University of Tehran
College of Engineering
School of Mechanical Engineering
Heat Transfer Course

Heat Transfer Project

Cross Flow Heat Exchanger: Tube Banks

Professor:

Dr. Ehsan Houshfar

Authors:

Masoume | Taherkhany | 810698269
Mohammad | Montazeri | 810699269

Deadline:

2023/6/29

Table of Contents

Table of Figures

List of Tables

Abstract

A heat exchanger is a system used to transfer heat between a source and a working fluid. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment.

Cross flow heat exchangers, as one of the most used types of these systems, are called the type of heat exchangers which have 90-degree angles for the direction of the fluids. They can be categorized as plate, shell and tube, one-phase, and multi-phase exchangers. Cross flow heat exchangers are used in cooling and ventilation systems where heat needs to be transferred from one airflow to another. They are made up of thin metal panels, usually aluminum, and thermal energy is exchanged through the panel. A traditional cross flow heat exchanger has a square cross-section.

Crossflow over tube banks is commonly encountered in practice in heat transfer equipment such as the condensers and evaporators of power plants, refrigerators, and air conditioners. In such equipment, one fluid moves through the tubes while the other moves over the tubes in a perpendicular direction. The tubes in a tube bank are usually arranged either in-line or staggered in the direction of flow.

In this project, a cross flow heat exchanger with staggered arrangement is being modeled by ANSYS Fluent once 2-dimensionally and once 3D. Then the reliability of their results is investigated via theorical and experimental equations. The following figure shows this heat exchanger inside an evaporator; so that the refrigerant fluid with a constant temperature is turned into saturated vapor and the air passing through the pipes is used for cooling and air conditioning.

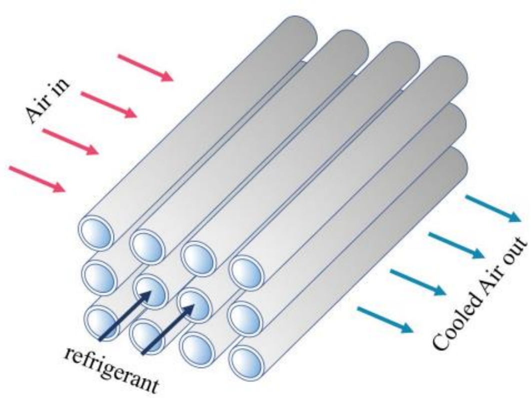


Figure 1: Cross flow over tube banks

Steps and Processes

The first step of this project was modeling the evaporator and designing it in ANSYS geometry. The 2D model of the system was given as figure 2. In this model all of the dimensions and coordinates are given, in addition to the air temperature and velocity at the inlet and also the constant temperature of refrigerant inside tubes.

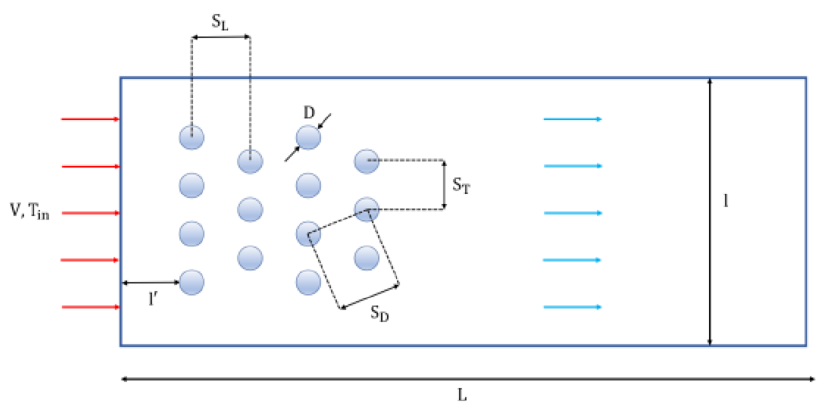


Figure 2: the 2D model used for the heat exchanger

This geometry was designed in the Design-Modeler environment using simple circle and rectangle commands. Then it was imported into Meshing environment. In the meshing procedure, at first an automatic mesh was generated by *Quadrilateral Dominant* method. Then the size of boundary elements near top and bottom plates were scaled down to help capturing the boundary layers. This procedure was also repeated around tubes via *inflation* command as well. For this matter, the thickness of the first layer was set to 1.2 mm and a 1.2 factor of growth rate was dedicated to the inflation method.

Details of "Inflation" - Inflation

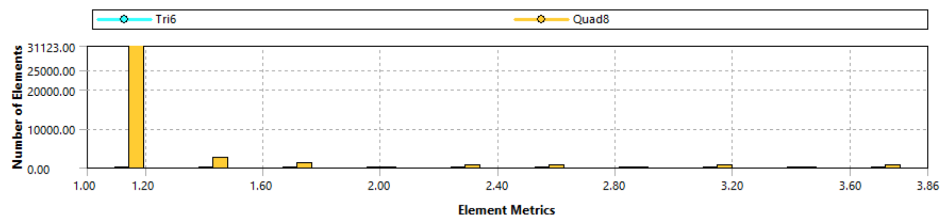
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Edges
Inflation Option	First Layer Thickness
First Layer Height	1.2e-003 m
Maximum Layers	12
Growth Rate	1.2
Inflation Algorithm	Pre

Figure 3: inflation settings

Then, after checking the quality of the mesh using some of its important metrics (see figure 4), it was assured that the created mesh was adequately fine.

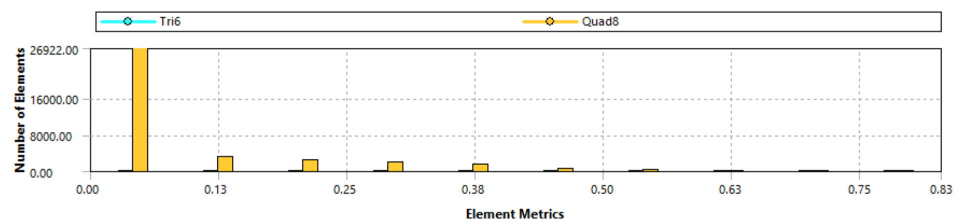
Table 1: Mesh quality control factors

Aspect ratio



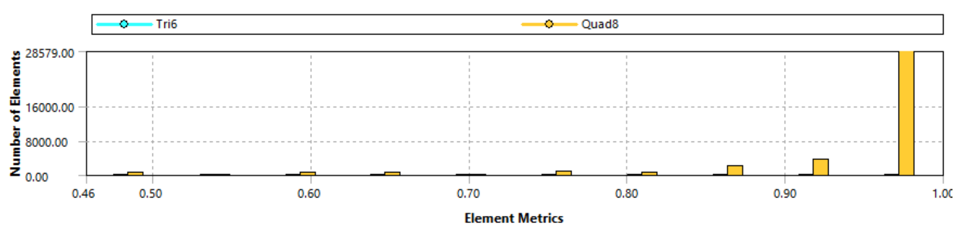
Mesh Metric	Aspect Ratio
Min	1.
Max	3.8638
Average	1.1884
Standard Deviation	0.44756

Skewness



Mesh Metric	Skewness
Min	1.3057e-010
Max	0.82968
Average	7.1779e-002
Standard Deviation	0.11816

Element quality



Mesh Metric	Element Quality
Min	0.45744
Max	1.
Average	0.95713
Standard Deviation	9.1997e-002

Finally, appropriate names were assigned to every boundary, so that they would be recognizable in setup procedures where specific boundary conditions are allocated to them.

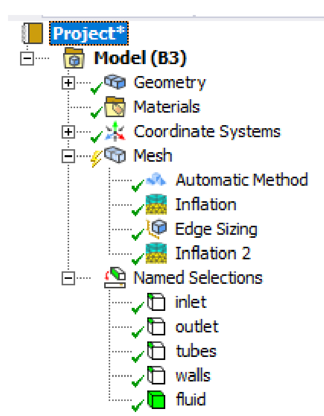


Figure 4: Tree diagram, showing the mesh methods and named selections

For this mesh, the element size was set to 0.006 m at the first try. To ensure *mesh independency*, in next runs the element size of mesh was reduced to 4.5 and then to 3 millimeters. More details about this topic are explained in «mesh independence study» section.

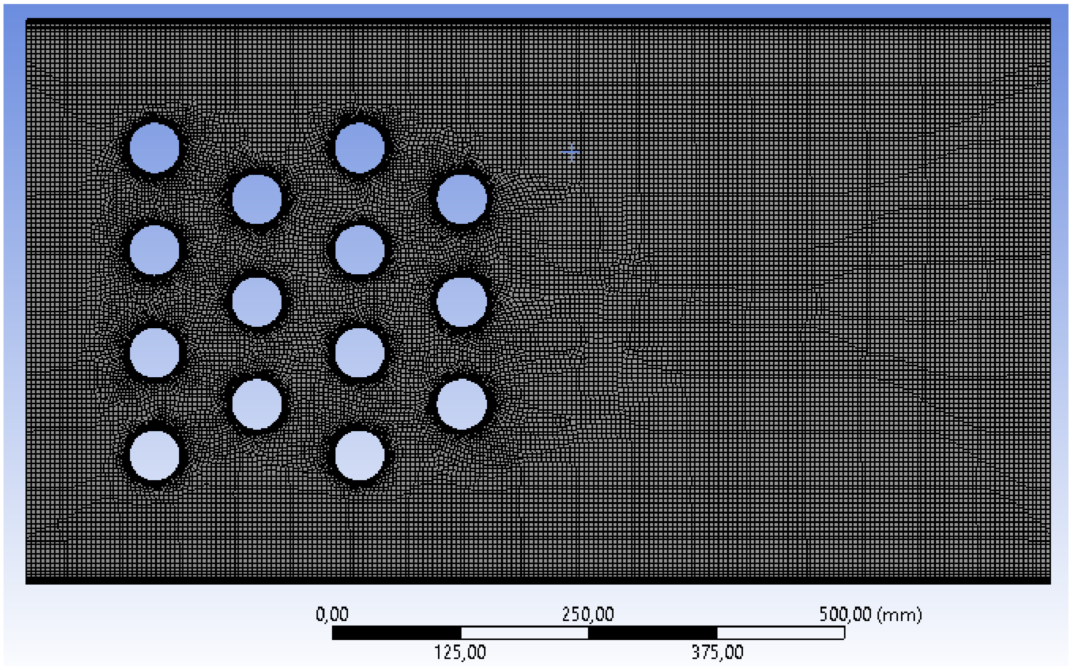


Figure 5: Final mesh

Next step after meshing is setup section which is done in Fluent environment. In this step, we choose to solve the system in planar, pressure based, and steady state. Then we turn on Energy model and set Viscous solver to *SST k-omega*. After defining the materials (air for fluid and copper for the tubes), we specify desired boundary conditions to the named selections we defined in meshing environment. These B.C.s are:

