OVERVIEW

A simple hot water tank, illustrated in Figures 1 through 3 below, consists of a main cylindrical tank and two small side pipes for the inlet and outlet. All solid surfaces of the system are thermally insulated, except that the temperature at the bottom of the main cylinder is externally maintained at a constant 70°C. The temperature of the water entering the inlet is set to 25°C. As cool water flows through the tank, it is heated up by the hot plate at the bottom. Thus, at the steady state we expect the temperature of the outflow to be higher than 25°C.

The goals of this project are to analyze:

1. How the steady-state temperature of the outflow is affected by the vertical positions of the inlet and outlet
2. How the steady-state temperature of the outflow is affected by the mass flow rate at the inlet
3. How the choice of “turbulence” vs. “laminar” model affects the outcome of the simulation

The properties of water are set to the following constant values:

Density: $\rho = 1000$ kg/m^3
Viscosity: $\mu = 8 \times 10^{-4}$ Pa-s
Conductivity: $k = 0.677$ W/m-K
Specific heat: $C_p = 4216$ J/kg-K

The “turbulence k-epsilon model” is used for all tasks except Task 4. “Energy equation” is turned on for all tasks.

For all tasks, the following geometric parameters are used (see Figures 2 and 3):

H = 1.2 m
D = 0.6 m
d = 0.04 m
L = 0.1 m

Inlet velocity is set to u = 0.05 m/s except for Task 3. The velocity and temperature at the inlet are uniform for all cases.

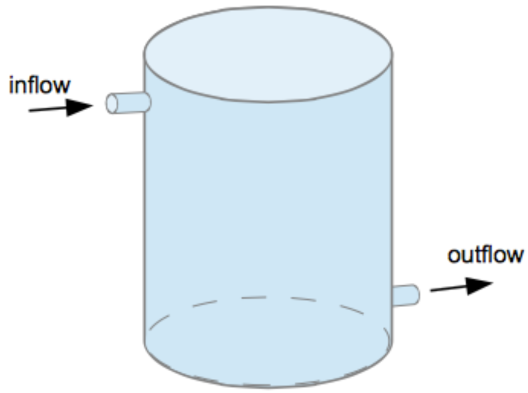


Figure 1: The water tank system which consists of a main cylinder and two circular side pipes for the inlet and outlet.

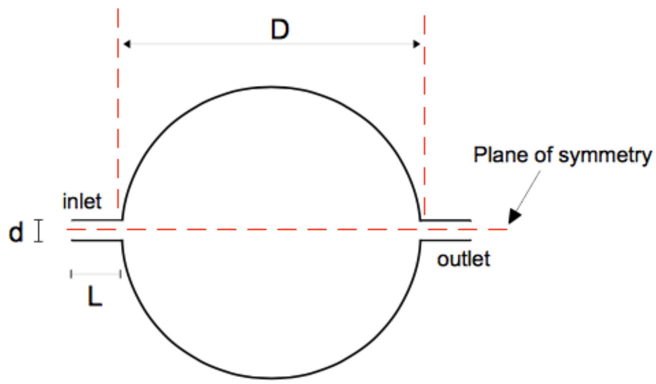


Figure 2: Top view of the water tank system. Key parameters: D is the diameter of the main cylinder; d is the diameter of both side pipes; L is the length of both side pipes.

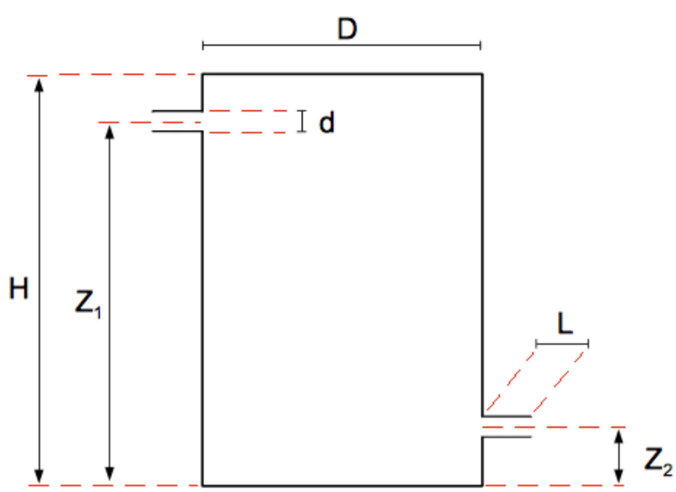


Figure 3: The vertical cross section of the water tank system along its plane of symmetry. Key parameters: H and D are the height and diameter of the main cylinder; Z_1 and Z_2 are the heights of the centers of the side pipes for the inlet and outlet, respectively; L is the length of both side pipes; d is the diameter of both side pipes.

TASK 1

Problem Statement:

Consider the nine combinations of the vertical positions of the side pipes with Z_1 = (0.2m, 0.6m, 1.0m) and Z_2 = (0.2m, 0.6m, 1.0m). Run Ansys-Fluent for those 9 cases to obtain the temperatures at the outlet. Summarize the results in a table.

Because the temperature and velocity at the outlet are generally not uniform, an averaging of temperature over the surface of the outlet is needed to fill the table. The most meaningful definition of the averaged temperature in this case is

$$T_{out} = \frac{\iint v_n T \, dA}{\iint v_n \, dA} \quad (1)$$

Procedure:

This task was more focused on relative results (i.e. which combination of inlet and outlet position gave the highest/lowest average outlet temperature) rather than determining the actual precise solution. Hence, a first-order solver was implemented and no mesh adaptation was used.

In all scenarios, a minimum of 600 iterations were calculated. In this project, calculations were considered to have converged when the continuity residual dropped permanently below 1 x 10^{-5}. Sometimes this required up to 900 iterations, but in most cases, 600 was sufficient to achieve convergence. An example of the residual plots is shown in Figure 4 below.

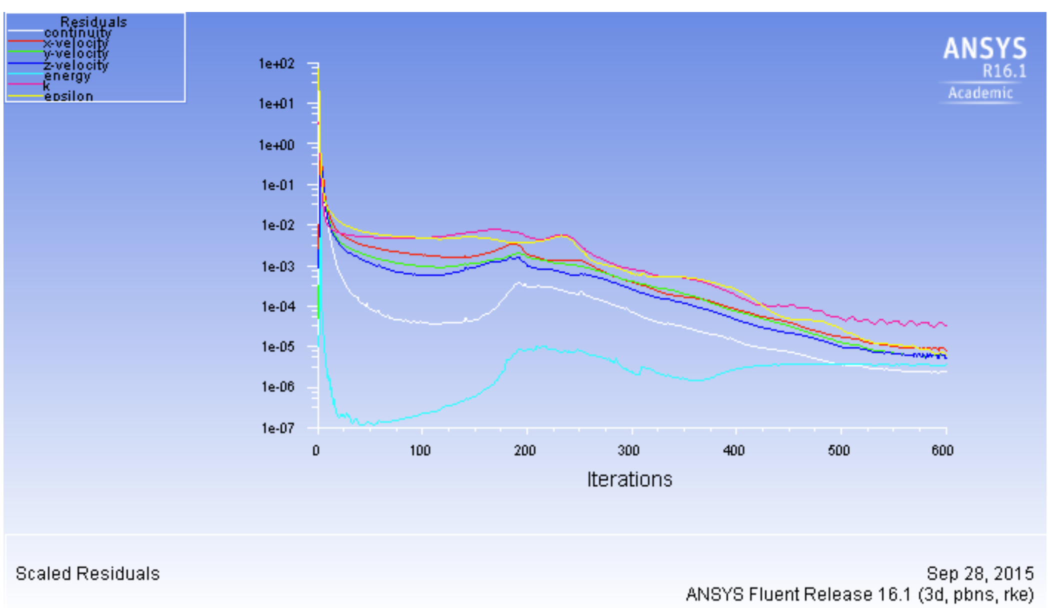


Figure 4: Plot of scaled residuals. Continuity is shown in white. Note that all plots began to flatten as the calculations converged.

