Sudden Expansion - Pre-Analysis & Start-Up

Authors: Yong Wang & Said Elghobashi, UC Irvine

Pre-Analysis & Start-Up

Preliminary Analysis

The computational domain consists of 2 parts: one for the small pipe and the other for the larger pipe. In the small pipe, the viscous boundary layer grows along the pipe wall starting at the inlet, and eventually a fully-developed velocity profile forms provided that the small pipe is long enough. As that flow passes through the expansion entrance, a recirculation zone forms in the corner of the larger pipe. The flow will become fully developed again after sufficient distance downstream from the recirculation zone.
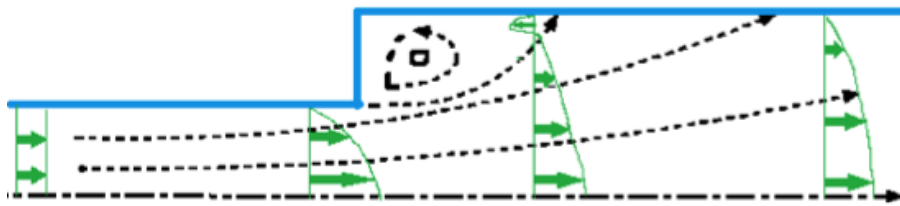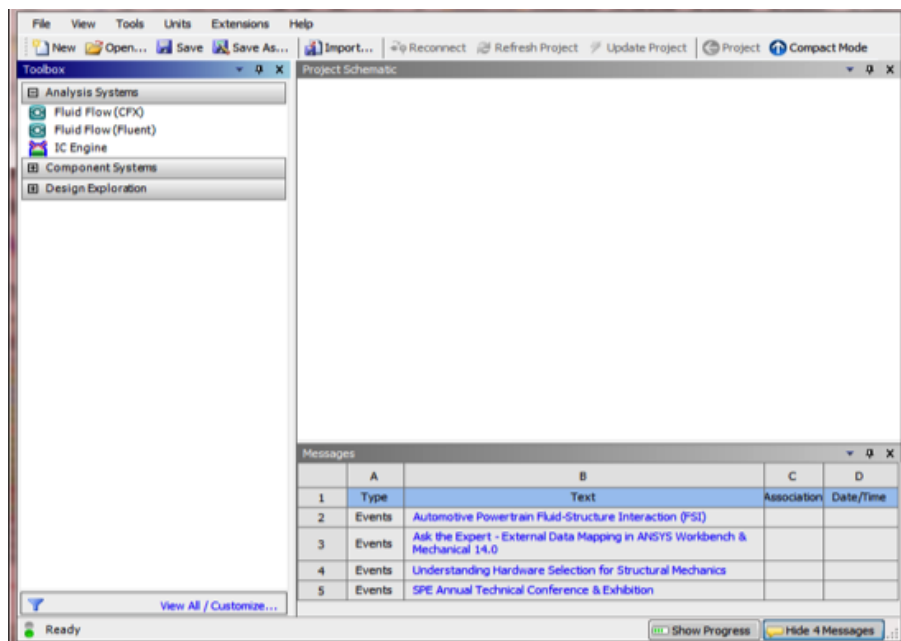

We'll create the geometry and mesh in ANSYS 14.0 which is the preprocessor for FLUENT, and then read the mesh into FLUENT.

Start ANSYS FLUENT

Prior to opening ANSYS, create a folder called *pipe* 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.

Go to Step 2: Geometry