Turbulent Pipe Flow - Pre-Analysis & Start-Up

Author: Rajesh Bhaskaran, Cornell University

Pre-Analysis & Start-Up

Preliminary Analysis

A turbulent flow exhibits small-scale fluctuations in time. It is usually not possible to resolve these fluctuations in a CFD calculation. So the flow variables such as velocity, pressure, etc. are time-averaged. Unfortunately, the time-averaged governing equations are not closed. (i.e. They contain fluctuating quantities which need to be modeled using a turbulence model.) No turbulence model is currently available that is valid for all types of flows and so it is necessary to choose and fine-tune a model for particular classes of flows.

In this exercise, you'll be turned loose on variants of the *k*- model. But in the real world, tread with great *caution*: you should evaluate the validity of your calculations using a turbulence model very carefully (which, ahem, means that there is no getting away from studying fluid dynamics concepts and numerical methods very carefully). FLUENT should *not* be used as a black box. The *k*- models consist of two differential equations: one each for the turbulent kinetic energy *k* and turbulent dissipation . These two equations have to be solved along with the time-averaged continuity, momentum and energy equations. So turbulent flow calculations are much more difficult and time-consuming than laminar flow calculations. This is an exercise to whet your appetite for turbulent flow calculations.

Start ANSYS FLUENT

Since the flow is axisymmetric, the geometry is a rectangle as in the Laminar Pipe Flow tutorial. We will first use a 100x30 mesh (i.e. 100 divisions in the axial direction and 30 divisions in the radial direction).

We could create this mesh from scratch, as in the Laminar Pipe Flow tutorial, but instead, we will modify the previous 100x5 to get the 100x30 mesh. This will introduce you to the art of modifying meshes in the ANSYS Workbench Mechanical Mesher.

Go to Step 2: Geometry

Go to all FLUENT Learning Modules