To further confirm convergence, the net mass flux through the tank was calculated and found to be effectively zero, as shown in Figure 5 below. To find this data, "Fluxes" was chosen from the "Reports" dropdown in Fluent.

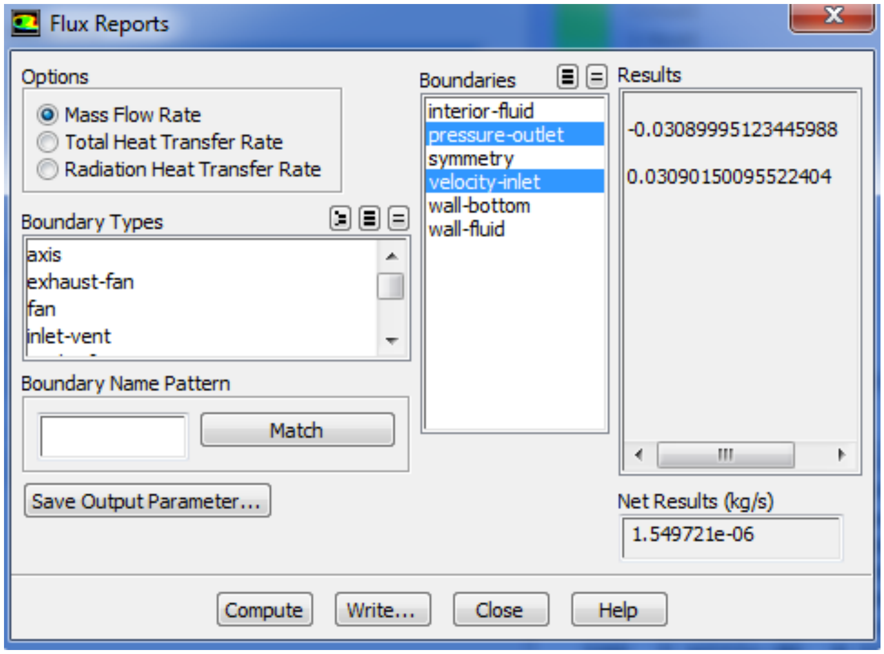


Figure 5: Mass flux report. Flux through inlet is positive, while flux through outlet is negative. Note that the net result is effectively zero.

To calculate the average outlet temperature through the method shown in Equation (1) above, the total heat transfer rate through the outlet was divided by the specific heat times the mass flux through the outlet. This works because the density and specific heat of the fluid were set to constant values; hence, they cancel out as shown below.

$$T_{out} = \frac{total\ heat\ transfer\ rate}{C_p\ (mass\ flux)} = \frac{\iint \rho\ C_p v_n T\ dA}{C_p \iint \rho\ v_n\ dA} = \frac{\iint v_n T\ dA}{\iint v_n\ dA}$$

To find the total heat transfer rate and mass flux, "Fluxes" was again chosen from the "Reports" dropdown in Fluent. Referencing Figure 5, it can be seen that instead of mass flow rate, one can choose "total heat transfer rate" in the "Options" section of the window.

An example calculation is shown below (using data from a simulation for (Z_1,Z_2) = (1.0m,1.0m)). Note that the "T" in Equation (1) is actually a ΔT, which means that Equation (1) actually calculates the average *change* in temperature at the outlet. Hence, the result must be added to the initial (inlet) temperature of 25°C to arrive at the average absolute temperature at the outlet.

$$T_{out} = T_{in} + \frac{\iint \rho\, C_p v_n T\, dA}{C_p \iint \rho\, v_n\, dA} = 298.15K + \frac{223.91133\, W}{\left(4216\, \frac{J}{kg\, K}\right)(.030911962\, \frac{kg}{s})} = 299.8681\, K$$

It could be assumed that this method is inaccurate, since the total heat transfer rate given by Fluent also includes the diffusive heat transfer. To check this inaccuracy, a more reliable method (that wasn't discovered until after all calculations had been performed in the above manner) can be used for comparison. This new method uses the "Surface Integrals" choice in the "Reports" dropdown in Fluent to report the value of the temperature flow rate. This value, divided by the mass flow rate also reported by Fluent, gives the absolute average temperature at the outlet. An example calculation is shown below, with data taken from the same simulation as in the calculation above.

$$T_{out} = \frac{temp\ flow\ rate}{mass\ flow\ rate} = \frac{\iint \rho\, v_n T\, dA}{\iint \rho\, v_n\, dA} = \frac{9.26951\, \frac{K\, kg}{s}}{.030911962\, \frac{kg}{s}} = 299.8681\, K$$

It can be seen that these two methods give the same result to four decimal places, suggesting that even if the first method is theoretically inaccurate, the diffusive heat transfer is so negligible that it hardly changes the result. Hence, the first method was used throughout the rest of this project.

Results:

The results from the method detailed above are displayed in Table 1 below.

Table 1: Average outlet temperature as inlet and outlet heights vary.

Outlet temperature (°C)		Inlet height Z_1 (m)		
		0.2	0.6	1.0
Outlet height Z_2 (m)	0.2	27.628	27.334	28.156
	0.6	28.195	26.941	27.222
	1.0	28.300	26.951	26.662

These results can be rearranged in order of decreasing outlet temperature to achieve a better understanding of how inlet and outlet height affect outlet temperature.

	Outlet temp (°C)	(Z_1,Z_2) (m)
Highest	28.300	(0.2,1.0)
	28.195	(0.2,0.6)
	28.156	(1.0,0.2)
	27.628	(0.2,0.2)
Median	27.334	(0.6,0.2)
	27.222	(1.0,0.6)
	26.951	(0.6,1.0)
	26.941	(0.6,0.6)
Lowest	26.662	(1.0,1.0)

Though these results were obtained with a first order solver and no mesh adaptation, the primary objective was to observe the relative influence of inlet and outlet height on outlet temperature. Hence, the results are still considered reliable in terms of which situation yields highest and lowest outlet temperature.

Because all data points only vary by less than 2°C, the precise order of the scenarios might vary slightly from person to person. Nevertheless, two general trends can be observed:

1. A lower value of Z_1 tends to give higher outlet temperature.
2. When Z_1 and Z_2 are further apart, this tends to give a higher outlet temperature.

The explanations for these trends are considered in the Results section of Task 2.

TASK 2

Problem Statement:

For the three cases from Task 1 with the highest, fifth highest (i.e. median), and lowest outlet temperature, plot the following:

1. The contour maps of temperature for the three horizontal (circular) cross sections at z = 0.2 m, 0.6 m, and 1.0 m (z is the vertical distance measured from the bottom of the tank)
2. The contour maps of temperature for the vertical cross section along the plane of symmetry (i.e. the cross section shown in Figure 3)
3. The contour maps of the velocity component parallel to the inlet velocity for the vertical cross section along the plane of symmetry (in this case, the x-velocity)

Use the contour maps generated for this task to interpret the outcome of Task 1. For example, if a particular choice of (Z_1,Z_2) yields the highest outlet temperature, explain why.

Procedure:

Based on the organized list of results from Task 1, it can be seen that the highest outlet temperature occurs for the scenario (Z_1,Z_2) = (0.2 m,1.0 m). (Through the rest of this report, scenarios will be listed simply as ([value],[value]). It is implicit that the order of these values is (Z_1,Z_2) and that the units are meters.) The median temperature occurs for the scenario (0.6,0.2). The lowest temperature occurs for the scenario (1.0,1.0).

All three of these scenarios were re-simulated using a second order solver (upwind, where applicable) and meshes adapted to the regions of large temperature gradient. Figure 6 below shows the changing to a second order solver. Figure 7 below shows a typical illustration of the region where the mesh was adapted; it was always near the base of the tank, since that’s where the temperature gradient was largest.

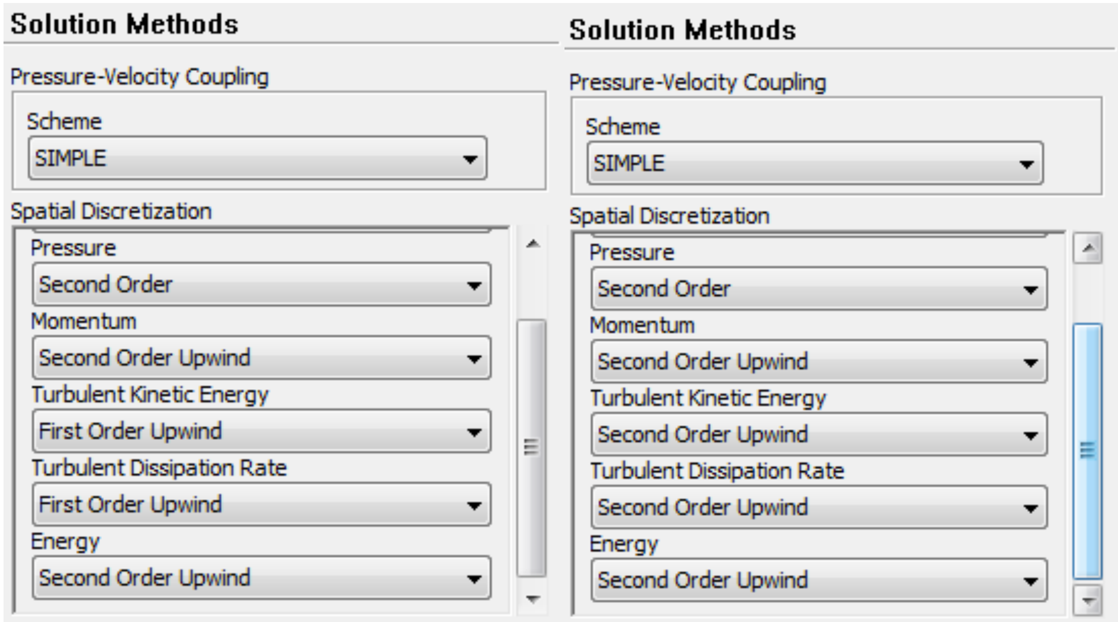


Figure 6: Change from first order solving methods (left) to second order (right).

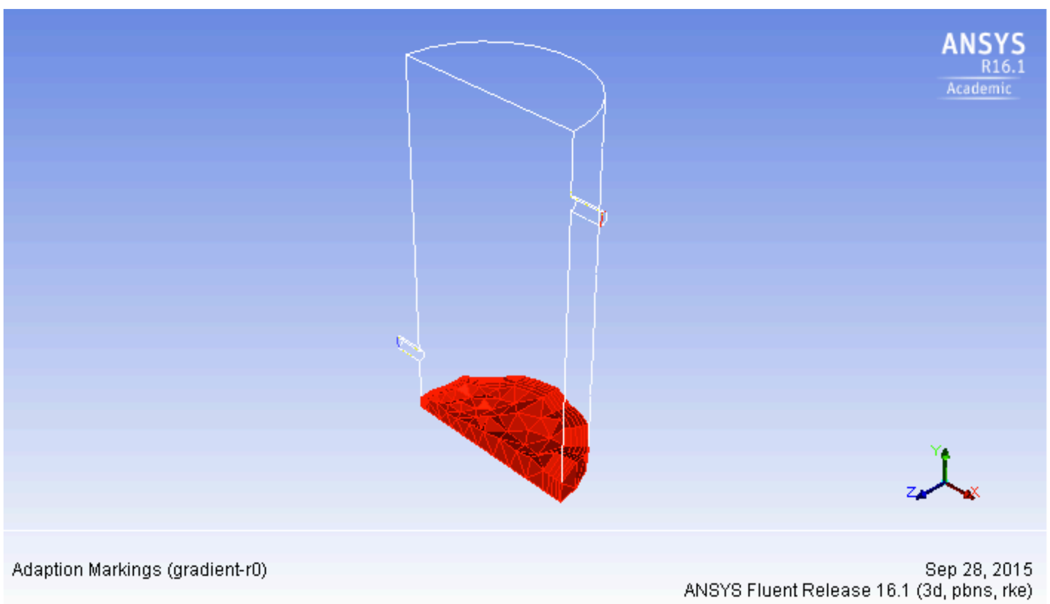


Figure 7: Region of mesh adaptation corresponding to high temperature gradient.

It should be noted that this method adjusted the final results of outlet temperature for all three situations. The new results, which are assumed to be more accurate, are summarized in Table 2 below.

Table 2: Average outlet temperature for three scenarios of interest using a second order solver and temperature gradient-based mesh adaptation.

Scenario (Z_1,Z_2) (m)	Average outlet temperature (°C)
(0.2,1.0)	29.192
(0.6,0.2)	29.179
(1.0,1.0)	27.567

All three outlet temperatures with this new method increased around 1 to 2°C. Interestingly, the median temperature scenario was much closer to the value of the max temperature after this new solving method was implemented. As was stated in the Results section of Task 1, it can be expected that the relative order of the scenarios with respect to outlet temperature could vary quite easily, since all data points are so close together. If all scenarios were re-run with this new solving method, (0.6,0.2) might no longer be the median, but it is expected that (0.2,1.0) would still be the maximum and (1.0,1.0) would still be the minimum.

Results:

Starting on the following page, Figures 8 through 10 display the requested contour plots for each of the three scenarios of interest. In each figure, the following system is used:

(a1) displays temperature for horizontal cross-section z = 0.2m
(a2) displays temperature for horizontal cross-section z = 0.6m
(a3) displays temperature for horizontal cross-section z = 1.0m
(b) displays temperature for vertical cross-section along plane of symmetry
(c) displays x-velocity for vertical cross-section along plane of symmetry

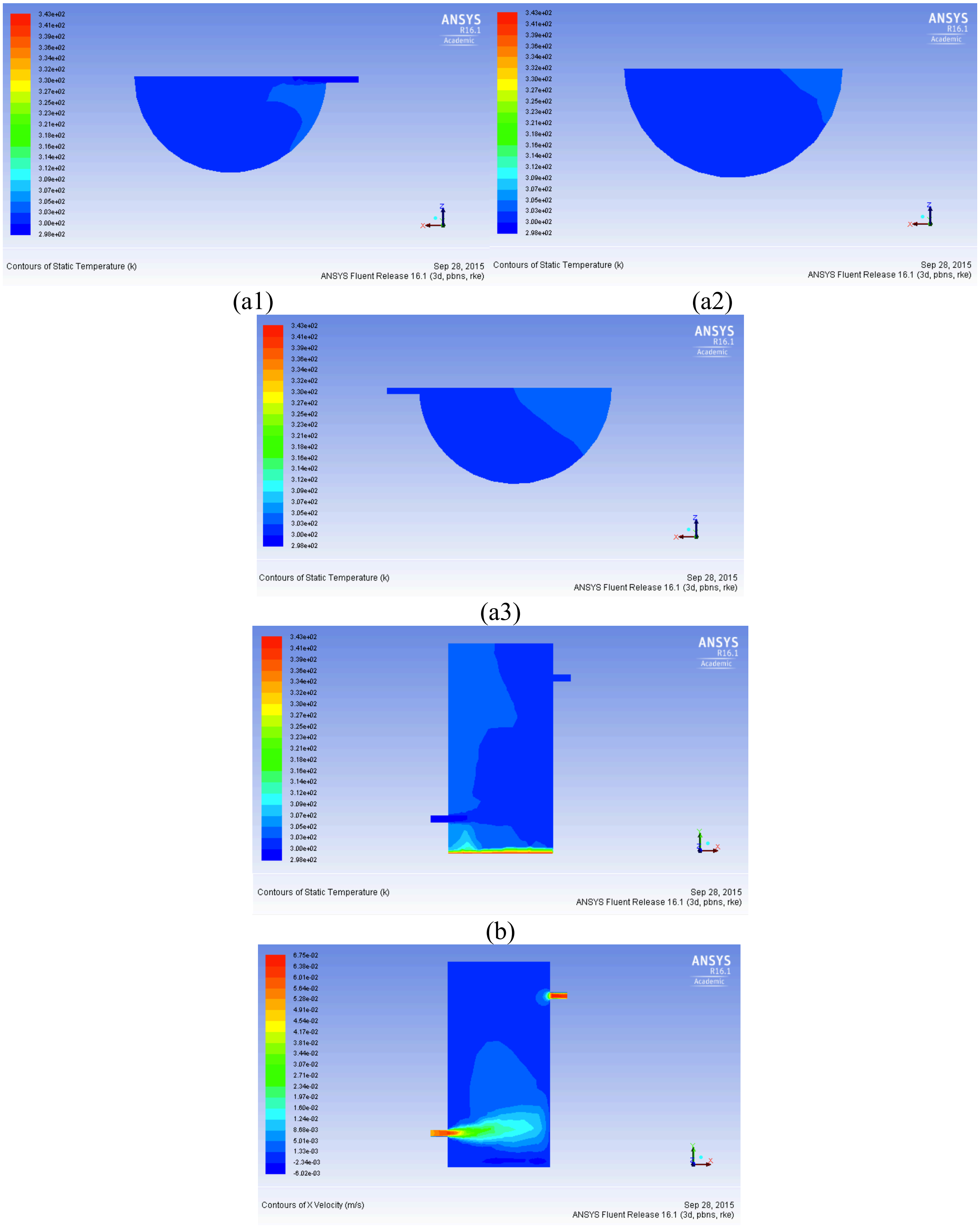
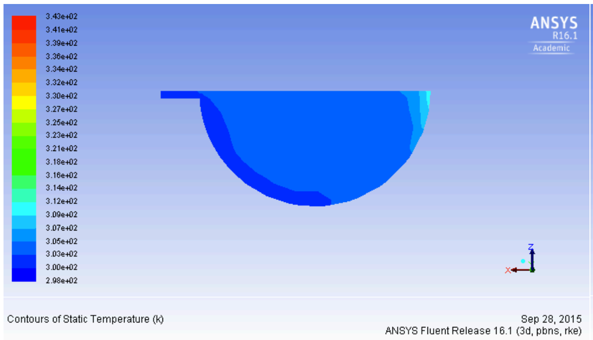
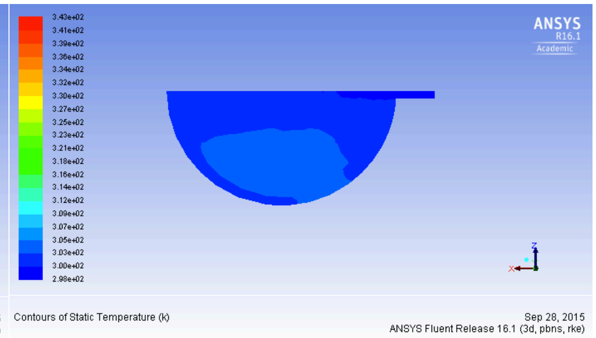
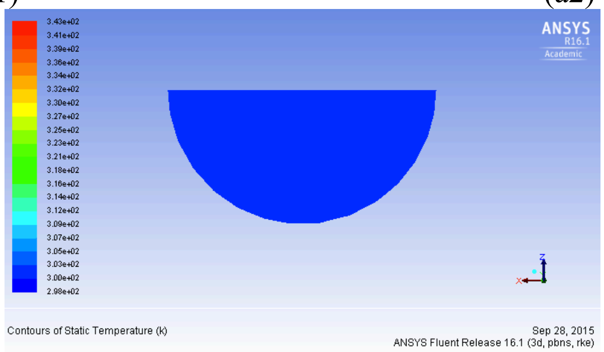


Figure 8: Contour plots for scenario (0.2,1.0).

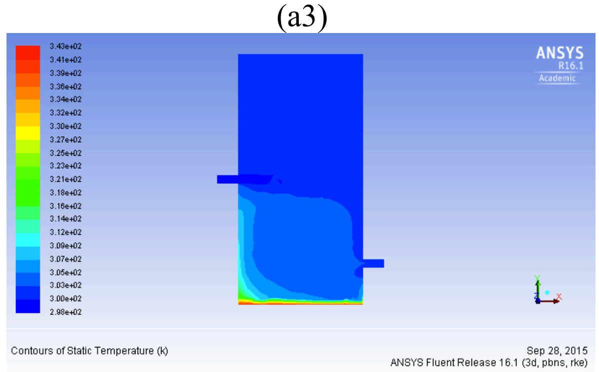




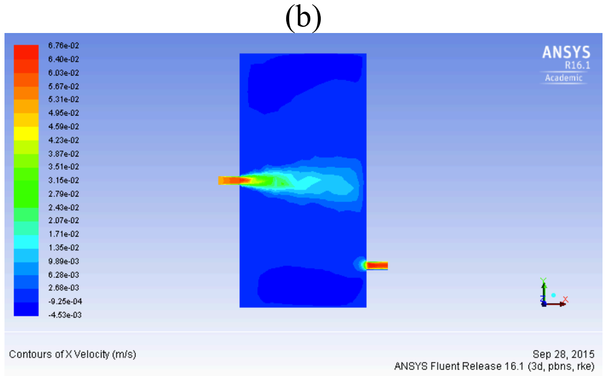
(a1) (a2)



(a3)

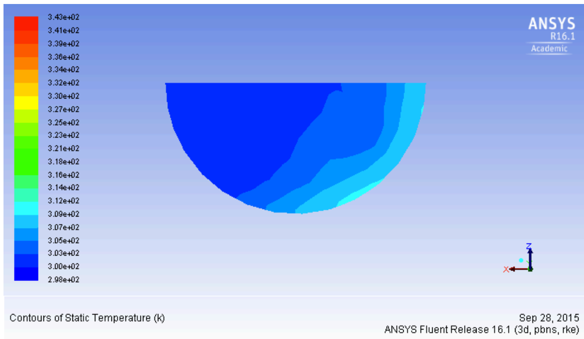


(b)

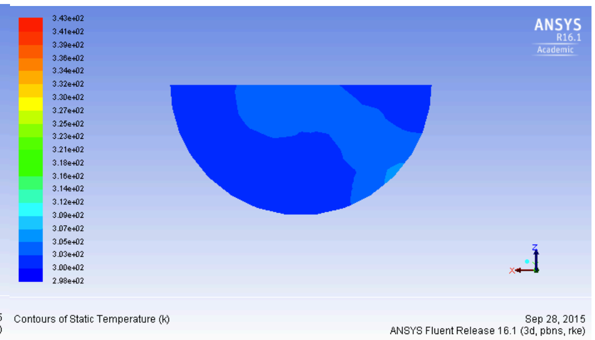


(c)

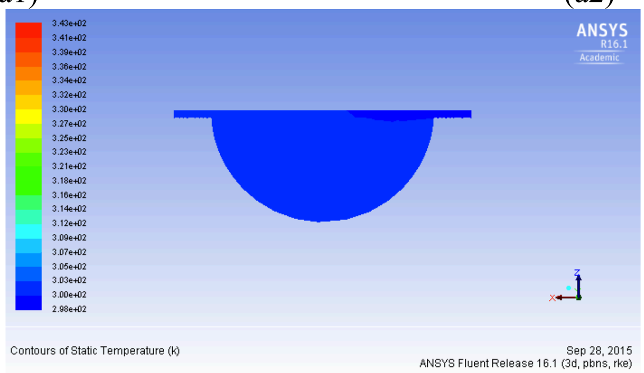
Figure 9: Contour plots for scenario (0.6,0.2).



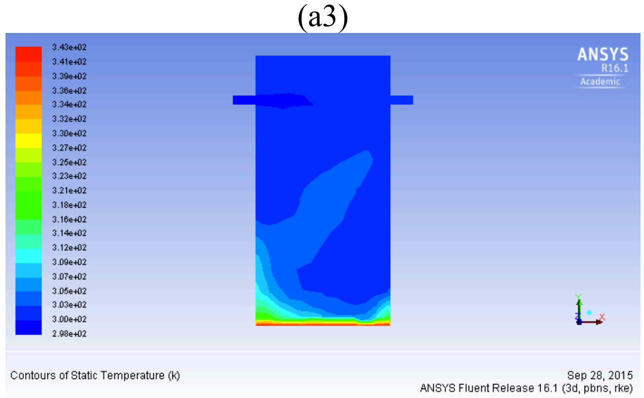
(a1)



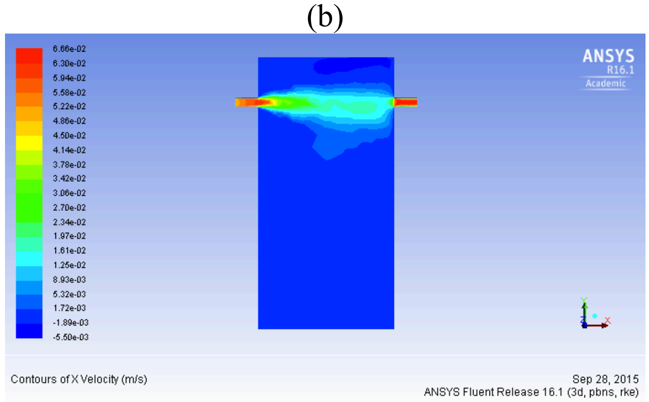
(a2)



(a3)



(b)



(c)

Figure 10: Contour plots for scenario (1.0,1.0).

Of the above figures, comparisons of sub-figures (b) and (c) are perhaps the most useful because they show where the fluid is traveling with the greatest velocity compared to the where the fluid experiences the largest temperature gradients. If one were to overlay the figures, it would be apparent that the scenario of highest outlet temperature has high-velocity contours that overlap or come close to the regions of highest temperature gradient. Conversely, the scenario of lowest outlet temperature has high-velocity contours that are far away from the regions of highest temperature gradient.

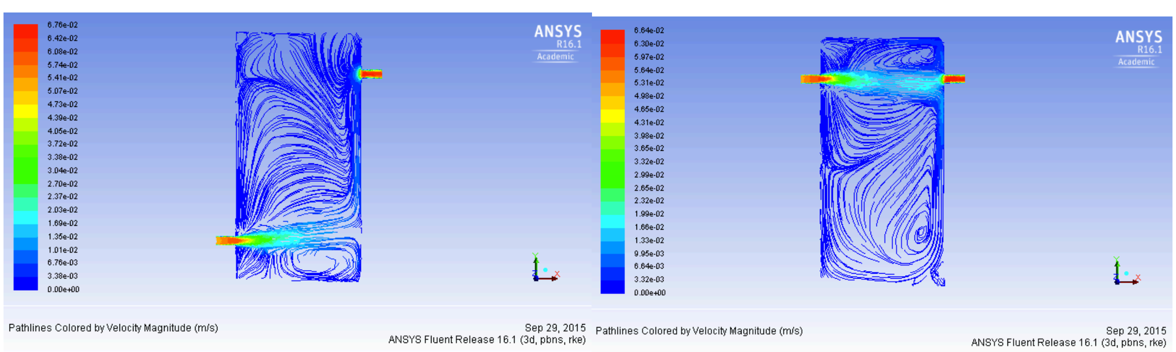
This trend is harmonious with the trends noted at the end of Task 1. As a reminder, these two trends were:

1. A lower value of Z_1 tends to give higher outlet temperature.
2. When Z_1 and Z_2 are further apart, this tends to give a higher outlet temperature.

Having the inlet closer to the heated base guarantees that the velocity of the water will be greater closer to areas of high temperature gradient, which causes greater heat flux and therefore higher overall rise in temperature. In addition, when the inlet and outlet are further apart, it takes more time for the water to flow through the tank, which allows heat transfer to have a greater effect.

Hence, the situation giving the highest outlet temperature would have the inlet closest to the base while the outlet is as far away as possible. Correspondingly, the situation giving the lowest outlet temperature would have the inlet and outlet both as far away from the base as possible. Both of these predictions are supported by the data from this task.

Similar reasoning can be used to explain why none of the scenarios increases the outlet temperature significantly compared to the inlet temperature. Figure 11 below displays path lines of the fluid for the scenarios of maximum and minimum outlet temperature. Velocity magnitude is represented by the color of the path lines.



(a) Scenario (0.2,1.0). (b) Scenario (1.0,1.0).

Figure 11: Path lines of fluid velocity.

Neither scenario exhibits many vertically oriented path lines that also correspond to high velocity near the base. Yet heat transfer is greatest when velocity is perpendicular to contour lines of temperature (which are predominantly horizontal near the base of the tank). Hence, no scenario exhibits significant change in temperature between the inlet and outlet. Indeed, this also explains why the data points are spread over such a small range of temperature.

TASK 3

Problem Statement:

For the case from Task 1 with the highest outlet temperature, repeat the simulation but with the inlet velocity set to 0.025 m/s, 0.1 m/s, and 0.2 m/s. Plot the steady-state outlet temperature as a function of inlet velocity for the four cases (including the original one from Task 1).

For the case with 0.2 m/s inlet velocity, make contour plots of temperature and velocity for the vertical cross-section in the same fashion as Task 2.

Procedure:

A second-order solver and mesh adaptation were again implemented for this task. Inlet velocity was varied by adjusting the boundary conditions in Fluent. All work was done using scenario (0.2,1.0) since that scenario exhibited the greatest outlet temperature in Task 1.

Results:

Table 3 below displays the data collected for this task. Figure 12 represents the same data in graphical form.

Table 3: Outlet temperature as inlet velocity varies.

Inlet Velocity (m/s)	Outlet Temperature (°C)
0.025	29.753
0.05	28.711
0.1	28.352
0.2	28.231

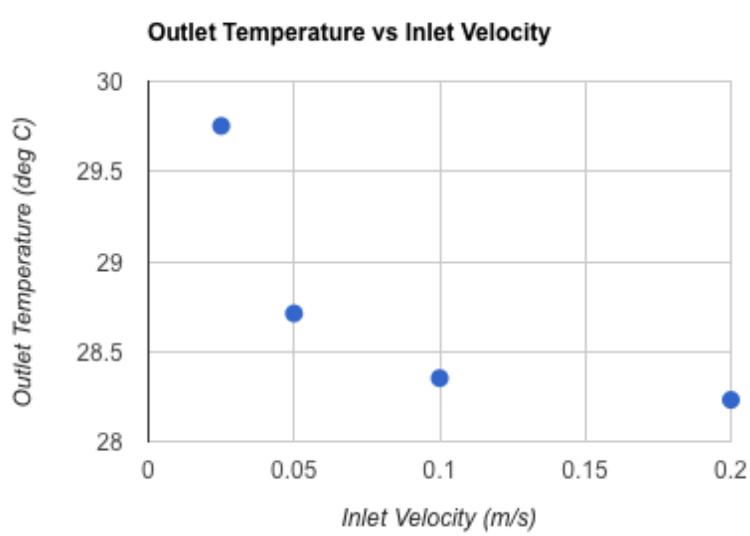
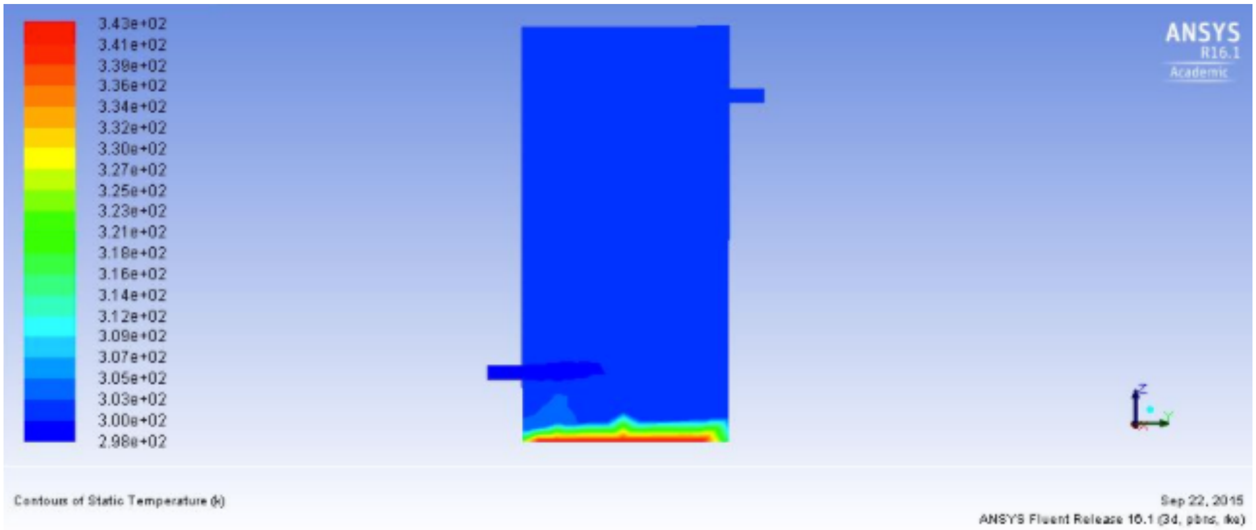
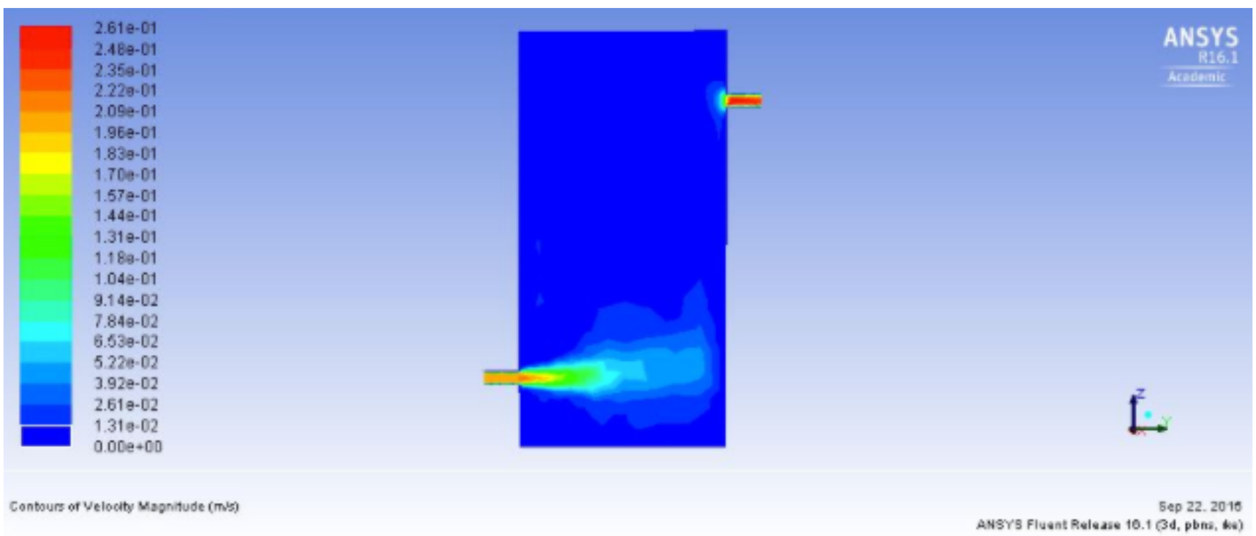


Figure 12: Plot of outlet temperature as a function of inlet velocity.

Figure 13 below displays (a) temperature and (b) x-velocity contours for the case where the inlet velocity was set to 0.2 m/s.



(a) temperature



(b) x-velocity

Figure 13: Contour plots for case of 0.2 m/s inlet velocity.

Figure 12 seems to suggest that outlet temperature decreases exponentially with inlet velocity. This makes sense because as inlet velocity increases, it becomes a more and more effective heat sink for the body of fluid as a whole. Figure 13 supports this notion as well, when compared to Figures 8b and 8c (the contour plots for temperature and velocity when inlet velocity was set to 0.05 m/s). Conversely, a slower inlet velocity functions as a weaker heat sink because the fluid has more time to stay in the tank while subject to heat transfer.

TASK 4

Problem Statement:

For the case from Task 1 with the highest outlet temperature, change the "turbulence k-epsilon" model to "laminar" and redo the simulation. Calculate the steady-state outlet temperature. Is the outlet temperature sensitive to the choice of the model? Does the behavior of numerical convergence (based on the diagram of "residual" vs. "number of iterations" provided by Fluent) change when the model is switched from turbulent to laminar?

Procedure:

The "Viscous" model from the "Models" dropdown in Fluent was changed from "turbulence k-epsilon" to "laminar." The simulation was then run under the same remaining parameters as in Task 1. All work was done using scenario (0.2,1.0) since that scenario gave the greatest outlet temperature in Task 1.

Results:

Table 4 below shows the difference in number of iterations and outlet temperature between the turbulent model and the laminar model.

Table 4: Turbulent vs Laminar Models.

	Turbulent Model	Laminar Model
Number of Iterations	800	114
Outlet Temperature (°C)	28.698	26.850

Though a difference of less than 2°C might normally be considered insubstantial, in the scope of this project, this is actually quite a large difference. Hence, it can be concluded that the outlet temperature is sensitive to the choice of the model.

In addition, the behavior of numerical convergence is also quite different between the two models. The turbulent model converged by 800 iterations, as can be seen by the residual plot in Figure 14 below. However, the laminar model concluded after just 114 iterations, never having converged, as shown in Figure 15. Instead, the error message shown in Figure 16 appeared, saying that "Divergence [was] detected in [the] AMG solver," specifically in calculating the temperature.

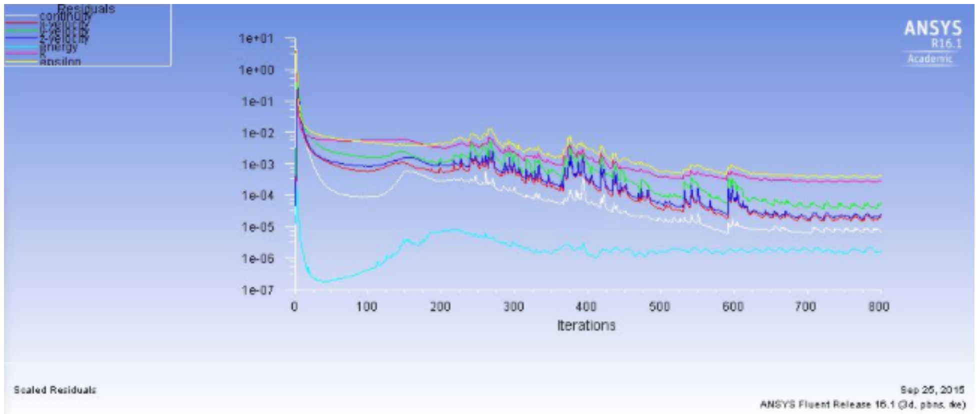


Figure 14: Residual plot for turbulent model, showing convergence around 650 iterations.

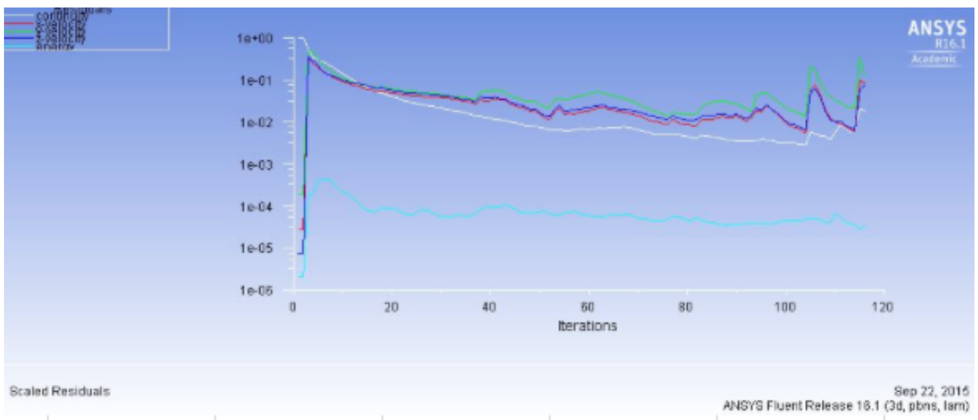


Figure 15: Residual plot for laminar model, showing divergence at 114 iterations.

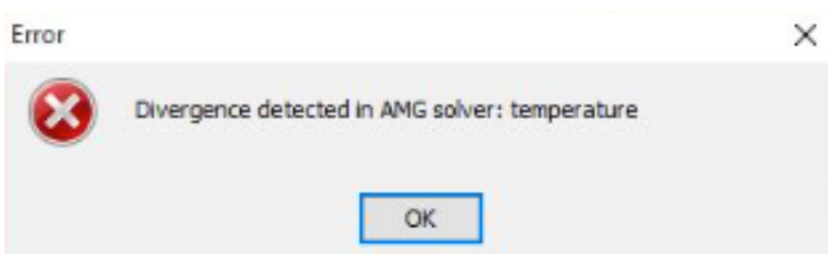


Figure 16: Error communicating that the laminar model reached divergence.

In summary, the choice of the model type, at least concerning turbulent versus laminar, makes a significant difference in the ability of system to successfully solve the simulation. It is possible that, were the laminar model to converge, the results would be much closer in value than 2°C. But from the information that could be collected, it would appear that the laminar model arrives at a substantially different solution from the turbulent model.

TASK 5

Problem Statement:

Consider the case from Task 1 with the highest outlet temperature.

(i) Use Fluent to report the average heat flux that comes out of the bottom boundary into the main water tank.

(ii) Next, replace the boundary condition, "temperature = 70°C," for the bottom boundary by a boundary condition with an imposed constant *heat flux* using the value from Part (i). Redo the simulation. Can the outlet temperature from the original simulation (that had an imposed constant temperature at the base) be recovered? If not, explain why.

