Turbulent Pipe Flow (LES) - Numerical Solution

Author: Ranjith Tirunagari, Cornell University
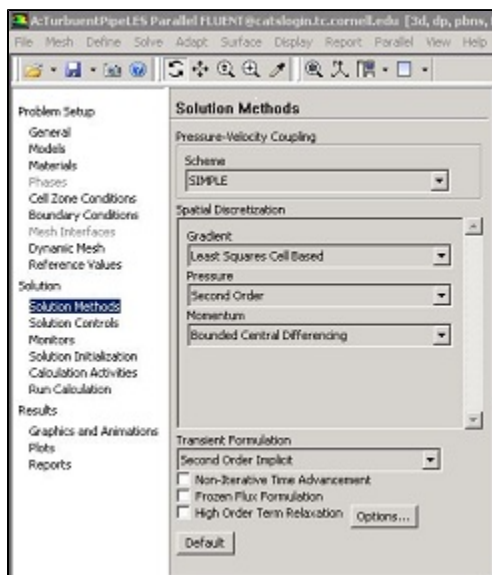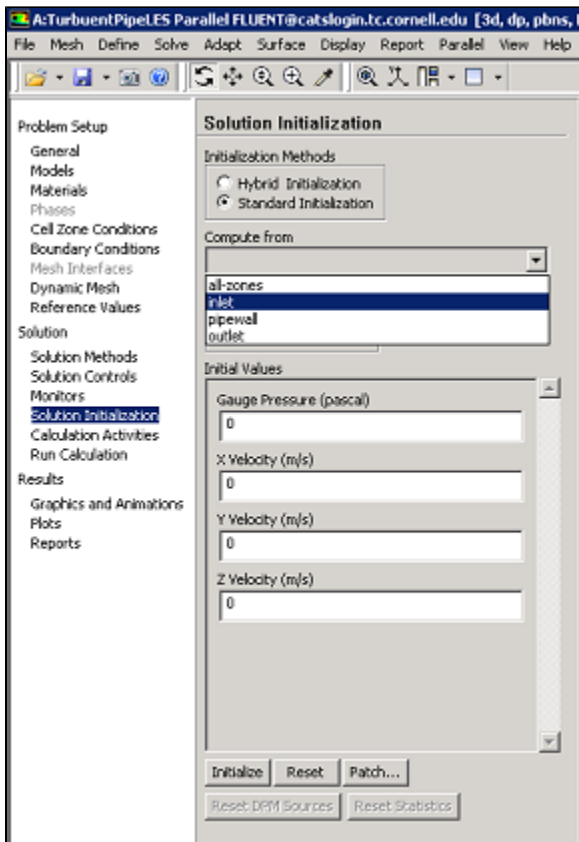
Numerical Solution

Solution Methods

Click on *Solution Methods* then click on *Momentum* and select *Bounded Central Differencing* as shown in the image below. Also choose, *Second order* for *Pressure* and *Second Order Implicit* under *Transient Formulation.* as

