Lab 6: Laminar Pipe Flow with Convection

**Objective:**

The objective of this laboratory is to introduce you to ANSYS ICEM CFD and ANSYS FLUENT by using them to solve for velocity and temperature profiles in developing laminar pipe flow with a constant surface heat flux. We will include conduction through the pipe wall in our simulation. Concepts introduced in this tutorial will include modeling axisymmetric flow, solid/fluid boundaries, and comparison to analytical results for validation.

**Background:**

*Computer Software*

Fluent Inc. was a company founded in 1988 in Lebanon, NH, as a spinoff of Creare Inc., an engineering consulting firm. Fluent Inc. marketed a collection of computational fluid dynamics (CFD) programs including FLUENT, a general purpose CFD solver, and associated utility programs. Fluent Inc. was acquired by ANSYS in 2006, a company founded in 1970 that develops computer-aided engineering (CAE) simulation software. The Mechanical Engineering and Aerospace Engineering Departments at Cal Poly together purchase academic teaching and research licenses for a range of ANSYS CFD software products. The latest version of this software is installed on the computers in Bldg. 192, Rooms 134. Older versions may be available in other computer laboratories. The software packages that we will use for this class include:

**ANSYS ICEM CFD** - pre-processing program used to generate the geometry and mesh for our CFD simulations.

**ANSYS FLUENT** - CFD solver and post-processing program that uses the finite-volume method (or control-volume method), the simplest implementation of the finite-element method. It can handle both structured and unstructured grids. We will introduce some of the particular numerical schemes that this code uses in class during the remainder of the quarter.

Alternatively, we could use the **ANSYS Workbench** platform which is a framework for the complete simulation process. For example, for a typical CFD analysis a project schematic is created that includes access to programs for geometry creation (using **ANSYS Design Modeler**), meshing (using **ANSYS Meshing**), CFD simulation (using **ANSYS FLUENT**), and post-processing. You may use this software for your final project if desired.

Finally, ANSYS offers additional grid generation programs (such as TurboGrid and Tgrid), general-purpose CFD solvers (such as CFX and FIDAP, which uses the finite-element method), and application based CFD solvers (such as Icepak for electronics cooling and Airpak for heating, ventilation, and cooling).
Problem Statement

For this lab we will consider air flowing through a plastic (PVC) pipe with a uniform surface heat flux on the outer surface as shown in Figure 1. The pipe has an inner diameter $D = 0.020$ m, length $L = 0.40$ m, and thickness $t = 0.005$ m. The inlet flow has uniform (or constant over the cross-section) velocity $U = 0.292$ m/s and temperature $T_{\infty,i} = 290$ K. The fluid exits into the ambient atmosphere at a pressure of 1 atm. The Reynolds number based on pipe diameter (from the above data) is

$$Re_D = \frac{\rho U D}{\mu} = 400.$$  \hfill (1)

Figure 1. Schematic of flow of air through a PVC pipe.

where $\rho$ is density and $\mu$ is absolute viscosity. For this Reynolds number the flow in the pipe will be laminar. Recall that the transition to turbulent flow typically occurs above a Reynolds number of approximately 2300. The total rate of heat transfer to the fluid at the outer surface of the pipe $q = 10$ W. This corresponds to a surface heat flux at the outer boundary of the pipe of

$$q_{s,o} = \frac{q}{A_{s,o}} = \frac{q}{\pi (D + 2t)L} = 265.3 \text{ W/m}^2.$$  \hfill (2)

For this flow, hydrodynamic and thermal boundary layers will grow along the pipe wall starting at the inlet. Eventually they will grow to fill the pipe completely (provided that the pipe is long enough). When this happens, the flow becomes fully developed. For hydrodynamically fully developed flow there is no longer any variation in the velocity profile in the axial $x$-direction. One can obtain an analytical solution to the governing equations in the fully developed region.
where $R$ is the pipe radius. The length of the development region for laminar flow can be estimated using the following correlation (experimentally developed equation)

\[
\frac{X_{fd, h}}{D} = 0.05 \text{Re}_D = 20
\]  

(4)

For thermally fully-developed flow with a constant surface heat flux there is no longer any variation in the shape of the temperature profile in the axial $x$-direction. Again, one can obtain an analytical solution to the governing equations in the fully-developed region as follows

\[
\left[ \frac{T_s - T(r)}{T_s - T_m} \right]_{x \geq x_{fd, t}} = \frac{6}{11} \left[ 3 + \left( \frac{r}{R} \right)^4 - 4 \left( \frac{r}{R} \right)^2 \right], \quad \left( T_s - T_m \right)_{x = L} = \frac{11}{48 \pi} \frac{q}{k L}
\]  

(5)

where $k$ is the thermal conductivity, $T_s$ is the surface temperature on the inside of the pipe wall, and $T_m$ is the mean temperature based on a mass-weighted average. The length of the development region for laminar flow can be estimated using the following correlation

\[
\frac{X_{fd, l}}{D} = 0.05 \text{Re}_D \text{ Pr} = 15
\]  

(6)

where Pr is the Prandtl number.

Based on an overall energy balance for the pipe, the mean temperature distribution for a uniform surface heat flux in both the developing and fully-developed region can be calculated from the following (which is valid for both laminar and turbulent flow):

\[
T_m(x) = T_{m, i} + \left( \frac{q}{\dot{m} c_p} \right) \frac{x}{L}
\]  

(7)

where $c_p$ is the specific heat and $\dot{m} = \rho U \left( \pi R^2 \right)$ is the mass flow rate. Finally, the heat transfer coefficient for our pipe flow is defined as

\[
h(x) = \frac{q_s}{T_s(x) - T_m(x)} = \frac{q/A_{ij}}{T_s(x) - T_m(x)} = \frac{q}{\pi D L [T_s(x) - T_m(x)]}
\]  

(8)

which can be used to calculate the Nusselt number based on diameter as follows:

\[
Nu_D(x) = \frac{h(x) D}{k}
\]  

(9)
For fully developed-laminar flow, substitute Equation (5) into Equation (8) to get the constant Nusselt number value of 48/11. You should have seen these equations in an undergraduate Heat Transfer course. Please refer to Chapter 8 of Introduction to Heat Transfer, by Bergman, et al. for more information on internal flow with heat transfer.

**Laboratory:**

**ANSYS ICEM CFD**

To run ICEM CFD, click on the ICEM CFD icon on the desktop. Figure 2 shows the main components of the graphical user interface (GUI) window that will open. Next, select **Help > Users Manual** from the Main Menu bar. This should open the ICEM CFD User’s Manual in your web browser. Review the Introduction to ANSYS ICEM CFD section to get oriented with the overall meshing process and the (GUI) before continuing. For our labs we will simulate 2-D cases. Thus, from the Computing the Mesh section, we will only be using the Shell Meshing modules. The Tetra, Hexa, and Prism modules are for 3-D meshing.
Next, you will create the mesh for the axisymmetric pipe flow problem. The mesh will be a two-dimensional rectangular slice of the domain as shown in Figure 1. For an axisymmetric flow the properties are constant in the \( \theta \)-direction and are only a function of \( x \) and \( r \). In ICEM CFD and FLUENT the \( r \)-coordinate will be represented by the \( y \)-coordinate. Note that in FLUENT the axis of rotation must be the \( x \)-axis for axisymmetric simulations.

Here are some notes for following the steps:

- Text in \textit{italics} indicates one of the GUI components indicated in Figure 2
- Text in \textbf{bold} font indicates the specific name for a tab, window, menu item, etc.
- LMB, MMB, and RMB are used for left, middle, and right mouse button clicks, respectively
- DEZ is used for the \textit{Data Entry Zone} where some options will be left as their default values and will not be explicitly noted in these instructions
- \textit{DCT} is used for the \textit{Display Control Tree}

\textit{Step 1. Select working directory and create new project}

\textit{Main Menu} - Create a folder for your project. Do not use a name with spaces, including all the directories in the path. From \textit{File} pull down menu, select \textbf{Change Working Directory} using LMB. In \textit{Browse for Folder} dialog box select the folder you just created and click \textbf{OK}. Verify that the new working directory has been set in the \textit{Message Window}.

\textit{Main Menu} - From \textit{File} pull down menu, select \textbf{New Project} using LMB. In \textit{New Project} dialog box create a new project. Again, do not use a name with spaces. Verify that your new project has been created in the \textit{Message Window}.

\textit{Step 2. Create points for geometry}

\textit{Function Tab} - From \textit{Geometry} select \textbf{Create Point} \( \bullet \) using LMB.

\textit{DEZ} - For \textbf{Create Point} enter the following:
- deselect \textbf{Inherit Part} (NOTE, this is only needed for Windows OS), in \textbf{Part} text edit box click LMB and enter PNT (replacing GEOM),
- select \textbf{Explicit Coordinates} \( xiz \) using LMB,
- under \textbf{Explicit Locations} ensure \textbf{Create 1 point} is selected from pull down menu, in \textbf{X}, \textbf{Y}, and \textbf{Z} text edit boxes ensure 0 is entered in each for point at (0, 0, 0),
- click \textbf{Apply} using LMB and verify the \textit{Message Done: points pnt.00},
- in \textbf{X} text edit box click LMB and enter 400 for point at (400, 0, 0),
- click \textbf{Apply} using LMB and verify the \textit{Message Done: points pnt.01},
- in \textbf{Y} text edit box click LMB and enter 10 for point at (400, 10, 0),
- click \textbf{Apply} using LMB and verify the \textit{Message Done: points pnt.02},
- in \textbf{Y} text edit box click LMB and enter 15 for point at (400, 15, 0),
- click \textbf{Apply} using LMB and verify the \textit{Message Done: points pnt.03},
- in \textbf{X} text edit box click LMB and enter 0 for point at (0, 15, 0),
- click \textbf{Apply} using LMB and verify the \textit{Message Done: points pnt.04},
- in \textbf{Y} text edit box click LMB and enter 10 for point at (0, 10, 0),
click **Apply** using LMB and verify the **Message Done: points pnt.05**, and click **Dismiss** using LMB.

**Utilities** - Select **Fit Window** using LMB to verify that 6 points have been created.

**DCT** - Expand **Geometry** and **Parts** menus by using LMB to change + to - for each. Under **Model**\|**Geometry** use RMB to click on **Points** and select **Show Point Names** using LMB. Verify that the points have been named pts.00, pts.01, pts.02, etc.

**Step 3. Create curves (or straight lines) for geometry**

**Function Tab** - From **Geometry** select **Create/Modify Curve** using LMB.

**DEZ** - For **Create/Modify Curve** enter the following:
- ensure **Inherit Part** is NOT selected,
- in **Part** text edit box click LMB and enter AXIS (replacing PNT),
- select **From Points** using LMB,
- select **Select location(s)** using LMB (may not be needed if “Select locations with left button …” is already displayed at bottom of **Graphics Window**),
- select pnt.00 and pnt.01 using LMB and then click MMB to create axis,
- verify the **Message Done: curves crv.00**,

in **Part** text edit box click LMB and enter WALL_INNER,
- select pnt.05 and pnt.02 using LMB and then click MMB to create inner pipe surface,
- verify the **Message Done: curves crv.01**,

in **Part** text edit box click LMB and enter WALL_OUTER,
- select pnt.04 and pnt.03 using LMB and then click MMB to create outer pipe surface,
- verify the **Message Done: curves crv.02**,

in **Part** text edit box click LMB and enter INLET,
- select pnt.00 and pnt.05 using LMB and then click MMB to create air inlet,
- verify the **Message Done: curves crv.03**,

in **Part** text edit box click LMB and enter OUTLET,
- select pnt.01 and pnt.02 using LMB and then click MMB to create air outlet,
- verify the **Message Done: curves crv.04**,

in **Part** text edit box click LMB and enter WALL_SIDES,
- select pnt.05 and pnt.04 using LMB and then click MMB to create left pipe end,
- verify the **Message Done: curves crv.05**,

select pnt.02 and pnt.03 using LMB and then click MMB to create right pipe outlet,
- verify the **Message Done: curves crv.06**, and
- click **DISMISS** using LMB.
NOTE: Different part names are necessary to set different boundary conditions in FLUENT. Because the boundary condition is the same for the left and right end of the pipe (insulated) they are given the same name (WALL_SIDES).

DCT - Under Model\Geometry use RMB to click on Points and unselect Show Point Names using LMB. Under Model\Geometry use RMB to click on Curves and select Show Curve Names using LMB and verify that seven curves have been created with names crv.00, crv.01, crv.02, etc. Verify that the Points, Curves, and Parts can be made visible and invisible by selecting the box next to them using the LMB.

Step 4. Create surfaces for air flow and pipe wall

Function Tab - From Geometry select Create/Modify Surface using LMB.

DEZ - For Create/Modify Surface enter the following:
- Ensure Inherit Part is NOT selected,
- In Part text edit box click LMB and enter FLUID (replacing WALL_SIDES),
- Select From Curves using LMB,
- Under Surf Simple Method ensure From 2-4 Curves is selected from pull down menu,
- Select curve(s) using LMB (may not be needed if “Select curves with left button…” is already displayed at bottom of Graphics Window),
- Select crv.00, crv.01, crv.03, and crv.04 using LMB and then click MMB to create the surface for the air flow,
- In Part text edit box click LMB and enter SOLID,
- Select curve(s) using LMB,
- Select crv.01, crv.02, crv.05, and crv.06 using LMB and then click MMB to create the surface for the pipe,
- Click DISMISS using LMB.

NOTE: Different surface names are necessary to set different material zones in FLUENT.

DCT - Under Model\Geometry use RMB to click on Surfaces and select Show Surface Names using LMB and verify that two surfaces have been created with names srf.00 and srf.01. Make the surfaces invisible by selecting the box next to Surfaces using the LMB.

Step 5. Mesh Edges and Surfaces

Function Tab - From Mesh select Curve Mesh Setup using LMB.

DEZ - For Curve Mesh Setup enter the following:
- In Method ensure General is selected from pull down menu,
- Select Curve(s) using LMB,
- Select crv.00, crv.01, and crv.02 using LMB and then click MMB,
- In Number of Nodes enter 201,
click Apply using LMB,
select Select Curve(s) using LMB,
select crv.03 and crv.04 using the LMB and then click the MMB,
in Number of Nodes enter 21,
in Bunching Law select Geometric 2 from pull down menu using LMB,
in Spacing 2 enter 0.2,
in Ratio 2 enter 1.1,
check Curve direction and verify arrows pointing up are shown for crv.03 and crv.04,
click Apply using LMB,

select Select Curve(s) using LMB,
select crv.05 and crv.06 using LMB and then click MMB,
in Number of Nodes enter 5,
click Apply using LMB, and
click Dismiss using LMB.

Note: Using a bunching law for the flow inlet and outlet allows us to decrease the size of the elements near the wall where the velocity and temperature gradients are the highest. The BiGeometric bunching law is used to set the initial node spacing on each end of the curve (using Spacing 1 and 2) are the node spacing growth rate (Ratio 1 and 2). To set the initial spacing and growth rate for just one side use the Geometric 1 or Geometric 2 bunching law that correspond to the start or end of the arrow indicated by the curve direction, respectively. Note that we also set the total number of nodes so the geometry is over-specified and the meshing program will need to adjust Spacing 2 and Ratio 2 to produce the final mesh.

Function Tab - From Mesh select Compute Mesh using LMB.

DEZ - For Compute Mesh enter the following:

- in Compute select Surface Mesh Only using LMB,
- check Overwrite Surface Preset/Default Mesh Type using LMB,
- in Mesh Type select All Quad from pull down menu using LMB,
- ensure that Overwrite Surface Preset/Default Mesh Method is NOT checked,
- click Compute using the LMB, and
- click Dismiss using LMB.

Utilities - Select Box Zoom using LMB and use the LMB to select an area near the inlet end of the pipe to inspect the mesh.

NOTE: You should produce a structured mesh with nodes concentrated along the inner wall.
Step 6. Save File and Export Mesh

Function Tab - From **Output Mesh** click **Select Solver** using LMB.

**DEZ** – For **Output Solver** insure that **ANSYS Fluent** is selected from the pull-down menu then click Apply and OK.

**Function Tab** - From **Output Mesh** select **Boundary conditions** using LMB.

In **Part boundary conditions** dialog box:
- expand **Edges** menu by using LMB to change + to -,
- expand **AXIS** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **axis** using the LMB,
  - click **Okay** using LMB to close the **Selection** dialog box,
- expand **INLET** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **velocity-inlet** using the LMB,
  - click **Okay** using LMB to close the **Selection** dialog box,
- expand **OUTLET** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **pressure-outlet** using the LMB,
- click **Okay** using LMB to close the **Selection** dialog box,
- expand **WALL_INNER** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **wall** using the LMB,
  - click **Okay** using LMB to close the **Selection** dialog box,
- expand **WALL_OUTER** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **wall** using the LMB,
  - click **Okay** using LMB to close the **Selection** dialog box,
- expand **WALL_SIDES** menu by using LMB to change + to -,
- click **Create new** to open the **Selection** dialog box,
  - under Boundary Conditions select **wall** using the LMB,
  - click **Okay** using LMB to close the **Selection** dialog box,
- click **Accept** using LMB.

**Function Tab** - From **Output Mesh** select **Write Input** using LMB.

In **Save** dialog box click **Yes** using LMB to **Save current project first**.

In **Open** dialog box click **Open** to select unstructured mesh with current project name.
In **ANSYS Fluent** dialog box enter the following:
- in **Grid dimension** select **2D** using LMB,
- in **Scaling** ensure **No** is selected,
- in **Write binary file** ensure **No** is selected,
- in **Ignore couplings** ensure **No** is selected,
- in **Boco file** retain the default file name,
- in **Output file** change the file from **fluent** to your project name (you do not need to add a file extension because “.msh” will be added automatically),
- and click **Done** using LMB.

NOTE: Verify that the “.msh” file has been created in the **Message Window**.

*Main Menu* - From **File** pull down menu, select **Exit** using LMB.
To run FLUENT, double click on the FLUENT icon on the desktop which will open the FLUENT Launcher Dialog Box. Select the two-dimensional (under Dimension select 2D), double precision (under Options select Double Precision) version of the code. Using double instead of single precision will help insure minimal round-off error. Under Display Options ensure all three options are checked, under Processing Options ensure Serial is selected, and then click OK. This will open the Graphical User Interface (GUI) for FLUENT. To get oriented with the GUI, select Help/User's Guide Contents from the Menu Bar. This should open the ANSYS FLUENT User's Guide in your web browser. Click on the following links: Part II: Solution Mode, 1. Graphical User Interface (GUI) and then 1.1 GUI Components to get to a description of the seven main components of the GUI shown in Figure 3. Review all of Section 1.1 before continuing.

Next, you will simulate the flow and heat transfer. Here are some notes for following the steps:

- Text in *italics* indicates one of the GUI components indicated in Figure 3
- Text in **bold** font indicates the specific name for a tab, window, menu item, etc.
- All mouse clicks are with the left mouse button by default except for the Graphics Window

![Figure 3. Graphical User Interface (GUI) components for ANSYS FLUENT 19.1.](image-url)
PREPROCESSING

Step 1. Read in the mesh you created using ICEM CFD

Menu Bar - From File pull down menu select Read and Mesh.

In Select File dialog box locate and select your mesh file (with the .msh ending) and then click OK. Ignore the warning in the Console regarding the axis boundary condition. We will fix this in a later step by setting the solver to axisymmetric as suggested.

Step 2. Display and Save Mesh Picture

Ribbon – Select the Viewing tab.

In the Mouse section, the functions for each button are shown and can be changed as desired.

Note: Your mesh should be displayed in the Graphics Window. You can dolly, zoom, and probe your mesh using the mouse buttons. Try to dolly and zoom the mesh. Also, use the mouse-probe button to get information about zones and boundaries that are displayed in the Console.

In the Display section click Views… to open the Views dialog box.

In Views dialog box select front and click Apply to return to the original view. Notice that you can save additional views. Under Mirror Planes select axis and select Apply to show top and bottom half of the pipe. Click Close to close the dialog box.

Menu Bar - From File pull down menu select Save Picture (or use picture button on toolbar).

In Save Picture Dialog Box select desired Format, Coloring, and other Options. Click Preview to review your figure and click Save to export a picture of your mesh.

Step 3. Check Mesh and Surface Names

Tree - Under Setup double click on General using the LMB.

Task Page - Under Mesh select Check to verify that the mesh is valid.

Note: The information about the mesh will be printed out in the Console. The most common error identified by the mesh check is negative volumes in the mesh which requires you to regenerate or repair the mesh.

Task Page - Under Mesh select Display to open the Mesh Display dialog box.

Note: By default all of the items under Surfaces are selected. Select the Deselect All Shown button in the upper right-hand corner with an “X” to deselect all the items, under Surfaces select int_solid and click Display using the LMB. The mesh for just the pipe wall should be displayed.
in the Graphics Window. Select each item under Surfaces in order and then click Display to verify each name. To get more information about items on any Task Page click Help located at the bottom of the dialog box. Click Close when finished.

**Step 4. Scale Mesh**

**Tree** - Under Setup ensure General is still selected.

**Task Page** - Under Mesh select Scale to open the Scale Mesh dialog box.

In Scale Mesh dialog box enter the following:
- under Mesh Was Created In select mm from the pull-down menu,
- click Scale using the LMB,
- verify under Domain Extents mesh is 0.4 m in x-direction and 0.015 m in y-direction,
- and click Close to close dialog box.

**Step 5. Set Units**

**Tree** - Under Setup ensure General is still selected.

**Task Page** - On bottom right select Units to open the Set Units dialog box.

In Set Units dialog box enter the following:
- under Quantities select temperature,
- under Units verify that either k (for Kelvin) or c (for Celsius) is selected, and
- click Close to close dialog box.

Note: By default, FLUENT will use the standard base SI units of m-kg-s-K. You can use the Set Units dialog box to change all of the units to a different base or you can change the units for an individual variable such as temperature as shown above.

**Step 6. Select Solver Settings**

**Tree** - Under Setup ensure General is still selected.

**Task Page** - Under Solver section enter the following:
- under Type select Pressure-Based,
- under Time select Steady,
- under Velocity Formulation select Absolute,
- under 2D Space select Axisymmetric (which fixes initial mesh warning), and
- make sure Gravity is NOT checked.

**Step 7. Select Models (Energy Equation and Laminar Flow)**

**Tree** - Under Setup double click on Models with the LMB. Also, click on the “>” symbol to the left of General to expand the menu.
Task Page - Under Models select Energy - Off and then click Edit to open the Energy dialog box. Check the box next to Energy Equation and then select OK. Under Models verify that it now shows Energy - On in both the Tree and the Task Page.

Task Page - Under Models select Viscous - Laminar and then click Edit to open the Viscous Model dialog box. Ensure that the box next to Laminar is selected and then select OK. Under Models verify that it still shows Viscous - Laminar.

Note: The other choices besides laminar flow are inviscid (without viscosity) and turbulent flow (which requires you to choose one of the 8 turbulence models offered in FLUENT). This is also where viscous heating can be turned on.

Task Page - Under Models verify that all the other models such as Multiphase, Radiation, Heat Exchanger, Species, Discrete Phase, Solidification and Melting, and Acoustics are Off.

Step 8. Select Material Properties

Tree - Under Setup double click on Materials with the LMB. Also, click on the “>” symbol to the left of Materials to expand the menu.

Note: The properties for air and aluminum at 300 K are already available. Additional material properties can be accessed for many substances using the FLUENT Database. For this problem we will use the default properties for air and enter our own properties for the PVC pipe.

Task Page - Under Materials select Solid and then click Create/Edit to open Create/Edit Materials dialog box.

In Create/Edit Materials dialog box enter the following:
- under Name enter plastic,
- under Chemical Formula enter pvc,
- under Properties enter the following:
  - density, $\rho = 1,400 \text{ kg/m}^3$,
  - specific heat, $c = 1,050 \text{ J/kg\text{•}K}$, and
  - thermal conductivity, $k = 0.16 \text{ W/m\text{•}K}$,
- click Change/Create and click No on the Question dialog box, and
- click Close to close the dialog box.

Task Page - Under Materials and Solid verify plastic has been added.

Step 9. Set Cell Zone Conditions

Tree - Under Setup double click on Cell Zone Conditions using the LBM.

Task Page - Under Zone select fluid and click Operating Conditions to open Operating Conditions dialog box.
In **Operating Conditions** dialog box verify the following:

- **Operating Pressure** is 101,325 Pa (standard atmospheric pressure),
- **Gravity** is NOT checked (assuming negligible forces due to gravity), and
- click **OK** to close dialog box.

**Task Page** - Under **Zone** select **solid** and click **Edit** to open **Solid** dialog box.

In **Solid** dialog box enter the following:

- under **Material Name** select **plastic** from the pull-down menu, and
- click **OK** to close dialog box.

**Step 10. Set Boundary Conditions**

**Tree** - Under **Setup** double click on **Boundary Conditions** using the LBM.

**Task Page** - Under **Zone** select **inlet**, verify **Type** is **velocity-inlet** in the pull-down menu, and select **Edit** to open the **Velocity Inlet** pop-up menu.

Note: All the boundaries you defined in ICEM CFD should be indicated in the **Task Page** under **Zone**. In addition, FLUENT automatically created **int_fluid** (air zone), **int_solid** (plastic zone), and **wall_inner-shadow** when you read in your mesh. The **wall_inner-shadow** boundary allows you to define separate boundary conditions for the fluid and solid sides at the air/plastic interface. For our case for heat transfer, we want to treat this as an interior boundary and **NOT** impose a thermal boundary condition. To do this we set the thermal condition to **coupled**.

In **Velocity Inlet** pop-up menu enter the following:

- select the **Momentum** tab and under **Velocity Magnitude (m/s)** enter **0.292**.
- select the **Thermal** tab and under **Temperature (K)** enter **290**, and
- select **OK**.

Repeat similar steps for each boundary condition given in the following table:

<table>
<thead>
<tr>
<th>NAME</th>
<th>TYPE</th>
<th>CONDITIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>axis</td>
<td>axis</td>
<td>Zone Name: axis</td>
</tr>
<tr>
<td>inlet</td>
<td>velocity-inlet</td>
<td>Velocity Magnitude (m/s): 0.292</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Temperature (K): 290</td>
</tr>
<tr>
<td>outlet</td>
<td>pressure-outlet</td>
<td>Gage Pressure (Pascal): 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Backflow Total Temperature (K): 300</td>
</tr>
<tr>
<td>wall_inner</td>
<td>wall</td>
<td>Thermal Conditions: Coupled</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Material Name: plastic</td>
</tr>
<tr>
<td>wall_outer</td>
<td>wall</td>
<td>Thermal Conditions: Heat Flux (W/m²): 265.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Material Name: plastic</td>
</tr>
<tr>
<td>wall_sides</td>
<td>wall</td>
<td>Thermal Conditions: Heat Flux (W/m²): 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Material Name: plastic</td>
</tr>
</tbody>
</table>
PROCESSING

Step 11. Set Solution Methods

Tree - Under Solution double click on Solution Methods using the LBM.

Task Page - Under Pressure-Velocity Coupling, Scheme ensure that SIMPLE (a segregated solver) is selected from the pull-down menu.

Task Page - Under Spatial Discretization set following options using pull-down menus:
- Gradient to Least Squares Cell Based,
- Pressure to Second Order,
- Momentum to Second Order Upwind, and
- Energy to Second Order Upwind.

Note: We are using second order schemes to improve the accuracy of our solution.

Step 12. Set Relaxation Parameters

Tree - Under Solution double click on Solution Controls using the LBM.

Note: The default relaxation parameters for each equation (which we will use for this simulation) are given by the Under-Relaxation Factors. As we learned from our earlier labs for conduction, these relaxation factors control the rate of convergence. If the problem is having difficulty converging due to nonlinearities these values can be decreased. If the problem converges easily, these values can be increased to speed up the rate of convergence.

Step 13. Set Convergence Criteria and Monitors

Tree - Under Solution double click on Monitors using the LBM. Also, click on the “>” symbol to the left of Monitors to expand the menu. Double click Residual using the LMB to open the Residual Monitors dialog box.

In Residual Monitors dialog box complete the following:
- under Options make sure Print to Console and Plot are selected,
- under Equations set the Absolute Criteria for all four equations to $10^{-6}$, and
- click OK to close the dialog box.

Note: The residual is the error in the current iteration obtained when the approximate solution is substituted back into the original governing equation and then scaled using the average value for the flow field. The residuals may not decrease by this much for every simulation for a variety of reasons. Iteration can be halted when the residuals reach this level or cease to decrease.

Step 14. Initialize Solution

Tree - Under Solution double click on Solution Initialization using the LBM.
**Task Page** - Under **Initialization Methods** select **Standard Initialization**. Under **Compute From** select **inlet** from pull-down menu and click **Initialize**.

Note: This is a better initial guess than the defaults values which will reduce the number of iterations until convergence.

**Step 15. Check Case Setup**

**Tree** - Under **Solution** double click on **Run Calculation** using the LBM.

**Task Page** - Click **Check Case** to determine if there are any significant problems with your model setup. An **Information dialog box** should open stating that there are **No recommendations to make at this time**. Click **OK** to close the dialog box.

**Step 16. Save Case and Data**

**Menu Bar** - From **File** pull down menu select **Write** and **Case & Data** to open the **Select File dialog box**.

In **Select File** dialog box enter **Case/Data File** name and click **OK** to save the solution setup (.cas file) and your initial guess (.dat file) and close the dialog box.

**Step 17. Run Calculation and Save Case and Data**

**Tree** - Under **Solution** ensure **Run Calculation** is still selected.

**Task Page** - under **Number of Iterations** enter **300** and then click **Calculate**. The solution should converge after about 200 iterations.

**Menu Bar** - From **File** pull down menu select **Write** and **Data** to open the **Select File dialog box**.

In **Select File** dialog box ensure **Data File** name is same as for Step 16 and click **OK**.

In **Question** dialog box click **OK** to overwrite the initial guess data file.

**POST PROCESSING**

**Step 18. Check Global Conservation of Mass and Energy**

**Tree** - Click on the “>” symbol to the left of **Results** to expand the menu. Under **Results** double click on **Reports** using the LBM.

**Task Page** - Under **Reports** select **Fluxes** and click **Set Up** to open **Flux Reports dialog box**.
In **Flux Reports** dialog box enter the following:
- under **Options** ensure **Mass Flow Rate** is selected,
- under **Boundaries** select **inlet** and **outlet**, and 
  click **Compute**.

Note: Verify that the net mass flow rate is less than 0.1% of that for the inlet to ensure mass is conserved globally.

In **Flux Reports** dialog box enter the following:
- under **Options** select **Total Heat Transfer Rate**,
- under **Boundaries** select **inlet**, **outlet**, **wall_outer**, and **wall_sides**, and
  click **Compute**.

Note: Verify that the net heat transfer rate is less than 0.1% of that for the top wall to ensure energy is conserved globally.

In **Flux Reports** dialog box click **Close** to close the dialog box.

**Step 19. Make Contour Plot of Temperature**

*Tree* - Under **Results** double click on **Graphics and Animations** using the LBM.

*Task Page* - Under **Graphics** select **Contours** and click **Set Up** to open **Contours dialog box**.

In **Contours** dialog box enter the following:
- under **Contours of** select **Temperature** from pull-down menu,
- under **Options** select **Filled**, unselect **Auto Range**, and unselect **Clip to Range**, 
- under **Min (k)** enter 290,
- under **Max (k)** enter 460,
  click **Colormap Options** to open the **Colormap dialog box**,
  - under **Type** select **float** from the pull-down menu,
  - under **Precision** enter 2,
  - under **Colormap Size** enter 17,
  click **Apply** and **Close** to close the dialog box.
  click **Display** to see contour plot in **Graphics Window**, and
  click **Close** to close the dialog box.

Note: In the *Tree* clicking on the “>” symbol to the left **Contours** to expand the menu. Your current contour plot should be named “contour-1”. You can save additional contour plots with different names as needed.

*Task Page* - Click **Views** to return to the **Views dialog box** used in Step 2.

Note: You can again choose from the standard or saved views or change the **Mirror Planes** setting. Select **Close** to close this **dialog box**.
Step 20. Make Velocity Vector Plots

Tree - Under **Results** ensure **Graphics and Animations** is selected.

Task Page - Under **Graphics** select **Vectors** and click **Set Up** to open **Vectors dialog box**.

In **Vectors** dialog box enter the following:
- under **Vectors of** ensure **Velocity** is selected from the pull-down menu,
- under **Color by** select **Temperature** from the pull-down menu,
- under **Options** unselect **Auto Range** and unselect **Clip to Range**,
- under **Min (k)** enter 290,
- under **Max (k)** enter 460, and
- click **Display** to see the plot in the **Graphics Window**.

Note: To see velocity vectors at specific axial locations you will next define lines in the domain.

In **Vectors** dialog box under **New Surface** pull-down menu select **Line/Rake** to open **Line/Rake Surface dialog box**.

In **Line/Rake Surface** dialog box enter the following:
- under **End Points** enter the following:
  - for x0 (m) enter 0.04,
  - for x1 (m) enter 0.04,
  - for y0 (m) enter 0.00, and
  - for y1 (m) enter 0.01,
- under **New Surface Name** enter **line-02**, and
- click **Create** to create a line across the air flow at \( x/D = 2 \).

Note: Repeat this procedure to define lines at \( x/D = 4, 6, 8, 10, 12, 14, 16, \) and 18 while changing the names to **line-02, line-04, line-06, line-08, line-10, line-12, line-14, line-16, and line-18**, respectively. When you are done click **Close** to close this dialog box and to return to the **Vectors Dialog Box**. You can verify that the lines have been created correctly by displaying them using the method shown in Step 3.

