Organisation of Islamic Cooperation

"Effect of holes placed on the perforated finned heat exchanger at different angles on the Nusselt and Reynolds numbers"

A thesis submitted to the department of Mechanical and Chemical Engineering (MCE), Islamic University of Technology (IUT), in the fulfillment of the requirement for degree of Bachelor in Mechanical Engineering.

Prepared By:

Muhammad Saad (131424)

Supervised By:Dr. Arafat Ahmed

Department of Mechanical and Chemical Engineering

Islamic University of Technology

CERTIFICATE OF RESEARCH:

The thesis title "**Effect of holes placed on the perforated finned heat exchanger at different angles on the nusselt and Reynolds numbers**" submitted by **MUHAMMAD SAAD** (**131424**), has been accepted as satisfactory in fulfillment of the requirement for the Degree of Bachelor of science in Mechanical and Chemical Engineering on November, 2017.

Signature of the Supervisor

_____

Dr. Arafat Ahmed

Department of Mechanical and Chemical Engineering.

CANDIDATE DECLARATION

It is hereby declared that this thesis or any part of it has not been submitted elsewhere for the award of any degree.

Signature of the Candidate.

_____

Muhammad Saad

 Student ID:131424

Department of Mechanical & Chemical Engineering (MCE)

Islamic University of Technology (IUT), OIC

Board Bazar, Gazipur Bangladesh.

Signature of the Supervisor

_____

Dr. Arafat Ahmed

Department of Mechanical & Chemical Engineering (MCE)

Islamic University of Technology (IUT), OIC

Board Bazar, Gazipur Bangladesh.

ACKNOWLEDGEMENT:

I express my heartiest gratefulness to Almighty Allah for His divine blessings, which made me possible to complete this thesis successfully.

First and foremost, I feel grateful and acknowledge our profound indebtedness to Dr. Arafat Ahmed, Department of Mechanical and Chemical Engineering, IUT. His endless patience, scholarly guidance, continual encouragement, constant and energetic supervision, constructive criticism, valuable advice at all stage has made it possible to complete this project. I would also like to offer thanks to all who contributed in many ways during the project work. I acknowledge my sincere indebtedness and gratitude to my parents for their love.

I seek excuse for any errors that might be in this report despite of our best efforts.

ABSTRACT :

In this research, 6mm diameter holes were opened on each circular fin on a heating tube in order to increase convective heat transfer. This is different from the other studies in the literature. These holes were placed on the circular fin in such a way as to follow each other at the same chosen angle. The holes created turbulence in a region near the heating tube surface on the bottom of the fin. Experiment were performed to analyze the effect of this turbulence on heat transfer and pressure drop. These experiment were performed at six different angular position in order to determine the best angular positioning. In addition, a perforated finned heater was compared with an imperforated finned heater to observe the differences. When Nu_d values were considered for the same airflow rate and the same Red, the Nu_d value of the heater with holes was found to be 2.43% higher than the Nu_d value of the heater without holes. However, there was a 3.84% decrease in pressure.

Key words: Heat exchanger, heat transfer, convection

CONTENTS:

INTRODUCTION

LITRATURE REVIEW

GEOMETRY

MESHING

RESULTS AND APPLICATIONS

CHAPTER 1:

INTRODUCTION

There are several method for making efficient use of energy but we will consider one method to make efficient use of energy is to improve heat transfer without imparting any energy on a fluid. Therefore, attempts have been made to apply methods. Such as shaping of the heat transfer area and orientation of the flow using different geometric profiles and designs.

These methods, which aim at improving heat transfer, have been analyzed analytically and experimentally by many researchers. Leu et al. (2001) also analyzed the performances of the air sides of plate-fin heat exchangers with round and oval shapes. Their results showed that the pressure drop increases with the fin angle while heat transfer and friction increase with fin length. Kayansayan's (1993) analysis found that boundary layer interaction between the fin and the plate leads to a reduction in the Nusselt number and in the convection and conduction value of the finned surface over an insulated cylinder.

