2D Steady Convection - Physics Setup

Author: Benjamin Mullen, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation **Exercises** Comments

Physics Setup

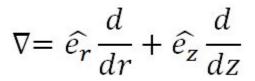
The geometry and the mesh have been set up to solve the problem using FLUENT but a few things need to be considered before we proceed.

Double Precision

Both single and double precision versions of FLUENT are available in workbench. Although single precision is sufficiently accurate in most cases, the disparate length scale in our model (long, thin pipe) may create large round-off error. Hence, we will use double precision to reduce this error.

Axisymmetric

When Axisymmetric is selected, FLUENT transforms the Cartesian coordinates to cylindrical polar coordinates. The gradient in the axis symmetric model now becomes:



Material Properties

The values entered in material properties will be applied to the constants in the governing equations.

Operating Condition

The absolute pressure is defined as the sum of the gauge pressure and the reference pressure:

 $p_{absolute} = p_{gauge} + p_{reference}$

In FLUENT, the reference pressure can be specified under operating condition. By default, the operating condition is 1 atm.

Open FLUENT

Make sure a check mark appears next to the mesh panel in workbench. Double click Setup



to open FLUENT.

Initial Settings

Before FLUENT launches, we will be prompted to set some options. In Options check the box next to Double Precision. .

ANSYS					
Dimension	Options				
② 2D	Double Precision				
O 3D	Use Job Scheduler				
Display Options	Processing Options				
📝 Display Mesh After Reading	Serial				
📝 Embed Graphics Windows	Parallel				
📝 Workbench Color Scheme					
Do not show this panel again					
Show More Options					

Once the options are set, click OK.

Problem Setup - General

Now, FLUENT should open. We will begin setting up some options for the solver. In the left hand window (in what I will call the *Outline* window), under *Pro blem Setup*, select *General*. The only option we need to change here will address the fact that pipe domain we created is axisymmetric. Under *2D Space*, click the radio box next to *Axisymmetric*.

Models

In the outline window, click *Models*. For viscous model, laminar is the default, so we don't need to change that. We will need to utilize the energy equation in order to solve for the temperature. Under *Models* highligh *Energy - Off* and click *Edit...*. Now, the *Energy* window will launch. Check the box next to *Ener gy Equation* and hit OK.

<u>File Mesh Define Solve</u>	e <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> e						
፤■┃≧ ▼ 🚽 ▼ 🞯 @ 🕼 🔂 🔁 € 🗶 🖉 🔍 🔚 ▼							
Meshing Mesh Generation	Models						
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Models Multiphase - Off Energy - On Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off						

Materials

In the Outline window, highlight Materials. In the Materials window, highlight Fluid, and click Create/Edit.... this will launch the Create/Edit Materials window; here we can specify the properties of the fluid. Set the Density to 1.2, the Specific Heat to 1000, the Thermal Conductivity to .02, and the Vis cosity to 1.8e-5.

Name air Chemical Formula		Material Type fluid •• FLUENT Fluid Materials ar ••				
					hoperbes	
Density (kg/m3)	constant		• Edt			
Cp (Specific Heat) (),Rg-k)	1.2 constant		• Edt			
	3000					
Thermal Conductivity (w/m-k)	constant 0.02		• Edt			
	constant		• Edt			
	1.8e-5					

Once finished, click Change/Create, then Close.

Boundary Conditions

Now we will specify the boundary conditions governing the problem. In the Outline window, highlight Boundary Conditions.

Operating Conditions

The default operating pressure in FLUENT is 1 atm, which is 101325 Pa. We can equate the operating pressure to the absolute pressure by setting the *gau ge pressure* to zero.

Centerline

Under Zone, highlight Centerline. Change the Type to axis. Confirm you are changing the selection, then leave the name as the default centerline.

Heated Wall

Under Zone, highlight heated_wall. The Type should have defaulted to wall. Next, click Edit.... Click the Thermal tab, and select the Heat Flux radio button. Change the Heat Flux (w/m2) to 37.5. Click OK.

Inlet

Under Zone, highlight *inlet*. The *Type* should have defaulted to *velocity-inlet*. Next, click *Edit...*. In the *Momentum* tab, change the *Velocity Specification Method* to *Components*, and specify the *Axial Velocity* to 0.1. Click *OK*

Isothermal Wall

Under Zone, highlight isothermal_wall. The Type should have defaulted to wall. Next, click Edit.... Click the Thermal tab, and select the Temperature radio button. Change the Temperature (k) to 300. Click OK.

Outlet

Under Zone, highlight outlet. The Type should have defaulted to pressure-outlet. Next, click Edit.... In the Momentum tab, ensure the Gauge Pressure is 0. Click OK.

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules