## 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  $T_{inlet} = 90 \degree C$ along the centerline.  $V_{inlet} = 20 \text{ cm}$ 

Cold water at 10 °C flows into a 5 cm  $\emptyset$ , 30 cm-long section of pipe at a volumetric rate of 40 cm<sup>3</sup> s<sup>-1</sup>. Hot water at 90 °C enters the mixing tee halfway up (15 cm along the pipe) via a 3 cm  $\emptyset$ , 15 cm-long section of pipe at a volumetric rate of 20 cm<sup>3</sup> s<sup>-1</sup>.

 $\rho_{\text{H2O}} = 1 \text{ g cm}^{-3}$  $\mu_{\text{H2O}} = 0.01 \text{ cP} = 0.01 \text{ g cm}^{-1} \text{ s}^{-1}$ 

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  $\text{Re}_D \leq 2300$ ). (Assume incompressibility and that fluid properties remain constant at their given room-temperature values for this calculation.)

$$\operatorname{Re}_{D} \equiv \frac{\rho v D}{\mu}$$

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.





ME 498CF1  $\cdot$  Fluent Lecture #1

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 densitybased 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  $\text{Re}_D$  above).

specific-heat sefora-boltzmann-constant surface-density surface-tension surface-tension-gradient temporature-difference temporature-difference temporature-inverse thermai-sconductivity thermain-tensity, they thermain-tensity, thermain-tensity, thermain-tensity, thermain-tensity, the thermain-tensity, thermain-tensity, thermain-tensity, the thermain-tensity, the tension te	Factor 1 Offset 273.15	default si british cgs

#### 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.

😫 BiFluent Fluent (3d, dp, pbns, lam) (ANSYS Academic Research)												3		
File N	vlesh Define Solv	/e Ada	pt Surface Displa	y Repor	rt Parallel V	ïew Help								
■    ❷ ▼ 🖬 ▼ 📾 ❷    ⑤ 수 Q Q ∥    Q 洗 開 ▼ 🗆 ▼														
Meshing Materials				1: Mesh	•									
Mesh Generation Materials												ANSYS		
Solution	Solution Setup									- 68				
Gener	General water-liquid													
Mode	Models Solid													
Mater	Materials aluminum									1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1				
Cell Z	nases 'ell Zone Conditions													
Bound	Boundary Conditions									and the second				
Mesh	Mesh Interfaces								100	and set				
Dyna	mic Mesh rence Values													
Solution														
Sde									1					
Sc Sc	Create/Edit Materi	1015						23						
Mc N	lame		M	aterial Typ	e		Order N	laterials by	Alta in	10 4 C				
SC	Sd water-liquid fluid					<ul> <li>Na</li> </ul>	me	and the second version of						
RUC	hemical Formula			uses the state	Mahawiala			emical Formula			Con.			
Rest	h2o <l></l>			uent riulu ustar-liquid	(h2ods)		Flu	ent Database		- 20	2020			
Gr	Gr Water-liquid (h20<1>			1(120(12)		i User-F	efined Database		100					
Pic					7									
Re Eluent Database Materials									X					
Propercies						Material Type					¥.			
1 11	Density	(kg/m3) constant			Fluent Fluid Mat	(b2ccbeib)		fluid	•				1	
1 11			998.2		vinyl-trichloros	illane (sicl3ch2ch)	<u>^</u>	Order Materials by				74	<u>~~</u>	
1 11					vinylidene-chlo	oride (ch2ccl2)	_	News				-		
Cp (Specific Heat) (j/kg-H) constant 4182 Thermal Conductivity (w/m-H) constant 0.6		constant		water-vapor (	h2o)		Chemical Formu	la						
		4182			wood-volatiles	rood_vol)	*					Jan 23. 7	Jan 23, 2014	2014
				•	m		•				ANSYS Fluent 14.5 (3d, dp, pbns, lam)			
		(w/m-k) constant								ŀ				_
			0.6		Copy Material	s from Case Delete								Ê.
1 11					Properties									
1 11	Viscosity	(kg/m-s)	constant			Density (kg/m3)	constant		View	Â				
1 11			0.001003							=				
			Į.				998.2							
						Cp (Specific Heat) (j/kg-k)	constant		View					
			Change/Create	Delete			- 44.00							
							4182							
					The	ermal Conductivity (w/m-k)	constant		View					
							conseane							
							0.6							
						Viscosity (kg/m-s)	constant		- View	e	·			
							consedite				vone.			
							0.001003				Done.			E
					1						Done.			
						New Edit	Save	Copy Close	Help					
									,					-
						4							÷.	

#### 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.
- 💶 Velocity Inlet × Zone Name inlet-y Momentum Thermal Radiation Species DPM Multiphase UDS Magnitude, Normal to Boundary • Reference Frame Absolute • Velocity Magnitude (m/s) 5 constant • Supersonic/Initial Gauge Pressure (pascal) constant • Turbulence Specification Method Intensity and Hydraulic Diameter • Turbulent Intensity (%) 5 P Hydraulic Diameter (m) P OK Cancel Help
- 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 ( $\mathbf{L}$ ) 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.

 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.