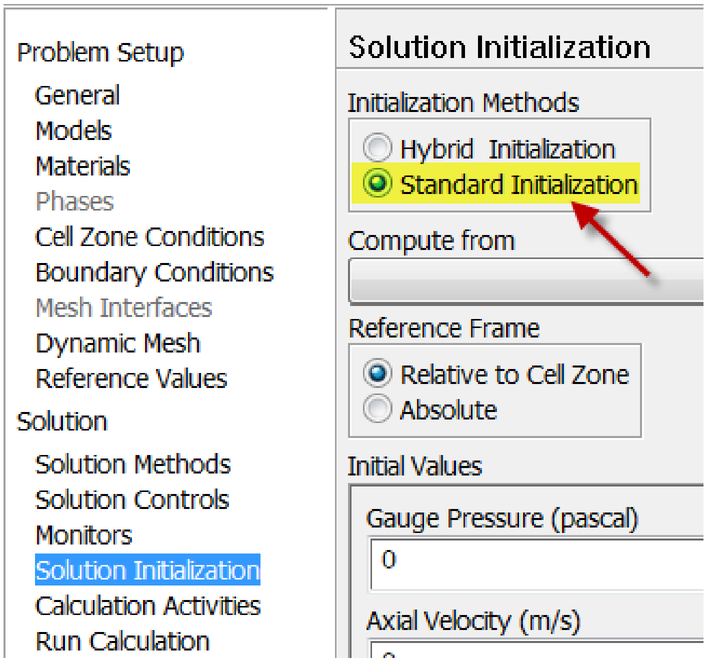


Click ***OK***.

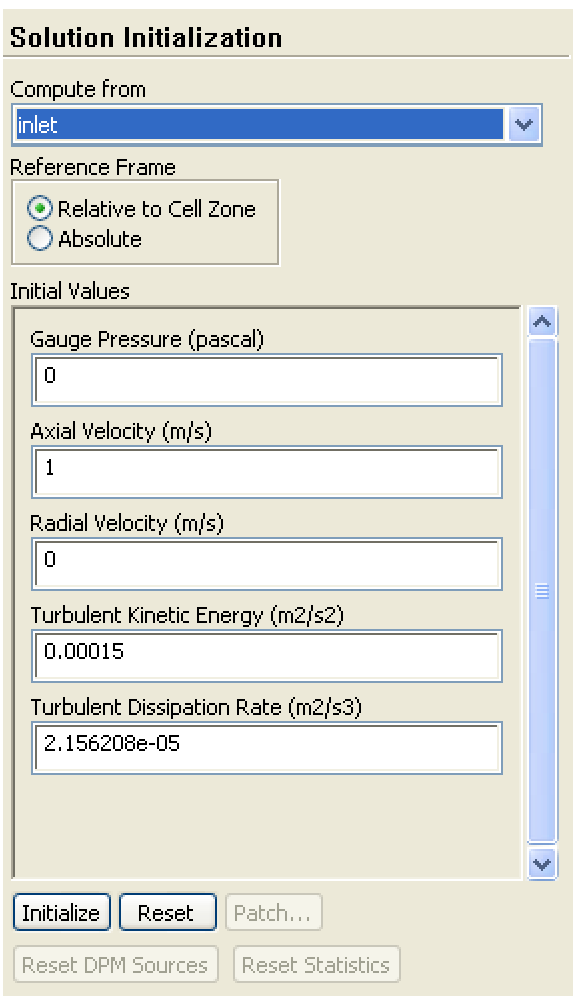
Set Initial Guess

We'll use an initial guess that is constant over the entire flow domain and equal to the values at the inlet:

Solution > Solution Initialization > Standard Initialization



In the *Solution Initialization* menu that comes up, choose ***inlet*** under ***Compute From***. The ***Axial Velocity*** for *all* cells will be set to 1 m/s, the ***Radial Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. The ***Turbulence Kinetic Energy*** and ***Dissipation Rate***(scroll down to see it) values are set from the prescribed values for the *Turbulence Intensity* and *Hydraulic Diameter* at the inlet.



Click ***Initialize*** (this is easy to overlook).

This completes the problem specification. Save your project.