Click Here for Higher Resolution

Set Initial Guess

Here, the flow field will be initialized to the values at the inlet. In order to carry out the initialization click on *Solution Initialization* then click on *Compute from* and select *inlet* as shown below.
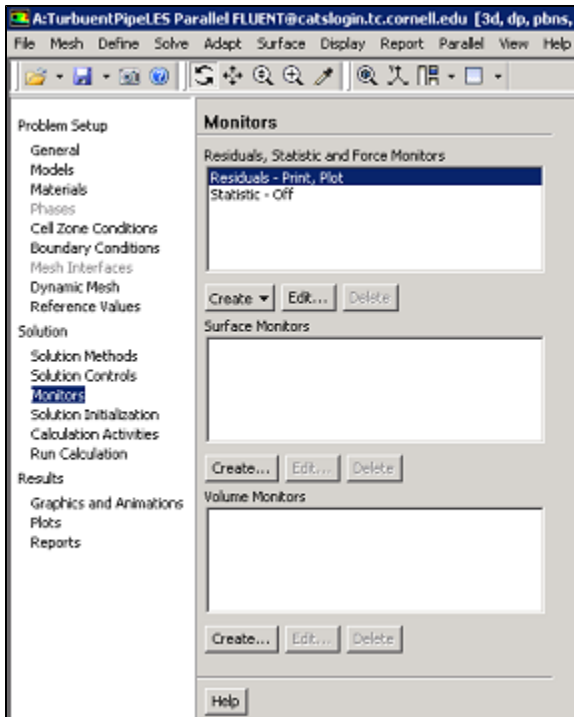
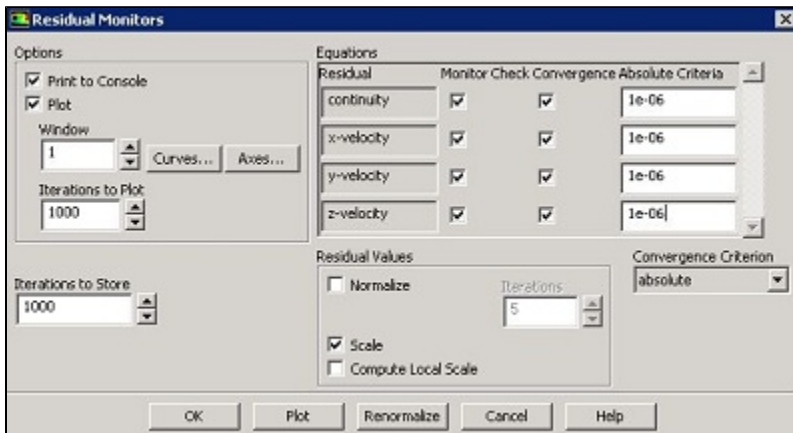Then click *initialize* to initialize the domain with an initial guess. This completes the initialization.

Set Convergence Criteria

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. In order to specify the residual criteria *(Click) Monitors > Residuals > Edit...*, as shown in the image below.

Next, change the residual under **Convergence Criterion** for **continuity**, **x-velocity**,and **y-velocity**, all to 1e-6, as can be seen below.
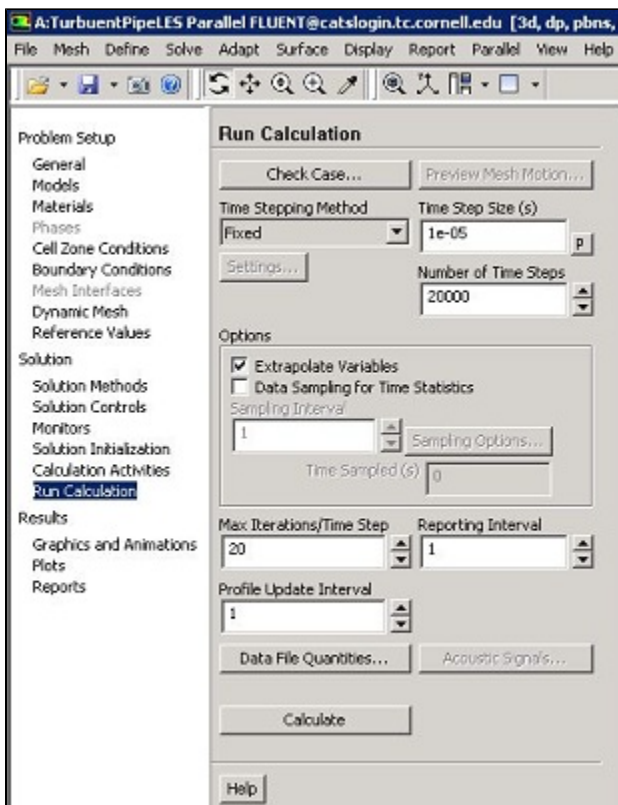
Lastly, click **OK** to close the **Residual Monitors** menu.