Table 2: Boundary conditions based on the project description's demand

Surface	Boundary Condition	Setting
Inlet	Velocity inlet	Velocity = 0.5 m/s
Outlet	Pressure outlet, no reverse flow	Gauge pressure = 0
Walls	Insulated walls	Heat flux = 0
Tubes	Isothermal	Temperature = 275 K

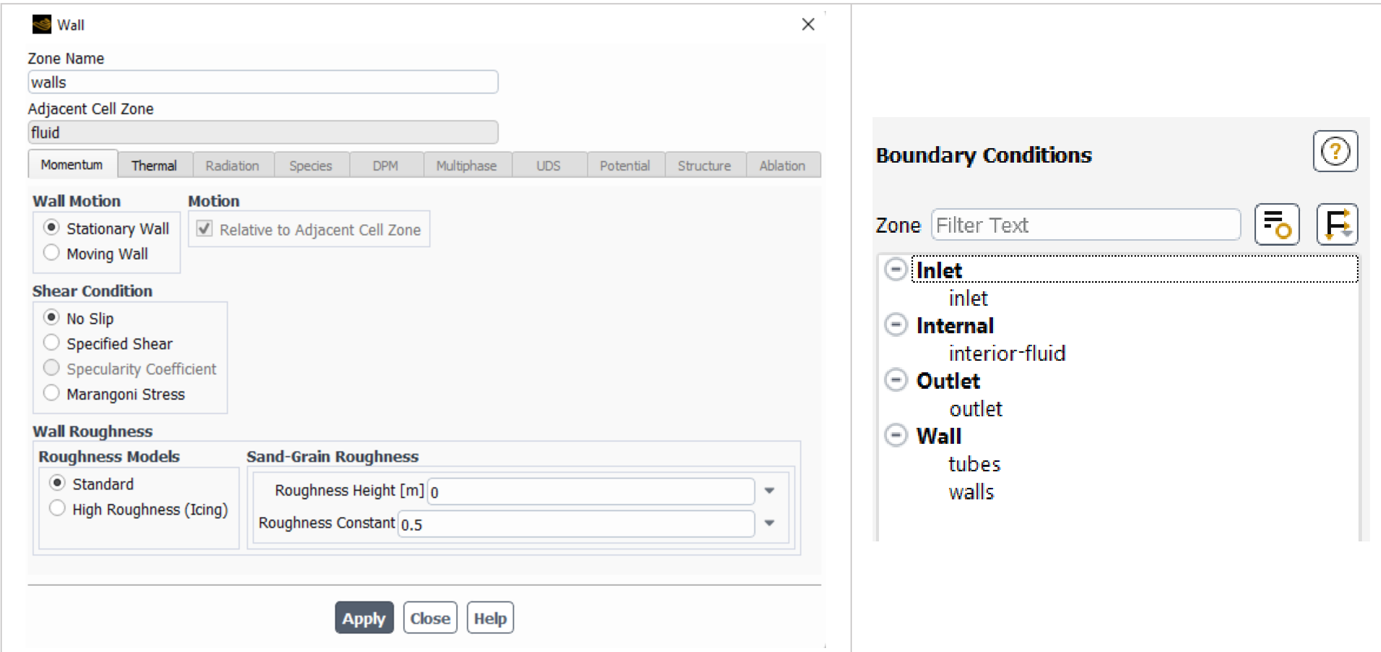


Figure 6: Overview of B.C.s and an example of adjusting wall B.C.

After boundary conditions, there are a few steps to adjust Solution via *Methods*, *Controls* and *Monitors* sections. The most important command, here, is setting residuals to less than 10^{-6} for all 6 equations. These governing equations of heat exchangers' analysis are:

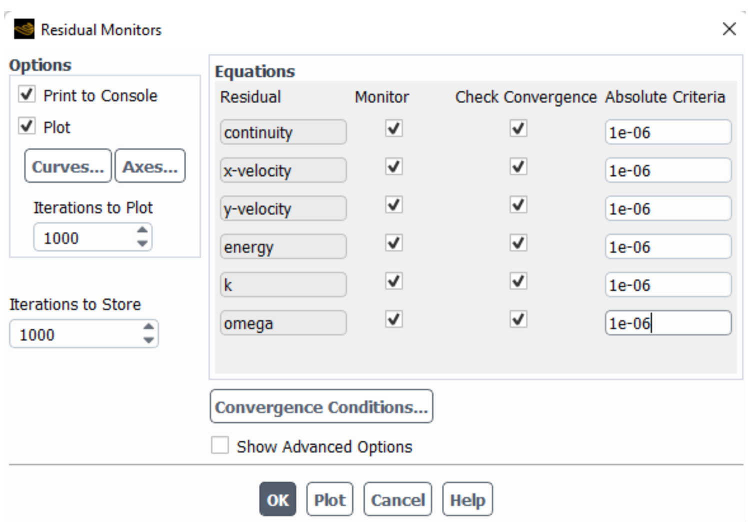


Figure 7: Residuals and equations

Finally, we initialize the setup and then run calculations with 1200 iterations. As a result, it's observed that all equations converge gradually and all of their residuals finally reach less than desired value (10^{-6}); so, it's assured that the obtained results are valid.

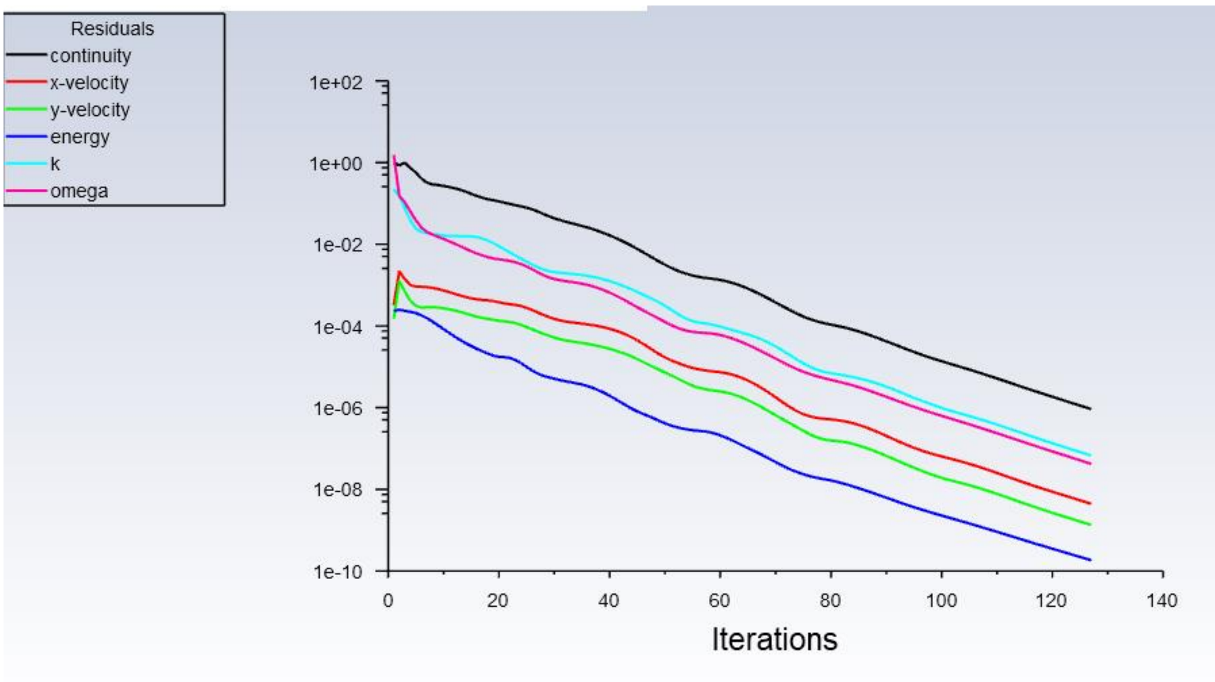


Figure 8: The convergence history for the governing equations

Results and Analysis

a) Contours

Now, with respect to the residual's plot (figure 8), we can plot the required contours as below:

