Turbulent Jet - Pre-Analysis & Start-Up

Authors: Lara Backer, Cornell University - taken in part from MAE 7340 Analysis of Turbulent Flows and Dr. S. Pope

Pre-Analysis & Start-Up

Preliminary Analysis

The turbulent equations that we will be solving are the Reynolds equations, where the Navier Stokes equations are transformed by substituting the velocities U(x,t) as the sum of mean velocity <U(x,t)> and the turbulent velocity fluctuations u(x,t). This is known as the Reynolds decomposition. Unfortunately, this substitution into the nonlinear term of the Navier Stokes equations results in additional terms known as the Reynolds stresses. These terms are crucial; they are what separate turbulent flow equations from the well known laminar form. These new Reynolds transport equations cannot be solved without closure models for the Reynolds stresses, because with solving for mean velocity, there will be more unknowns than there are equations.

The k- model is one of the most widely used closure models in which two additional equations are solved for variables k (turbulent kinetic energy) and (turbulent dissipation). This model is complete, in that these two variables can be manipulated to describe a a turbulent length and time scale. The assumptions made in using this model are that the two k and transport equations are valid, and the turbulent viscosity hypothesis, that the Reynolds stresses are proportional to the mean rate of strain (similar to the Newtonian hypothesis relating stress and strain that is used in the Navier Stokes equations).

This is a more advanced tutorial than the other turbulent flow tutorials on this site, and requires you to export data for analysis and equation manipulation for the Results section onward.

Starting Fluent

While the actual jet inflow is much smaller than the mesh, the additional jet area at the inlet allows for entrainment of air near the inlet, and radial dispersal of the fluid into the surroundings as X increases.

The geometry and mesh have already been created, and can be read directly into Fluent as case and data files. **Download them here**. This tutorial will focus on the analysis of laminar and turbulent jets, and specification of the turbulent k-epsilon model.

Once you have downloaded the files, open Ansys FLUENT. Select double precision and the **2D** case (the case and data files cannot be read in if you select the 3D case).

Go to Step 2: Laminar Setup and Solution

Go to all FLUENT Learning Modules