Turbulent Pipe Flow (LES) - Verification & Validation

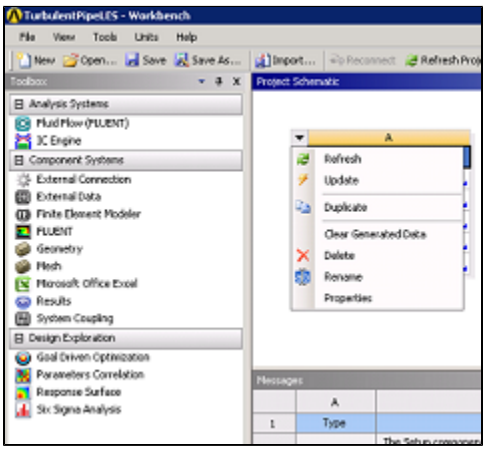
Author: Ranjith Tirunagari, Cornell University

Verification & Validation

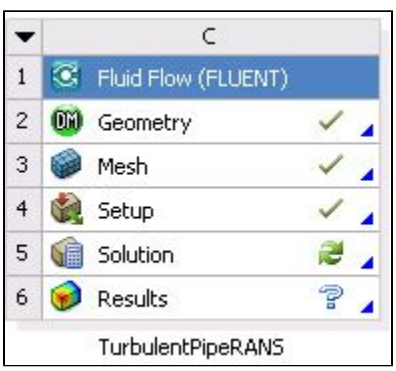
In order to verify the LES solution to the turbulent pipe flow problem, we will compare it against the solution obtained using standard k-e model.

Turbulent Pipe with k-e model

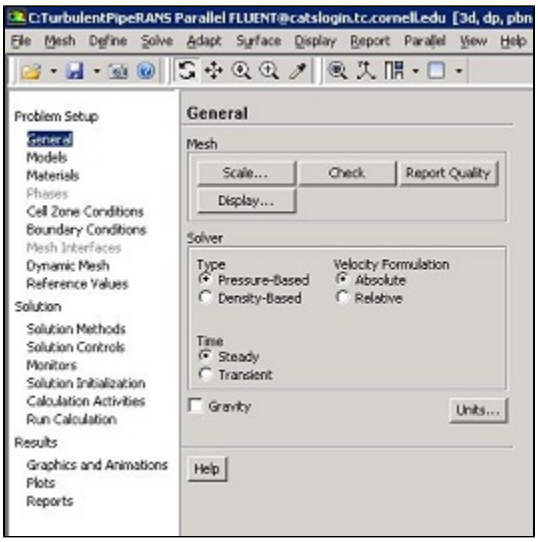
In the ***Workbench Project Page,*** left-click on the downward pointing arrow on the top-left corner of the ***TurbulentPipeLES*** project and select ***Duplicate*** fro m the drop down menu as shown in the figure below.



Higher Resolution Image
A new project will be created. Rename this new project as ***TurbulentPipeRANS*** as shown in figure below.

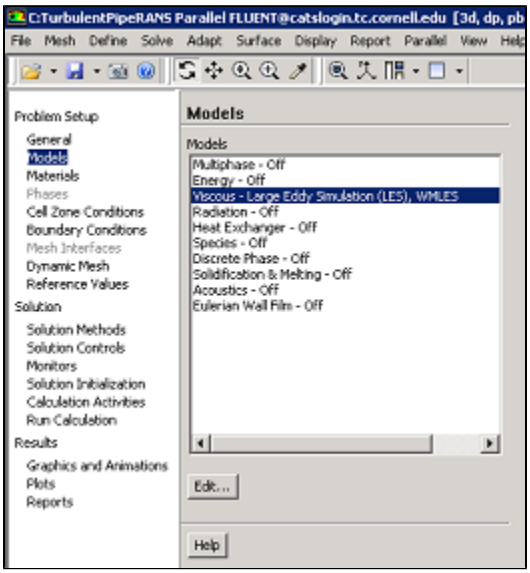


Next, double click on the ***Solution*** cell of the ***TurbulentPipeRANS*** project. This will launch the FLUENT solver. Here we will change the viscosity model to k-e model. In the FLUENT window, click on ***General*** tab on the left hand side pane. Choose ***Steady*** for ***Time***. Click ***OK*** for the pop-up warning.