Execute Calculation

Note that, we have to run the transient simulation to a statistically stationary state and then collect statistics. Click on **Run Calculation** then choose the **Time Step Size(s)** as 1e-05 and the **Number of Time Steps** as 20000. Also choose **Extrapolate Variables** option (refer to FLUENT documentation for information) and leave the **Max Iterations/Time Step** as 20 (default). The corresponding image is shown below.

As a safeguard save the project now. Now, click on **Calculate** in order to run the calculation. The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated. The residuals decrease during the inner iterations and jump again when we advance by one time step as shown in image below.

From the above inputs, it is clear that 20 inner iterations are carried out to advance the simulation by one time step and we requested for 20,000 time steps. So, typically these simulations are very long even if we use 4 processors. To reduce the simulation time one can opt to High Performance Computing (HPC) resources. The calculations performed for this tutorial are based on the simulations run on an HPC cluster using 48 cores.
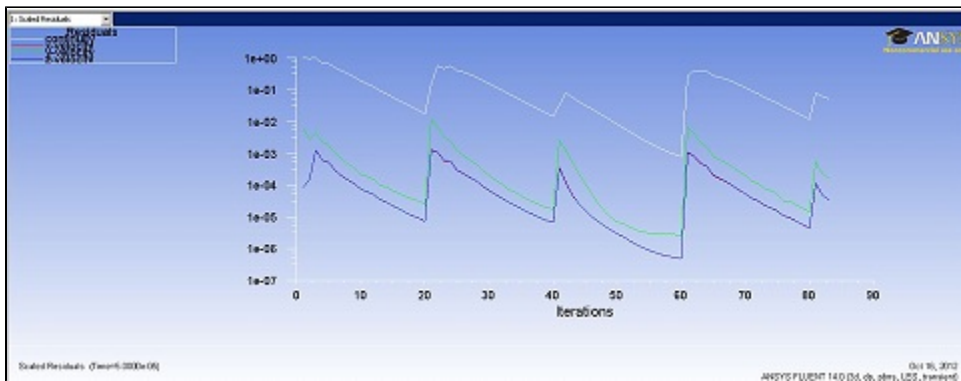
In this simulation, it is verified that the statistically stationary state is reached by advancing the simulation from 0s to 0.1s in physical time i.e. using only 10,000 time steps with 1e-05s time step size with 20 inner iterations.When statistically stationary state is reached, statistics have to be collected by advancing the physical time from 0.1s to atleast 0.15s. Use 5,000 time steps with 1e-05 time step size with 20 inner iterations. This is done as shown in the figure below. Select the *Data Sampling for Time Statistics* to start collecting the statistics. You will see the physical time for which the statistics have been collected across *Time Sampled (s)*.
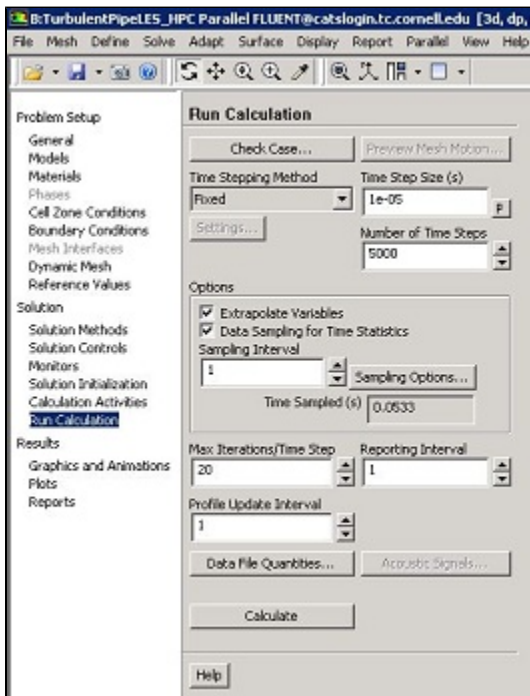
Collect the statistics for 0.05s (this corresponds to marching from 0.1s to 0.15s in time) as shown in the above figure. At this point save the project once again.

Go to Step 6: Numerical Results