Their results showed that the pressure drop increases with the fin angle while heat transfer and friction increase with fin length. Karabacak (1992) showed the effects of fin parameters on the heat transfer that occurs in the form of radiation and free convection in a horizontal cylindrical heater with rectangular sections and circular fins. Karabacak (1996) also investigated the experimental relationships between heat flow, Nusselt number and temperature difference in a finned heater. Xiao and Tao (1990) studied a naphthalene sublimation technique to examine the heat transfer coefficients and friction factors for wavy plate-fin heat exchangers with different fin lengths. They applied thermal performance assessments for the same pumping power and the same pressure drop and found that both the Sherwood number and the friction factor increase drop decrease with an increase in fin spacing. Kayansayan's (1993) analysis found that boundary layer interaction between the fin and the plate leads to a reduction in the Nusselt number and in the convection and conduction value of the finned surface over an insulated cylinder. Yan and Sheen (2000) performed an experiment to determine the heat transfer and pressure drop characteristics of a plate (wavy and partitioned) fin heat exchanger. Yakut and Sahin (2004) experimentally investigated the flows that cause the vibration characteristics of the turbulators used to increase the heat transfer in heat exchangers. They found that the Nusselt number increases with the Reynolds number.

The goal of the present study is to reduce thick of the boundary layer that is formed on the circular fins placed on the heating tube, thus increasing heat transfer through convection in this area.

Parallel and counter flow arrangement

Hot and cold fluid both flow parallel to each other in same direction. Hot fluid flowing through inner tube and cold fluid is flowing through the annulus.as it flow parallel we call parallel flow. And heat transfer taking place to the surface wall of inner tube from hot fluid to the cold fluid across the surface of the inner tube. If we reverse the flow of any one of the fluid the flow will be said to counter flow arrangement.

Chapter 2:

LITERATURE REVIEW

Heating systems with finned tubes used in practice are usually liquid/gas heat exchangers. Therefore we set up one fluid is flowing outside the tube and the other fluid is flowing inside the tube, typically this type of heat exchanger used for gas and liquid, liquid will flow inside the tube cause the liquid have already high heat transfer coefficient. On gas side we don't have high heat transfer coefficient in order to reduce the thermal resistance we put fins on gas side.

In this study, the external diameter of the body into which the finned tube was placed was 154 mm, the heating tube external diameter was kept fixed at 29 mm (d), the fin diameter was 87 mm (D) and the fin thickness was 0.5 mm (t). However, the distance between the fins was varied to be one of five different values (s); these were 4, 8, 10, 12 and 15 mm. The heating tube length was kept fixed at 900 mm. In accordance with the goal of the experimental study, convective heat transfer was intended to be increased by reducing thick of the boundary layer on the fins by using 6 mm diameter holes that were opened at different angles on the circular fin that was placed on the heating tube. These same diameter holes on each fin could follow each other at the same selected angle.

The research we continued for six different angular positions. These angles were 0, 15, 30, 45, 60 and 90°.the circular finned heating tubes that had 6 mm-diameter holes at above angles.Flow of the heating fluid (water) was kept fixed in the experiments while water temperatures were varied, thus allowing the experiment to be conducted at seven different temperatures. These temperatures were 40, 45, 50, 55, 60, 65 and 70° C. Air flow was varied to be 0.04 kg/s.

Chapter 3:

<u>*GEOMETRY*</u>

In this project, we have used a air as a fluid.The steps of modelling the fins and tubes within Ansys DesignModeler will be discussed farther:

Step1: Launch ANSYS ,by going to the start-up menu and double clicking on workbench file in the ANSYS 13.0 folder, Go to Analysis Systems and double click, then double click on Geometry.
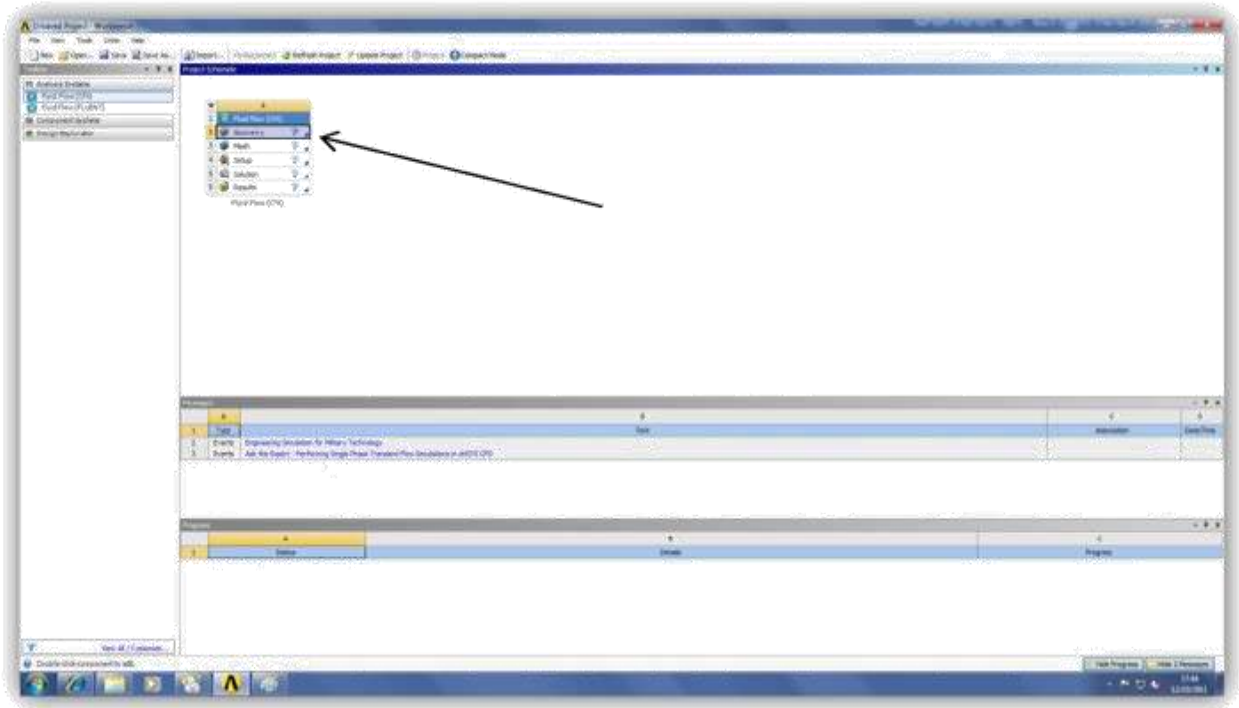

Step2: Select the unit, in here the units selection depends on what kind of units will fit you job. In our project we used mille meter.

Step3: Click on the icon that has the Cartesiancoordinate'sicon. Onceit'sclicked you can see where the plotting plane is.

Step4: Looking at the window you will see the Cartesian coordinates system is shown on the screenwith a blue arrow referring to the x axis, green arrow referring to the z axis, red arrow referring to the y axis. A note to the user that 2d sketching is conducted on the specified xy plane as long as the user specifies some other plotting plane (xz plane ,yz plane)
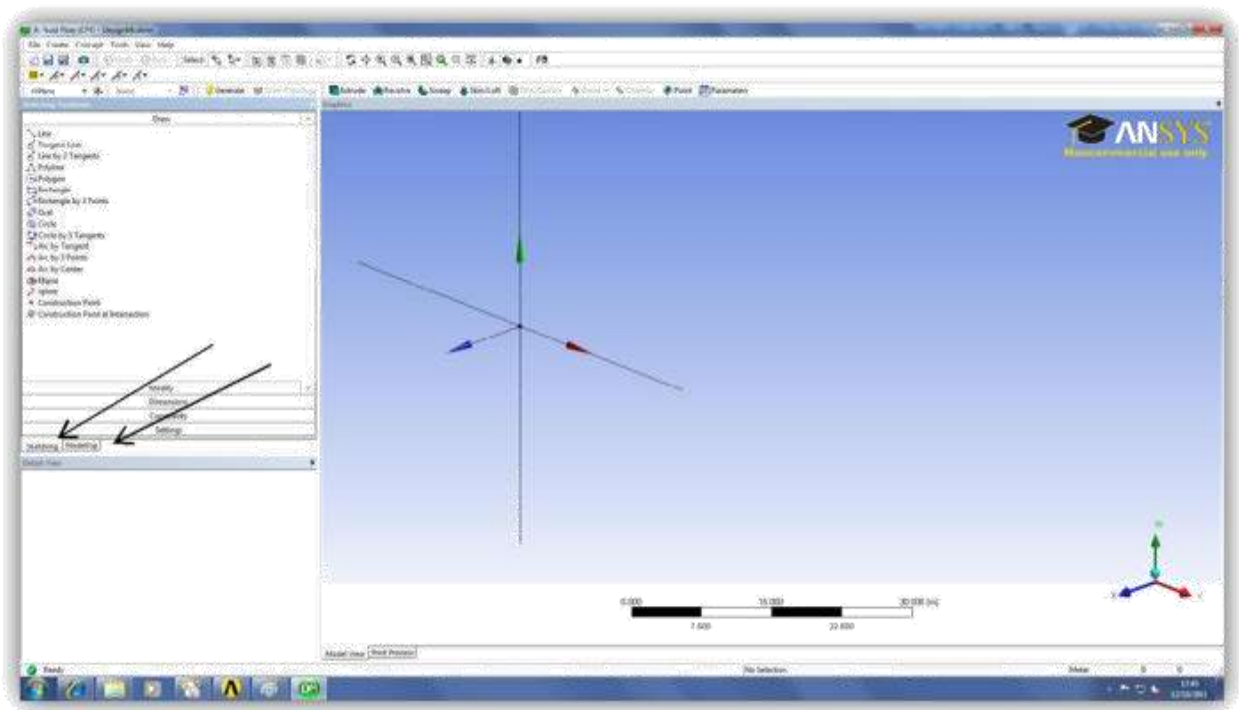
The user can access the modeling tools by pressing on the Sketching tab or he can accessthe modeling planes by going to Modeling.

5

Step5: Select the Line and start sketching the fins and body profile, the line command is done by pressingtwice, once you have drawn the first line, press again on the last plotted point and continue till you plot all the profile, remember to make sure that the profile contour is closed. There is a second alternative which consist to click-on on the rectangle icon and draw your rectangle of the desired length and the radius of the pipe. Note that it's is advisable to always start in 0,0 coordinates. In other words, the origin of your Cartesian coordinates system.
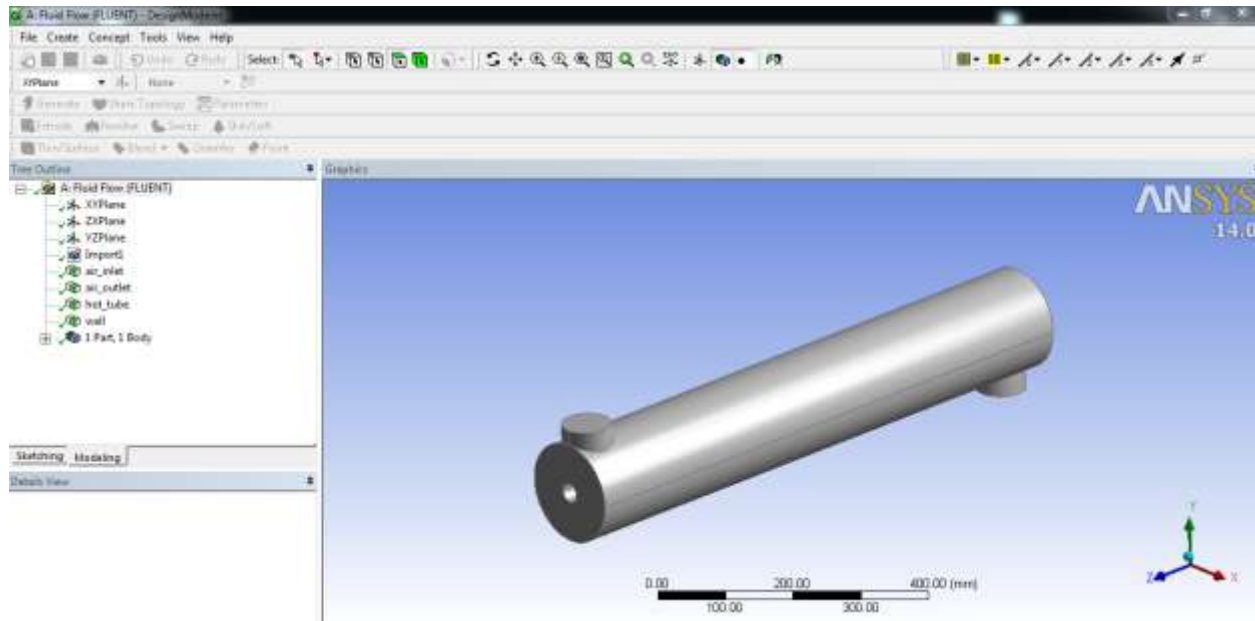
Step6: Finally select the revolving command to revolve the obtained figure to finally end up with your pipe with desired length and diameter.It is also to note that the model can be imported to ANSYS Workbench from other modeler like SolidWorks and some other Cad program.
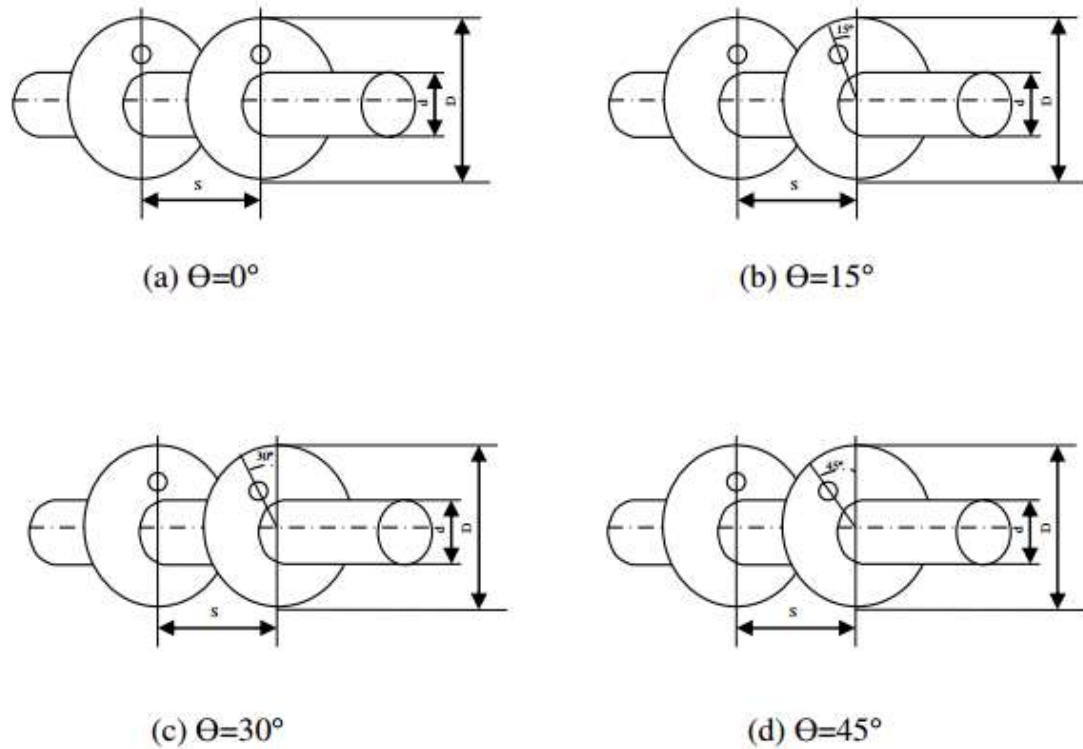
(a) Θ=0°

(b) Θ=15°

(c) Θ=30°

(d) Θ=45°

Figure . Finned heating tubes where it was opened 6 mm-diameter holes at the angular locations
=0°, =15°, =30°, =45°, =60° and =90°

Heat transferred by convection

The convective heat transfer is given by,

$$Q_c = h_0 (\eta_0 A_0)(T_w - T_0)$$

Here, $A_o = A_w + \Sigma A_f$: total heat transfer surface and o: represents the efficiency of the finned surface. It can be calculated with the equation (Yakar, 2007)

$$\eta_o = 1 - \frac{\Sigma Af}{A}(1-\eta_f)$$

Fin efficiency η_f was given in reference (Yakar, 2007) according to the fin parameters. Moreover, Aw is the area of the outer surface of heating tube and ΣA_f is the total surface area of the fin. The convective heat transfer coefficient ho is determined by the following equation when total energy input Q_t is known

$$h_o = \frac{Q_t - \Sigma Q_t}{\eta_o A_0 (T_w - T_0)}$$

η_o in equation (12) contains a heat convection coefficient that must be solved for by iteration. Iteration is continued until the o, f and ho values converge to within 1%. Depending on the ho value obtained by iteration, the Nusselt number is determined as follows (depending on the outer diameter of the heating tube)

CHAPTER 4:

MESHING

For both CFD (fluid) and FEA (solid) modelling, the software performs the computations at a range of discrete locations within the domain. The purpose of meshing is to decompose the solution domain into an appropriate number of locations for an accurate result.
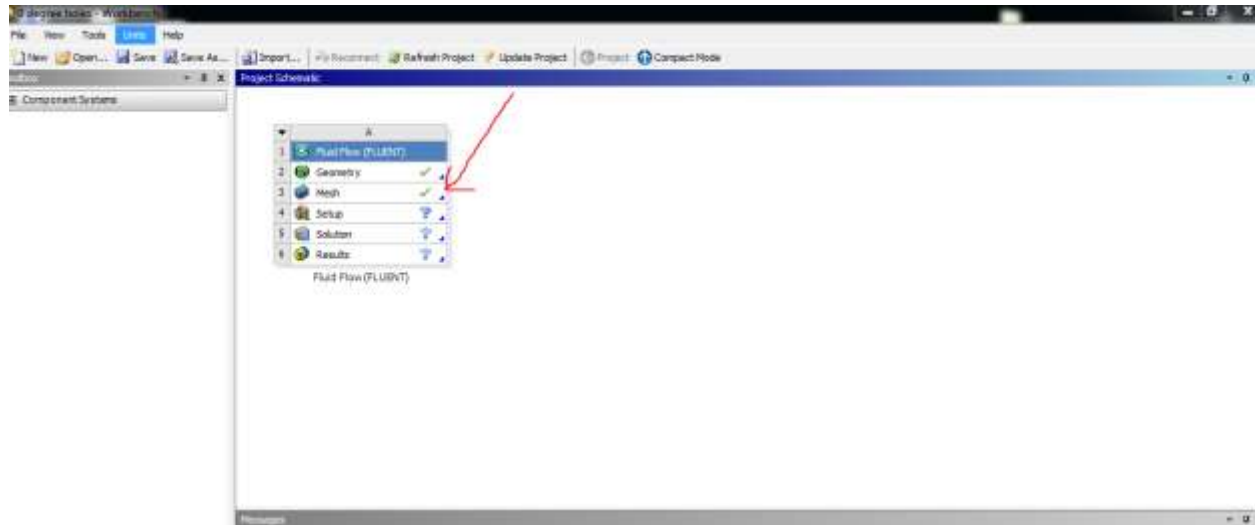
Types of mesh:

There are basically two types of meshing. Free and Mapped Meshing.

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or mapped mess is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it. A mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this type of mesh you mustbuild the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh.

