2D Steady Convection - Physics Setup

Author: Benjamin Mullen, Cornell University

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:

$$\nabla = \hat{e}_r \frac{d}{dr} + \hat{e}_z \frac{d}{dz}$$

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.

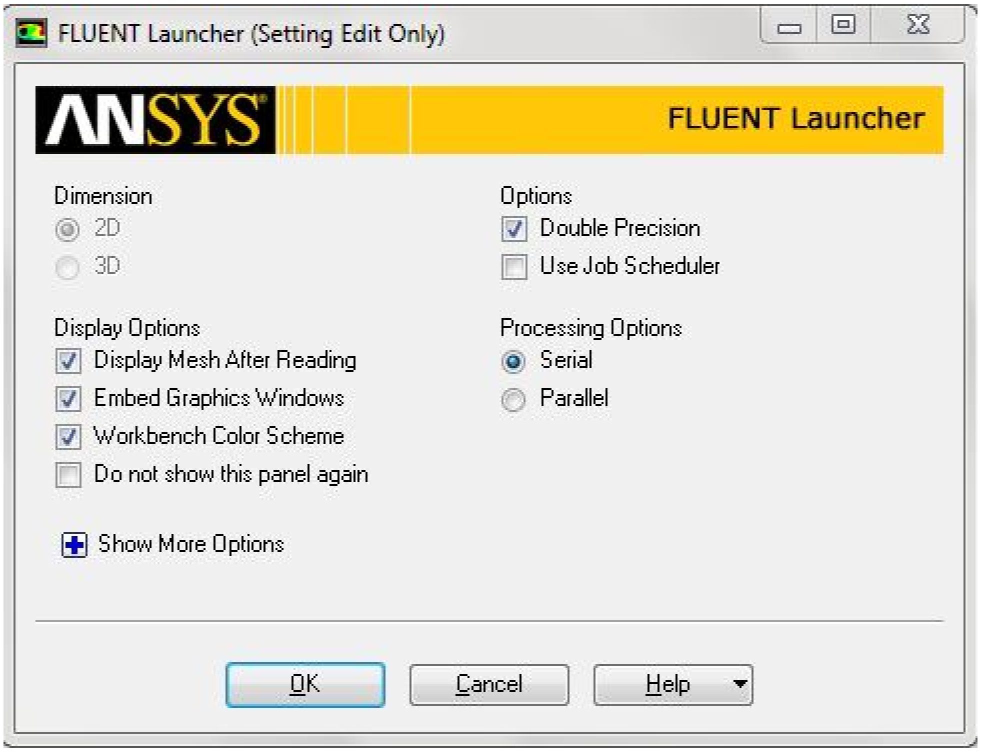
Open FLUENT

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***. .



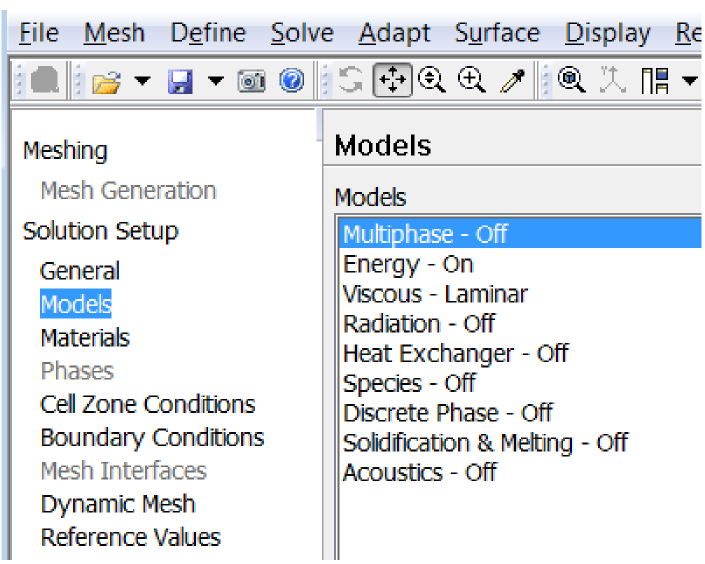
Once the options are set, click OK.

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 ***Problem 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***.

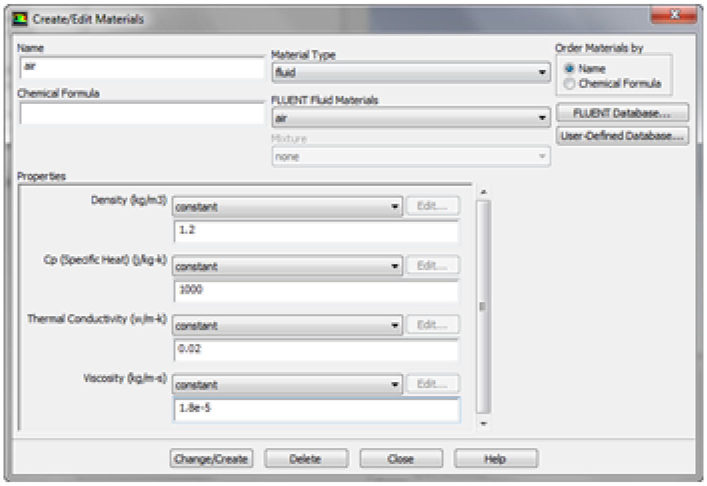
Models

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 ***Energy Equation*** and hit OK.



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 ***Viscosity*** to `1.8e-5`.



click here to enlarge

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 ***gauge 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