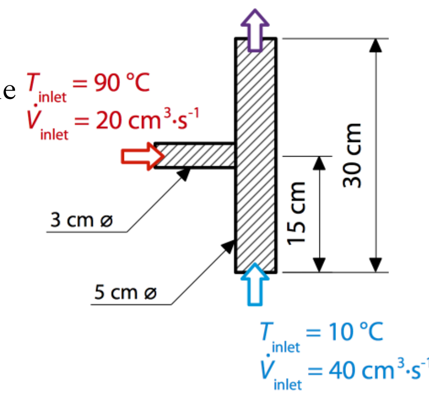
Mixing Tee

This exercise focuses on the basic Fluent workflow. We start by simulating a mixing tee *in media res*: the geometry and mesh are already provided, and we walk through the stages of setting up and executing the job, as well as plotting the results.

Problem Description

We will simulate the flow of water through a mixing tee at two different inlet temperatures. We wish to determine features of the resulting temperature distribution as well as the axial velocity profile along the centerline.



Cold water at 10 ˚C flows into a 5 cm ⌀, 30 cm-long section of pipe at a volumetric rate of 40 $cm^3\ s^{-1}$. Hot water at 90 ˚C enters the mixing tee halfway up (15 cm along the pipe) via a 3 cm ⌀, 15 cm-long section of pipe at a volumetric rate of 20 $cm^3\ s^{-1}$.

$\rho_{H2O} = 1\ g\ cm^{-3}$

$\mu_{H2O} = 0.01\ cP = 0.01\ g\ cm^{-1}\ s^{-1}$

1. Determine the Reynolds number of the flow at its fastest point (presumably the outlet) and thus whether turbulence will need to be considered in the solution (laminarity requires $Re_D \leq 2300$). (Assume incompressibility and that fluid properties remain constant at their given room-temperature values for this calculation.)

$$Re_D \equiv \frac{\rho v D}{\mu}$$

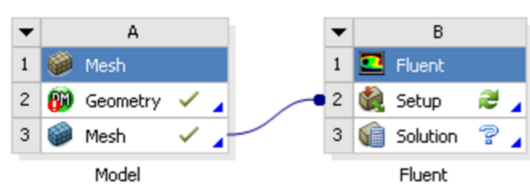
Calculate the Reynolds number of the outlet flow. _________ Is the flow laminar?

Getting Started

Download the ANSYS project `mixing_tee`.

Start ANSYS Workbench. Open the project `mixing_tee` from the *Toolbar*. (Ignore any warnings at this point.)

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 already complete.

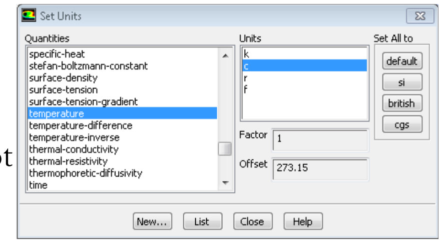


- Double-click *Setup* to launch Fluent and start setting up the problem. Select `double precision` and click OK.

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 require definition each time you set up a model in Fluent. If we look at the main menu, we see these exact tasks under the *Define* and *Solve* menus. The order of this menu suggests the order in which we 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 us to complete when defining the model.

Mesh



Fluent calculations are always in SI units. We can scale the coordinates of our mesh: for example, if we define your geometry in millimeters, scale down by a factor of 1000 to get it in meters.

Here we also select steady-state or transient flow and whether or not we 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 `c` (Centigrade).

Model

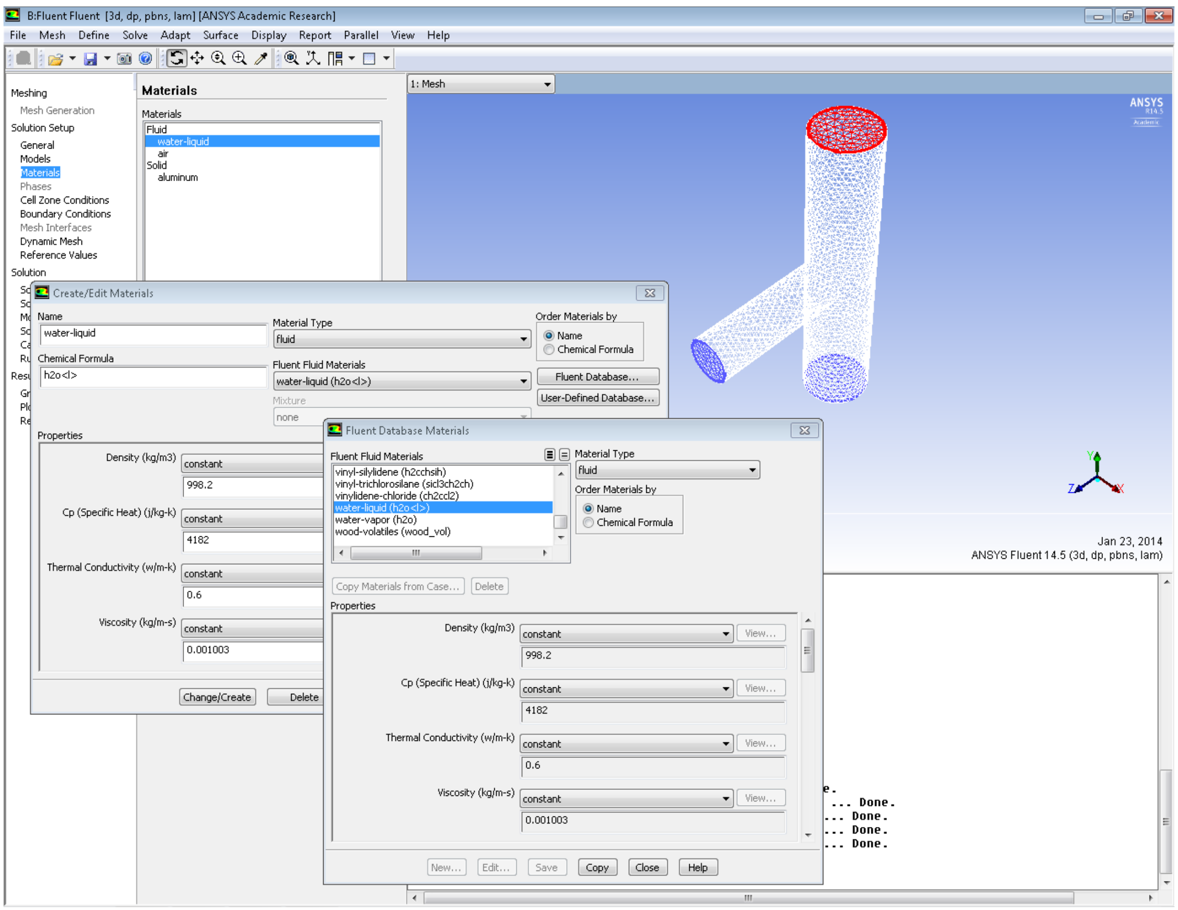
Here we specify the physics we want to include in our model.

- Double click on *Energy* and click the check box to enable the solution of the energy equation.
- Double click on *Viscous* and make sure that `laminar` is selected (no turbulence model, in accordance with our calculation of Re_D above).

Materials

Here is where we 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

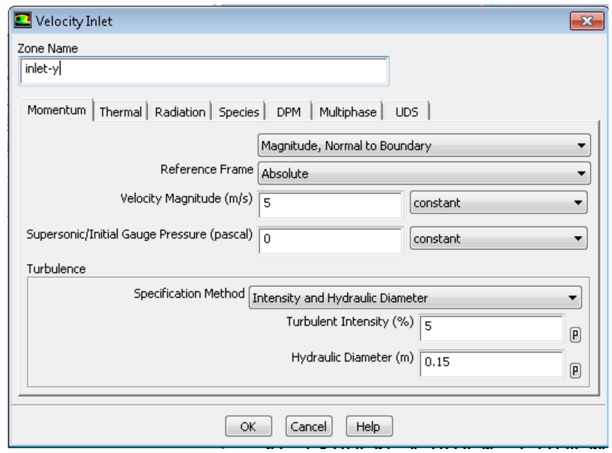
Multiphase flow will remain disabled, as we are working with a single phase simulation.

Cell zone conditions

Here is where we define cell zone properties—for instance, set up porous zones or include multiple fluids.

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

Boundary Conditions



Here we apply system boundary conditions to the list of surfaces in the mesh; in this case, `inlet-y`, `inlet-z`, etc. Note that we can specify boundary conditions for each equation in the simulation: momentum, energy, and so forth.

- Select `inlet-y`, and click *Edit...*.
- On the *Momentum* tab, enter a velocity magnitude of your inlet velocity (not volumetric flow rate!) in m/s: `0.02037`.
- On the *Thermal* tab, enter `10` for temperature.
- Set the boundary conditions for `inlet-z` accordingly and accept default settings for other boundaries:

  `0.02829` for velocity

  `90` for temperature
- Click *Edit* and examine the settings of `outlet-y` and `wall-solid` without making changes.

Mesh Interfaces, Dynamic Mesh, Reference Values

We will revisit these as needed.

Solution Methods

Here we can select appropriate numerical methods to solve your problem.

- Choose First Order Upwind for Momentum and Energy.

Solution Controls

Here we control the iterative solution process, as you may have learned in ME 412 or another CFD class.

Monitors

Here we define our 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. (What do these monitors check for?)

Initialization

Here is where we define our initial conditions and initial guess of the solution for our iterative solution method.

- Click *Initialize* to seed the mesh with an initial guess for the iterative solution process.

Calculation Activities

Here is where we set up autosave/checkpoint data, animation, and any other tasks that need to happen as the model is being solved. Right now we don't need to worry about these.

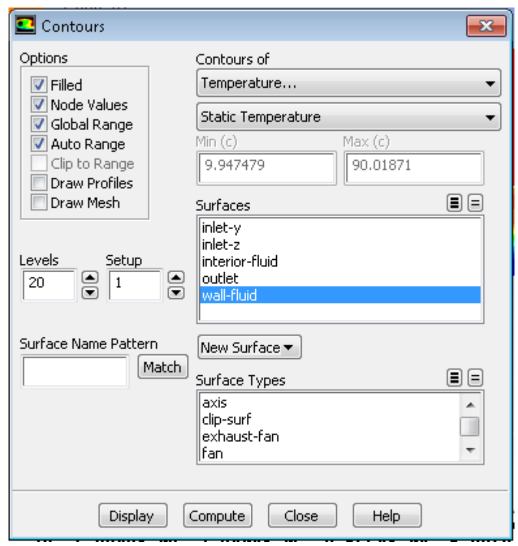
Run Calculation

- Enter `200` in Number of Iterations
- Click *Calculate*.

Fluent will display the monitored variables in a plot format by iteration. This tells you about the convergence behavior (and whether your simulation is converging at all!).

Postprocessing

There are several tools that we use to query our results, be they intermediate or final. This is where we make contour plots or extract data to make *x*–*y* plots.



For example, let's make a temperature contour plot on the pipe wall.

- Select *Graphics and Animation* at left.
- Double-click *Contours*.
- Check the *Filled* box.
- Choose a contour of `Temperature, Static Temperature` on the surface of wall-solid.
- Then click *Display*. The contour will appear in the graphics window. You can use the *Toolbar* to rotate, zoom, and move the contour.

You can also try *Vectors*, which produces a three-dimensional plot of `Velocity, Velocity Magnitude` with the default settings.

Let's additionally make some two-dimensional plots:

- Select *Plots* at left.
- Double-click `XY Plot`.

If we select an existing surface and plot a feature, we get an aggregate plot of values along that surface—that is, the value *at each cell*. This is a bit messy and illegible.

What we would like isn't an aggregate plot like this, but a centerline (℄) plot. In order to do this, we need to create a path along which we would like data.

- In the *Solution XY Plot* dialog box, click *New Surface* and select *Line/Rake Surface*.
- Create a line from `(0, -15, 0)` to `(0, 15, 0)` and name it `centerline`. Click *Create* and then *Close*.

- Set the plot direction to `(0, 1, 0)` (*i.e.*, the vertical y direction).
- Select `Temperature, Static Temperature` and `centerline`, then click *Plot*.

We performed the foregoing postprocessing in the Solution tool, rather than the standalone Results tool in the ANSYS Workbench. The Results tool is well-adapted for creating reports, and we will discuss it in a later class.

Conclusion

You have seen the typical workflow for a finite-volume or finite-element package, whether CFD or structural. ANSYS, ABAQUS, Fluent, COMSOL, and other packages all follow a similar workflow, stepping from definitions and meshing through solutions and postprocessing.

2. Submit your centerline temperature plot to `me498admin@mechse.illinois.edu`, subject "`flec01 lab`". This is generally how you will submit soft copies of lab materials or homework exercises.

   You may retain this hard copy.