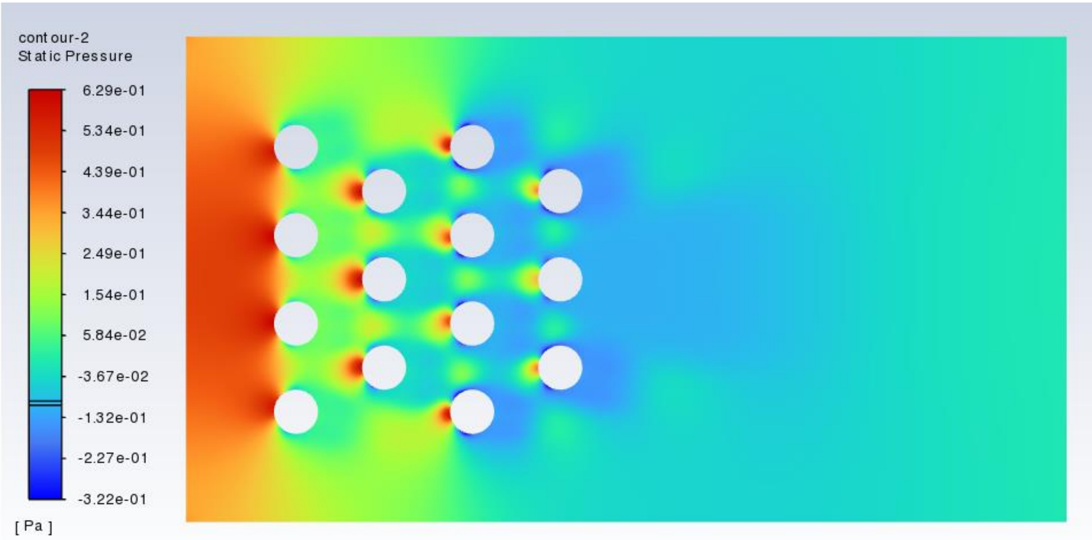


Figure 9: Static Pressure Contour

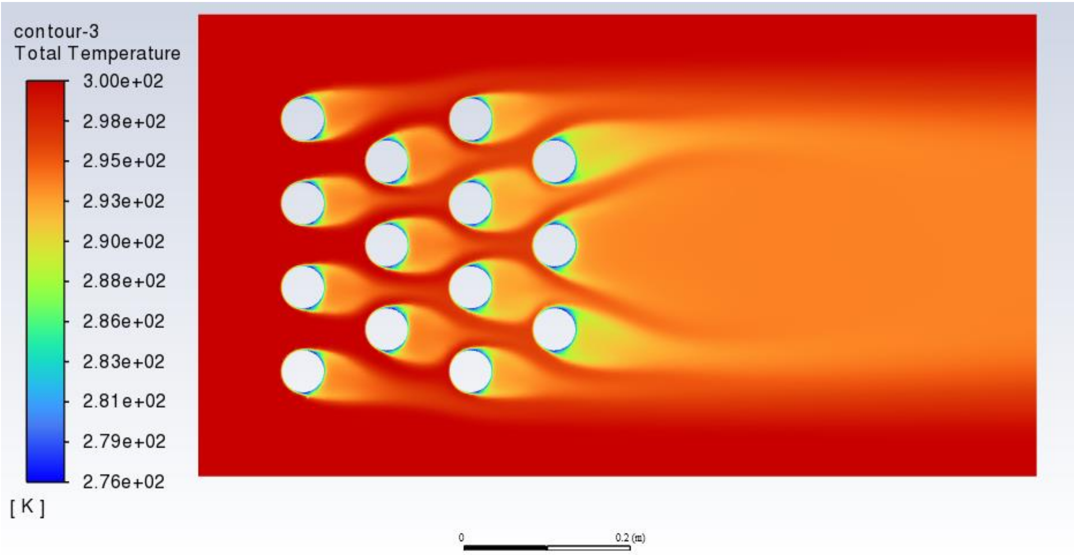


Figure 10: Temperature Contour

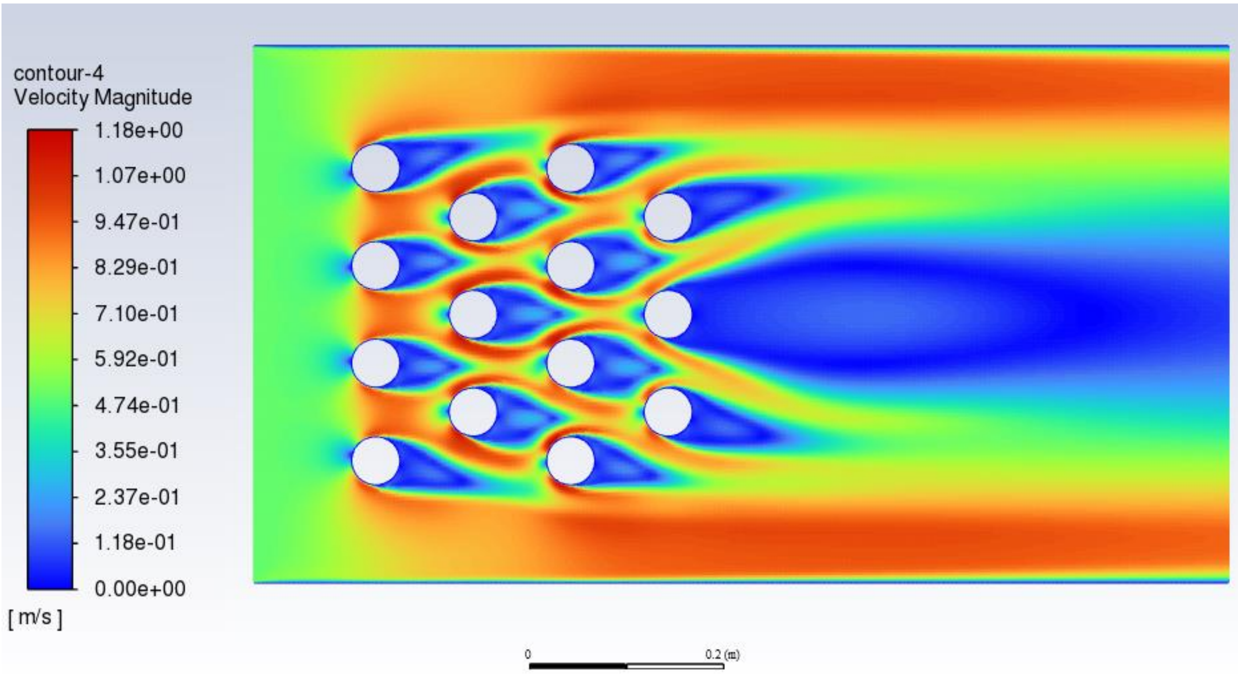


Figure 11: Velocity Contour

According to these contours, we can find out some important parameters of this evaporator. For instance, the maximum velocity of air is around 1.18 m/s which happens between second row tubes. Also, due to the huge wake formation behind each tube, it's observed that the velocity is reduced quite suddenly in these wake regions.

Another parameter is the exit temperature of air which is around 294 Kelvin. Since the temperature difference between air and refrigerant was not very much $(300 - 275 = 25K)$, this small temperature reduction in the air is acceptable; especially because only a few tubes were used in the heat exchanger.

In addition, from pressure contour it's concluded that the pressure is reduced in the direction of flow. Since the fluids tend to flow from high pressure towards low pressure regions, this result is just as expected. This phenomenon is justified by the head loss theory that comes from conservation of energy:

$$h_l = \frac{\Delta P}{\rho} = f.\frac{L}{D}.\frac{\bar{V}^2}{2}$$

Also, it's obvious that the pressure gradient before and after every tube is considerably huge; this is the main reason of drag force acting on cylinders inside a flow.

b) Values and Plots

The most important factors of a heat exchanger are its exit temperature, exit velocity, pressure and its total heat transfer. According to the obtained results, the mentioned parameters can be reported as below for the outlet's center point ($x = 1m, y = \frac{0.55}{2} = 0.275m$).

Table 3: Final results

Exit temperature	Exit velocity	Inlet gauge pressure	Total heat transfer
294 K	**0.1 m/s**	**0.5 Pa**	**873.5 W/m**

Note that since the flow is evacuated to the atmosphere, the outlet pressure is equal to atmospheric pressure. Thus, the inlet gauge pressure is reported instead. Also, the magnitude of exit velocity is quite small; that's because the center point of exit plane is suited in a large wake region (see figure.11).

The heat transfer is reported in W/m. That's because of the 2D modeling assumption. In fact, it can be put in this way that in this analysis, the width of the heat exchanger is assumed to be 1 meter.

These values are interpreted from the following diagrams that are plotted based on the center line of the heat exchanger. This line is, in fact, the vertical symmetry line of the system and is shown below:

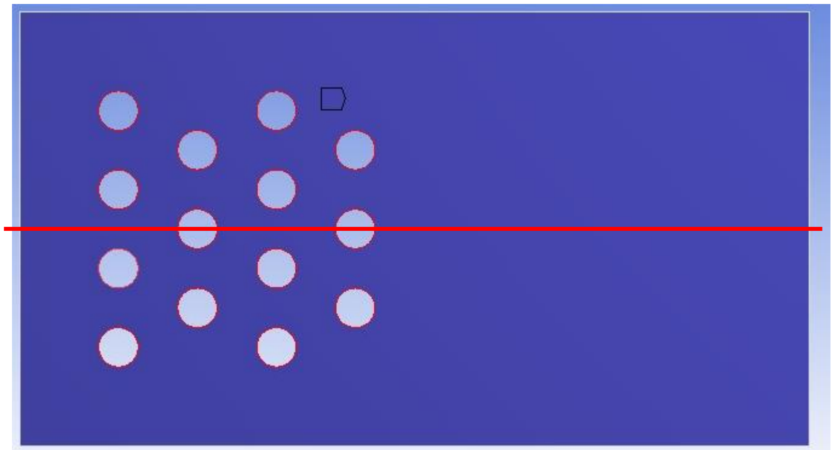


Figure 12: X axis for the following diagrams

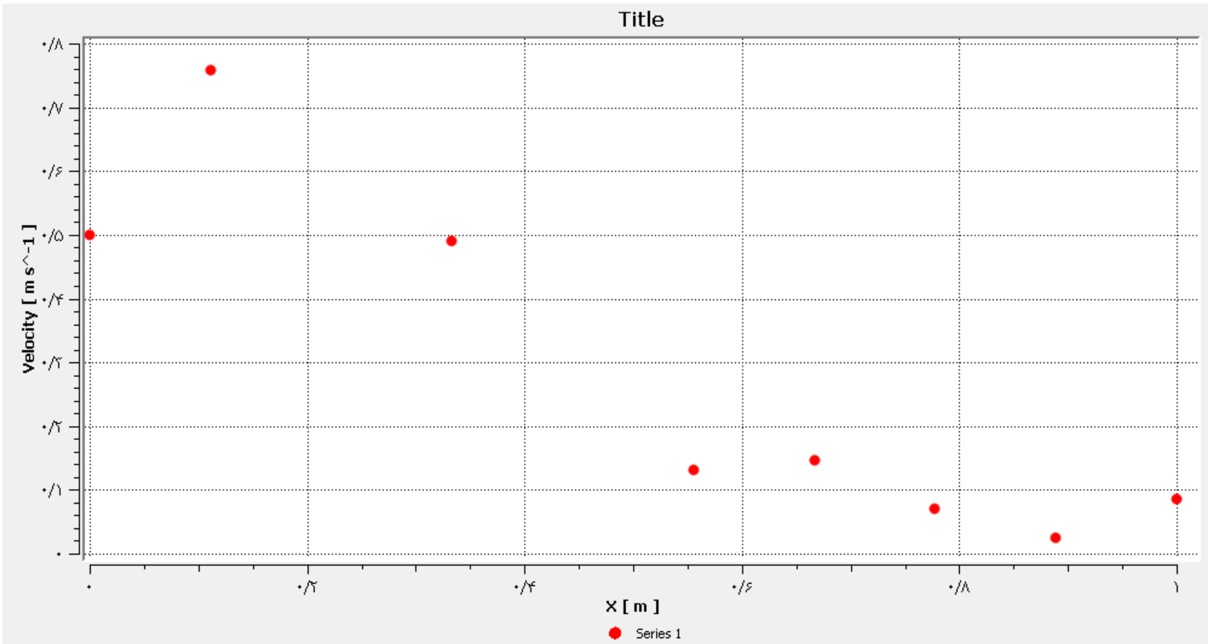


Figure 13: Velocity diagram

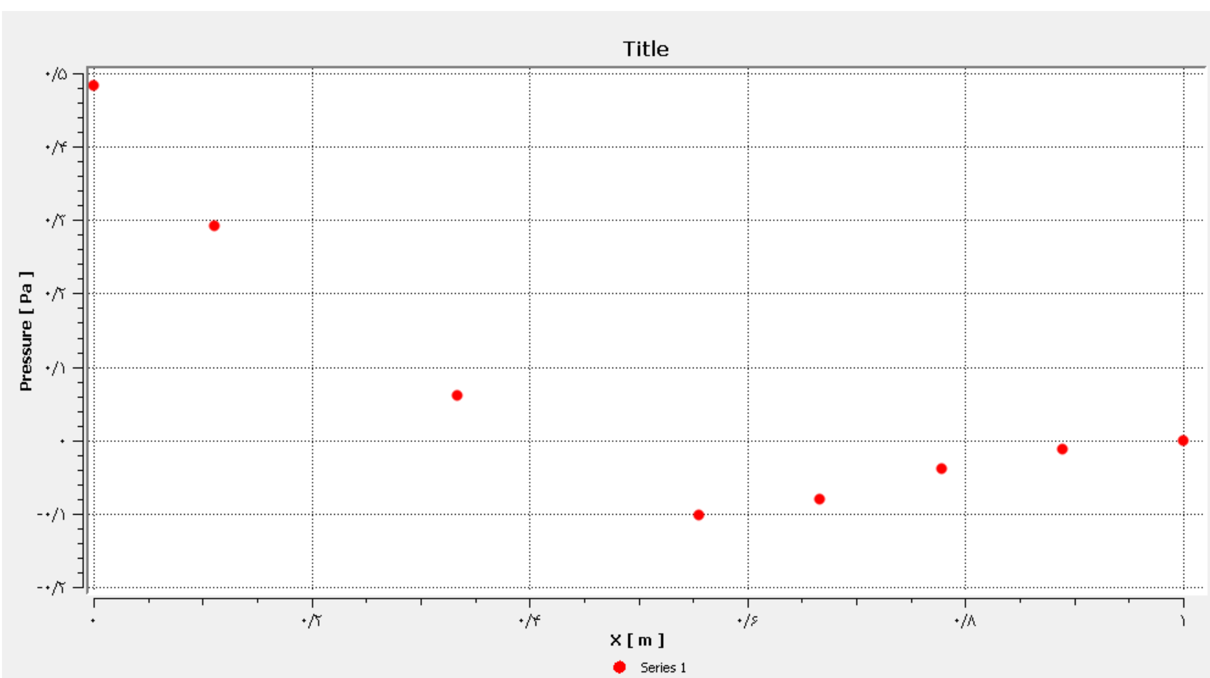


Figure 14: Pressure diagram

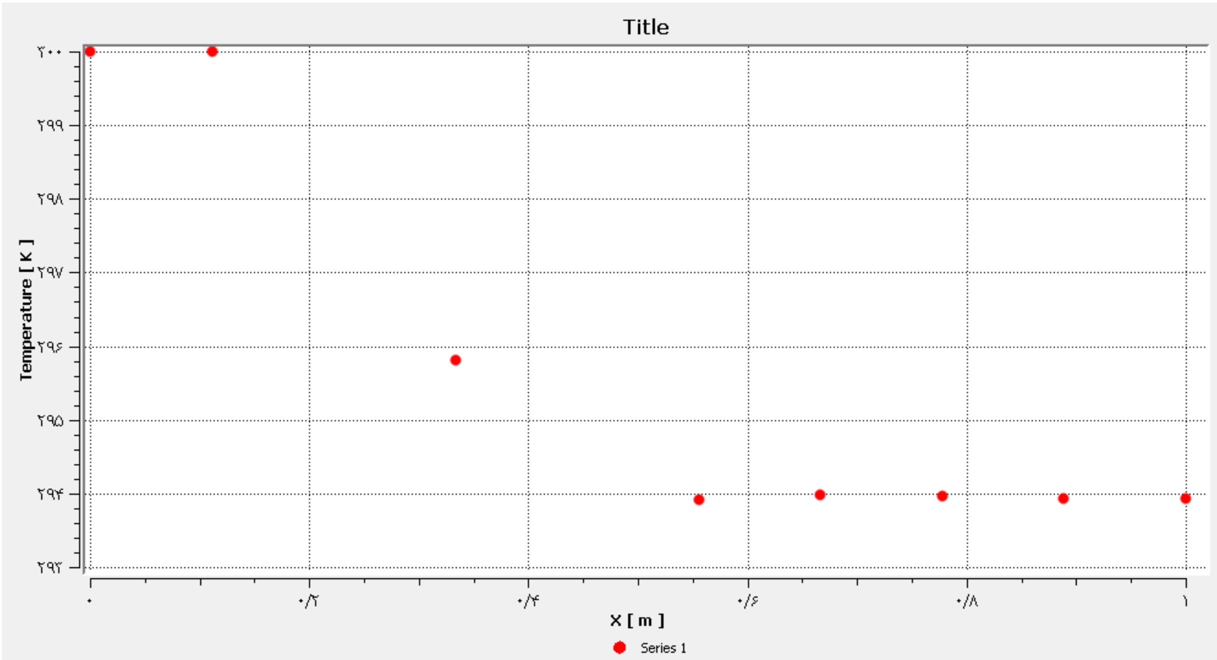


Figure 15: Temperature diagram

The total heat transfer of the tubes is calculated and shown in *reports* section of the Fluent. Since the top and bottom walls are insulated, the only heat transfer happens between the refrigerant and the air; thus, the total heat transfer of the system is equal to that of tubes.

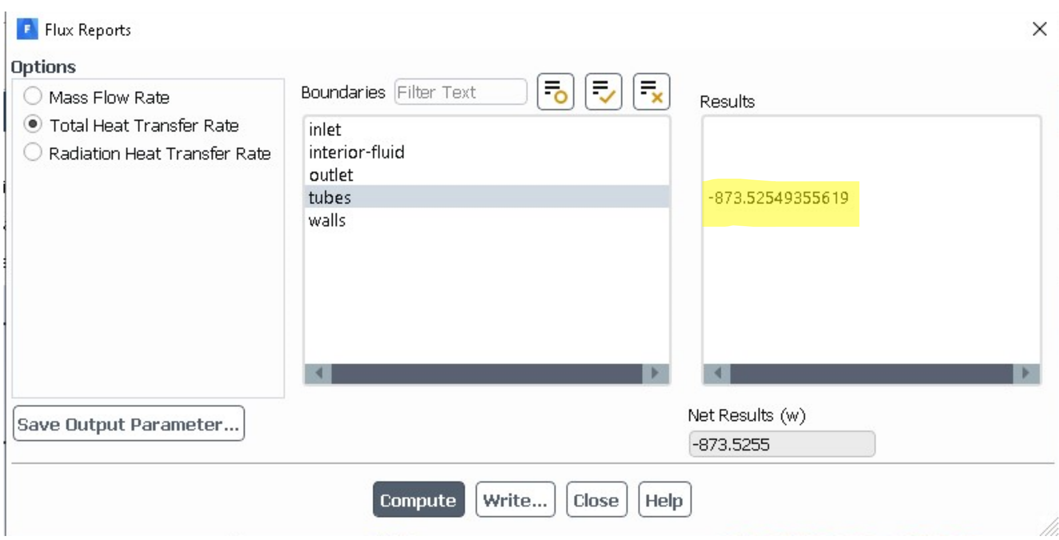


Figure 16: Total heat transfer rate of the system

c) Heat Transfer of Each Tube

To find out the heat transfer of each tube, separately, there has been a modification in the mesh. As a matter of fact, the tubes have been named individually at the final step of meshing. The names of tubes are as:

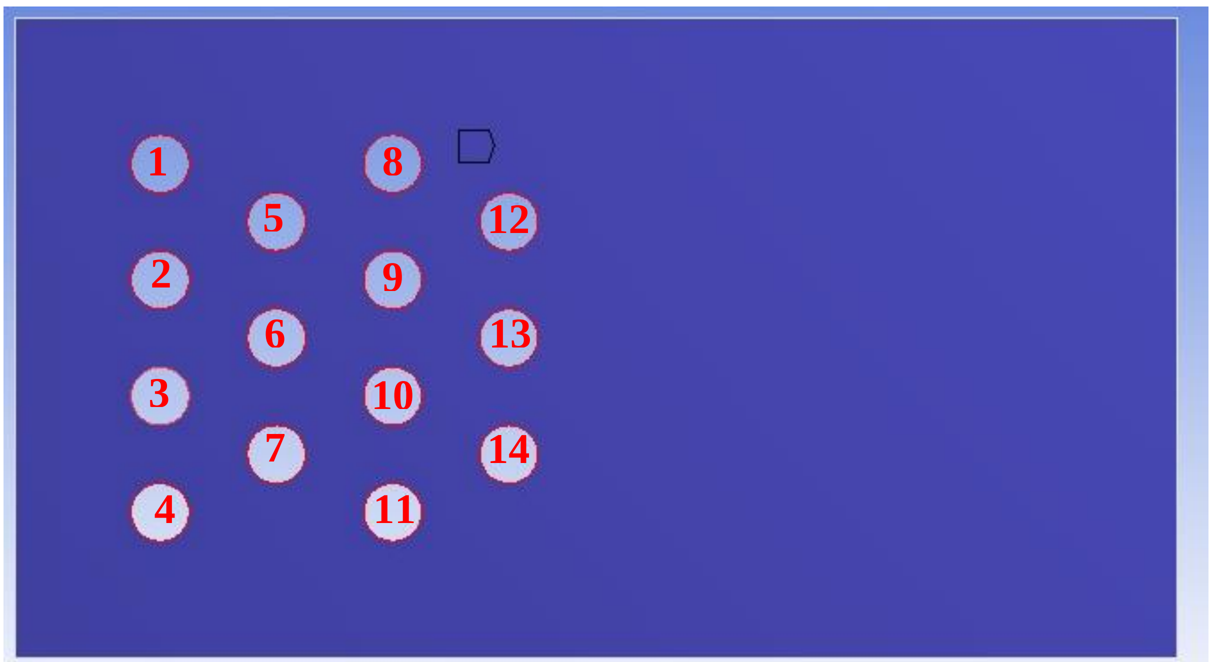


Figure 17: Tube names

With these named selections, after running the analysis once more, we'll get the following results in W/m unit:

Table 4: Heat transfer of each tube separately

Tube 1	Tube 2	Tube 3	Tube 4	Tube 5	Tube 6	Tube 7
61.4	60.7	61.1	61.3	65.6	66.4	65.4
Tube 8	Tube 9	Tube 10	Tube 11	Tube 12	Tube 13	Tube 14
68.8	61.2	60.6	67.9	53.5	51.0	53.9

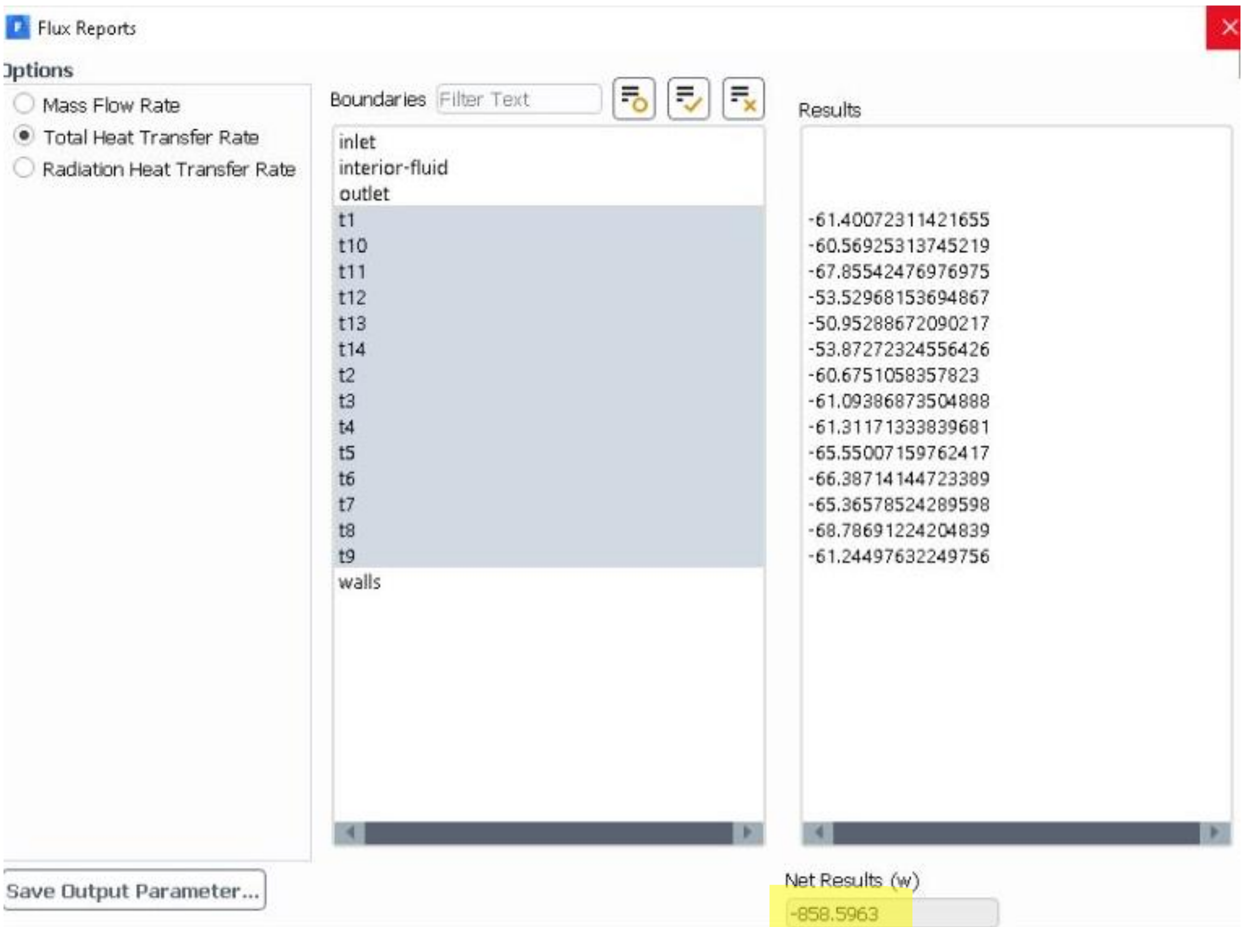


Figure 18: Heat transfer of each tube in Fluent

As expected, the heat transfer of all tubes in total is almost as much as what we had calculated before. (873.5 W/m ≈ 858.6 W/m)

Mesh Independence Study

Mesh convergence study (also known as a mesh independence or mesh independence study) is used to investigate whether simulation results are independent of the underlying mesh or not. It is done by running simulations with different mesh resolutions and checking if the results change. This type of investigation is critical in CFD due to the mechanics of CFD simulations and the underlying mathematics. A mesh sensitivity study consists of running the same simulation using grids with different resolutions and analyzing how much the converged solution changes with each mesh.

In this project, the first analysis was based upon a mesh with 6 millimeters element size. After that, the same processes were repeated with finer mesh networks with 4.5 mm and 3 mm element size. By comparing the results of all these 3 runs, it was observed that the results do not alter much from 4.5 mm to 3 mm. From a practical perspective this means that when the mesh size has reached 4.5mm, the results change very little with additional refinement. Therefore, it is concluded that the mesh independency is achieved at 4.5 mm element size for mesh.

In the following, the obtained results for each mesh are gathered around so that the comparison between them gets easier.

