Forced Convection of cold water over submarine

Problem Specification

In our problem, the submarine is located in the underwater of the sea where the water flow velocity is 1 m/s and the water temperature is 240 K. Assume that the surface of the submarine is a 90° arc with the radius of 1 m. The temperature inside the submarine is kept at 300 K. The material of the submarine is steel. Analyze the heat loss of the submarine with and without considering the effect of wall thickness of the submarine.

Analyze:
The properties of the water at 240 K could be summarized as:
\[ \mu = 1.78 \times 10^{-3} \text{kg/(m \cdot s)} \]
\[ k = 0.58 \text{W/(m \cdot K)} \]
\[ C_p = 4187 \text{J/kg \cdot K} \]
\[ \rho = 980 \text{kg/m}^3 \]

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]
\[ \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = F_x - \frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \]
\[ \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = F_y - \frac{\partial P}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \]
\[ \frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{\lambda}{\rho c_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial w^2} \right) \]

\[ \text{Re}_L = \frac{\rho u d}{\mu} = \frac{980 \times 1 \times 14.14}{1.78 \times 10^{-3}} = 7.78 \times 10^5 \]
A turbulence model shall be used in the simulation.

**Create Geometry in GAMBIT**

**Start GAMBIT & Select Solver**

Specify that the mesh to be created is for use with FLUENT 6:

Main Menu > Solver > FLUENT 5/6

**Create Vertices**

We will treat this problem as a 2-dimensional problem. Let's begin by creating the vertices that define our flow region.

Operation Toolpad > Geometry Command Button ➔ Vertex Command Button ➔ Create Vertex

Note that the *Create Vertex* button has already been selected by default. After you select a button under a sub-pad, it becomes the default when you go to a different sub-pad and then come back to the sub-pad.

Input: Global x=7.07, y=0.0. apply to create Vertex 1: (0, 7.07)

Repeat this process to create other vertices:
Vertex 2: (0, 7.07)
Vertex 3: (14.14, 7.07)
Vertex 4: (0, 17.07)
Vertex 5: (14.14, 17.07)

Operation Toolpad > Global Control > Fit to Window Button

This fits the vertices we have created to the size of the Graphics Window.

Create Edges
An edge is created by selecting two vertices and creating a line between them.
Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge, then select create arc.

Click the up arrow button next to the vertices box in the Create real circular arc window.

Select Vertex 1 for center, vertex 2 and vertex 3 for two end points. Then, click on apply. Click Close.

Then select Create Straight Edge window to create straight edges.
To form a face out of the area enclosed by the four lines, we need to select the four edges that enclose this area. This is done in much the same way as when we selected the vertices.

Click the up arrow button next to the vertices box in the Create Face From Wireframe window.

Then push the All right arrow button to bring these vertices into the Picked column.

Click Close. Then click Apply in the Create Face From Wireframe window to create the face. The edges and vertices will become blue, indicating that they now form a face.
Step 2: Mesh Geometry in GAMBIT

We'll now create a mesh on the rectangular face with 100 divisions in the vertical direction and 30 divisions in the horizontal direction. We'll first mesh the four edges and then the face. The desired grid spacing is specified through the edge mesh.

Mesh Edges

Operation Toolpad > Mesh Command Button \(\text{[Edges]}\) > Edge Command Button \(\text{[Edges]}\) > Mesh Edges \(\text{[Mesh Edges]}\)

Mesh Strategy

In creating this mesh, it is desirable to have more cells near the plate (Edge 1) because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field.

Click the up arrow button \(\text{↑}\) next to the Edges box in the Mesh Edges window. Select edge Edge.2.

Then push the right arrow button \(\text{→}\) to bring this vertex into the Picked column. Notice that the arrow on the selected edge should be pointing upwards. An upwards pointing arrow indicates the direction of closely spaced nodes to widely spaced nodes. Remember, we will need more closely spaced nodes near the boundary layer in order to resolve it accurately.

The proper arrow direction is necessary to ensure a proper mesh. Select Edge.4 in the Mesh Edges window. The arrow on this edge is pointing downwards, which needs to be changed. Shift + Middle-click on the selected edge to change the direction of the arrow to upward.

Under Type, select Successive Ratio, if it is not already selected. Set Ratio to 1.08. Under Spacing, select Interval Count. Set Interval Count to 100 and then click Apply.
Select Edge 1 and Edge 3 in the Mesh Edges Window. The direction of the arrows on these edges is irrelevant because the divisions will be the same length. Leave the Successive Ratio set to 1 and set the Interval Count to 30. Click Apply.

Mesh Face

Operation Toolpad > Mesh Command Button ➔ Face Command Button ➔ Mesh Faces

Shift left-click on the face or use the up arrow next to Faces to select the face. Click Apply.
Specify Boundary Types in GAMBIT

Create Boundary Types
We'll next set the boundary types in GAMBIT. The left edge is the inflow of the flow field, the right edge the outflow, the top edge the open top of the flow field, and the bottom edge the plate.

Operation Toolpad > Zones Command Button ➔ Specify Boundary Types Command Button ➔ This will bring up the Specify Boundary Types window on the Operation Panel. We will first specify that the left edge is the inflow. Under Entity:, pick Edges so that GAMBIT knows we want to pick an edge (face is default).

Now select the left edge by Shift-clicking on it. The selected edge should appear in the yellow box next to the Edges box as well as the Label/Type list under the Edges box. Next to Name:, enter inflow. For Type:, select VELOCITY_INLET. You may have to move the Specify Boundary Types box up in order to see the bottom of the list and select VELOCITY_INLET.

Click Apply. You should see the new entry appear under Name/Type box near the top of the window.
Repeat this process for the other three edges according to the following table:

<table>
<thead>
<tr>
<th>Edge Position</th>
<th>Name</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Left</td>
<td>inflow</td>
<td>VELOCITY_INLET</td>
</tr>
<tr>
<td>Right</td>
<td>outflow</td>
<td>PRESSURE_OUTLET</td>
</tr>
<tr>
<td>Top</td>
<td>top</td>
<td>SYMMETRY</td>
</tr>
<tr>
<td>Bottom</td>
<td>plate</td>
<td>WALL</td>
</tr>
</tbody>
</table>

You should have the following edges in the Name/Type list when finished:

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>inflow</td>
<td>VELOCITY_INLET</td>
</tr>
<tr>
<td>out</td>
<td>PRESSURE_OUTLET</td>
</tr>
<tr>
<td>sym</td>
<td>SYMMETRY</td>
</tr>
<tr>
<td>plate</td>
<td>WALL</td>
</tr>
</tbody>
</table>

Save and Export
Main Menu > File > Save
Main Menu > File > Export > Mesh...
Type in submarine.msh for the File Name:. Select Export 2d Mesh because this is a 2 dimensional mesh. Click Accept.
It is important to check that submarine.msh has been created in your working directory. GAMBIT may periodically fail to write the .msh file. If this should happen, simply try writing the .msh file to another directory and then coping it into your working directory.

Set Up Problem in FLUENT
Launch Fluent
Start > Programs > Fluent Inc > FLUENT 6
Select the 2ddp version and click Run.
The "2ddp" option is used to select the 2-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.
Import Grid
Main Menu > File > Read > Case...
Navigate to the working directory and select the submarine.msh file. This is the mesh file that was created using the preprocessor GAMBIT in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:

Check and Display Grid
First, we check the grid to make sure that there are no errors.
Main Menu > Grid > Check
Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:
Main Menu > Grid > Info > Size
The following statistics should appear:
Display the grid:
Main Menu > Display > Grid...
Make sure all 5 items under *Surfaces* are selected.

Then click Display. The graphics window opens and the grid is displayed in it. Your grid should look like this:

Define Solver Properties
Main Menu > Define > Models > Solver
We'll use the defaults of 2D space, segregated solver, implicit formulation, steady flow and absolute velocity formulation. Click OK.
We are interested in solving the temperature distribution, so we need to solve the energy equation. Select the Energy Equation and click OK to exit the menu.

Under \textit{Model}, select the k-epsilon turbulence model. We will use the Realizable model in the k-epsilon Model box. The Realizable k-epsilon model produces more accurate results for boundary layer flows than the Standard k-epsilon model.

Click OK.
Define Material Properties

Main Menu > Define > Materials...

Change the fluid materials to water from the database.
Select water from the drop list of fluid materials. Then click on copy. Change the density to 980, thermal capacity to 4187, thermal conductivity to 0.58, viscosity to 0.00178. Click on Change/create.
**Define Operating Conditions**

Main Menu > Define > Operating Conditions...

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the Operating Pressure.

Click Cancel to leave the default value in place.

![Operating Conditions Window](image)

**Define Boundary Conditions**

We'll now set the value of the velocity at the inflow and pressure at the outflow.

Main Menu > Define > Boundary Conditions...

We note here that the four types of boundaries we defined are specified as zones on the left side of the **Boundary Conditions Window**. There are also 2 zones default-interior fluid, used to define the interior of the flow field. We will not need to change any setting for these 2 zones.