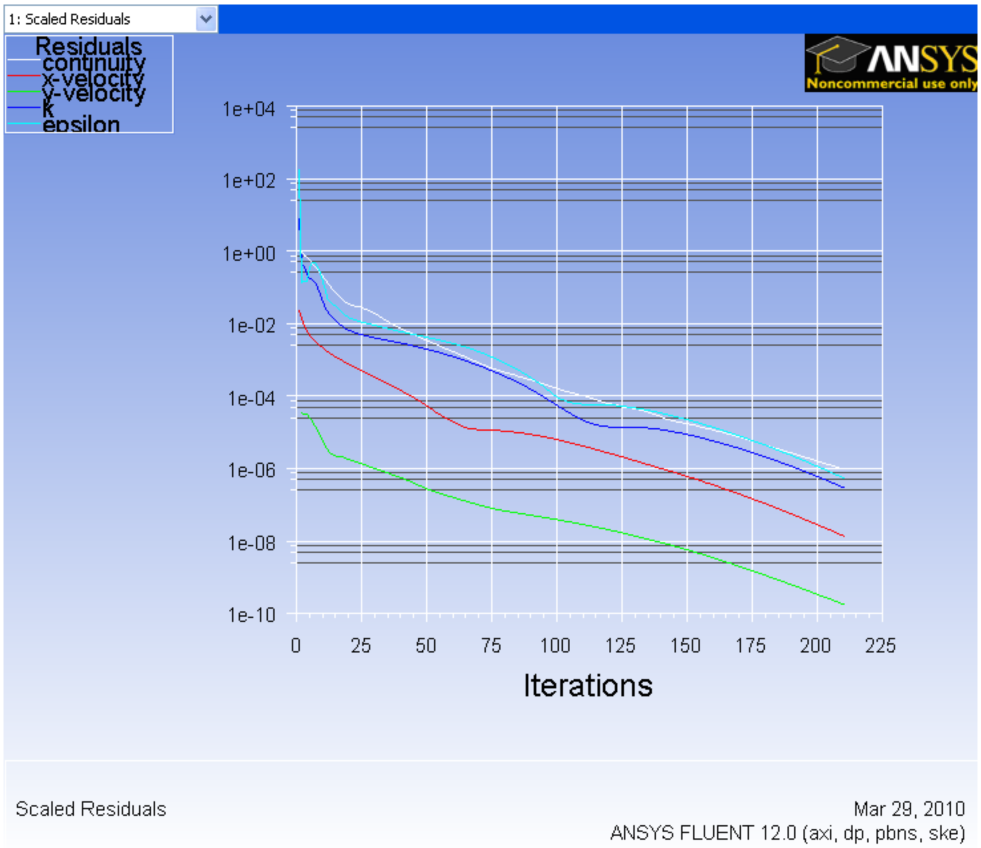
Iterate Until Convergence

Solve for 700 iterations.

Solution > Run Calculation

In the *Iterate* menu that comes up, change the ***Number of Iterations*** to `700`. Click ***Calculate***.

The solution converges in a total of about 220 iterations. You may get a different number of iterations to convergence depending on your mesh and software version.



Click here to see a higher resolution image.

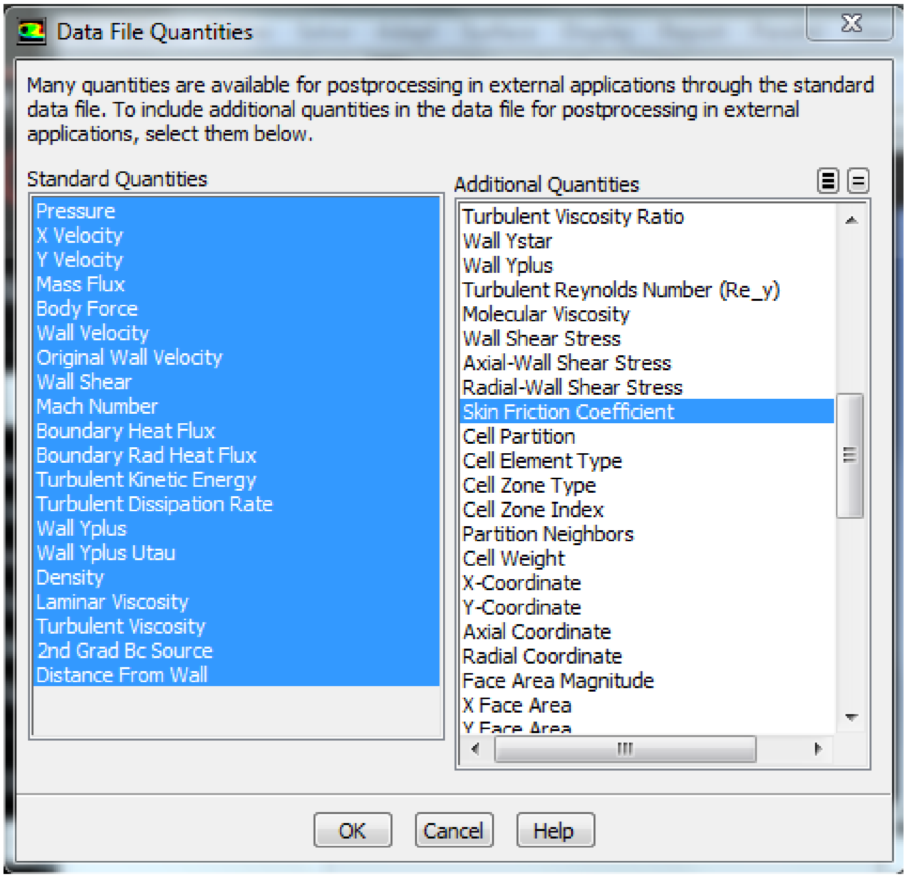
We need a larger number of iterations for convergence than in the laminar case since we have a finer mesh and are also solving additional equations from the turbulence model.

Setup Data Export

In addition to the standard data quantities, we would also like to view the results for the Skin Friction Coefficient. This quantity is not transferred to the post-processor by default; so we have to do it manually.

File > Data File Quantities

Under ***Additional Quantities***, select ***Skin Friction Coefficient***, which should be roughly half way down. Your window should now look like this:



Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules