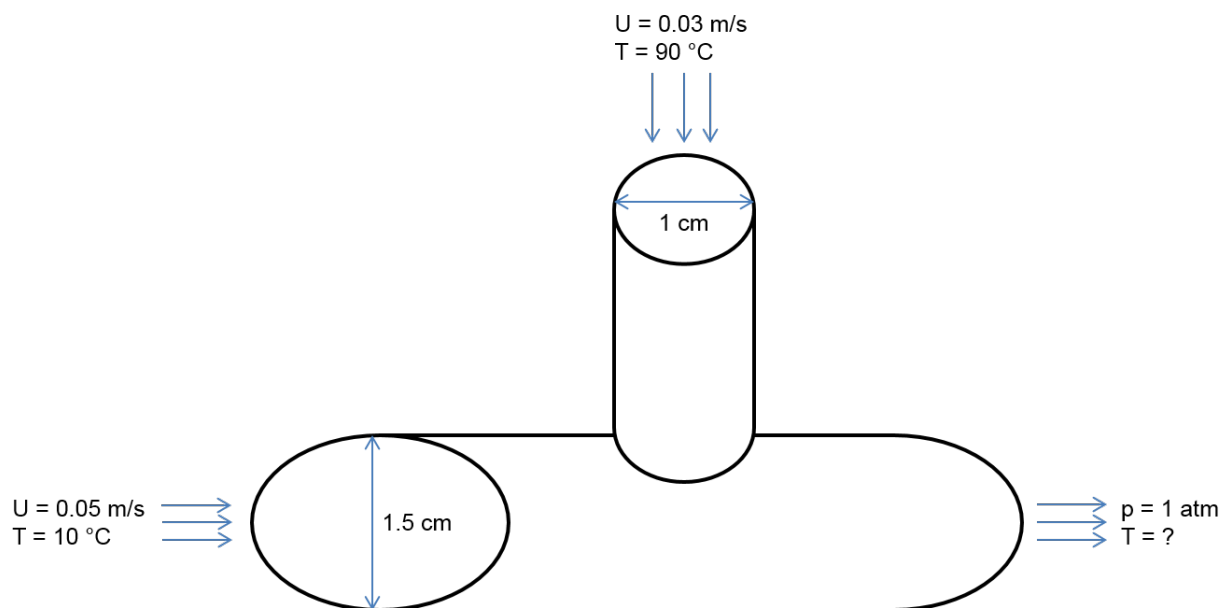


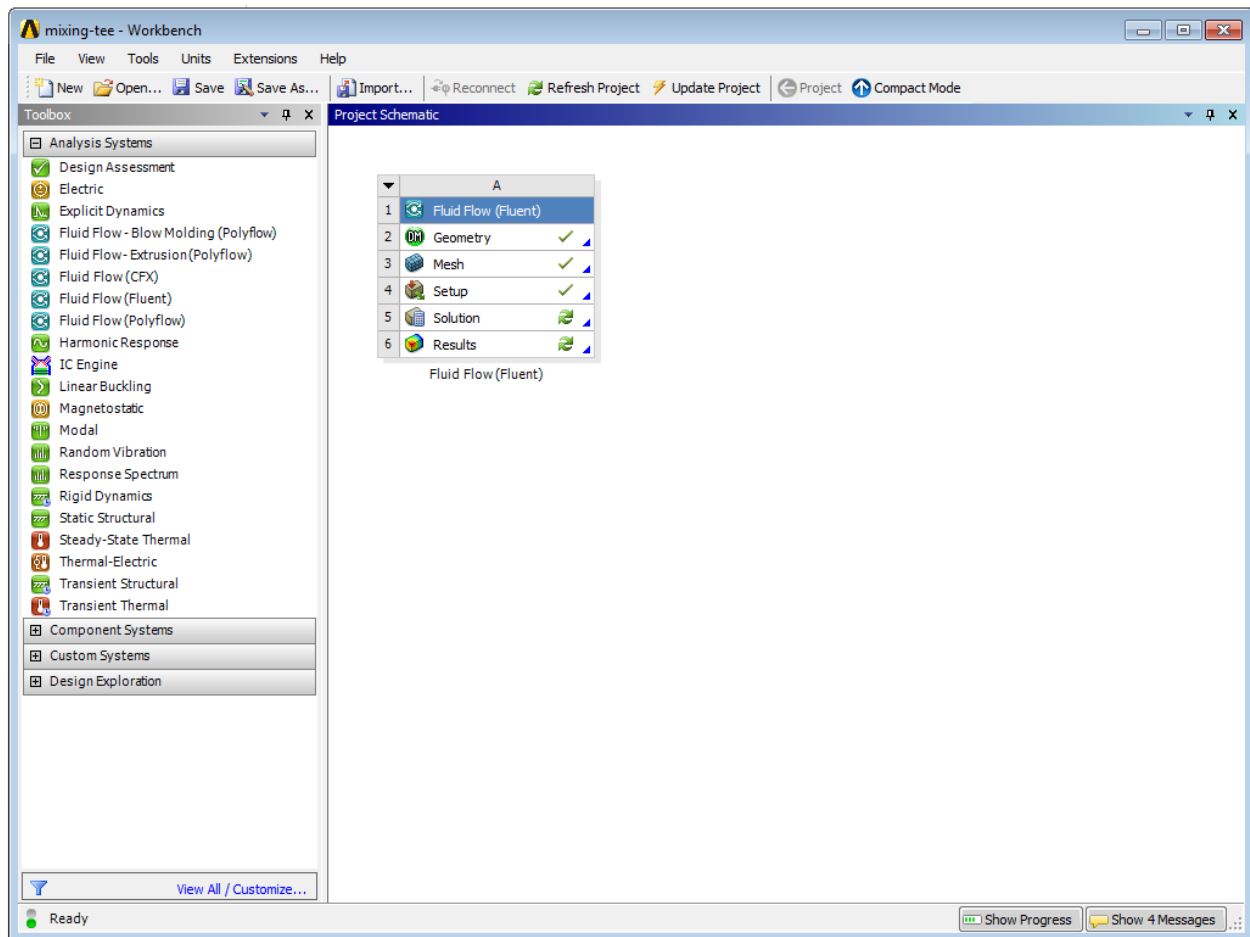
## Introduction

The purpose of this laboratory exercise is to provide an introduction to the commercial computational fluid dynamics (CFD) software FLUENT. FLUENT is comparable to many other commercial CFD codes in that it uses the finite-volume method to solve the equations that govern the physics behind many problems. FLUENT is not the only commercial code used in industry, though it is fairly popular. The real trick to learning any piece of software is to know 1) what are its capabilities and limitations and 2) where in the interface the various features are hidden. This is the first of many exercises that will introduce you to the software. We begin by simulating a mix-tee.

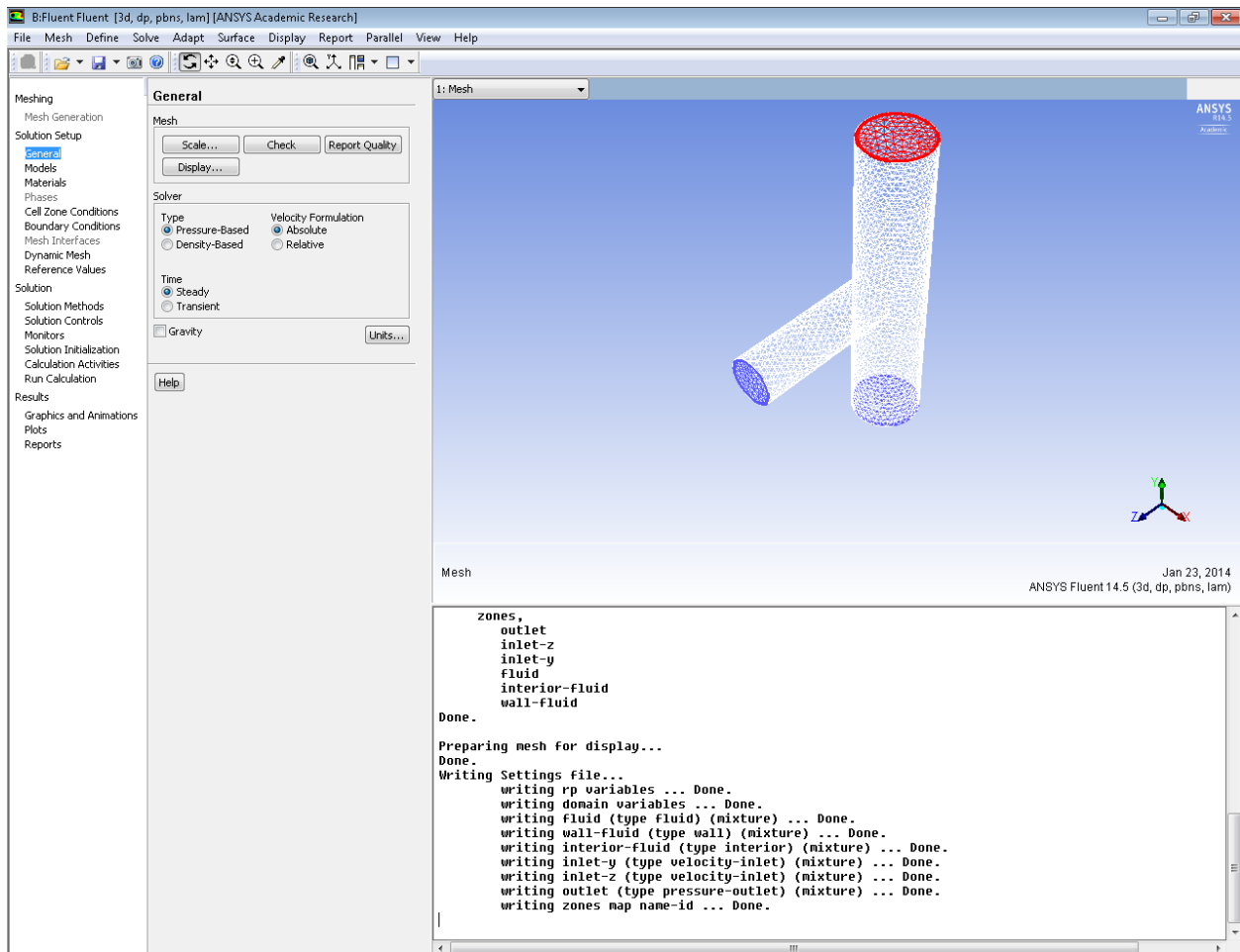
## Problem description



Start ANSYS Workbench. Close the “Getting Started” box if popped up. Open the project “lab 01 – mixing-tee” from Toolbar.



A list of tasks is loaded in the Project Schematic side of the interface. Note that the Geometry and Mesh are marked by green checks, meaning they are ready. **Double-click “Setup” to launch FLUENT and start setting up the problem. Choose double precision and click OK.**



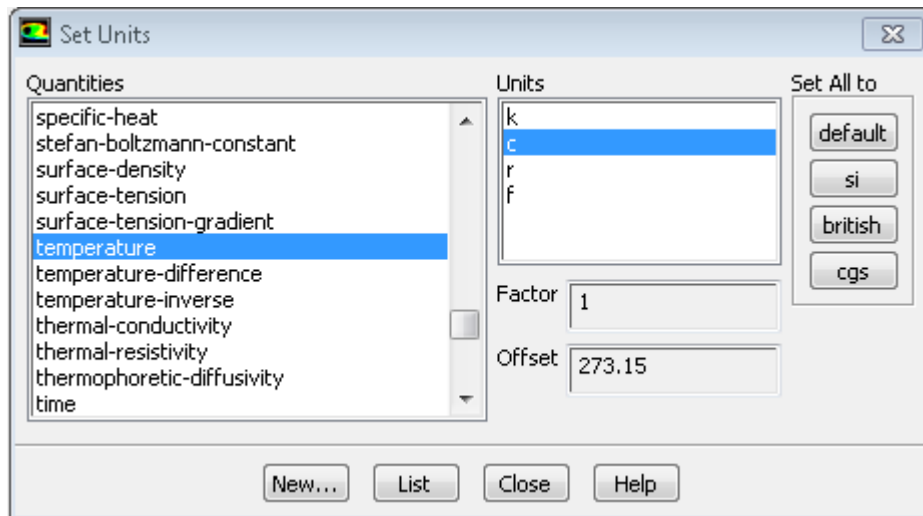
A wireframe of the mesh will load in the graphics window, and some details of the mesh will print to the console window. The left side of the interface contains a list of items that you will have to define each time you set up a model in FLUENT. In fact, if you look at the main menu, you will see these exact tasks under the “Define” and “Solve” menus. The order of this menu suggests the order in which you should complete the required tasks.

Notice when you click on the different entries in the left pane, how the next section over changes and reveals sub-options and more tasks for you to complete when defining the model.

## General

Here is where you can scale the coordinates of your mesh, for example if you defined your geometry in millimeters, you can easily scale it down by a factor of 1000 to get it to meters. FLUENT requires standard SI units for all dimensions and properties. Here you define steady or transient, whether or not you want a pressure-based solver (incompressible flow) or a density-based solver (compressible flow). You can also define your out-of-plane assumption when working with 2D models (planar vs axisymmetric).

Click on “Units” and set the unit of temperature to Celsius.

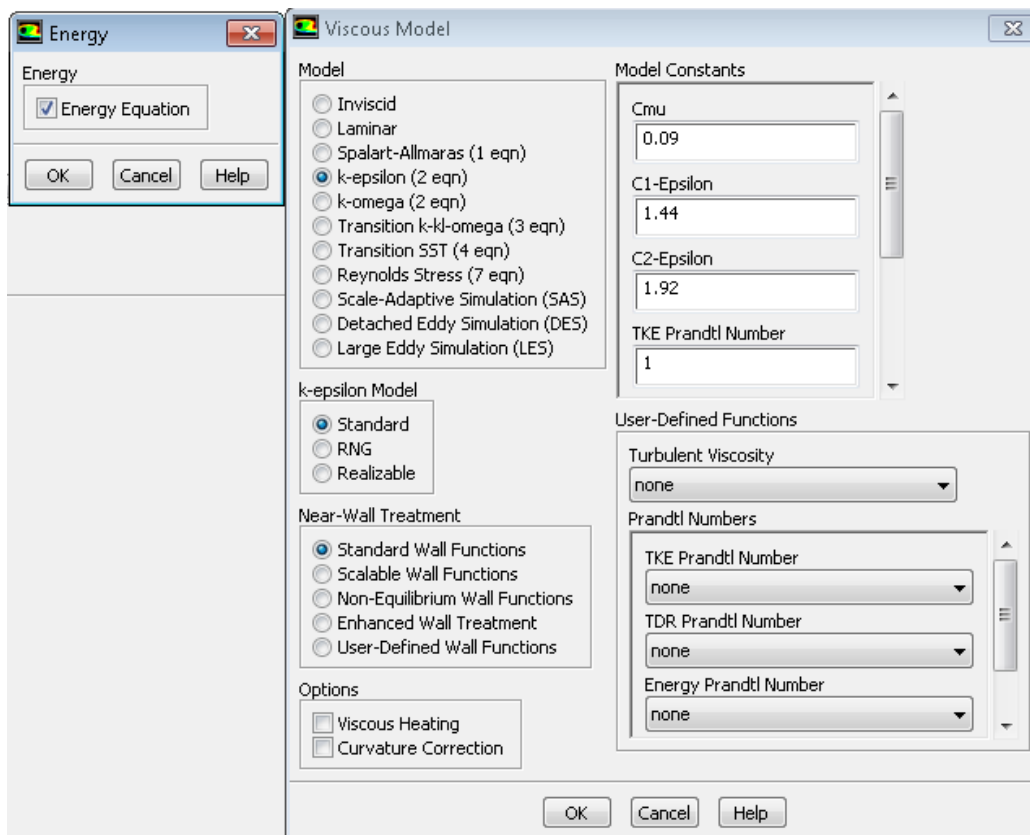


## Models

Here is where you tell FLUENT what physics you want to include in the model, and various related options.

Double click on “Energy” and click the check box to enable the solution of the energy equation.

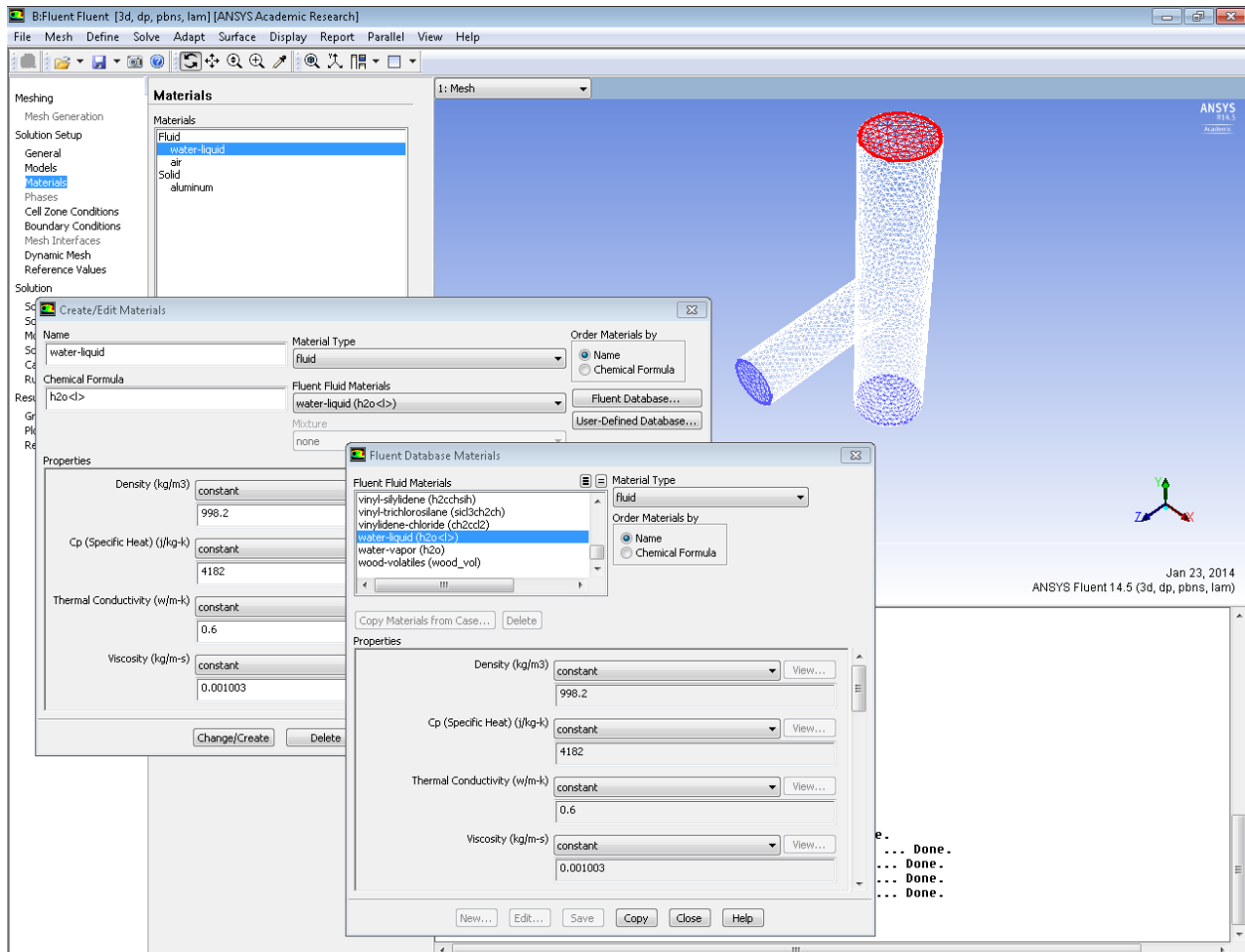
Double click on “Viscous” and examine the viscous models available in FLUENT. We will keep the default “laminar” for this tutorial.



## Materials

Here is where you define your material properties. The default fluid is always air. You could create new material by typing in its physical properties. Alternatively, FLUENT has most commonly used materials in its database. You simply need to copy them from there.

Click “Create/Edit...”, and then “FLUENT Database”. Find “water-liquid (h2o<l>)” from the list and click “Copy”. Close the windows and see water-liquid appear in the available fluid list.



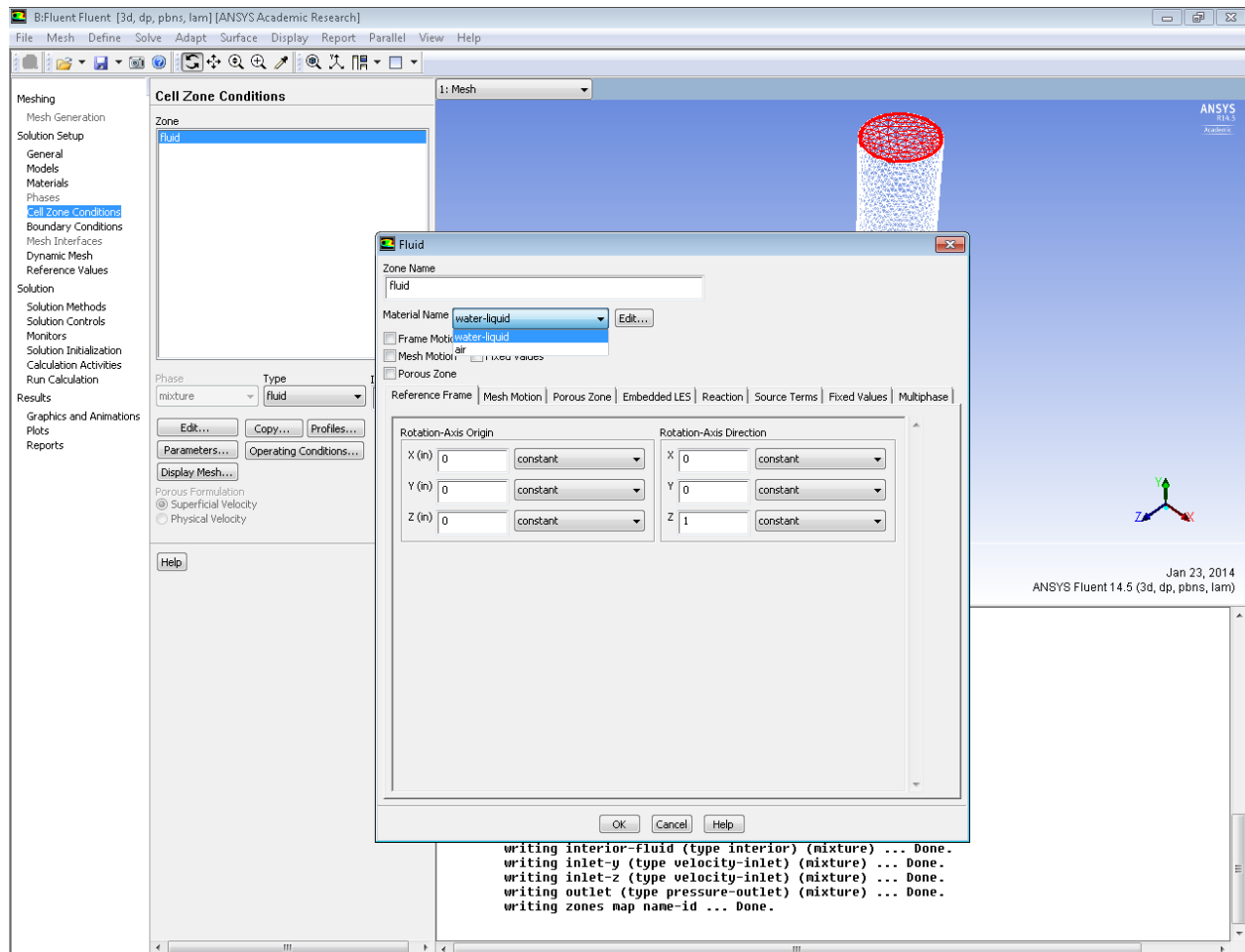
## Phases

Phases is grayed out for now- since we are working with a single phase simulation, you don't need to do anything here. We will revisit this later as needed.

## Cell zone conditions

Here is where you define cell zone properties. For instance, define the material in the zone, setup porous zones, etc.

Double click the zone called “fluid”. Change “Material Name” to “water-liquid”.

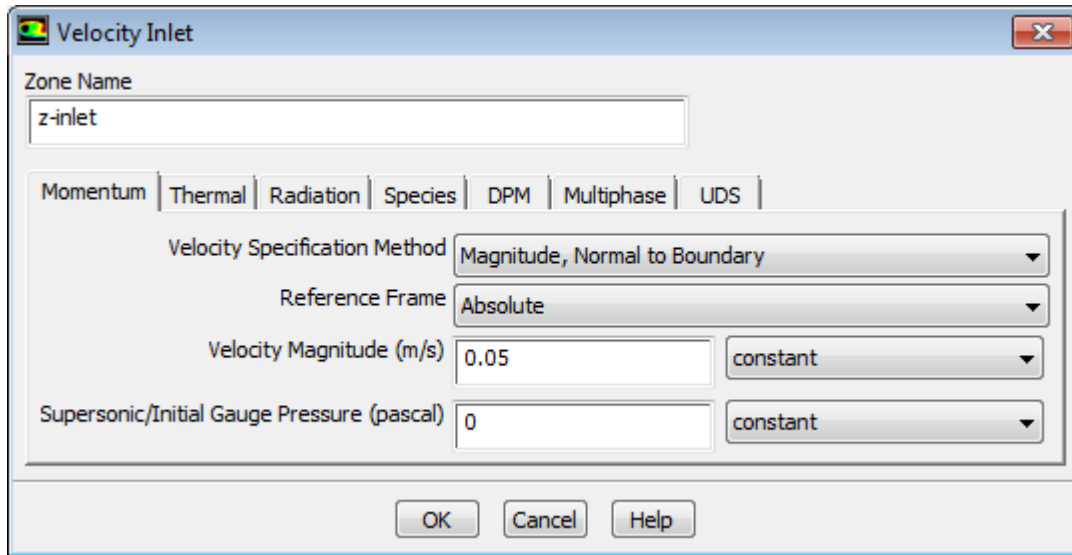


## Boundary Conditions

Here is where you apply your boundary conditions – you should see the list of “inlet-y”, “inlet-z” etc. Note that you can specify boundary conditions for the momentum equation, for the energy equation, etc.

Select “z-inlet”, and click “Edit...”. Enter velocity magnitude of “0.05” under the “Momentum” tab. Go to the “Thermal” tab and enter 10 for temperature.

Set the boundary conditions for “y-inlet” accordingly: “0.03” for velocity, and 90 for temperature. Accept default settings for other boundaries.



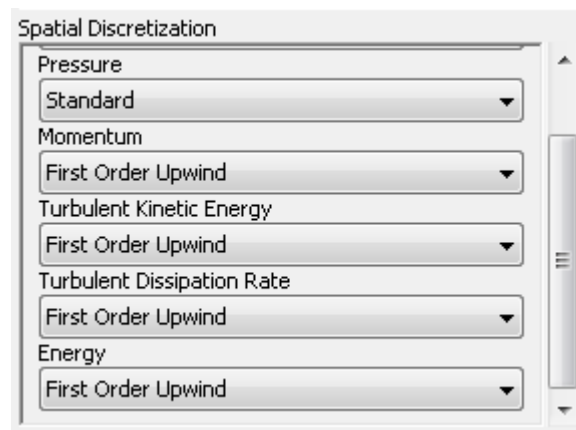
## Mesh Interfaces, Dynamic Mesh, Reference Values

We will revisit these as needed.

## Solution Methods

Here you can control the numerical methods used to solve your problem.

Choose “First Order Upwind” for Momentum, and Energy.



## Solution Controls

Here is where you control your iterative solution process. Again, Professor Vanka will give you the theory here.

## Monitors

Here is where you define your convergence criteria and can set up a monitor to watch the value of certain quantities, like the integrated composition over the domain, or the flux through a surface.

## Initialization

Here is where you define your initial conditions and initial guess of the solution for your iterative solution method.

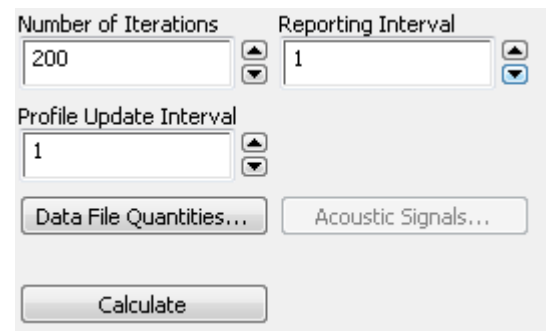
### Calculation Activities

Here is where you can set up autosaves, animations, and any other tasks that need to happen as the model is being solved.

### Run Calculation

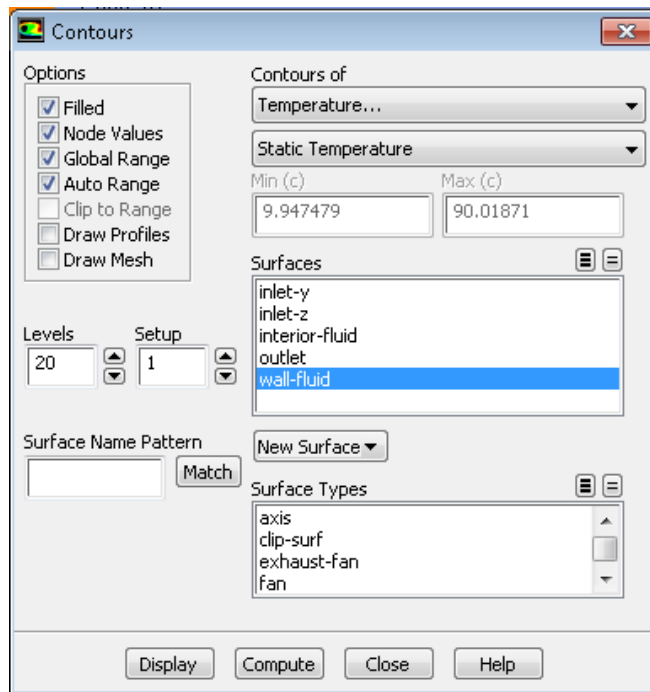
The “go button” and its related options.

Enter 200 in “Number of Iterations” and click “Calculate”.



The screenshot shows the 'Run Calculation' dialog box. It has two input fields at the top: 'Number of Iterations' with the value 200 and 'Reporting Interval' with the value 1. Below these is the 'Profile Update Interval' field with the value 1. There are two buttons: 'Data File Quantities...' and 'Acoustic Signals...'. At the bottom is a large 'Calculate' button.

### Results (and suboptions)



The screenshot shows the 'Contours' dialog box. On the left, under 'Options', there are checkboxes for 'Filled' (checked), 'Node Values' (checked), 'Global Range' (checked), 'Auto Range' (checked), 'Clip to Range' (unchecked), 'Draw Profiles' (unchecked), and 'Draw Mesh' (unchecked). Below these are 'Levels' (20) and 'Setup' (1). There is a 'Surface Name Pattern' field and a 'Match' button. On the right, under 'Contours of', there is a dropdown menu showing 'Temperature...'. Below that is another dropdown menu showing 'Static Temperature'. There are two input fields for 'Min (c)' (9.947479) and 'Max (c)' (90.01871). Below these are two lists: 'Surfaces' (inlet-y, inlet-z, interior-fluid, outlet, wall-fluid) and 'Surface Types' (axis, clip-surf, exhaust-fan, fan). At the bottom are buttons for 'Display', 'Compute', 'Close', and 'Help'.

These are several tools that you use to query your results, be they intermediate or final. This is where you can make contour plots, or extract data and make x-y plots. For example, let's make a temperature contour plot on the pipe wall.

Double click “Contours”. Check the “Filled” box. Choose a contour of Static Temperature on the surface of “wall-fluid”. Then click “Display”. The contour will appear in the graphics window. You can use the Toolbar to rotate, zoom, and move the contour.