Move down the list and select inflow under Zone. Note that FLUENT indicates that the Type of this boundary is velocity-inlet. Recall that the boundary type for the inflow was set in GAMBIT. If necessary, we can change the boundary type set previously in **GAMBIT** in this menu by selecting a different type from the list on the right. Click Set....

Enter 1 for Velocity Magnitude. This sets the velocity of the fluid entering at the left boundary to a uniform velocity profile of 1m/s. Set Temperature to 240K. Change Turbulence Specification Method to Intensity and Viscosity Ratio. Set Turbulence Intensity to 1 and Turbulent Viscosity Ratio to 1. Click OK.
Choose outflow under Zone. The Type of this boundary is pressure-outlet. Click Set.... The default value of the Gauge Pressure is 0. The (absolute) pressure at the outflow is 1 atm. Since the operating pressure is set to 1 atm, the outflow gauge pressure = outflow absolute pressure - operating pressure = 0. Because we do not expect any backflow, we do not need to set any backflow conditions. Click Cancel to leave the defaults in place.

Click on submarine under Zones and make sure Type is set as wall. Click Set.... Because we have a heated isothermal surface, we need to set the temperature. On the Thermal tab, select Temperature under Thermal Conditions. Change Temperature to 300. The material selected is inconsequential.
because the surface has zero thickness in our model, thus the material properties of the surface do not affect the heat transfer properties of the surface. Click OK.

The last boundary condition to set is for the top of the flow field. Click on top under Zones and make sure Type is set as pressure-outlet. Click Set... to see that there is no need to change anything for this boundary. Click OK. And we need to make sure the fluid in the computational domain is the material that we set before. Click on the fluid, and set. From the drop list of the material name, find the water-liquid and select it. You could make sure it is the material you just set by click on edit. Then click on OK.

Click Close to close the Boundary Conditions menu.
Solve the problem
We'll use a second-order discretization scheme.
Main Menu > Solve > Controls > Solution...
Change Density, Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate, and Energy all to Second Order Upwind. Leave Pressure and Pressure-Velocity Coupling set to the default methods (Standard and SIMPLE, respectively). The other Pressure and Pressure-Velocity Coupling methods are useful for flows with particular characteristics not present in our problem.

Click OK.

Set Initial Guess
Initialize the flow field to the values at the inflow:
Main Menu > Solve > Initialize > Initialize...
In the Solution Initialization window that comes up, choose inflow under Compute From. The X Velocity for all cells will automatically be set to 1 m/s, the Y Velocity to 0 m/s and the Gauge Pressure to 0 Pa. These values have been taken from the inflow boundary condition.
Click Init. This completes the initialization. Then click Close.

Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We will iterate until the residual for each equation falls below $1e^{-6}$.

Main Menu > Solve > Monitors > Residual...

Under Options, select Print and Plot. This will print the residuals in the main window and plot the residuals in the graphics window as they are calculated.
Click OK.
This completes the problem specification. Save your work:
Main Menu > File > Write > Case...
Type in submarine.cas for Case File. Click OK. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Iterate Until Convergence
Start the calculation by running 10,000 iterations. The solution will converge before 10,000 iterations are performed, which will stop the iteration process.
Main Menu > Solve > Iterate...
In the Iterate Window, change the Number of Iterations to 10000. Click Iterate.

The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated.

The residuals fall below the specified convergence criterion in approximately 143 iterations.
Save the solution to a data file:

Main Menu > File > Write > Data...

Enter submarine.dat for Data File and click OK. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

**Analyze Results**

**Velocity at end of the submarine**

Main Menu > Plot > XY Plot...

Under *Options*, unselect Position on X Axis and select Position on Y Axis. Under *Plot Direction*, enter 0 in the X box and 1 in the Y box. This tells FLUENT to plot a vertical rather than horizontal profile.

Under X Axis Function, pick Velocity... and then in the box under that, pick X Velocity. Finally, select outflow under Surfaces since we are plotting the velocity profile at the outflow. De-select plate under Surfaces.
Click on Axes... in the Solution XY Plot window. Select X in the Axis box. In the Options box select Major Rules to turn on the grid lines in the plot. Click Apply. Then select the Y in the Axis box, select Major Rules again, and turn off Auto Range. In the Range box enter 0.1 for the Maximum so that we may view the velocity profile in the boundary layer region more closely. Click Apply and Close.

Uncheck Write to File. Click Plot.
Contours of Velocity Magnitude (m/s)
Heat Flux

Effect of wall thickness of submarine

Go back to boundary condition definition

Set the wall thickness of submarine with 0.3m and the material of submarine is aluminum.
Initialize the problem again and make the iteration.
Calculate heat flux again.

\[
\frac{546590.9}{4019025.5} \times 100\% = 13.6\%
\]