Mesh control:

The default mesh controls that the ANSYS program uses may produce a mesh that is adequate for the model you are analyzing. In this case, you will not need to specify any mesh controls. However, if you do use mesh controls, you must set them before meshing solid model.

Mesh controls allow you to establish such factors as the element shape, midsize node placement, and element size to be used in meshing the solid model. This step is one of the most important of your entire analysis, for decisions you make at this stage in your model development will profoundly affect the accuracy and economy of your analysis.

For further Mesh details, consult ANSYS website or any book about the software.

Now we would like to give a brief discussion about how the meshing has been done on our particular project.

Step1:Launch your workbench as discussed in previous parts.

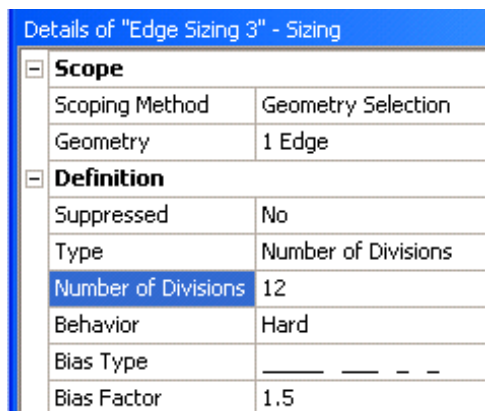***Step2:***Check on the toolbox and double click on "fluent flow"

Step3:After completing the step2, a window normally pops up on the project schematic and then double click on "Mesh".

Step4:Define your element sizing, depending on the number of element desired from the window at the down part of the toolbox. We had 30000 elements on our projects.

Step5:This final step consist to just generate your mesh after defining your mesh elements and others, by just clicking on "generate Mesh" on the top left corner of your workbench.

However, some setting has got to be mad for a better mesh, as we used mesh control, we allocated every edge the desired number of element: for the Wall and the Axis we allocated 10000 elements and for inlet and outlet we got 30 elements. And never forget to click on "hard" for the behavior and "number of division" on the type, as shown on the image below.

Note that the image was not taken from our project meshing.

Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	12
Behavior	Hard
Bias Type	_____ ___ _ _
Bias Factor	1.5

Setup:

This part is basically the most important one. This is where the simulation process takes its information from. It is quite different from one project to another, e.g.; one system or material to another. However we will discuss according to our project.

Step1:Adding constraints to ANSYS Design Modeler Parameters in ANSYS Workbench.

In this step, you will start ANSYS Workbench, open the projectile file, review existing parameters, create new parameters, and add constraints to existing ANSYS Design Modeler parameters.

1.Start ANSYS Workbench by clicking the windows start menu, then selecting the workbench option in the ANSYS14.0 program group.

Start ➡**Programs** **ANSYS14.0** ➡**Workbench14.0**

This will displays the ANSYS workbench application window, which has the **toolbox** on the left and the **project schematic** to its right. Various supported applications are listed in the toolbox, and the components of the analysis system are in the Project schematic.

2.Open the project file for your defined project. Parameter already has a FLUENT-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in ANSYS FLUENT.

3.Review the parameters that have already been defined in ANSYS Design Modeler.

a.Click the **Parameter Set** bar in the ANSYS Workbench Project schematic.

b.Select the **outline** option under the View menu.

ViewOutline ➡

This displays the Outline of all Parameters view in ANSYS Workbench, where you can view an outline of all parameters associated with the current project.

c.Select Properties option under the View menu

View Properties➡

This displays the Properties view in ANSYS Workbench

d.Select the Files option under the View menu (optional)

View ➡Files

This displays the Files view in ANSYS Workbench, where you can view the files associated with the current project.

The Project Loaded into ANSYS Workbench Displaying Parameter
Outline, Properties, and Files View
Note the types of files that have been created for this project. Also not the states of the cells for Fluid Flow analysis system. Since the geometry and mesh have already been defined, the status of the Geometry and Mesh is up to date. Also since the ANSYS FLUENT setup is not complete,

the setup cell's state is refresh required and since the simulation has yet to be performed, with no corresponding results the state for the solution and results cells is unfulfilled, with no corresponding results, the state for the solution and results cells is unfulfilled.

