

# Turbulent Pipe Flow (LES) - Pre-Analysis & Start-Up

Author: Ranjith Tirunagari, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

## Pre-Analysis & Start-Up

### Preliminary Analysis

In Large-Eddy Simulations, the instantaneous velocity  $\mathbf{U}(\mathbf{x},t)$  is decomposed into a filtered component  $\langle \mathbf{x},t \rangle$  and a residual component  $u'(\mathbf{x},t)$ . The filtered velocity component represents the large scale unsteady motions. In LES, the large scale turbulent motions are directly represented whereas the effects of small scale turbulent motions are modeled. The filtered equations for the filtered velocity can be obtained from the Navier-Stokes equations. The non-linear convective term in the momentum equation introduces a residual stress tensor which is due to the residual motions. Closure is needed for this residual stress tensor and hence require modeling. There are simple to complex models in FLUENT which we will use.

Since we are solving for  $\langle \mathbf{x},t \rangle$ , the LES is an unsteady simulation where we march in time. In order to collect statistics like the mean and the root mean square (r.m.s.) velocities, we need to first reach a statistically stationary state. In comparison, simulation using k- model solves only for the mean velocity.

More information on the LES can be found in Turbulent Flows book by Pope[2000].

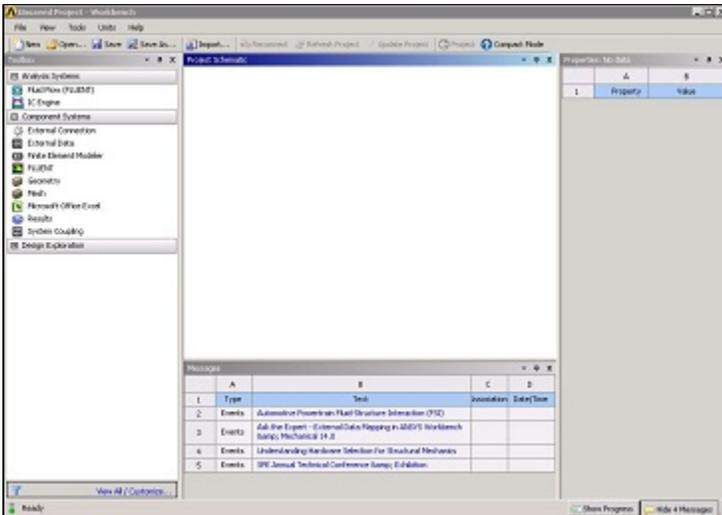
### Start ANSYS FLUENT

Since LES is a three-dimensional unsteady simulation, the computational domain is a full pipe.

Prior to opening ANSYS, create a folder called *turbulent\_pipe\_LES* in a convenient location. We'll use this as the working folder in which files created during the session will be stored. For this simulation Fluent will be run within the ANSYS Workbench Interface. Start ANSYS workbench:

**Start> All Programs> Ansys 14.0> Workbench 14.0**

The following figure shows the workbench window.



[Higher Resolution Image](#)

[Go to Step 2: Geometry](#)

[Go to all FLUENT Learning Modules](#)