a) Results for 6mm element size mesh

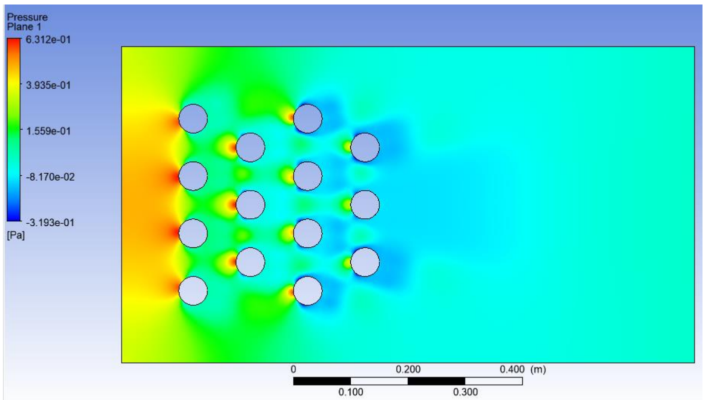


Figure 19: Pressure Contour for 6 mm mesh

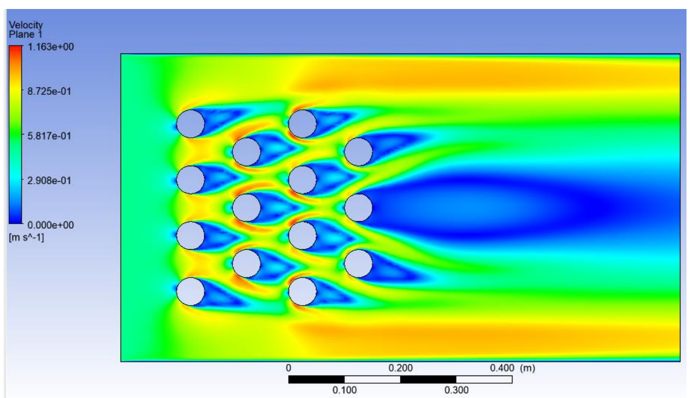


Figure 20: Velocity Contour for 6 mm mesh

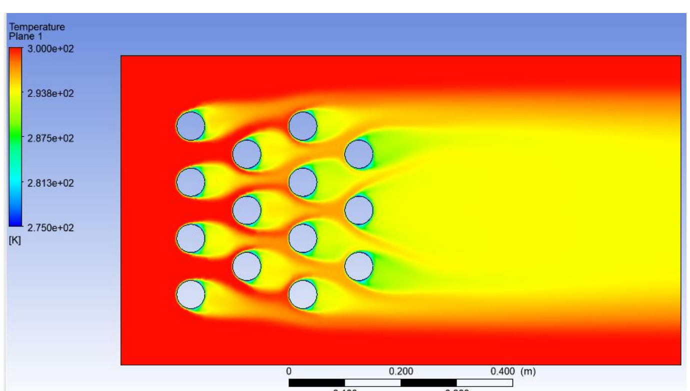


Figure 21: Temperature Contour for 6 mm mesh

b) Results for 4.5mm element size mesh

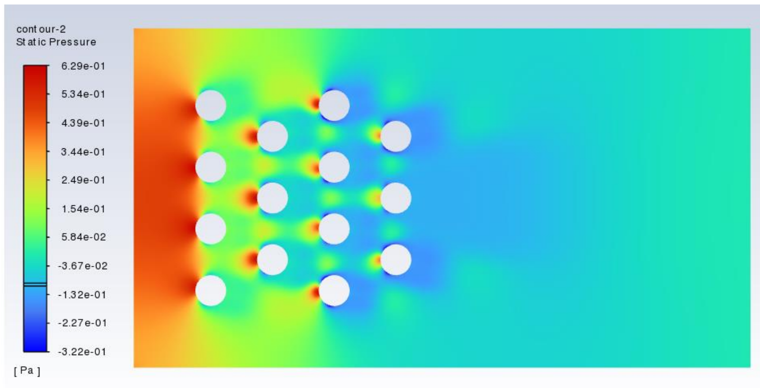


Figure 22: Pressure Contour for 4.5 mm mesh

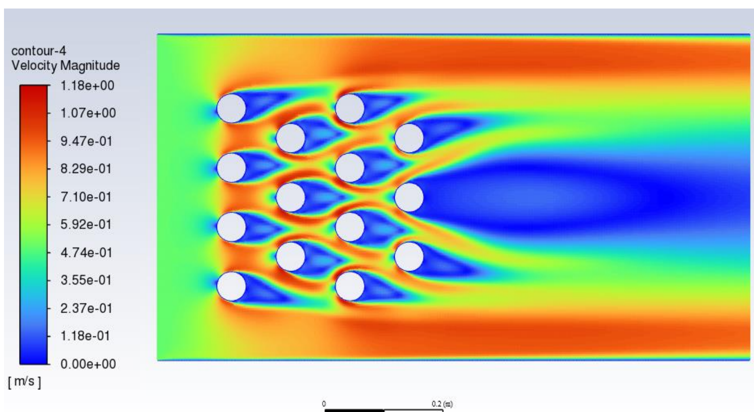


Figure 23: Velocity Contour for 4.5 mm mesh

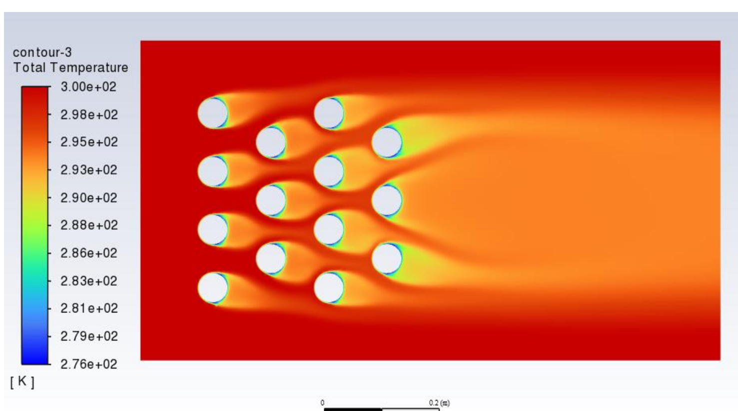


Figure 24: Temperature Contour for 4.5 mm mesh

c) Results for 3mm element size mesh

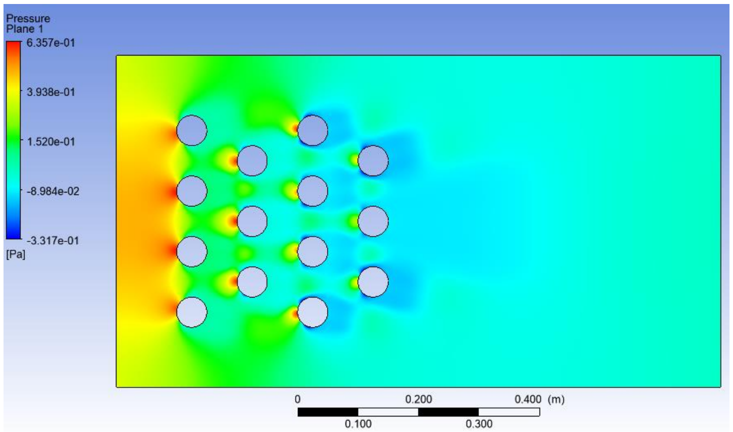


Figure 25: Pressure Contour for 3 mm mesh

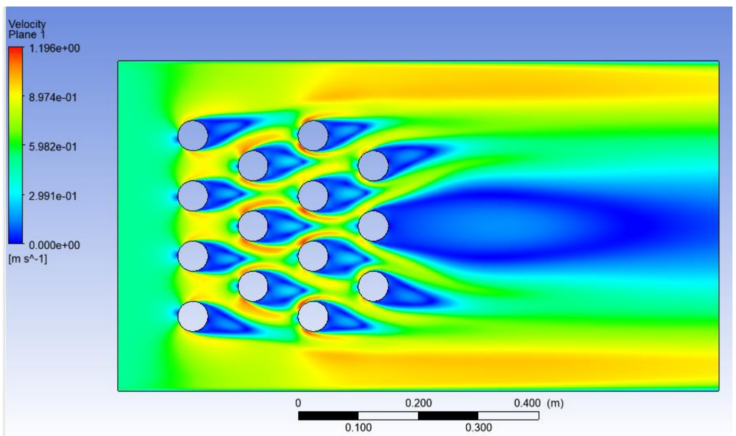


Figure 26: Velocity Contour for 3 mm mesh

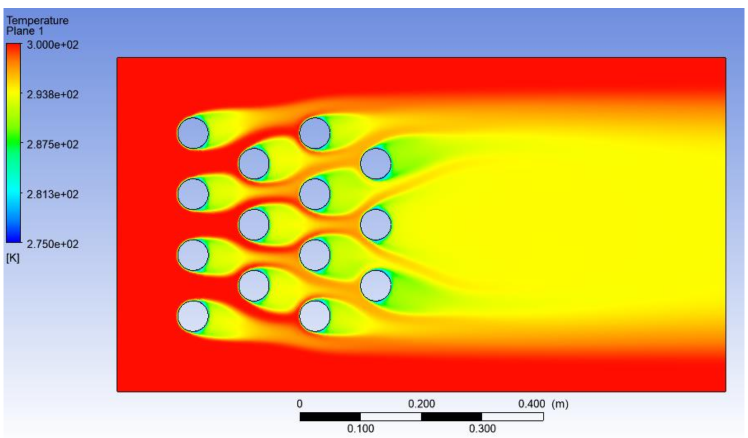


Figure 27: Temperature Contour for 3 mm mesh

d) Final comparison

In conclusion, we interpret that after 4.5 mm mesh size, the results would have less than 5 percent difference, which is a good engineering approximation. For better examination of the convergence of results to the actual solution, and to avoid the complexity in analysis of contour images, we use graphs of the results instead.

Since the purpose of the system is to cool down the air flow, the best parameter to consider can be the temperature. For this purpose, a line is passed through the middle of the heat exchanger, where the symmetry line passes. By plotting the temperature alternations of air while advancing in the direction of the flow, we can achieve a proper diagram to compare 3 mesh sizes. This is done for all three mesh sizes and the results are as below:

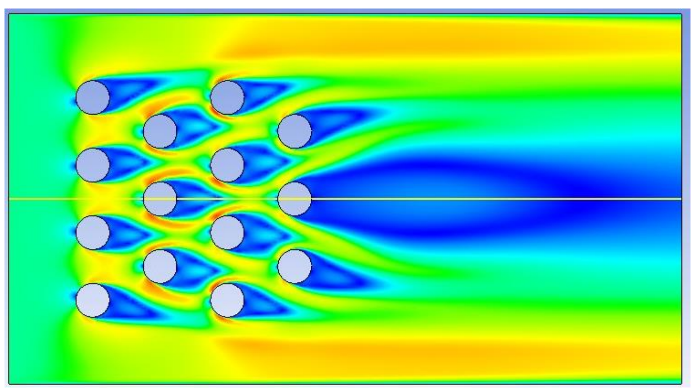
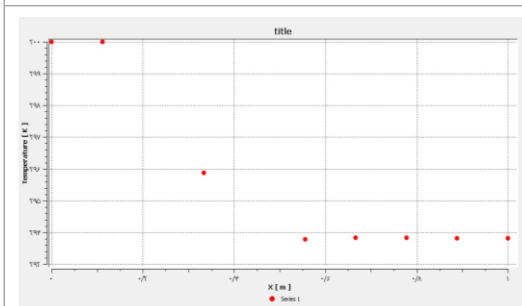
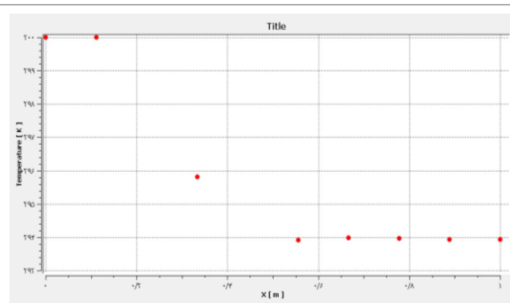
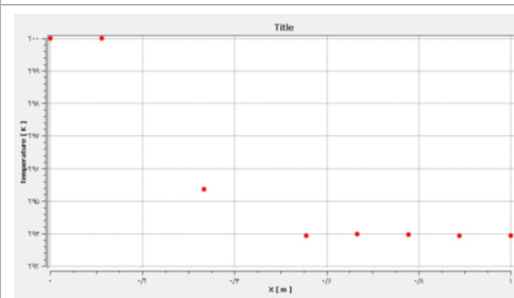


Figure 28: The center line to which the next diagrams belong

Table 5: Comparison between temperature diagrams of 3 different mesh sizes

6 mm	4.5 mm	3 mm
		

As it's visible, the exit temperature is calculated to be 294 K in both 4.5mm and 3mm sizes, which is a little more than the calculated temperature in 6mm mesh. Therefore, the mesh independency is achieved at 4.5 mm element size; thus, refining the mesh would only result in more time consumption and other expenses, while it wouldn't affect the result considerably. So, all of the values reported as final results, are based on 4.5 mm mesh.

Validation of Results with Equations

In this subject it's intended to compare the former modeling results with experimental equations given.