In **Vectors** dialog box enter the following:
- under **Surfaces** select **inlet, outlet, line-02, line-04, etc.**,
- under **Scale** enter 4,
- under **Skip** enter 1, and
- click **Display** to see the plot in the **Graphics Window**, and
- click **Close** to close the dialog box.

Note: You should be able to see the velocity and temperature distributions developing from uniform at the inlet to fully-developed profiles (parabolic for velocity and quartic for temperature) by the exit.
Step 21. Make XY Plots

Tree - Under Results double click on Plots using the LBM.

Task Page - Under Plots select XY Plot and click Set Up to open Solution XY Plot dialog box.

In Solution XY Plot dialog box enter the following:
under Plot Direction for X enter 0 and for Y enter 1,
under Y Axis Function select Velocity from the pull-down menu,
under Surfaces select inlet, outlet, line-02, line-04, etc., and
click Plot to display velocity profiles in the Graphics Window,
under Options select Write to File
click Write to open the Select File dialog box,
under XY File enter a name with the “.xy” extension and
click OK to save the file and close the dialog box,
click Close to close the dialog box.

Use the same Solution XY Plot dialog box to make an xy-plot of temperature profiles and write the data to a file for the same locations. To do so, under Y Axis Function, instead of Velocity select Temperature from the pull-down menu.

Use the same Solution XY Plot dialog box to make an xy-plot of wall surface temperature and write the data to a file. To do so, under Plot Direction under X enter 1 and under Y enter 0. Also, under Surfaces select only WALL_INNER.

Step 22. Calculate Mean Temperature Along Pipe

Tree - Under Results double click on Reports using the LBM.

Task Page - Under Reports select Surface Integrals and click Set Up to open Surface Integrals dialog box.

In Surface Integrals dialog box enter the following:
under Report Type select Mass-Weighted Average from pull-down menu,
under Field Variable select Temperature,
under Surfaces select inlet, outlet, line-02, line-04, etc.,
click Compute to display mean temperature at each axial location in Console,
select Write to open the Select File dialog box,
under Surface Report File enter a name with the “.srp” or “.txt” extension and
click OK to save the file and close the dialog box,
click Close to close the dialog box.
Step 23. Check and Print Out Simulation Report

Ribbon – From Solving tab in the Run Calculation section click the Input Summary… button.

In Input Summary dialog box enter the following:
- under Report Options select Models,
- select Print to display the model settings in the Console.
- verify that model settings are correct,
- select all of the Report Options,
- click Save to open the Select File Dialog Box,
  - enter a file name under Report Summary File with the “.sum” extension and
  - click OK to save the file,
- click Close to close the dialog box.

Note: You can also select each of the Report Options individually and Print in sequence to verify all your model settings are correct.
Assignment:

Submit the following as a single pdf document:

1. Plot of the mesh.

2. Summary report file for the FLUENT simulation.

3. Contour plot of temperature.

4. Vector plot with vectors displayed at $x/D = 0, 2, 4, 6, 8, 10, 12, 14, 16, 18, \text{ and } 20$.

5. Export the velocity profile data into Excel or MATLAB and make a plot of $u/U$ versus $r/R$ at $x/D = 0, 2, 4, 6, 8, 10, 12, 14, 16, 18, \text{ and } 20$. Also, add a calculated profile for hydro-dynamically fully-developed flow. Make sure to use line colors and symbols that can be easily seen (the defaults in Excel are terrible).

6. Export the temperature profile data into Excel or MATLAB and make a plot of $(T - T_e)$ versus $r/R$ at $x/D = 0, 2, 4, 6, 8, 10, 12, 14, 16, 18, \text{ and } 20$. Also, add a calculated profile for thermally fully-developed flow.

7. Export the surface temperature data into Excel or MATLAB and make a plot of surface temperature versus axial location, $x$. On the same plot add the mean temperature from (1) the results of your simulation (FLUENT Step 22) and (2) calculating values using the overall energy balance relationship. You should expect these two values for the mean temperature to be very close indicating that energy is being conserved at each axial location.

8. Make a plot in Excel or MATLAB of $Nu_D$ versus $x$. Include a line indicating the fully-developed value for $Nu_D$ for laminar flow.

Answer the following questions:

1. Calculate maximum possible Mach and Eckert numbers for this case. Explain if it is reasonable to model this flow as incompressible with negligible viscous dissipation.

2. Briefly discuss the steps that were taken to reduce each type of error (discretization, round-off, and iteration) and insure good accuracy. Refer to the figures and describe how the solution has been validated. Give an estimate of the uncertainty in the results based on the data.

3. Briefly discuss where the flow is hydro-dynamically fully-developed. Refer to the figures to support your discussion. Compare your results to the entry-length prediction given by the correlation in the Background section.

4. Briefly discuss where the flow is thermally fully-developed. Refer to the figures to support your discussion. Compare your results to the entry-length prediction given by the correlation in the Background section.