Forced Convection Laminar Flow

Modified by Robert P. Hesketh, Chemical Engineering, Rowan University Fall 2006

In this problem you will simulate forced convection in pipe flow. The fluid exits an unheated section of pipe at a temperature of 20°C (293 K) and flows into a section of pipe with a constant wall temperature of 40°C (313 K). The heated section of pipe has a length of 0.1 m and the radius of the pipe is 0.0254 m. The average velocity of the entering fluid is 0.02 m/s.

Modeling Using the Graphical User Interface

1. Open COMSOL.
2. Select **axial symmetry (2D)** from the **Space dimension** list.
3. Select the application mode **Chemical Engineering Module>Momentum balance> Non-Isothermal Flow (steady-state)**.
4. Click the **Multiphysics** button and click the **Add** button to add the application mode to the model.
5. Select the application mode **Chemical Engineering Module>Energy balance>Convection and Conduction.** Click **Add** and then **OK**.

OPTIONS AND SETTINGS

1. Enter the following variable names in the **Constants** dialog box under the **Options** menu. You will need to use the units of Kelvin since we have a correlation in Kelvin for the viscosity and heat capacity of water.

NAME	EXPRESSION
kc	0.6
Twall	40+273=313
Tin	20+273=293
uavg	0.02

GEOMETRY MODELING

1. Make pipe with length 0.1 m and radius 0.0254 m. You can do this by adding a rectangle.
2. Click the **Zoom Extents** button.

PHYSICS SETTINGS

Open the **Expressions** dialog box from the **Options>Expressions** menu, and enter the following variable names and expressions: (**The temperature in these expressions has units of Kelvin)**

NAME	EXPRESSION
eta1	0.11021-9.386e-4*T+2.684e-6*T^2-2.569e-9*T^3
cp	4190.86-0.977*T+0.019*T^2-5.6e-5*T^3
vin	uavg*2*(1-(r/0.0254)^2)
rho	823.9933+1.415411*T -0.0028125 *T^2

Boundary Conditions

1) Choose **1 Non-Isothermal Flow (ns)** from the **Multiphysics** menu.

a) In the **Boundary Settings** dialog box, select the inlet to the pipe to have a parabolic velocity profile. Since we are assuming that this flow originates from a pipe that is not heated to a section of the pipe that is heated, then the entering velocity will have a parabolic velocity profile. Select the **Inflow/Outflow velocity** from the **Boundary condition** list. Set the parabolic velocity profile by specifying the expression vin given in the expressions that you defined above. Another method that COMSOL uses to specify a condition at a boundary is to use the bézier curve parameter, s, which varies between 0 and 1 on a given boundary.
b) Select **Normal flow/Pressure** to obtain the fully developed flow boundary condition for the outlet. Set **p0** to 0
c) Specify the boundary at r=0 as a symmetry axis.
d) Specify no-slip at the wall.

2) Click **OK**.

Subdomain Settings

1. Enter material properties according to the following table in all the subdomains in the **Subdomain Settings** dialog box under the **Physics** menu:

SUBDOMAIN	1
ρ	rho
η	eta1

2. Switch to the **Init** tab and enter uavg for the **v(t_0)** initial value. We have not done this in the past, but this will help with the giving the solver an initial value to start the solution.
3. Click **OK**.

Boundary Conditions

1) Switch to the **2 Convection and Conduction (cc)** application mode.
2) Open the **Boundary Settings** dialog box from the **Physics** menu and specify conditions according to the following table.
   a) Inlet – Temperature of Tin
   b) Outlet – Convective flux
   c) Pipe wall – Twall

3) Click **OK**.

Subdomain Settings

1. Open the **Subdomain Settings** dialog box from the **Physics** menu, select subdomain 1 and specify values according to the following table. (note u and v are predefined variables in COMSOL)
2. Select the **Artificial Diffusion** check box and select the **Streamline**

SUBDOMAIN	1
k (isotropic)	kc
ρ	rho
C_p	cp
Q	0
u	u
v	v

Diffusion check box with default settings and click **OK**.

This adds a weak term contribution that adds explicit streamline diffusion to the dependent variable *T* only. This stabilizes the temperature field by damping oscillations produced by the discretization. For a more comprehensive explanation of streamline diffusion stabilization consult the *COMSOL User's Guide* .

3. Switch to the **Init** tab and enter Tin for the **T(t** $_0$**)** initial value.
4. Click **OK**.

MESH GENERATION

1. Select **Mesh Parameters** from the **Mesh** menu and select **Fine** in the **Predefined mesh sizes** list.
2. Click **OK**.
3. Click **Initialize Mesh** in the Main toolbar.
4. You may want to return to this menu to refine your mesh in the region in which the temperature is changing. Warning don't make it too fine or the computer will run out of memory.

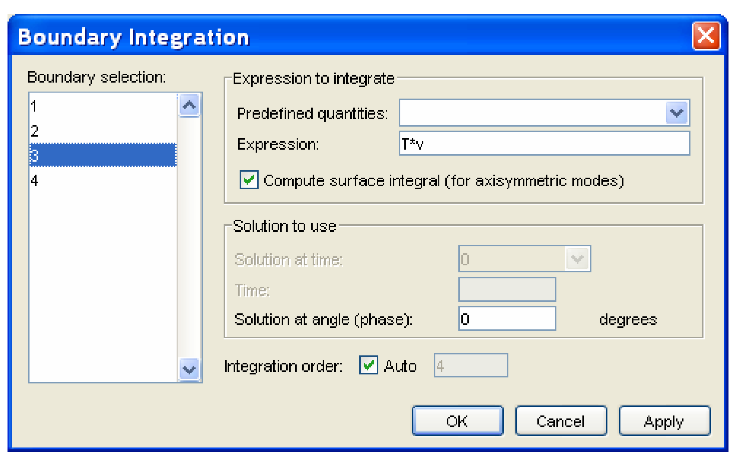
COMPUTING THE SOLUTION

Click the **Solve** button in the Main toolbar to solve the problem.

POSTPROCESSING AND VISUALIZATION

1) Click the **Plot Parameters button to open the Plot Parameters dialog box.**
2) On the **Surface** page, select **Temperature** from the **Predefined quantities** list, and click **OK** .
3) Make a second surface plot of velocity and add arrows to this plot showing the magnitude of the velocity.
4) Now check your inlet velocity and exit velocity by making a cross-section plot of the inlet and outlet
5) Make a second plot giving the temperature profile at the inlet, outlet and several other intermediate points along the length of the pipe.
6) To calculate the integral average temperature, $\overline{T} = \frac{\int Tuds}{\int uds}$, proceed as follows:

   a) In the **Postprocessing** menu, select **Boundary Integration** and then select exit boundary and type T*v (notice that v is the x-direction velocity) in the **Expression** edit field. Also select the compute Surface integral (for axisymmetric



modes). Click **OK**. The value will appear in the message log.

b) Repeat the procedure, but this time select **x-velocity (ns)** in the **Predefined quantities** list. Divide the first value with the second

7) Now run this model again with the wall temperature at 90°C. Determine outlet average temperature. Comment on how this improves the heating of the fluid.
8) If you reduced the channel to that of a micro-reactor would you see similar behavior? Change your volume to give a radius of 0.254 mm and length of 0.01 m and run the model using 90°C as the wall temperature. Calculate the outlet average temperature.
9) Now run the model again, but this time specify a boundary condition of a constant heat flux. This would be similar to a wall that was an electrical heater or even closely wrapped electrical heat tape. Assume that the constant wall flux was equivalent to a 800 W heating tape that can evenly cover the surface of the pipe: 800W/(2πR*L). For this problem use the "heat flux boundary condition and specify q_0. Do not use the convective flux boundary condition. This is used in the case where you were solving for the
10) Comment on how the COMSOL results compare with a plug flow model of this problem (you can solve these and obtain an analytical solution and also compare numbers if you have time). Compare the average outlet temperature values found from Comsol solution for each of the above cases. Comment on why or why not a plug flow model would be suitable for use in modeling these cases.

Constant Wall Temperature: $\dot{m}C_p \dfrac{dT}{dz} = 2\pi R h_{laminar}\left(T_{wall} - T\right)$

$$N_{Nu} = \frac{h_{laminar} D}{k} = 1.86\left(N_{\mathrm{Re}} N_{\mathrm{Pr}} \frac{D}{L}\right)^{1/3}\left(\frac{u_b}{u_w}\right)^{0.14}$$ Equation 4.5-4 in Geankoplis

Constant heat flux: $\dot{m}C_p \dfrac{dT}{dz} = 2\pi R q_{wall}$

Submit:

1. **Average Temperature Calculation**
2. Contour Plots (for constant wall Temperature (40 & 90°C), microreactor, and wall flux) Add arrows on top of plot to signify velocity.
   2.1. Temperature -
   2.2. velocity
3. **Cross Section Plots** (for constant wall Temperature (40 & 90°C), microreactor, and wall flux) **:**
   3.1. Temperature
   3.2. velocity
4. Comment on the results from 40 and 90°C. Comment on how heating the wall to a higher temperature improves the heating of the fluid.
5. Comments on reducing the channel to that of a micro-reactor. What are the advantages of this type of system?
6. Comment on how the COMSOL results compare with a plug flow model of this problem (Solve the above equations and obtain an analytical solution. Compare the outlet temperature obtained from the plug flow models with the average outlet temperatures obtained from the COMSOL simulations.