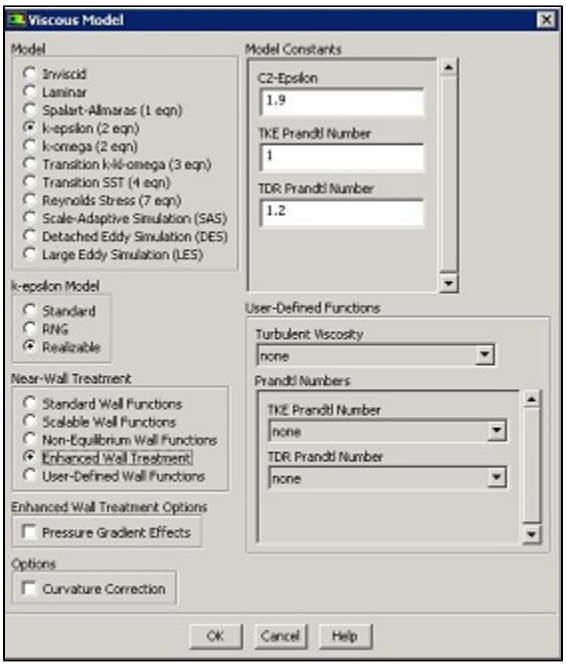
Higher Resolution Image

Next click on ***Models>Viscous*** on the left hand side pane as shown below. Choose ***Edit...***.



Higher Resolution Image

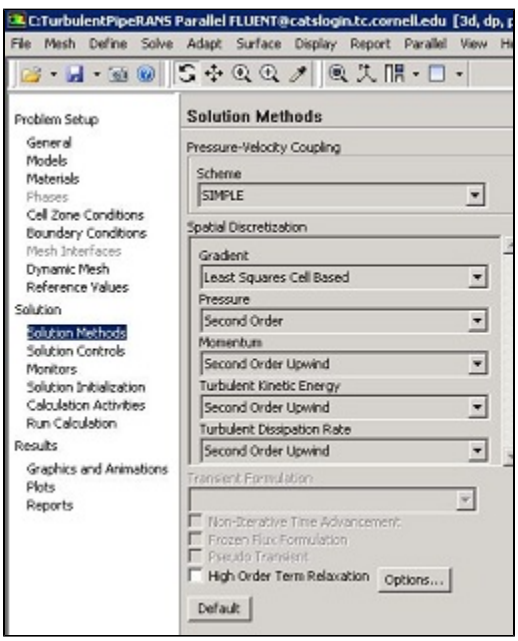
In the ***Viscous Model*** window, choose ***k-epsilon (2 eqn)*** for ***Model***. Choose ***Realizable*** for ***k-epsilon Model***. Choose ***Enhanced Wall Treatment*** for ***Near-Wall Treatment***. Click ***OK*** as shown in the figure below.



Higher Resolution Image

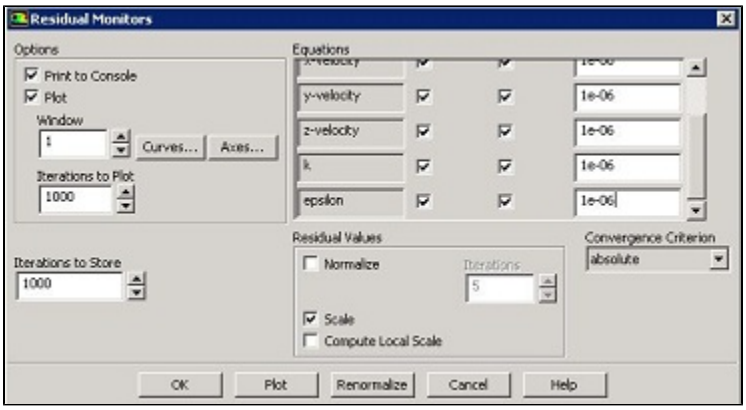
Click on ***Boundary Conditions*** tab on the left hand side pane. Choose ***Outlet*** and click ***Edit...***.Change the ***Turbulence Specification Method*** to ***Intensity and Hydraulic Diameter***. Set ***Backflow Turbulent Intensity*** to ***10%*** and ***Backflow Hydraulic Diameter*** to ***0.0127m***. Click ***OK***.

Next, click on ***Solution Methods*** tab on the left hand side pane. Choose ***Second Order Upwind*** for ***Turbulent Kinetic Energy*** and ***Turbulent Dissipation Rate***.



