ME485: Computational Fluid Dynamics

CHT analysis of a U-tube heat exchanger

03/01/2026

Daniyaal Rashid Ali Hakeem Muhammad Huzaifa Khan Ghory

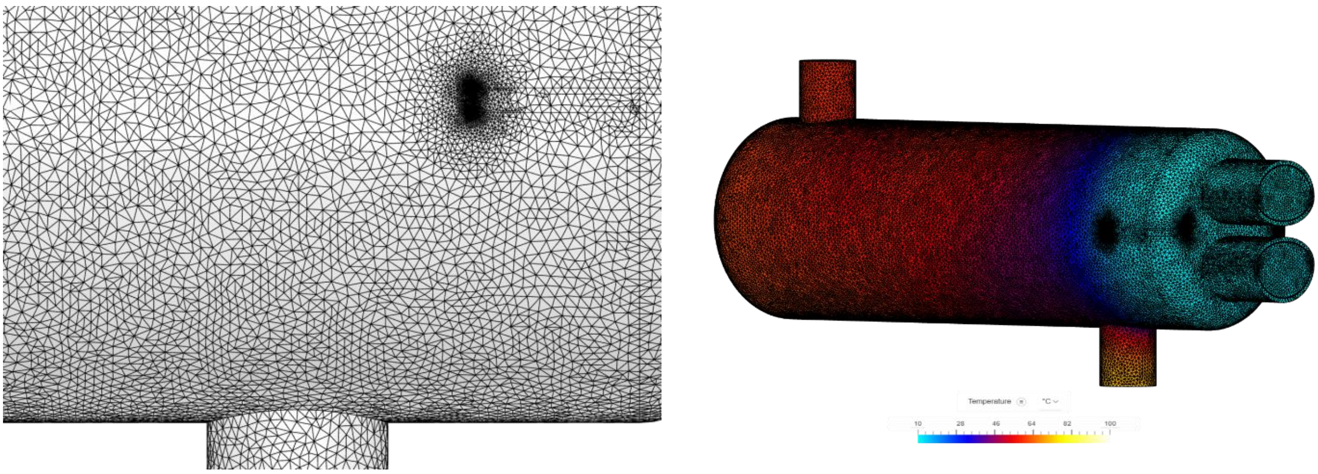
Abstract

In this study, a conjugate heat transfer (CHT) analysis of a U-tube heat exchanger is performed using the finite volume method (FVM) on the SimScale platform. Hot air flows through the steel casing while cold water circulates inside the U-tubes. Average inlet pressures and outlet temperatures are monitored to assess solution convergence and energy balance. A mesh independence study is conducted using three mesh fineness levels to determine the optimal mesh that provides mesh-independent results. The effects of zero, two, and four internal baffle plates on flow physics, pressure losses, and heat transfer performance are investigated. In addition, the water-side geometry is modified by varying the number and diameters of the tubes, leading to changes in the flow structure. The results indicate that a mesh fineness level of 5 provides an optimal balance between accuracy and computational cost. While variations in the number of baffle plates have a limited influence on the overall performance, modifications to the number and diameters of the water pipes significantly affect the flow behaviour and heat transfer characteristics.

1. Methodology

1.1. Meshing

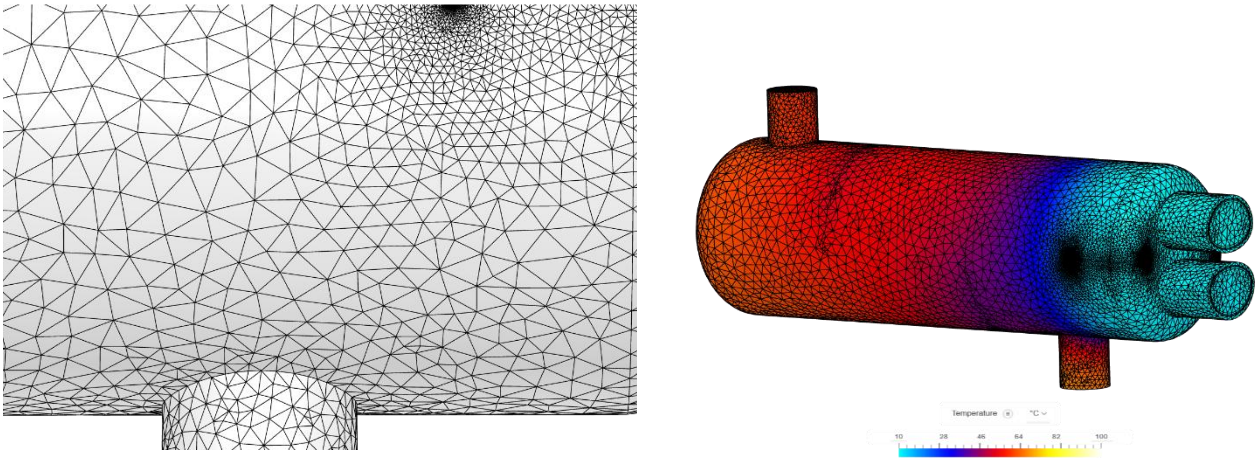
For the U-tube heat exchanger, a mesh is generated with a generic fineness level of 5. While the global control parameter 'fineness' may not be the best choice to generate and compare a mesh, it is sufficient for the scope of this study. The mesh at fineness level 5 has triangular cells with 1.6 million cells. Once the mesh is created, the simulation can be performed.



(a) (b)

Figure 1. Mesh created at fineness level 5 with triangular cells. (a) Close-up view. (b) Whole body view

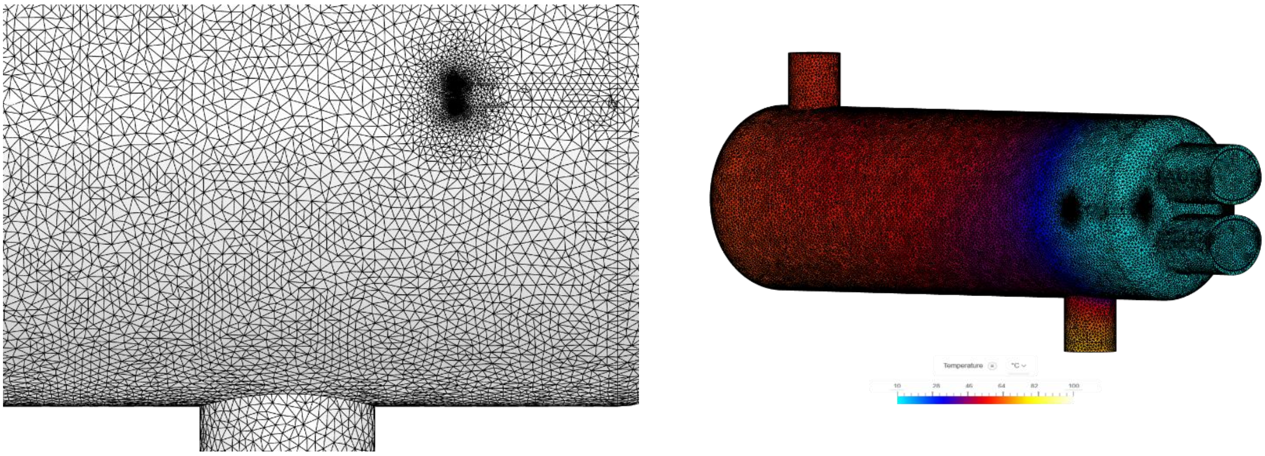
The mesh is now re-generated with more coarser cells. For this a fineness level of 3 is utilized. The total number of cells are now 1.1 million.



(a) (b)

Figure 2. Mesh created at fineness level 3 with triangular cells. (a) Close-up view. (b) Whole body view

The mesh is now re-generated with finer cells. For this a fineness level of 7.6 is utilized. The total number of cells for this fineness level are 7.1 million.



(a) (b)

Figure 3. Mesh created at fineness level 7.6 with triangular cells. (a) Close-up view. (b) Whole body view

Table 1. Number of cells produced by the mesh for fineness levels and the corresponding air outlet temperatures and air inlet pressures

Fineness Level	Number of Cells (millions)	Air Outlet Temperature (K)	Percentage difference	Air Inlet Pressure (Pa)	Percentage difference
2	0.997	320.17	0	101402.6	0
3	1.1	319.37	-0.25	99323.5	-2.05
5	1.6	319.81	0.138	101403.8	
7.6	7.1	321.22	0.44	101408.6	
9	19.2	317.15	-1.27	101407	

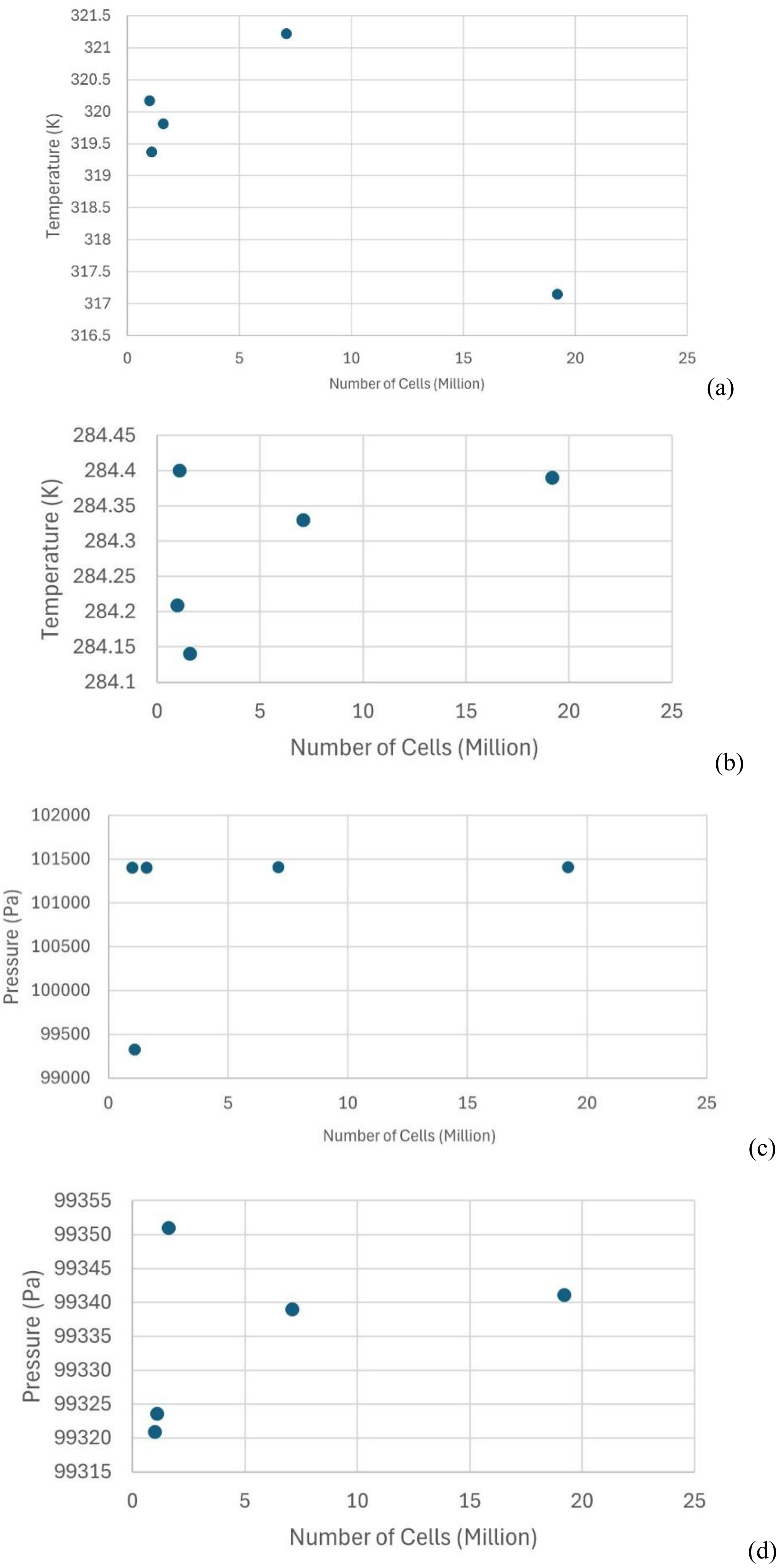
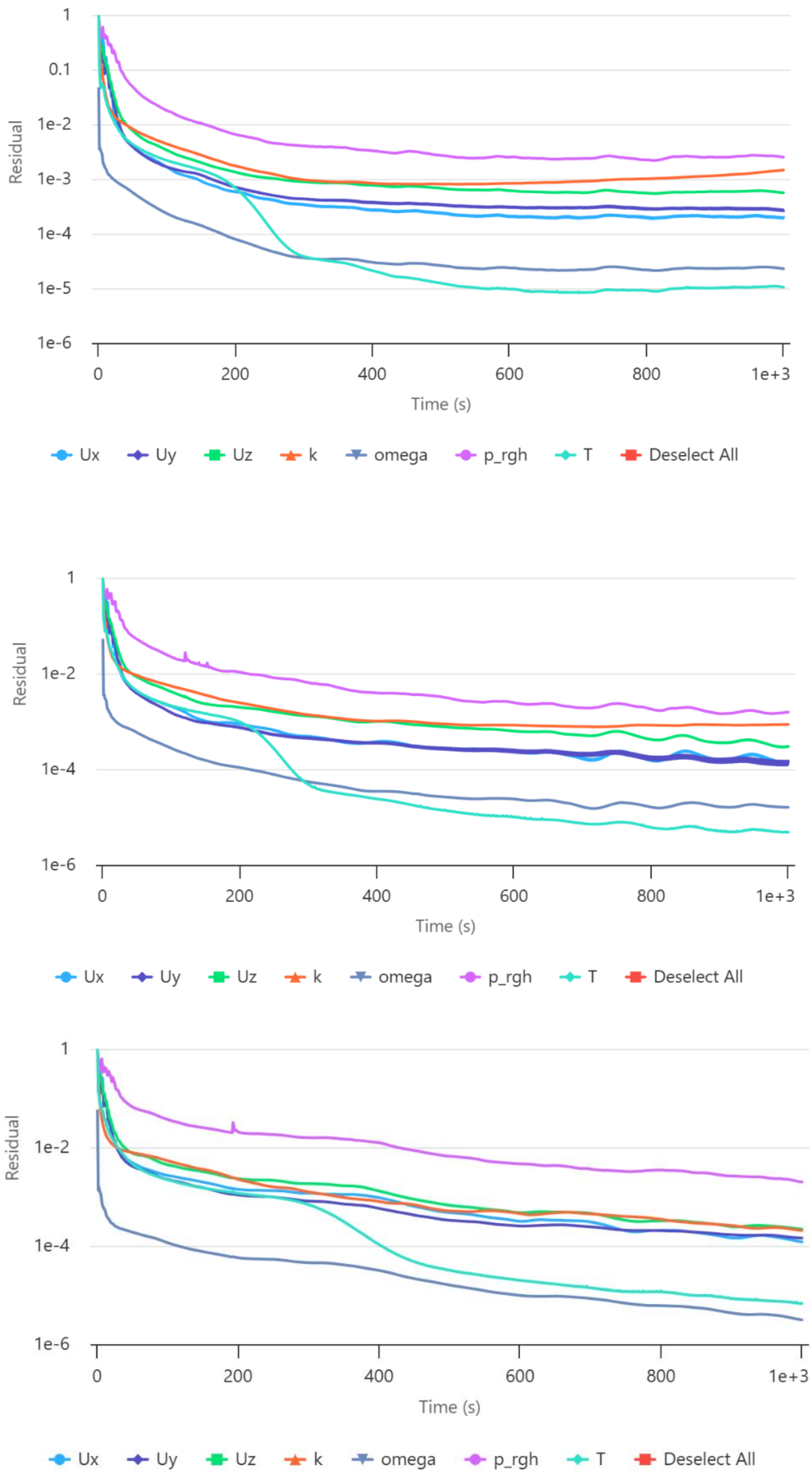


Figure 4. (a) The plot of average outlet air temperature vs number of cells. (b) plot of average outlet water temperature vs. number of cells. (c) plot of average inlet air pressure vs. number of cells. (d) plot of average inlet water pressure vs. number of cells

Table 1 and Figure 4 show the results of the mesh independence study. The most reasonable fineness level is 5. The outlet air temperature difference between fineness 5 and the finer values is low compared to the increase in the number of cells and the computing power needed behind it. The total cell count go from 1.6 million cells for fineness 5 to 7.1 million for fineness 7.6 and 19 million cells for fineness 9. However, the inlet air pressure values after fineness level 5 stay nearly constant which is a sign of mesh independence.



(a)

(b)

(c)

Figure 5. (a) Convergence plot for fineness level 3. (b) (a) Convergence plot for fineness level 5. (c) (a) Convergence plot for fineness level 7.6.

It can be observed through the convergence plot that the finer the mesh, the quicker the convergence. For the first 200-400 iterations, error rates drop to $1 * 10^{-2}$ quickly for a mesh fineness level of 7.6

compared to a mesh fineness level of 3. All the solutions converge and fineness of 7.6 has the least oscillations. One thing to note however is that the temperature residuals drop to a lower error faster for a coarser mesh since it has a lesser number of nodes.

1.2. Boundary Conditions

The main aim for this heat exchanger is to cool hot air through convection from the pipes. For this reason, inlet water and air temperatures and outlet pressures as atmospheric pressure for both air and water are fixed. Since SimScale is being used for the analysis, specific conditions like no-slip at tube and shell walls are not specified because SimScale automatically understands the geometry and applies these conditions. The inlet and outlet faces, however, are manually selected and the values assigned as shown.

For air,

$$V_{air_{in}} = 10\frac{m}{s}$$

$$T_{air_{in}} = 100\ °C$$

$$p_{air_{out}} = 101325\ Pa$$

For water,

$$V_{water_{in}} = -0.05\frac{m}{s}$$

$$T_{water_{in}} = 10\ °C$$

$$p_{water_{out}} = 101325\ Pa$$

2. Results

2.1. Baffles

Baffles are thin plates places inside the casing of the heat exchanger and play a significant role in the overall heat transfer as well as the fluid dynamics. The three main purposes of baffles are enhancing turbulence and mixing which creates more vortices, force the fluid to take longer path hence increase heat transfer, and create separation and reattachment which thins or resets boundary layers. In this study, 3 separate simulations are performed, one with no baffles, one with 2 and one with 4 baffles to observe their effect.

2.1.1. Zero Baffles

When zero baffles are placed, the air flows through the case without much turbulence and mixing with low pressure gradients.

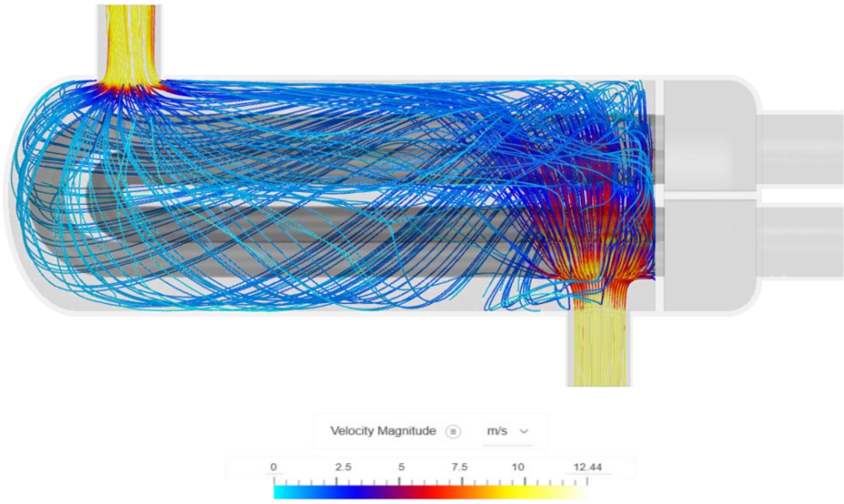
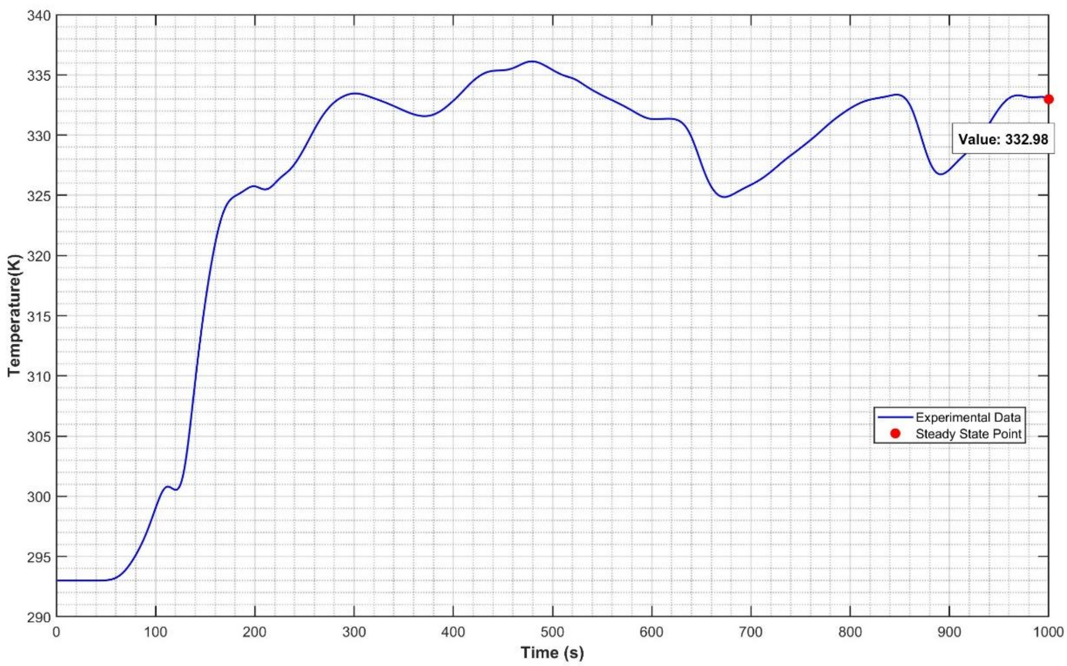
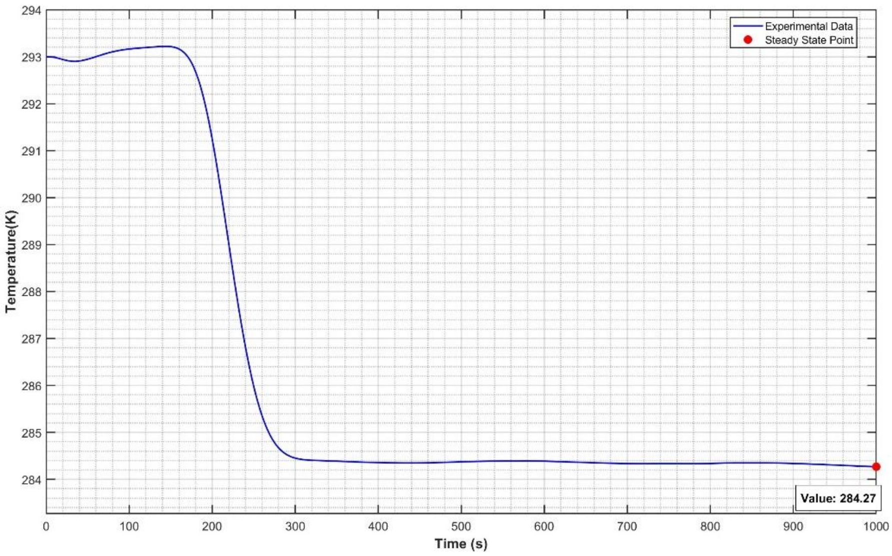


Figure 6. Flow of air in the casing with no baffles and low turbulence and pressure gradients.

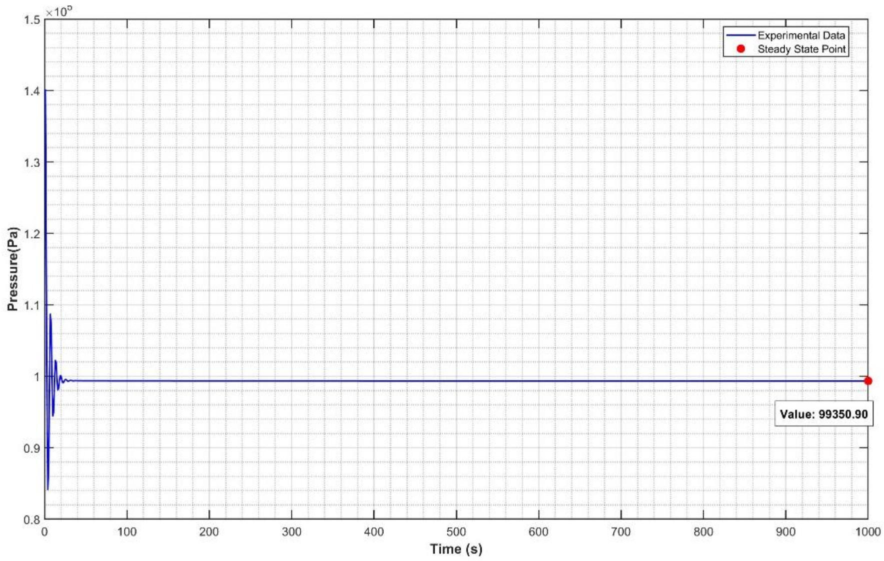
Now simulations can be performed with no baffles.



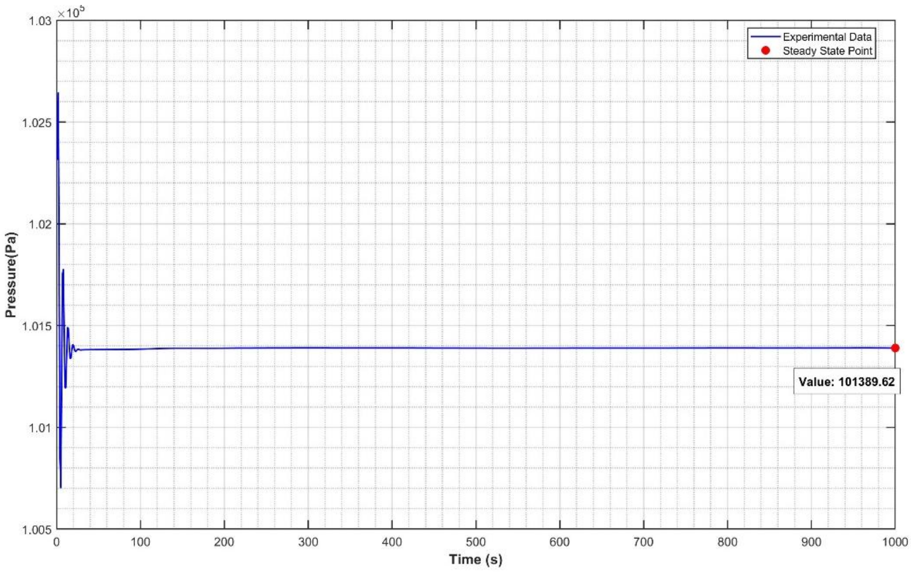
(a)



(b)



(c)



(d)

Figure 7. For zero baffles (a) Average air outlet temperatures over 1000 iterations. (b) Average water outlet temperatures over 1000 iterations. (c) Average water inlet pressures over 1000 iterations. (d) Average air inlet pressures over 1000 iterations

It is observed that the air outlet temperature variates significantly over the iterations until it smoothens out at around 900 iterations to a value of 332.98 K. The oscillations in the outlet temperature for the air can be attributed to the uninterrupted flow of air through the shell as seen in Figure 6. Some path lines can be seen reaching the outlet much quicker than other which leads to a difference in temperature, since the probe measures average temperature oscillations are observed.

2.1.2. Two baffles

Two baffles are the most common in a U-tube heat exchanger. The effect of two baffles on the air and water outlet temperatures can now be observed. Two baffles are expected to create significant turbulence and low-pressure gradients. They are also expected to reduce the oscillations at the air outlet temperature since the flow can no longer reach the outlet from various paths, instead the flow now follows a certain path around the baffles.