Procedure:

The same procedure as in Task 1 was used to find the total heat transfer through the base of the tank (from the "Reports" dropdown, "Fluxes" was chosen, followed by "Total Heat Transfer Rate"). This value was then set as the new boundary condition for the base (constant heat flux was chosen in place of constant temperature for the boundary condition). The simulation was redone using a second order solver and a mesh adapted to the regions of high temperature gradient (as detailed in Task 2). The new outlet temperature was calculated through the same procedure as in Task 1.

All work was done using scenario (0.2,1.0) since that scenario gave the greatest outlet temperature in Task 1.

Results:

Figure 18 below shows Fluent's report of the total heat transfer rate and mass flow rate at the inlet, outlet, and tank base.

```
        Total Heat Transfer Rate                  (w)
-------------------------------- --------------------
                 pressure-outlet           -541.75293
                  velocity-inlet      -0.00084852136
                     wall-bottom            541.47253
                ---------------- --------------------
                             Net          -0.28124403

                  Mass Flow Rate               (kg/s)
-------------------------------- --------------------
                 pressure-outlet         -0.030915109
                  velocity-inlet          0.030901501
                     wall-bottom                   -0
                ---------------- --------------------
                             Net       -1.3608485e-05
```

Figure 18: Reports of heat flux and mass flux through inlet, outlet, and tank base.

The above data shows that the average heat flux that comes out of the bottom boundary into the main water tank is approximately 541.473 W. Once this value was imposed as the new boundary condition at the base, the newly calculated outlet temperature was 29.157°C. This is only a 0.12% difference from the outlet temperature calculated using a constant temperature boundary constraint, which was 29.192°C. Hence, it appears that the original outlet temperature can successfully be recovered even when the boundary condition is changed to a different – yet thermodynamically equivalent – type.

Interestingly, however, the temperature contour plot on the symmetry plane from the constant heat flux simulation (Figure 19 below) changed substantially compared to those from the constant temperature simulation (Figure 20 below).

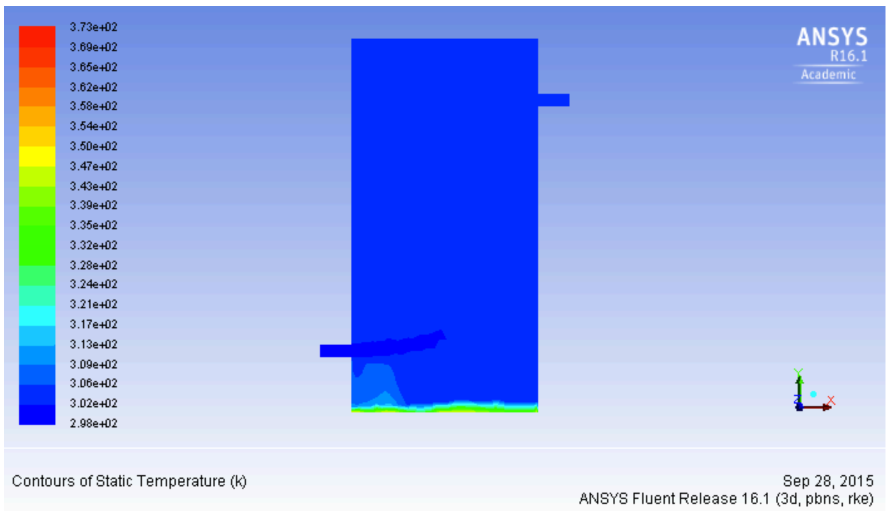


Figure 19: Temperature contours from constant heat flux simulation.

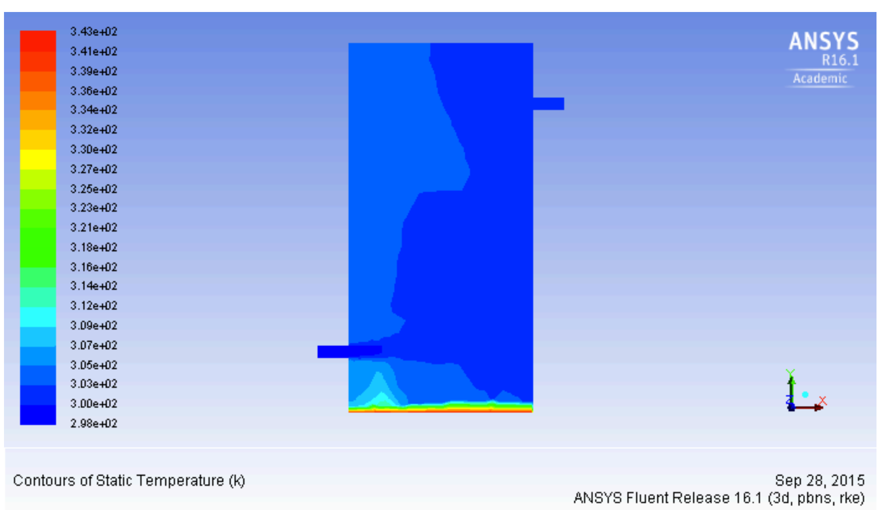


Figure 20: Temperature contours from constant temperature simulation.

There are two apparent differences between these contour plots. First, the temperature appears more uniformly cool above the inlet in the constant heat flux simulation. Second, the temperature gradient at the base in the constant heat flux simulation appears much steeper than that of the constant temperature simulation: the colors change from red to blue much quicker (indeed, it is nearly impossible to see any red at all in the constant heat flux simulation).

To compensate for this steeper temperature gradient near the base, further mesh adaptations were performed, but the outlet temperature and temperature contours hardly changed even after two additional mesh adaptations. Furthermore, the maximum temperature in the tank found by the constant heat flux simulation was approximately 100°C (373 K) compared to a max temperature of 70°C (343 K) in the constant temperature simulation. This is likely the primary reason why the temperature gradient was steeper near the tank base for the constant heat flux simulation.

For these reasons, there does appear to be something inherently different about the solutions based on the type of boundary condition imposed at the tank base. Nevertheless, the outlet temperature stayed nearly constant despite the changing boundary condition.

Therefore, it seems safe to assume that some aspects of the simulation solutions are sensitive to boundary conditions, while others are not. Based on the above information, it seems logical to conjecture that regions further away from the changing boundary condition will remain less affected, while regions closer to the boundary condition will be affected more substantially. This could be seen as a "ripple effect," where a local change affects regions around it, but with magnitude that decreases radially from the location of the changed boundary condition.

CONCLUSIONS

In summary, the following major conclusions were gathered from the results of this project:

1. Fluid body geometry, especially with regards to locations of inlets and outlets, has a significant impact on the behavior of a thermo-fluid system.
2. Flow direction relative to temperature gradient has a significant impact on the efficacy of heat transfer. Specifically, greater heat transfer occurs when the fluid flows perpendicular to temperature contours.
3. Boundary conditions can have significant impacts on the behavior of a thermo-fluid system.
    a. Inlet velocity can control how effective that inlet is as a heat source/sink.
    b. Theoretically equivalent thermodynamic boundary conditions may change simulation results in some ways while having little effect on other results of the simulation.
4. Choice of model (in this case, laminar versus turbulent) can have a significant impact on both the simulation results and the ability of the software to solve the simulation at all.