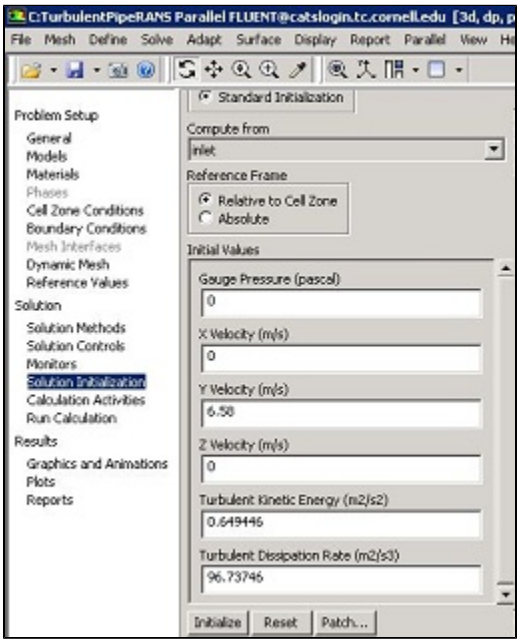
Higher Resolution Image

Click on ***Monitors > Residual*** and click ***Edit...***. Set ***1e-06*** for ***k*** and ***epsilon***. In order to get faster convergence, choose ***1e-05*** for ***continuity*** and click ***OK***.



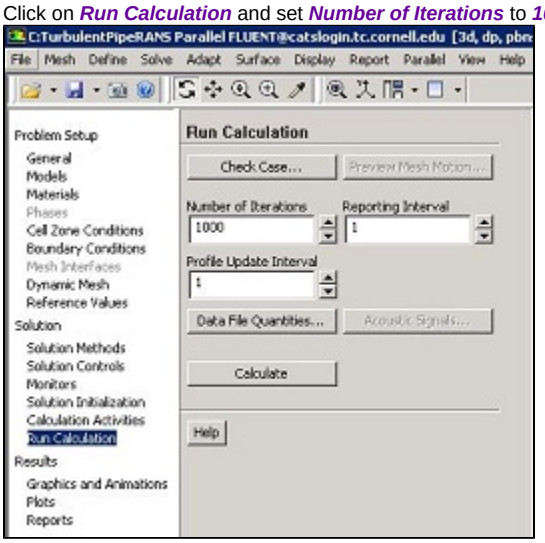
Higher Resolution Image

Click on ***Solution Initialization*** and click ***Compute from*** and choose ***inlet***.Click ***initialize*** as shown below.



Higher Resolution Image

Click on ***Run Calculation*** and set ***Number of Iterations*** to ***1000***.Click ***Calculate*** as shown below.

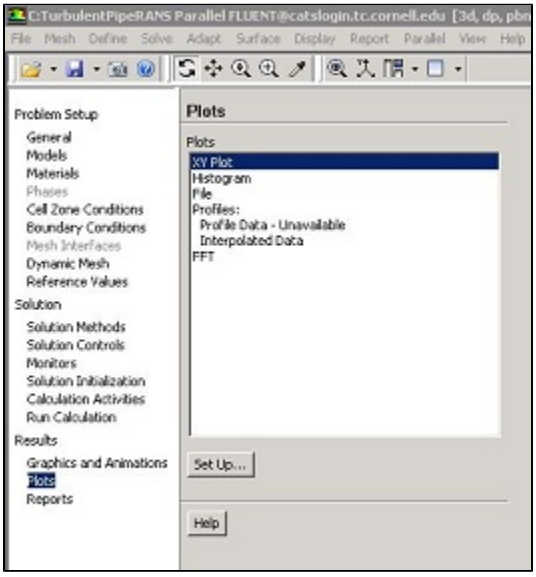


Higher Resolution Image

The solution converges in about 600 iterations.

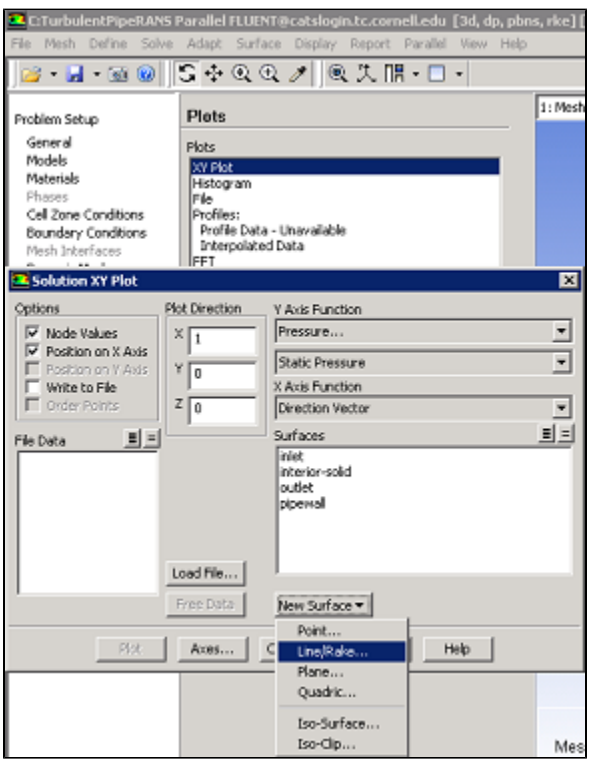
Create Line

Click on ***Plots > XY Plot > Set Up...*** as shown below.



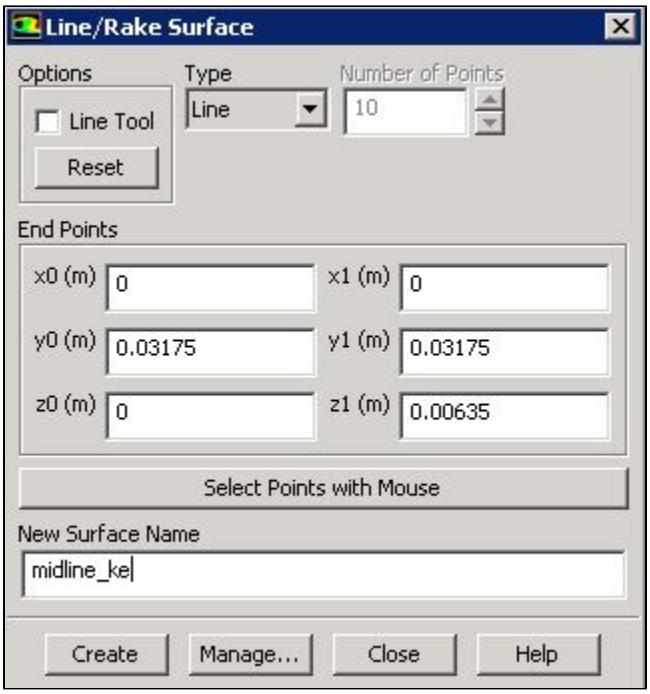
Higher Resolution Image

In the ***Solution XY Plot***, choose ***New Surface > Line/Rake...*** as shown below.

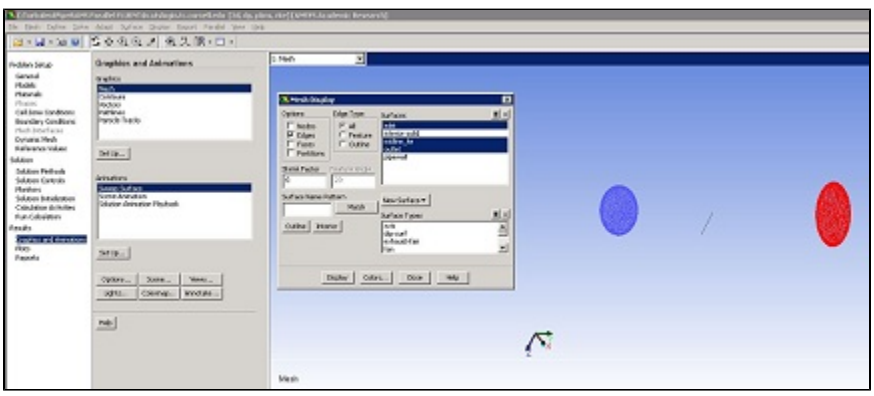


Higher Resolution Image

Choose the coordinates of the two points that define the line as below and name the line as ***midline_ke*** and click ***Create*** and ***Close*** the window.

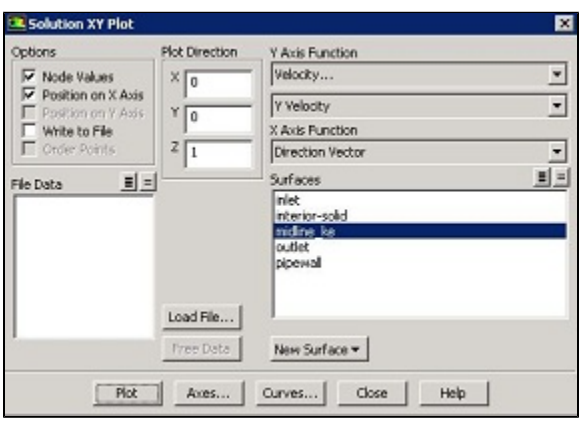


The line can be viewed by clicking ***Graphics and Animations > Mesh > Set Up...***.