Since $S_D > \frac{S_T+D}{2}$, in order to obtain maximum velocity, we'll have:

$$V_{max} = \frac{S_T}{S_T - D}V = \frac{10}{10-5} \times 0.5 = 1\frac{m}{s}$$

The Nusselt number is also calculated using this equation:

$$\overline{Nu_D} = C_1 C_2 {Re_{D,max}}^m Pr^{0.36} \left(\frac{Pr}{Pr_s}\right)^{0.25}$$

Now we find $Re_{D,max}$ and the $\frac{S_T}{S_L}$ ratio:

$$Re_{D,max} = \frac{\rho V_{max} D}{\mu} = \frac{1.1614 \times 1 \times 0.05}{1.846 \times 10^{-5}} = 3145.72$$

$$\frac{S_T}{S_L} = 1$$

According to the tables in appendix, we'll find the constants as:

$$C_1 = 0.35$$

$$C_2 = 0.89$$

$$m = 0.6$$

So, we'll have:

$$\overline{Nu_D} = 0.35 \times 0.89 \times 3145.72^{0.6} \times 0.707^{0.36} \times \frac{0.707}{0.7135}^{0.25} = 34.4261$$

The average heat transfer coefficient is obtained by:

$$\bar{h} = \overline{Nu_D}\frac{k}{D} = 34.4261 \times \frac{26.3 \times 10^{-3}}{0.05} = 18.1081\ \frac{W}{K.m^2}$$

In order to find the outlet temperature, the following equation is used:

$$T_S - T_O = (T_S - T_I)\exp\left(\frac{-\pi D N \bar{h}}{\rho V N_T S_T c_p}\right) = (275-300)\exp\left(\frac{-\pi \times 0.05 \times 14 \times 18.1081}{1.1614 \times 0.5 \times 3.5 \times 0.1 \times 1007}\right)$$

$T_S - T_O = -20.58\ K$

$T_0 = 295.58\ K$

The logarithmic mean temperature difference (LMTD) is calculated like:

$$\Delta T_{lm} = \frac{(T_S - T_i) - (T_S - T_O)}{\ln\left(\frac{T_S - T_i}{T_S - T_O}\right)} = \frac{(275 - 300) - (275 - 298.58)}{\ln\left(\frac{275 - 300}{275 - 298.58}\right)} = -22.7184\ K$$

And finally, the total heat transfer ratio for unit length of the tube banks will be:

$$\mathrm{q}' = \mathrm{N}\left(\bar{h}\pi D \Delta T_{lm}\right) = 14 \times 18.1081 \times \pi \times 0.05 \times (-22.7184) = \mathbf{904.6874}\,\frac{\boldsymbol{W}}{\boldsymbol{m}}$$

If we assume that the heat exchanger is 1.5 m wide, it means the tubes will be 1.5 meter long. So the heat transfer ratio will be:

$$3 \times q' = 3 \times 904.6874 = \mathbf{1357.0311}\ \boldsymbol{W}$$

So, the results, based on the dimensions of our CFD modeling will be:

Table 6: Validation of results

	$\dot{Q}$	
	2D	**3D**
Experimental equations	904.7 W/m	1357.0 W
Modeling results	873.5 W/m	1275.8 W

In the next section, we will analysis the 3D model of the system. In the end, by comparing the 3D model's results with the calculated values in this section, we'll examine that the error is less than the engineering approximation; thus, the results are validated and trustworthy.

3D Modeling

As the extra part of this project, the heat exchanger has been modeled in 3-dimensional format. For this matter, all of the steps explained in «Steps and Procedures» have been repeated for 3D geometry and mesh. In order to avoid prolixity of script, we abstain from explaining the processes once more. The only point in this modeling is to minimize the solving expenses, we've modeled only upper half of the system. Because we're dealing with a symmetric system, the results for the lower half would be the same. Also, it must be mentioned that the width of the evaporator has been assumed 1.5 meters in this analysis. Here are the results:

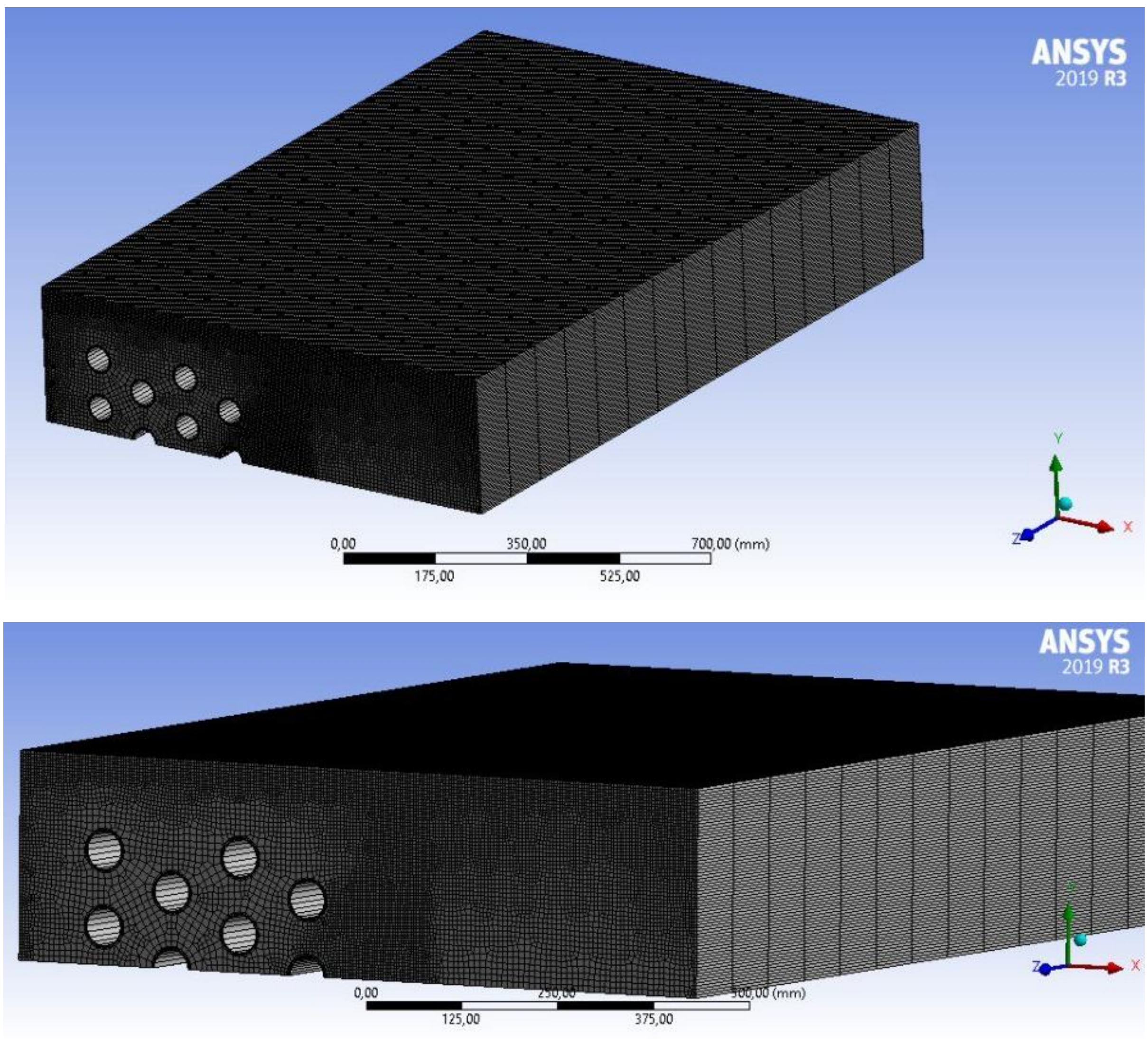


Figure 29: 3D mesh

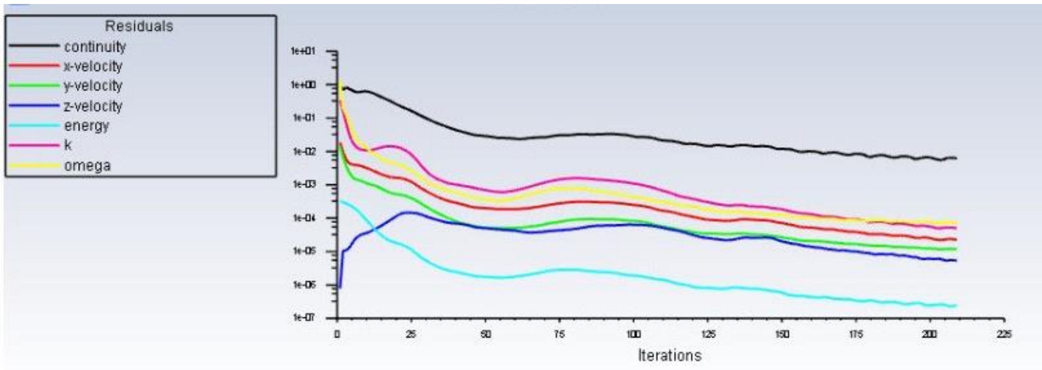


Figure 30: 3D residuals (convergence history)

