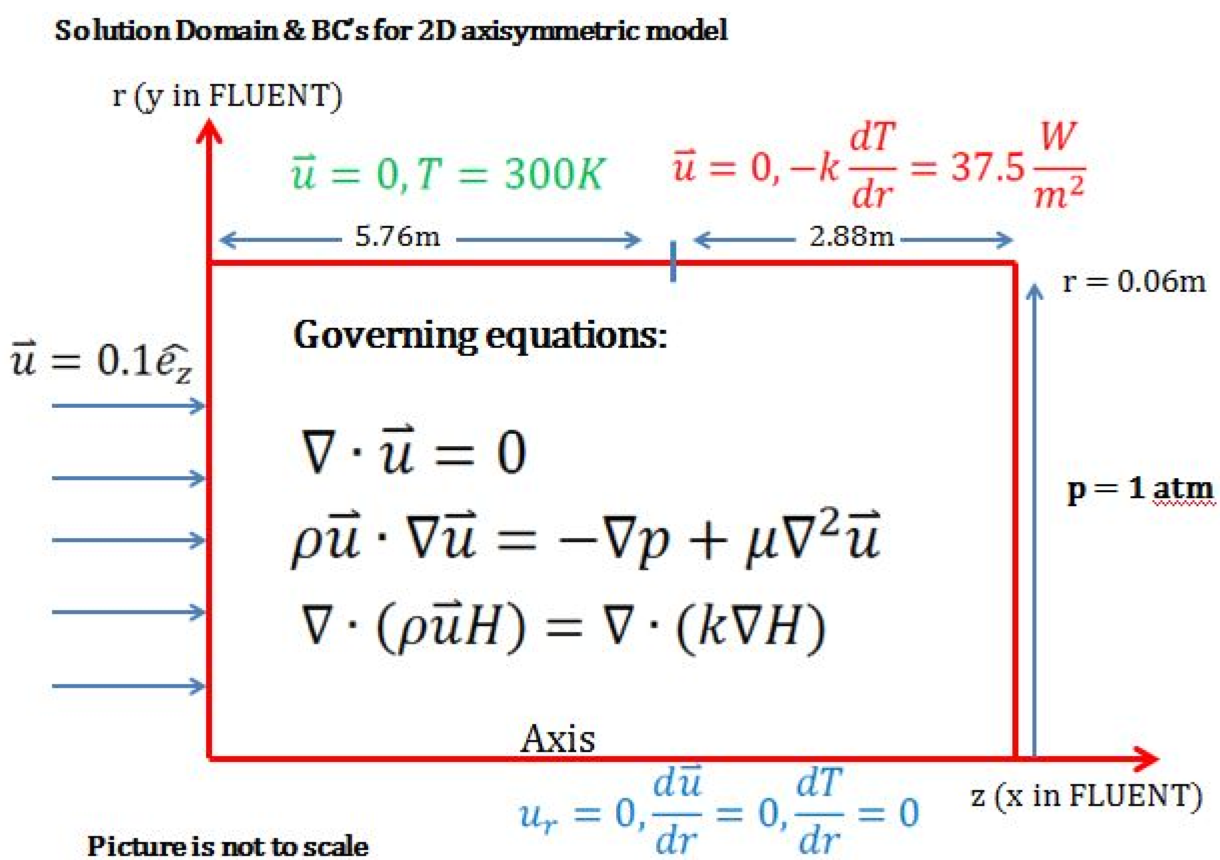
2D Steady Convection - Pre-Analysis & Start-Up

Author: Benjamin Mullen, Cornell University

Pre-Analysis & Start-Up

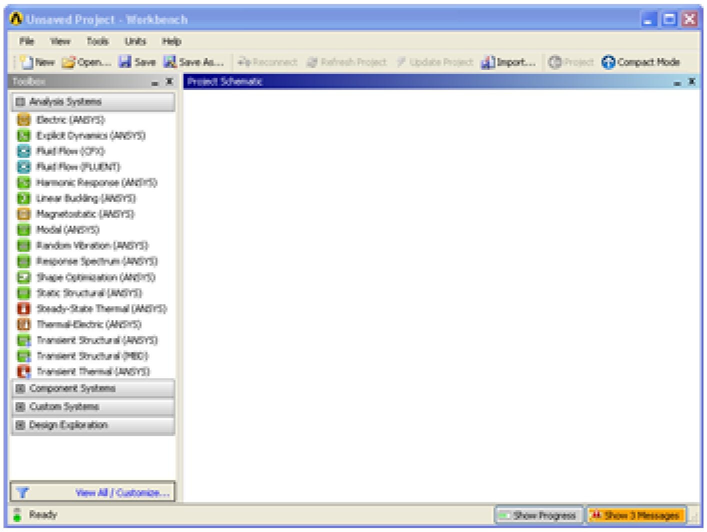
Boundary Value Problem

We will assume that the flow is axisymmetric. This means that in cylindrical coordinates (*r*, , *z*), there is no variation in the circumferential direction (). So the geometry in the simulation is a rectangle as shown below. Revolving the rectangle 360 degrees about the axis gives the full 3D pipe geometry. We need to solve a boundary value problem. The governing equations for this problem are conservation of mass, conservation of momentum (in *r* and *z* directions), and conservation of energy. The governing equations and boundary conditions are shown below. We will solve this boundary value problem numerically using the ANSYS FLUENT solver.



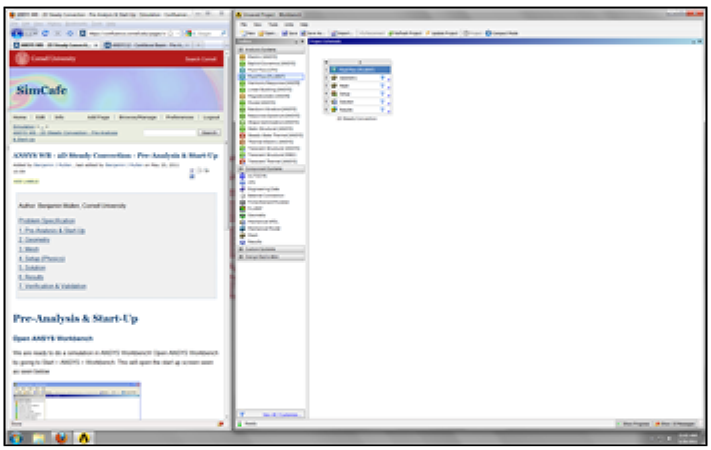
Open ANSYS Workbench

We are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen seen as seen below



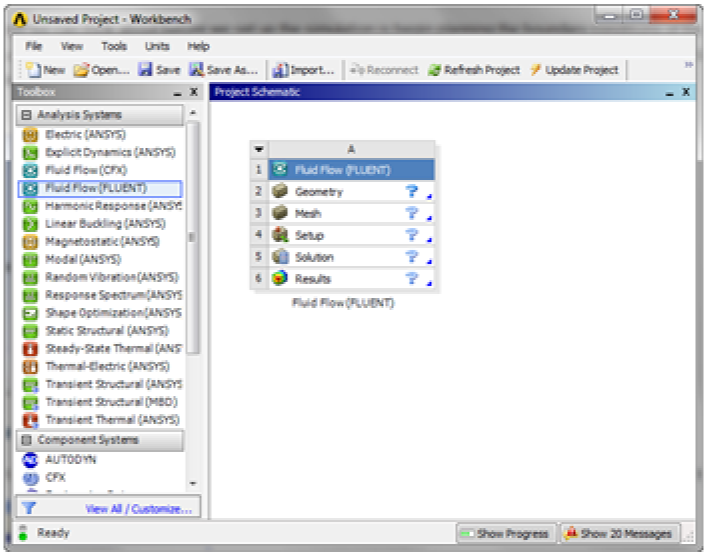
Screen Management

This tutorial is designed such that the user can have both ANSYS Workbench and the tutorial open. As shown below, this online tutorial should fill approximately 1/3 of the screen, while ANSYS Workbench fills the remaining 2/3 of the screen.



Setup Project

To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We will be using FLUENT to complete the simulation. Load the ***Fluid Flow (FLUENT)*** box by dragging and dropping it into the Project Schematic.



Right click the top box of the project schematic 

and go to ***Rename***, and name the project `2D Steady Convection`. You are ready to create the geometry for the simulation.

Go to Step 2: Geometry

Go to all FLUENT Learning Modules