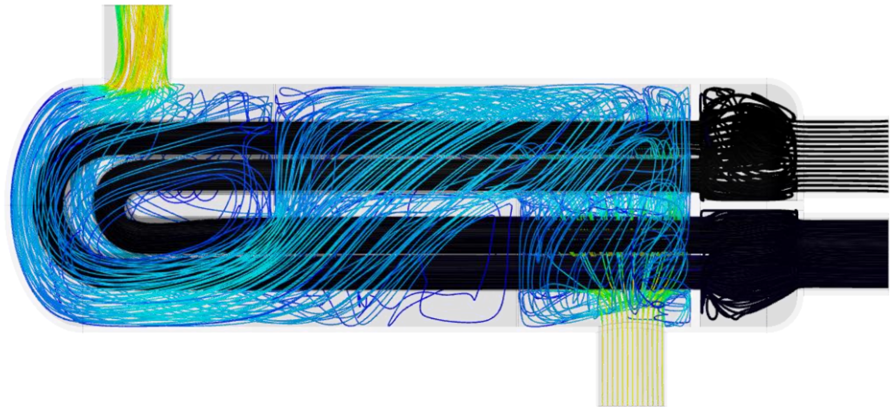
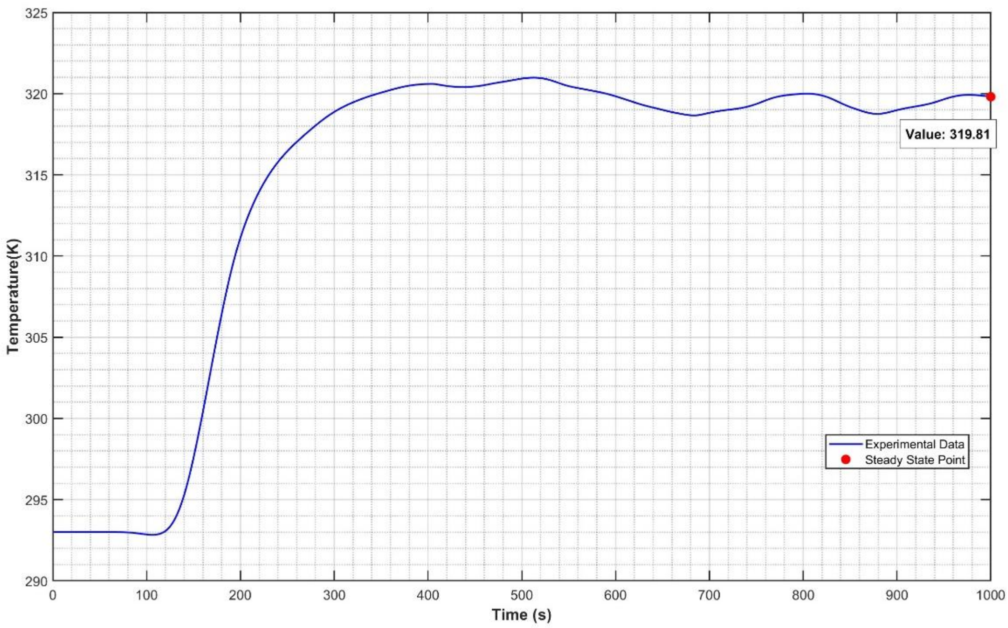
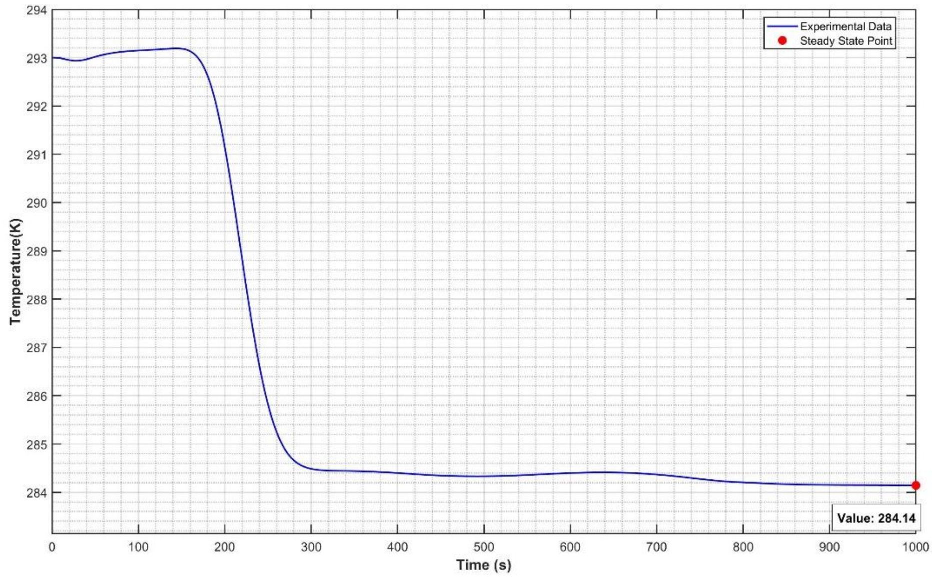


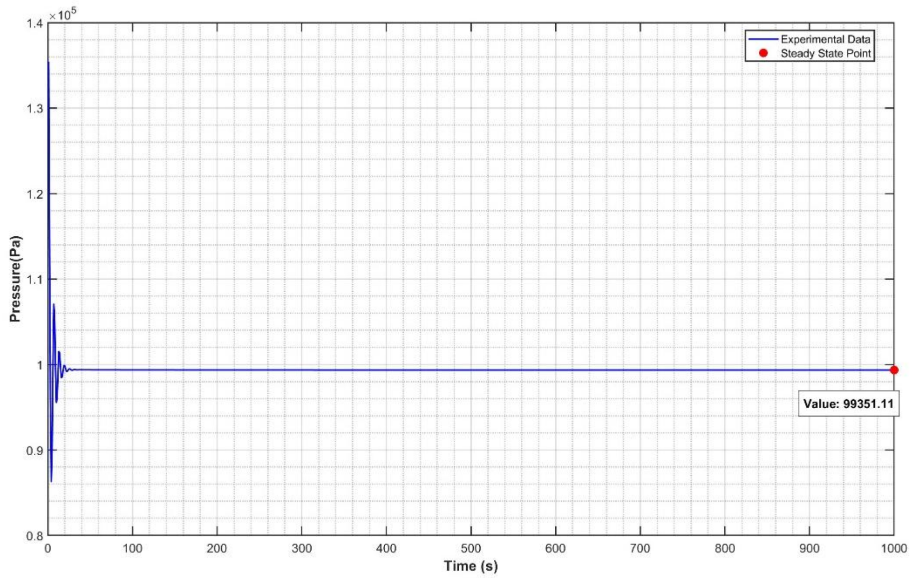
Figure 8. Flow of air in the casing with 2 baffles and medium turbulence and pressure gradients.



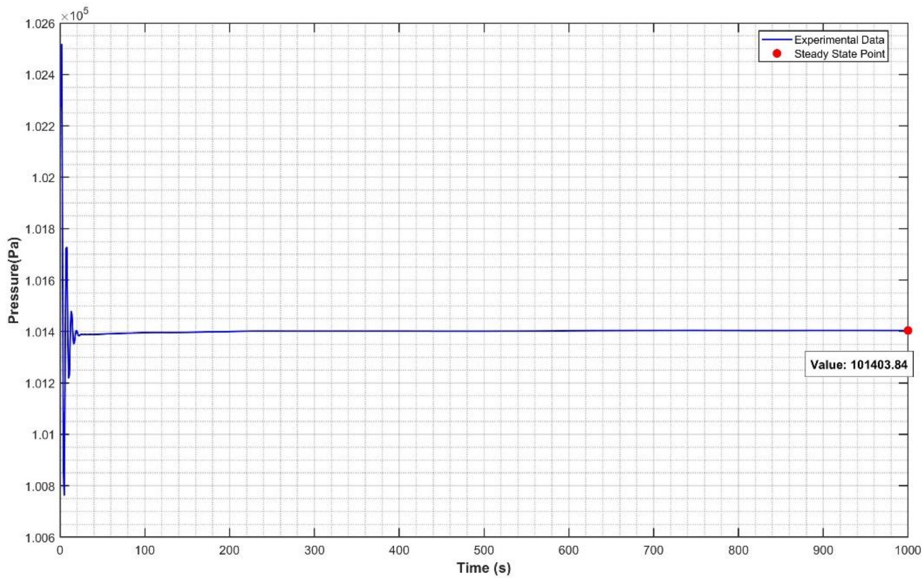
(a)



(b)



(c)



(d)

Figure 9. For 2 baffles (a) Average air outlet temperatures over 1000 iterations. (b) Average water outlet temperatures over 1000 iterations. (c) Average water inlet pressures over 1000 iterations. (d) Average air inlet pressures over 1000 iterations

It is observed, as expected, that the outlet air temperature has dropped significantly due to the presence of two baffles. For zero baffles, the outlet air temperature was 332.98 K and for 2 baffles it became 319.81 K. That is a difference of 13.17 K (°C)

2.1.3. Four Baffles

The number of baffles is now changed to 4. This introduces significant turbulence since the baffles are set up in a way that makes the flow move up and down 4 times. Figure 10 shows the baffle placement and the flow of air within the casing. The air takes up a much longer path, which leads to more contact with the pipes.

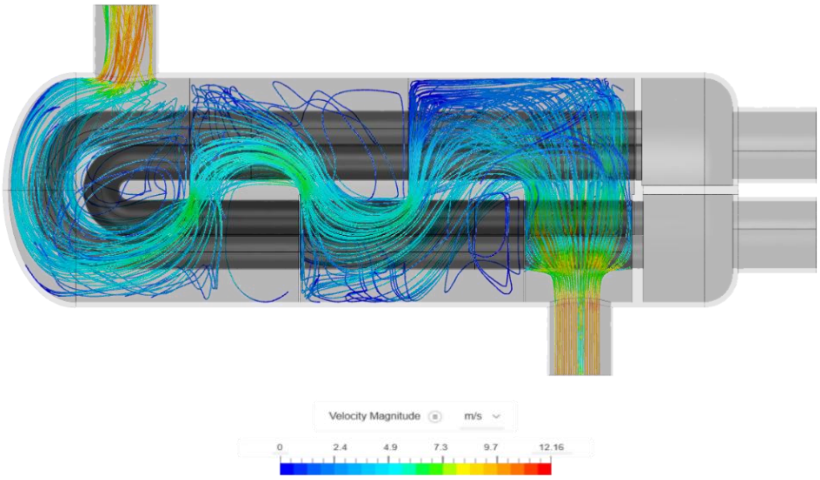
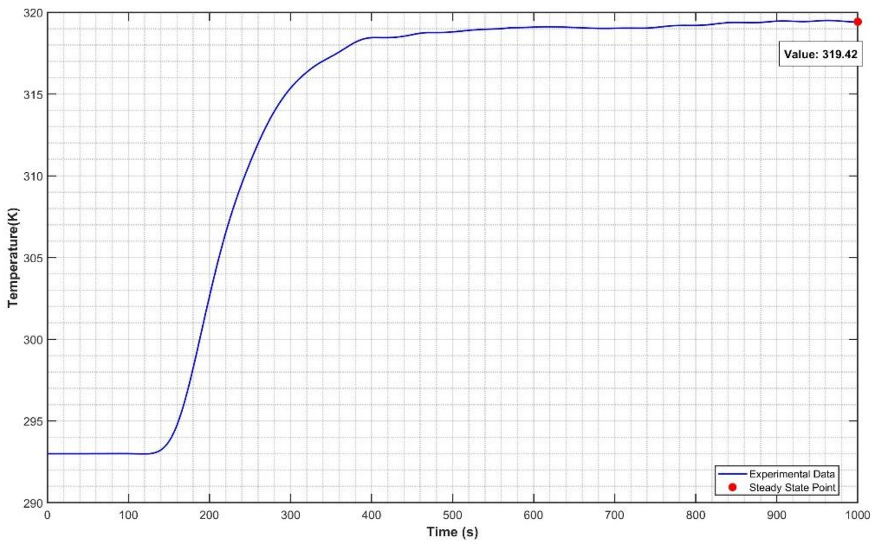
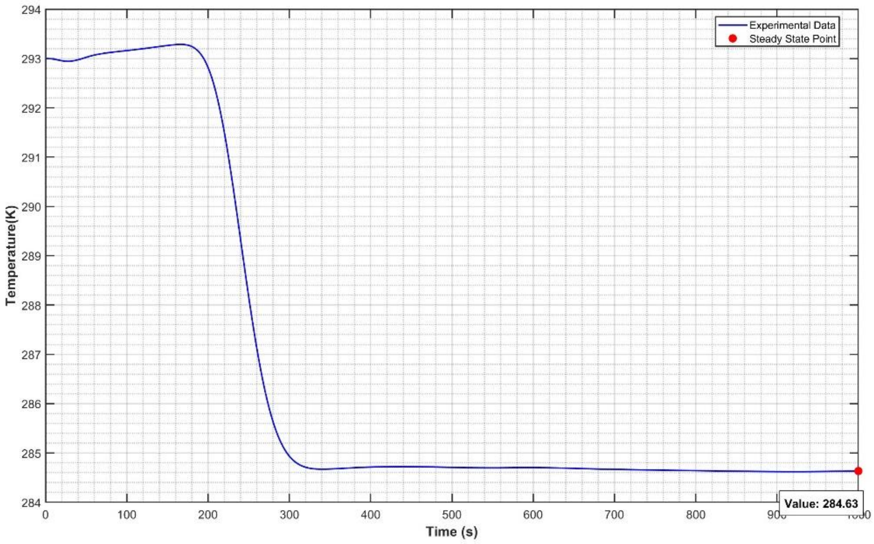


Figure 10. Flow of air in the casing with 4 baffles and high turbulence and pressure gradients.

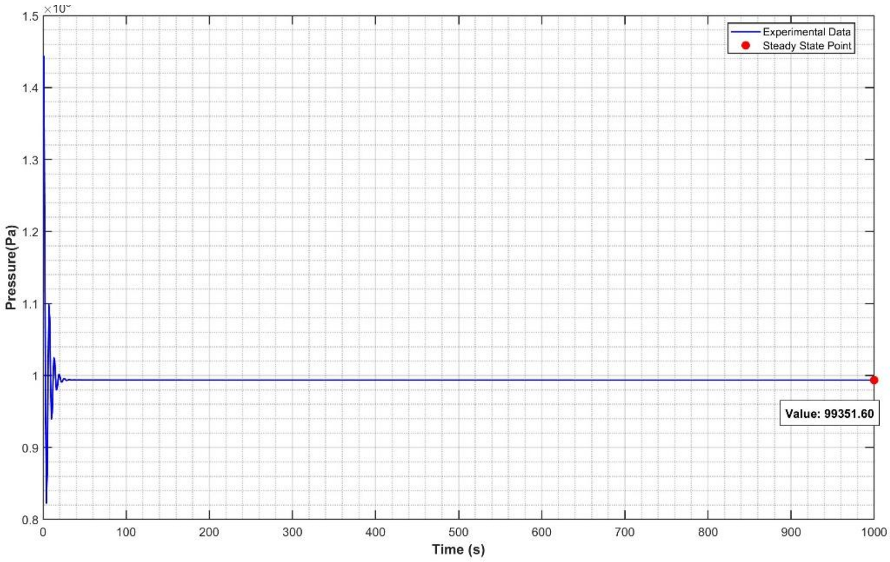
Now flow over 4 baffles is simulated.



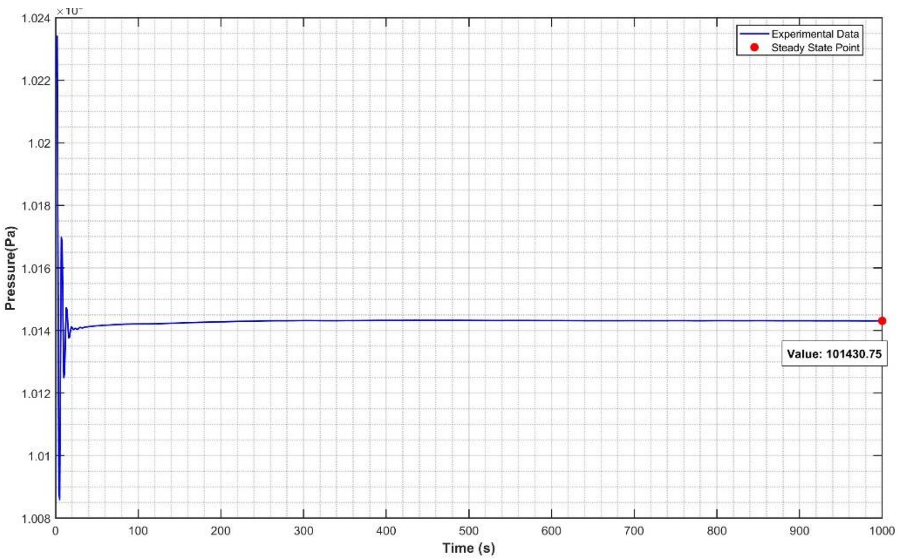
(a)



(b)



(c)



(d)

Figure 11. For 4 baffles (a) Average air outlet temperatures over 1000 iterations. (b) Average water outlet temperatures over 1000 iterations. (c) Average water inlet pressures over 1000 iterations. (d) Average air inlet pressures over 1000 iterations

It is observed that the final air temperature at the outlet is 319.42 K. This is only a 0.39 K (°C) difference from 2 baffles. Therefore, it is apparent that the number of optimum baffles is 2 and any more addition of baffles does not make any significant difference to the final air temperature and the overall heat exchanger efficiency.

Table 2. The outlet temperatures and inlet pressures for zero, two and four baffles placed inside the casing

Number of Baffles	Outlet Air Temperature (K)	Outlet Water Temperature (K)	Inlet Water Pressure (Pa)	Inlet Air Pressure (Pa)
0	338.92	284.27	99350.90	101389.62
2	319.81	284.14	99351.11	101403.80
4	319.42	284.63	99351.60	101430.75

Air has already reached its final possible temperature of 319.81 K for 2 baffles and so increasing the amount of mixing or turbulence will not cause any significant temperature change. Another reason is due to the non-linear relationship between heat-exchanger effectiveness (ε) and number of transfer units (NTU). The baffles are also transfer units.

$$\varepsilon = \frac{1 - \exp[-NTU(1 - C_r)]}{1 - C_r \exp[-NTU(1 - C_r)]}$$

According to the following equation, heat-exchanger effectiveness (ε) reaches a point of diminishing returns as ε reaches the value of 1 for increasing NTU and this shows why the change in temperature was not significant (Bergman et al., 2011, p. 689).

2.2. Quantity and geometry of tubes

Another parameter that can be altered is the diameter of the tubes and their quantity to see their effect on the heat transfer and Fluid dynamics. To do this, the diameter of the pipes is changed from 34 mm to 25.5 mm and 2 additional tubes are added. In the original geometry when obtaining streamlines, recirculation zones were observed near the entrance of the water pipes as shown in Figure 12. Since there were recirculation zones, the new tubes were added on those areas so as to reduce the recirculation and introduce pathways for the water to flow through smoothly. The size of the tubes is chosen through CAD design on Fusion 360. Since the original tube diameter was too big to fit 7 pipes, the diameter was reduced to 25.5 mm.

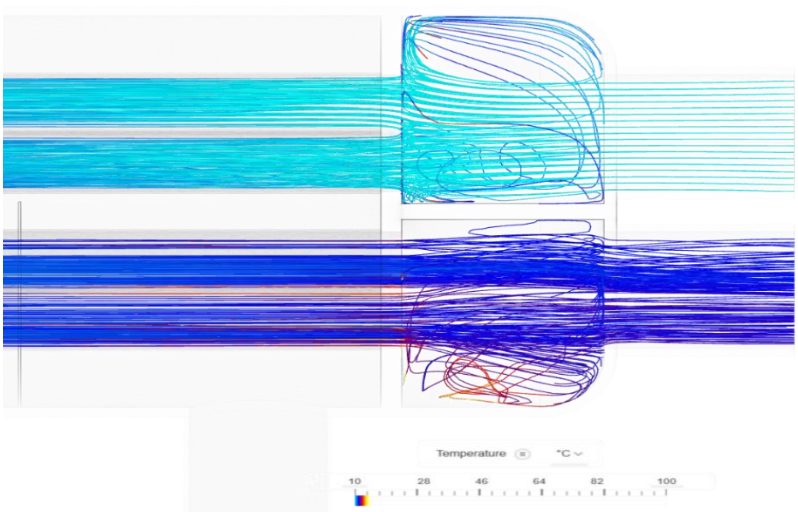
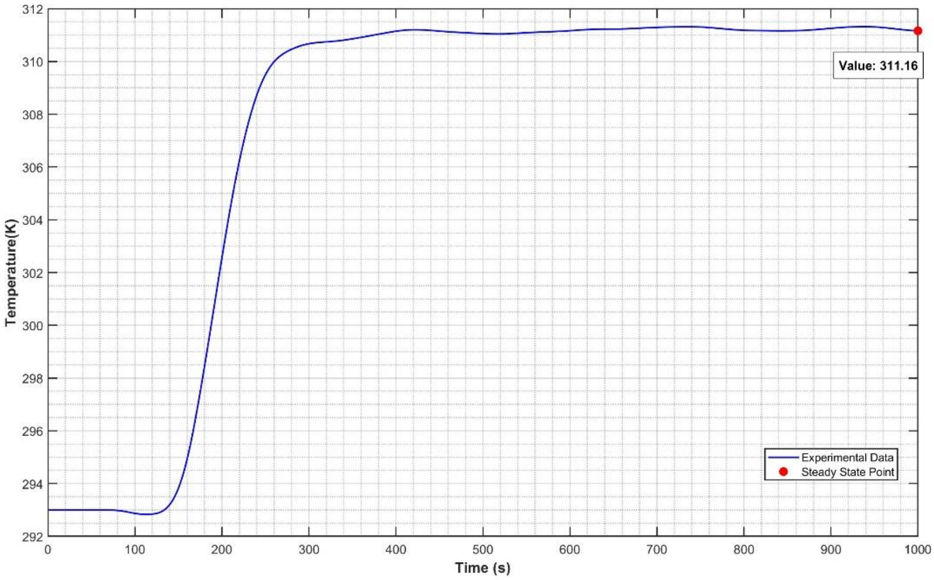
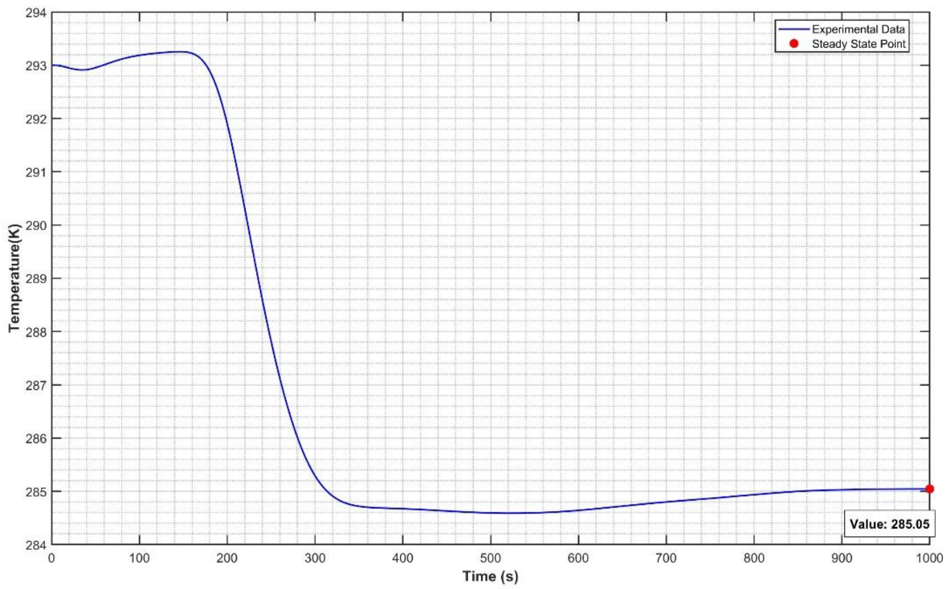


Figure 12. Recirculation zones forming at the corner of the entrance of the tubes.

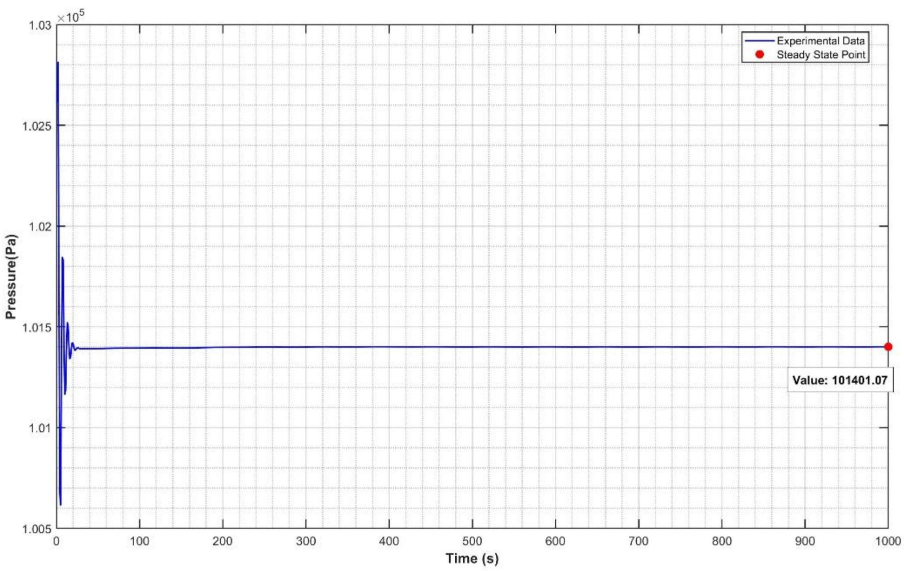
A simulation with this new geometry is performed, with the same boundary conditions and 2 baffles.



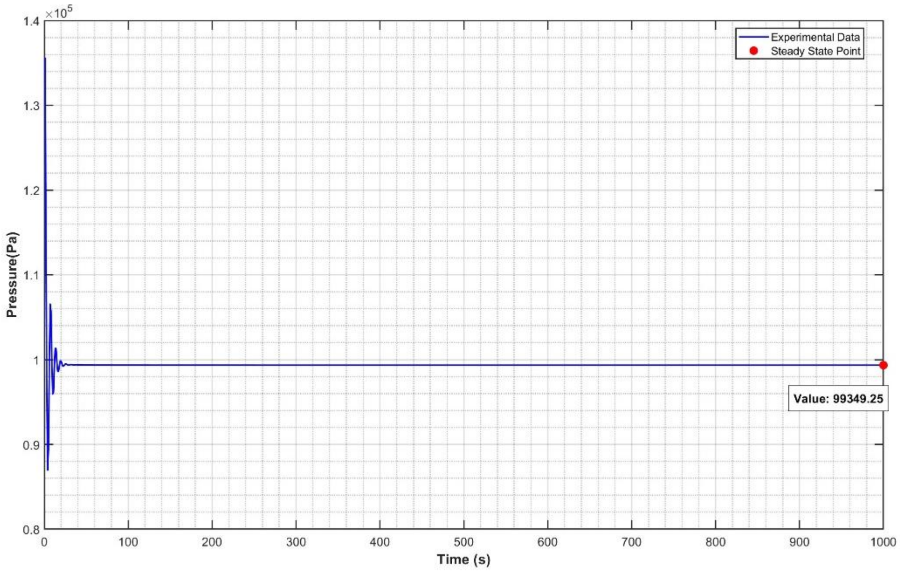
(a)



(b)



(c)



(d)

Figure 13. 7 tubes and 25.5 mm diameter (a) Average air outlet temperatures over 1000 iterations. (b) Average water outlet temperatures over 1000 iterations. (c) Average air inlet pressures over 1000 iterations. (d) Average water inlet pressures over 1000 iterations

Increasing the number of pipes caused a significant decrease in the outlet air temperature. The temperature for 2 baffles with 5 tubes at a tube diameter of 34 mm was 319.81 K and the temperature for outlet air for same number of baffles, 7 tubes and 25.5 tube diameter is 311.16 K. That is an 8.65 K (°C) difference. This is due to the increase in the surface area of the tubes and the increase in the amount of water flowing through the tubes.

2.3. Symmetry

Alternatively, the CHT analysis was performed using a half-geometry model by applying symmetry conditions at the center plane. This technique is valid because both the physical geometry and the boundary conditions for the air and water flow paths are symmetrical. By treating the center plane as a symmetry boundary, the simulation achieves the same results without compromising accuracy.

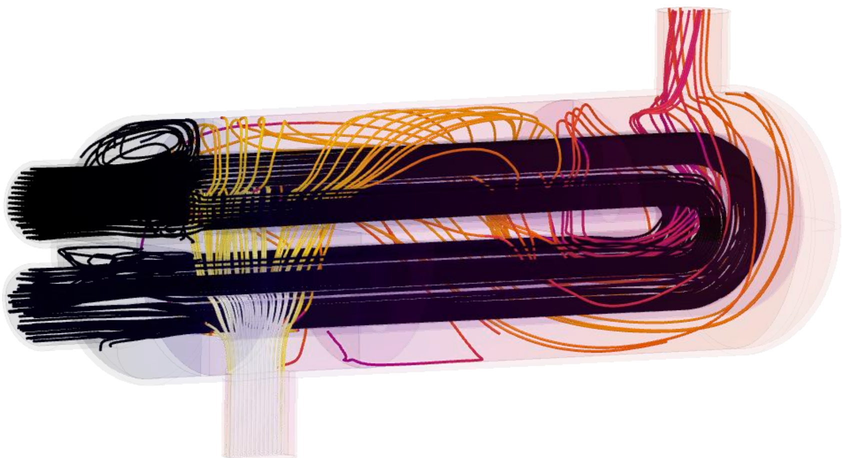
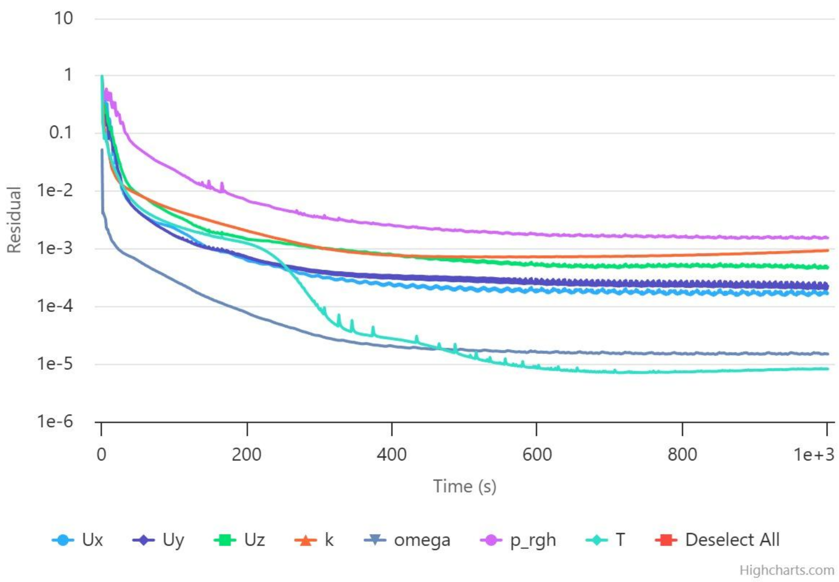
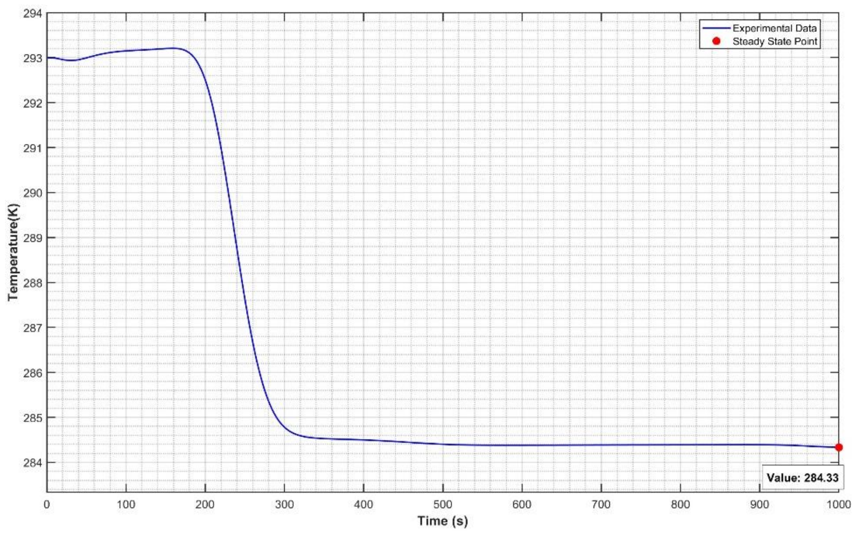


Figure 14. Modelled half geometry with symmetry plane.

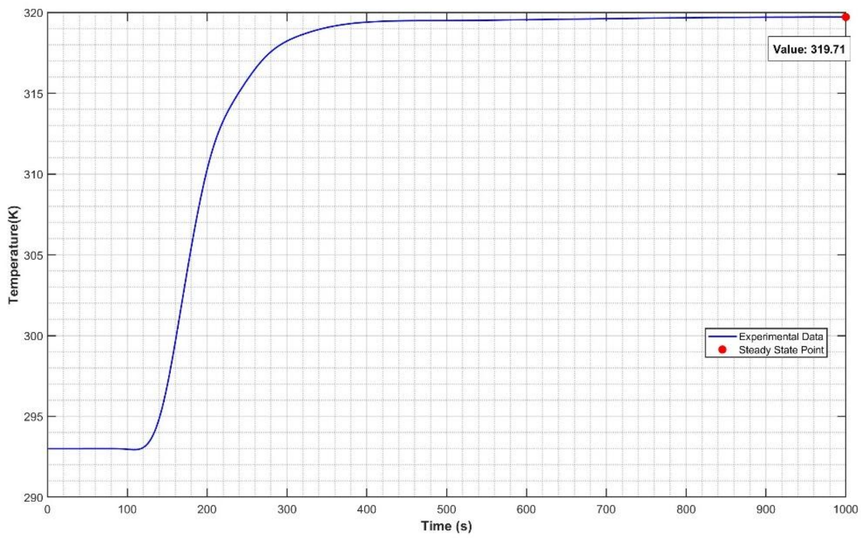
The residual plots, air outlet temperature and water outlet temperature can also be obtained for half geometry to further prove our geometry.



(a)



(b)



(c)

Figure 15. (a) Residual plots of the half geometry for 1000 iterations. (b) Water outlet temperature for the half geometry for 1000 iterations. (c) Air outlet temperature for 1000 iterations.

2.4. Heat Transfer

The heat transfer rate between water and air can be calculated using the inlet and outlet temperatures for each respectively. To do that, we need the air and water flow areas and the flow rates.

$$A_{air} = \frac{\pi}{4} D_{inlet}^2 \tag{1}$$

$$A_{air} = \frac{\pi}{4}(0.025)^2 = 0.001963\ m^2$$

$$\dot{m} = \rho_{\mathrm{air}} * A_{air} * V_{air} \tag{2}$$

$$\dot{m_{air}} = 0.946 \times 0.001963 \times 10 = 0.0186\ \mathrm{kg/s}$$

$$A_{water} = N_{pipes} \times \frac{\pi}{4} D_{pipe}^2 \tag{3}$$

$$5 \times \frac{\pi}{4}(0.034)^2 = 0.004539\ \mathrm{m}^2$$

$$\dot{m} = \rho_{water} * A_{water} * V_{water} \tag{4}$$

$$m_{\dot{w}ater} = 999.7 \times 0.004539 \times 0.05 = 0.2269\ \mathrm{kg/s}$$

Now heat transfer can be calculated,

$$\dot{Q}_{air} = \dot{m}_{air} * c_{p,air} * \left(T_{in,air} - T_{out,air}\right) \tag{5}$$

$$\dot{Q}_{air} = 0.0186 \times 1009 \times (373.15 - 319.81) = 1001\ \mathrm{W}$$

$$\dot{Q}_{water} = \dot{m}_{water} * c_{p,water} * \left(T_{out,water} - T_{in,water}\right) \tag{6}$$

$$\dot{Q}_{water} = 0.2269 \times 4193 \times (284.14 - 283.15) = 942\ \mathrm{W}$$

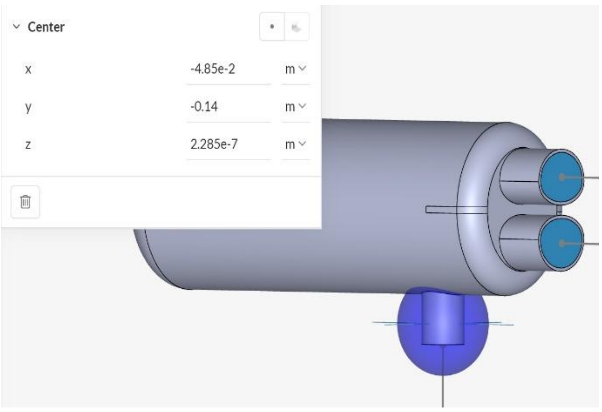
The heat transfer for the casing can be found using the following formula:

$$\dot{Q}_{loss} = \dot{Q}_{air} - \dot{Q}_{water} \tag{7}$$

$$\dot{Q}_{loss}\ (\dot{Q}_{casing}) = 1001 - 942 = 59\ \mathrm{W}$$

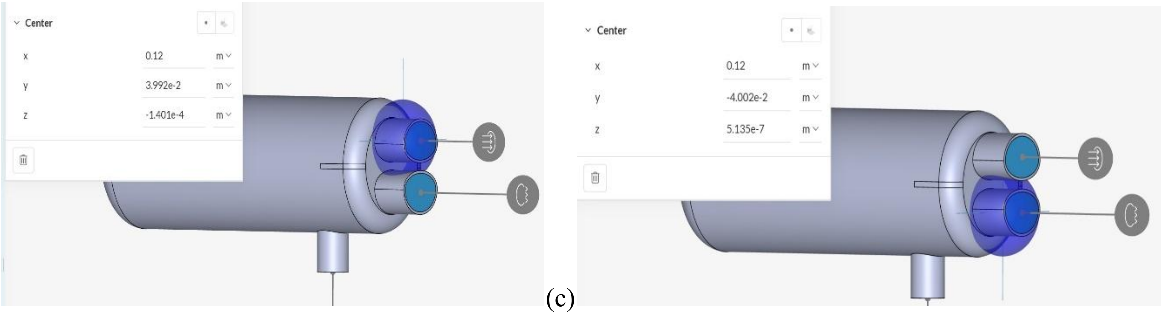
2.5. Residual Plots

The error vs iterations are plotted to understand convergence as SimScale uses 1000 iterations without any verification behind the choice. To confirm a simulation is performed with 2000 iterations. To obtain monitoring plots probe points were placed at appropriate points at the water and air inlet and outlets.