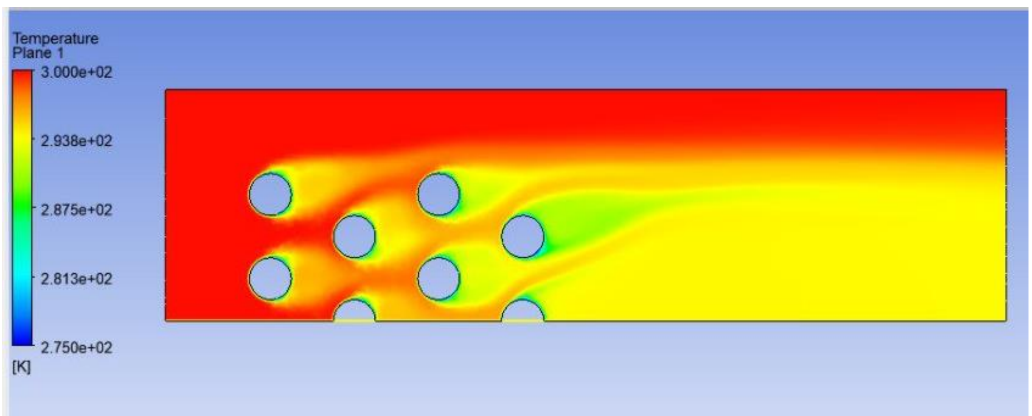


Figure 31: 3D temperature contour

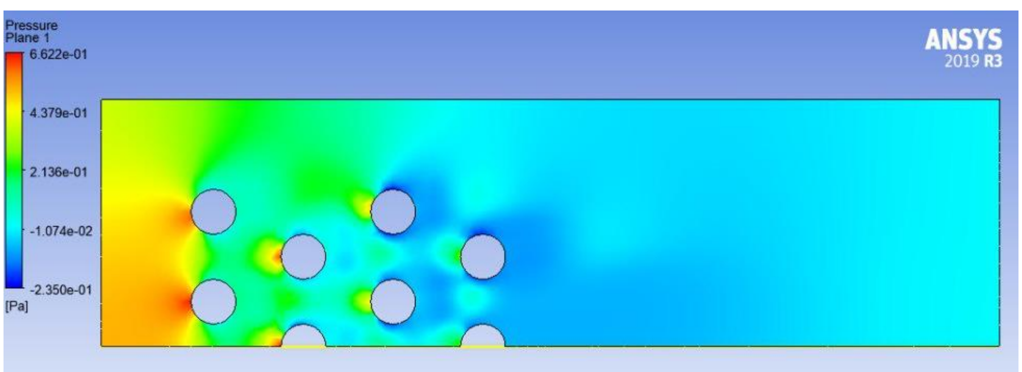


Figure 32: 3D pressure contour

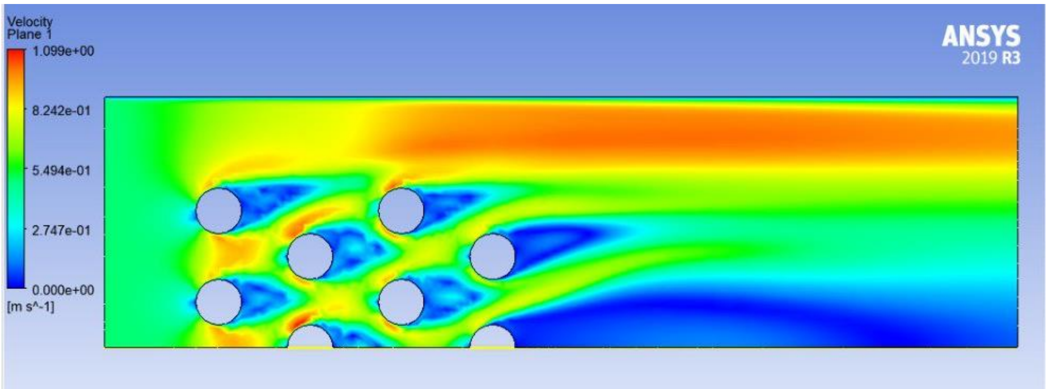


Figure 33: 3D velocity contour

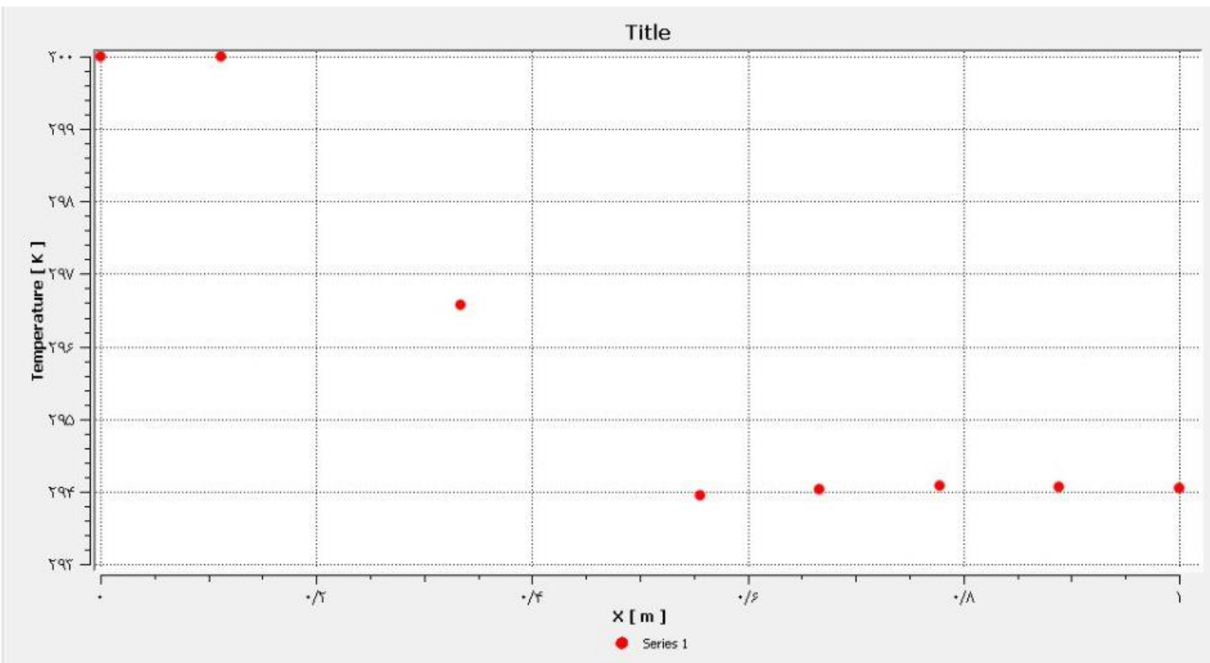


Figure 34: 3D temperature diagram of the centerline

The total heat transfer of this system is calculated to be **637.9 W** (see figure 35). since it’s the half of the system, this value must be doubled to give the actual heat transfer. So, we get:

$$\mathbf{2 \times 637.9} = 1275.8\ W$$

If we refer to the total heat transfer of the 2D model, we'll find the calculated value as **858.6 W/m**. If we multiply this value into the width of the evaporator which we assumed to be 1.5 meter, we must get the same value as what we got in this 3D analysis.

$$\mathbf{1.5} \times \mathbf{858.6} = 1287.9\ W$$

In the last section «Validation of Results with Equations», the theorical heat transfer of the system has been calculated to be **1357.1 W**. As we can see here, both values are very close to **1357.1W** which is calculated using the theorical equations. This means our CFD modeling and analysis for both 2D and 3D sections was precise enough to call it successful and acceptable.

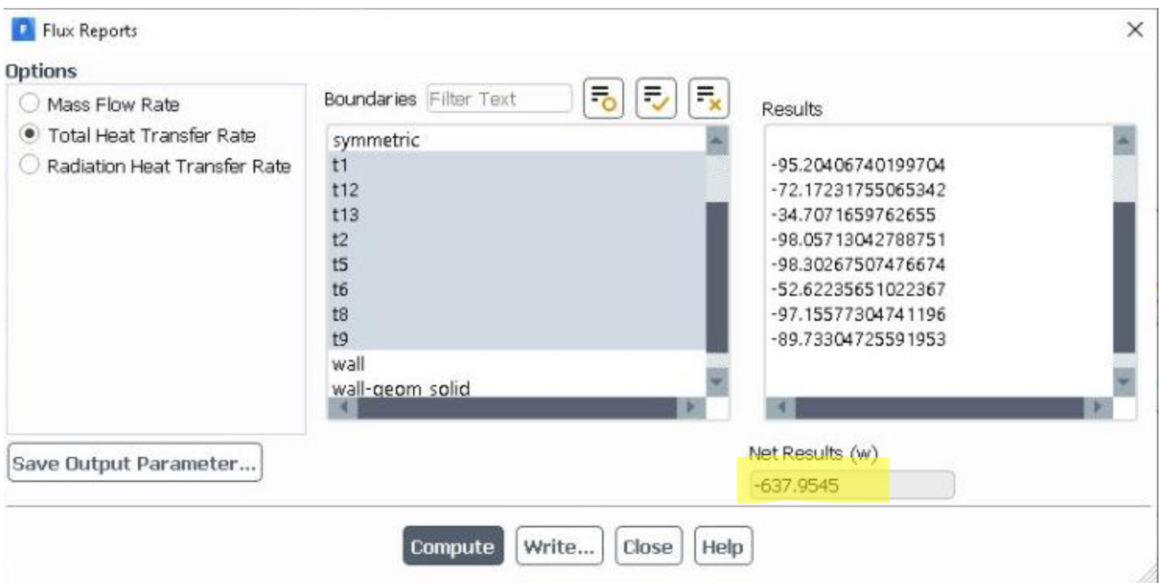


Figure 35: 3D heat transfer of each tube

Appendix

Required tables for experimental equations of heat transfer:

Table 7: Constants of Nusselt equation

Conguration	$Re_{D,\max}$	C_1	m
Aligned	$10–10^2$	0.80	0.40
Staggered	$10–10^2$	0.90	0.40
Aligned	$10^2–10^3$	Approximate as a single (isolated) cylinder	
Staggered	$10^2–10^3$		
Aligned $(S_T/S_L > 0.7)^a$	$10^3–2 \times 10^5$	0.27	0.63
Staggered $(S_T/S_L < 2)$	$10^3–2 \times 10^5$	$0.35(S_T/S_L)^{1/5}$	0.60
Staggered $(S_T/S_L > 2)$	$10^3–2 \times 10^5$	0.40	0.60
Aligned	$2 \times 10^5–2 \times 10^6$	0.021	0.84
Staggered	$2 \times 10^5–2 \times 10^6$	0.022	0.84

[a]For $S_T/S_L < 0.7$, heat transfer is inefficient and aligned tubes should not be used.

Table 8: Correction factor of Nusselt equation

N_L	1	2	3	4	5	7	10	13	16
Aligned	0.70	0.80	0.86	0.90	0.92	0.95	0.97	0.98	0.99
Staggered	0.64	0.76	0.84	0.89	0.92	0.95	0.97	0.98	0.99

References

(1) Heat exchanger - Wikipedia. https://en.wikipedia.org/wiki/Heat_exchanger.

(2) Cross Flow Heat Exchangers: All Practical Guides You Should Know - Linquip. https://www.linquip.com/blog/cross-flow-heat-exchangers/.

(3) Understanding Heat Exchangers - Types, Designs, Applications and .... https://www.thomasnet.com/articles/process-equipment/understanding-heat-exchangers/.

(4) 12 Different Types of Heat Exchangers & Their Application [PDF]. https://www.theengineerspost.com/types-of-heat-exchanger/.

(5) Cross-flow heat exchangers: what are the operating principles?. https://www.oesse.com/en/blog-en/cross-flow-heat-exchangers-what-are-the-operating-principles/.

(6) Cross-flow heat exchanger in buildings | Grundfos. https://www.grundfos.com/solutions/learn/research-and-insights/cross-flow-heat-exchanger.

(7) (PDF) Mesh Sensitivity & Mesh Independence Study - ResearchGate. https://www.researchgate.net/publication/339289671_Mesh_Sensitivity_Mesh_Independence_Study.

(8) Independence Study - an overview | ScienceDirect Topics. https://www.sciencedirect.com/topics/engineering/independence-study.

(9) (PDF) Mesh independence - ResearchGate. https://www.researchgate.net/publication/331313077_Mesh_independence.

(10) P. J. Roache. "Perspective: A Method for Uniform Reporting of Grid Refinement Studies". J. Fluids Eng. Sep 1994, 116(3): 405-413 (9 pages).

(11) NASA. Examining Spatial (Grid) Convergence.

(12) Richardson Lewis Fry and Gaunt J. Arthur 1927. VIII. The deferred approach to the limit. Philosophical Transactions of the Royal Society of London.