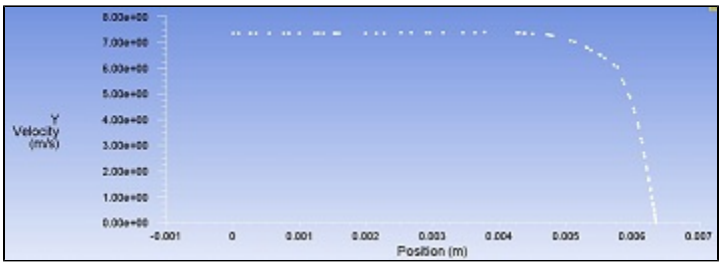


Higher Resolution Image

Go back to the ***Solution XY Plot*** window by clicking ***Plots > XY Plot > Set Up...***. In this window, choose ***Velocity...*** and ***Y Velocity*** for ***Y-Axis Function***. Also, choose ***0, 0, 1*** as the ***Plot Direction***. Finally, choose ***midline_ke*** under ***Surfaces***. and click ***Plot***.

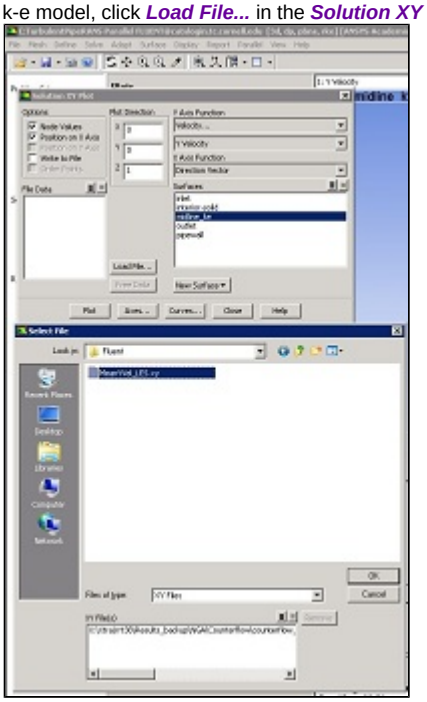


Higher Resolution Image



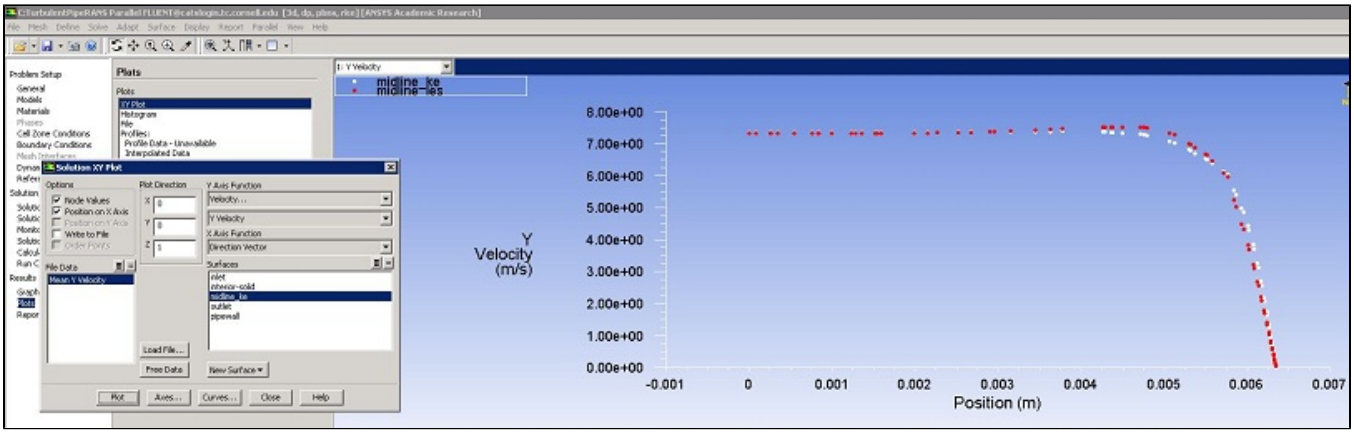
Higher Resolution Image

To compare the solution from LES simulation with that from the k-e model, click ***Load File...*** in the ***Solution XY Plot*** and choose the saved file ***MeanYVel_LES.xy*** from the previous section as shown in the figure below.



Click Here for Higher Resolution

In the ***Solution XY Plot*** select ***Mean Y Velocity*** under ***File Data*** and click ***Plot***. The figure below shows the comparison of the mean Y velocity from k-e and LES simulations, both the results are very close which validates the solution procedure for the LES simulation.



Click Here for Higher Resolution

Go to Exercises

Go to all FLUENT Learning Modules