Boussinesq Flow

Natural convection flows are encountered in various engineering applications. In this lab, you will solve a steady state natural convection problem in a 2D square cavity. The flow is induced by thermal differential between the left and right walls that are held at constant hot and cold temperatures, respectively. The top and bottom walls are insulated (zero heat flux). The fluid near the left wall ascends because it is hotter than the ambient temperature, while the fluid near the right wall descends because it is cooler, hence generating a clockwise vertex. The phenomenon is characterized by Rayleigh number, Ra = $g\rho\beta(T_H - T_C)L^3 / \mu\alpha$. In this formula,

L	Length of plate	g	Acceleration due to gravity	μ	Dynamic viscosity
T_H	Temperature of hot edge	β	Thermal expansion coefficient	α	Thermal diffusivity
T_C	Temperature of cold edge	ρ	Fluid density		

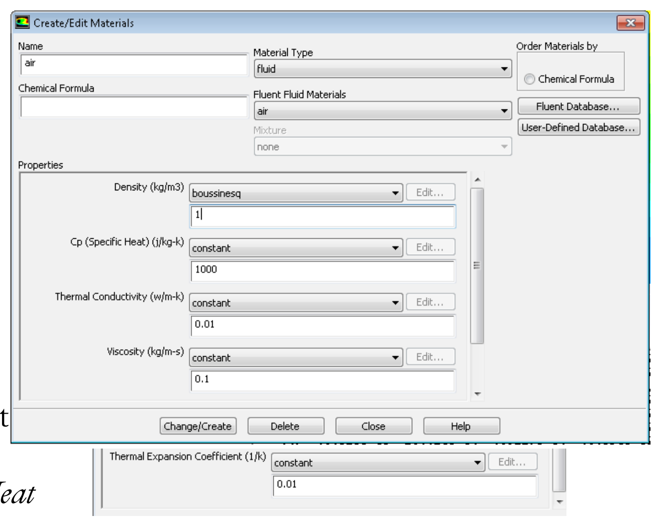
Geometry, Mesh

- Create a new 1 m × 1 m square geometry with a 0.01 m mesh. Create named selections for the wall cells, called `topwall` (top), `hotwall` (left), `bottomwall` (bottom), `coldwall` (right).

Setup

- You will solve a steady-state laminar natural convection model with Ra = 10,000.
  - Start Fluent with "double precision".
  - Change temperature unit to "degrees centigrade".

  Materials
  - Go to *Materials* menu and change the density option to "boussinesq" as at right. The density is the ρ_0 in the Boussinesq approximation.



  Boundary Conditions
  - Set the boundary conditions for the four walls:
    - The top and bottom walls have zero heat flux.
      In order to set these, click on *Edit* after selecting "bottomwall". Select the *Thermal* tab and select the "Heat Flux" option. Enter 0 for *Heat Flux* and *Heat Generation Rate*.
      Set the same boundary condition for the top wall.
    - The left and right walls have constant temperature at 1 and 0, respectively. (This is arbitrary and will scale.)
      In order to set these, click on *Edit* after selecting "leftwall". Select the *Thermal* tab and then select *Temperature*. Enter 1 for *Temperature*.
      Set the boundary condition of the right wall accordingly.

Boundary Conditions—Operating Conditions
  - Check the *Gravity* box and enter –1 in the *y*-direction. (You may also put –9.8 if preferred.)

- ◦ Enter 0.5 for the *Operating Temperature* that corresponds to T_0 in the Boussinesq approximation.
- ◦ You do not need to specify operating density when using the Boussinesq approximation. Leave the *Specified Operating Density* unchecked.
- Initialize and start the calculation. If your setup is correct, it should converge within 500 iterations.

Postprocessing

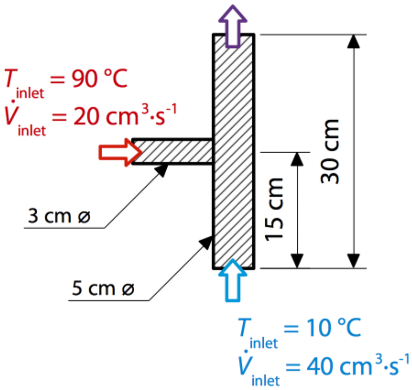
- Make a contour plot of temperature, of pressure, and of velocity.
- Plot the y-velocity along the horizontal centerline and x-velocity along the vertical centerline.
- Do the results make sense? Compare your results to a benchmark solution[de Vahl Davis 1983]. The materials are different, so the magnitude will not match. However, the location of velocity maxima should be the same (in his notation, u_{max} is the maximum x-velocity; z is the y-axis; w_{max} is the maximum y-velocity; and x is the x-axis).
- Is it a reasonable simulation? Why or why not?

	Ra			
	10^3	10^4	10^5	
$\lvert\psi_{mid}\rvert$	1·174	5·071	9·111	16·32
$\lvert\psi\rvert_{max}$	—	—	9·612	16·750
x, z	—	—	0·285, 0·601	0·151, 0·547
u_{max}	3·649	16·178	34·73	64·63
z	0·813	0·823	0·855	0·850
w_{max}	3·697	19·617	68·59	219·36
x	0·178	0·119	0·066	0·0379
$\overline{\mathrm{Nu}}$	1·118	2·243	4·519	8·800
$\mathrm{Nu}_{\frac{1}{2}}$	1·118	2·243	4·519	8·799
Nu_0	1·117	2·238	4·509	8·817
Nu_{max}	1·505	3·528	7·717	17·925
z	0·092	0·143	0·081	0·0378
Nu_{min}	0·692	0·586	0·729	0·989
z	1	1	1	1

Conclusion

- Submit your contour and centerline temperature plots to `me-498admin@illinois.edu` with subject "flec03 lab".