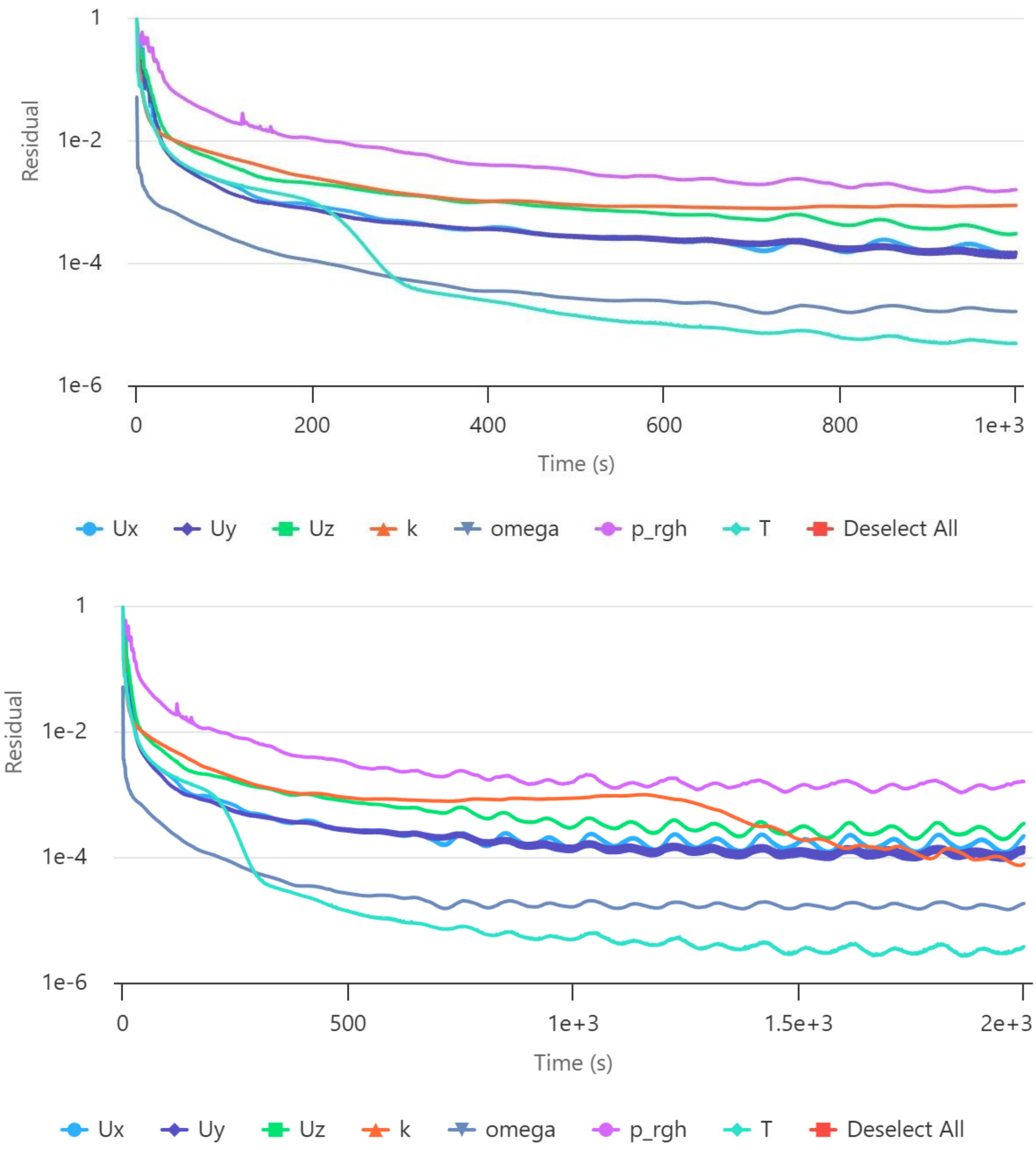
(a)

(b)



(c) (d)

Figure 15. (a) Air inlet probe location. (b) Air outlet probe location. (c) Water inlet probe location. (d) Water outlet probe location.



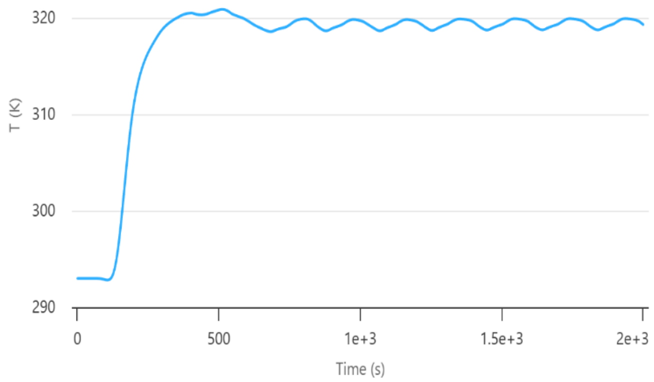
(a)

(b)

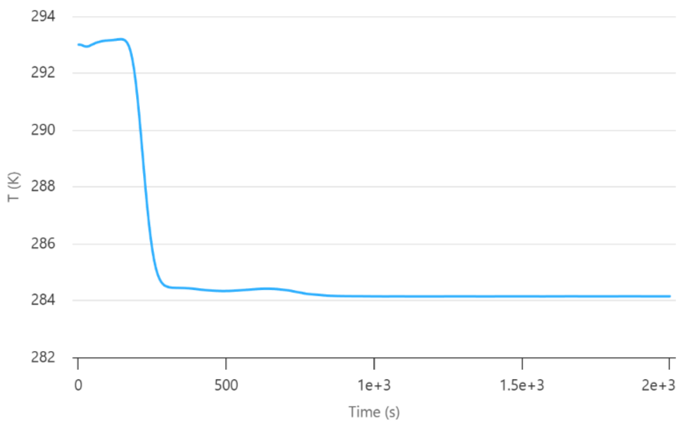
Figure 16. (a) Residual plots for U_x, U_y, U_z, k, ω, T and p vs 1000 iterations performed. (b) Residual plots for U_x, U_y, U_z, k, ω, T and p vs 2000 iterations performed.

Figure 15 is a very clear exhibition of the oscillatory nature of unsteady problems like the one being studied here. It can be concluded from the figure that stopping the solution at 1000 iterations is enough since the residuals simply start oscillating from the 800 iterations mark and continue in this manner till the 2000 iterations mark.

This can also be confirmed through the average air and water outlet temperatures.



(a)



(b)

Figure 17. (a) Average air temperature at the outlet oscillating around 319 K after 700 iterations. (b) Average water temperature at the outlet staying constant around 284 K after 700 iterations.

3. Conclusion

The overall object of this study was to perform a fully fledged conjugate heat transfer CFD analysis using the SimScale Software. Initially, the analysis was performed simply using the parameters and geometry provided by SimScale, however, to properly verify and understand the working of the U tube Heat Exchanger several parameters were monitored and modified.

In SimScale the number of iterations is set to 1000 by default and the mesh refinement to 5. To properly verify the use of these parameters, the number of iterations was increased to 2000 and the effects were analyzed. This led to confirmation that 1000 iterations are sufficient for the CHT U-Tube Heat Exchanger case. Additionally, a mesh independence study was performed by using several meshes ranging from 997k cells to 19.2 million cells. This also confirmed the use of refinement level 5 which corresponded to 1.6 million cells.

The effect of baffles on the heat exchanger was investigated by using different heat exchanger geometries with no baffles, two baffles and 4 baffles. Using the temperature at the air outlet calculations were performed to relate the baffles to the rate of heat transferred. This investigation led to confirmation that the two baffles are the most viable choice in terms of heat exchanger efficiency.

Furthermore, the number of pipes was changed as recirculation zones were observed at the pipe entrances located behind the main water inlet. This led to a big improvement compared to the change in the number of baffles. The main reason for this is the massively increased surface area of heat transfer which increasing the number of baffles did not achieve. From the monitoring plots obtained this is verified and the number of recirculation zones significantly reduced. This shows that geometric modifications to the water flow were more significant than modifications to the air flow aimed at increasing turbulence.

Finally, to reduce the overall computing cost and time of the simulation a symmetry condition was imposed on the x-y plane in the middle of the geometry. The results were investigated and the overall results corroborated the use of the symmetry plane. This result is significant as it cuts down on the computational cost and more refined meshes can therefore be utilized to more accurately capture the nature of the flow.

References

Bergman, T. L., Lavine, A. S., Incropera, F. P., & DeWitt, D. P. (2011). *Fundamentals of heat and mass transfer* (7th ed.). John Wiley & Sons.

SimScale. (2025, November 19). *Tutorial: Conjugate heat transfer in a U-tube heat exchanger*. https://www.simscale.com/docs/tutorials/cht-heat-exchanger/