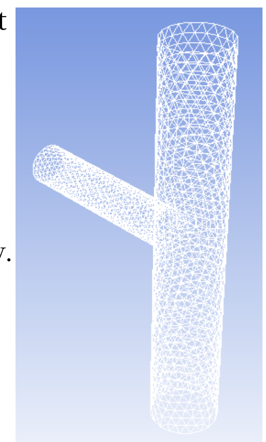
You may retain this hard copy.



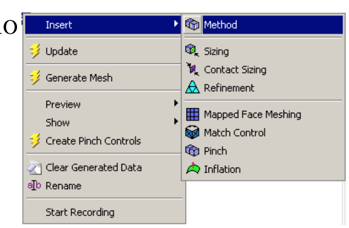
Mixing Tee Completion

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}.



- In the ANSYS Workbench, load the mixing tee geometry you created last time. Note that whereas we formerly had only question marks next to the steps in the project schematic, the *Geometry* entry now has a check mark and the *Mesh* entry has a pending symbol. Double-click on *Mesh* to enter the *ANSYS Meshing* tool.
  Meshing
- Your geometry should be displayed in an isometric three-dimensional view. You may adjust your unit system back to "cgs" by selecting the appropriate option under the *Unit* menu.
  Note that you can select various pieces of the geometry independently by clicking on each surface. This is a result of having your cursor in *Face* detection mode.
- We will use this to define inlets, outlets, and boundary conditions eventually, but for now we need to do only two things: create named regions (for BC definition later) and create a meshing strategy for each segment of the project.
- Select *Model→Mesh* in the *Outline* tree at left.
- In the *Details* pane, verify that the *Physics Preference* is "CFD" and the *Solver Preference* is "Fluent".
- Right-click on *Mesh* and select *Insert→Method* (make certain that no faces are selected or this won't appear).
  "Automatic Method" appears as an option.



- In the *Details* pane, click *Geometry* and then select the mixing tee object. Click *Apply*.
- Click either *Update* or *Generate Mesh*.
  A progress meter appears briefly while the mesh is generated.
- We now need to create named areas to refer to in creating boundary conditions.
  - Click the *Select Face* button.
  - Rotate the pipe up slightly and select the bottom *y*-inlet face. Right-click on it and select *Create Named Selection*. Name this face "inlet-y".
  - Repeat this process for "inlet-x" and "outlet-y".
- Return to the ANSYS Workbench by closing this window.
- The *Mesh* entry often does not automatically update. Select *Update Project* to force it to update. This takes a few moments. (Incidentally, save often!)

This returns you to the starting point of the mixing tee lab in the first lecture. But this time, we will solve the problem with varying temperature.

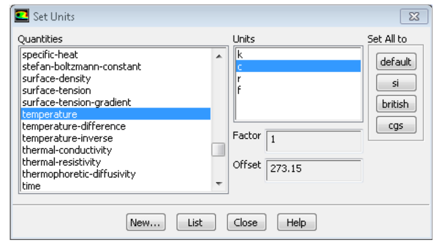
Setup

- Open *Setup*. Select "Double Precision".

A number of warnings and messages appear as the mesh is loaded and compared.

Mesh

- Click on *Units* and set the unit of temperature to "c".



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.

Materials

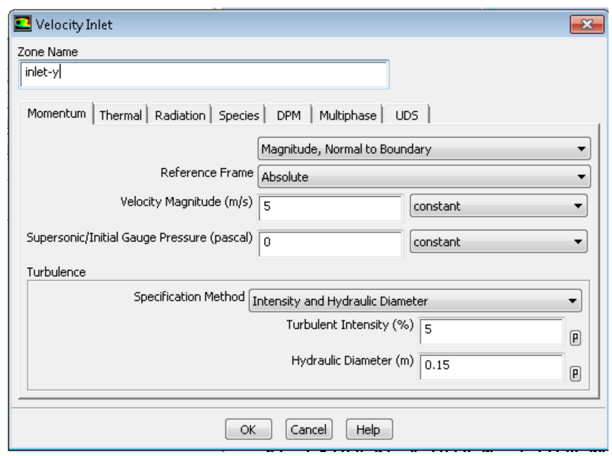
- Click *Create/Edit...*, and then *Fluent Database*.
- Find "water-liquid (h2o<l>)" from the list and click *Copy*.
- Close the windows and see "water-liquid" appear in the available fluid list.

Cell zone conditions

- 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.

Solution Methods

- Choose "First Order Upwind" for *Momentum* and *Energy*.

Initialization

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

Run Calculation

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

Fluent displays the monitored variables in a plot format by iteration.

Postprocessing

- Generate a filled contour plot of "Temperature, Static Temperature" on the surface of "wall-solid".
- Generate a vector plot of "Velocity, Velocity Magnitude".
- Generate an XY Plot of the centerline temperature:
    - 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*.

Conclusion

1. Submit your contour and centerline temperature plots to `me-498admin@illinois.edu` with subject "flec03 lab".

You may retain this hard copy.