*Step2:*Setting up the CFD Simulation in ANSYS fluent.

Now that you have edited the parameters for the project, you will set up a CFD analysis using ANSYS Fluent. In this step, you will start ANSYS Fluent, and begin setting up the CFD simulation.

1.Start the ANSYS FLUENT.

In the ANSYS Workbench project schematic, double click the Setup cell in the ANSYS FLUENT fluid flow analysis system. You can also right click the setup cell to display the context menu where you can select the edit option.

When ANSYS Fluent is first started, fluent launcher is displayed, allowing you to view and/or set certain ANSYS Fluent start-up option.

Fluent launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

This figure shows the ANSYS FLUENT Launcher.

Note that the dimension 2D or 3D is automatically set by the software according to your geometry, and it is unchangeable.

Make sure to enable these options on the launcher based on your project.

2.Set up your models for the CFD simulation.

a.Enable heat transfer by activating the energy equation.
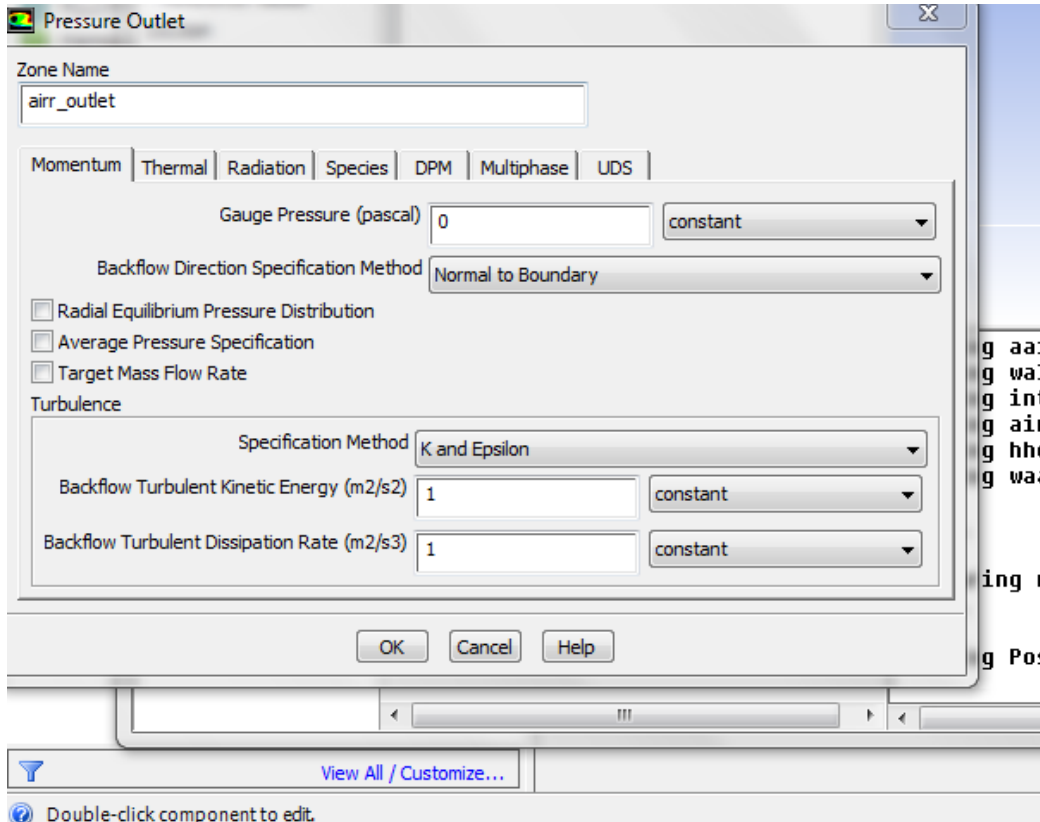
Models ➡ Energy ➡ Edit
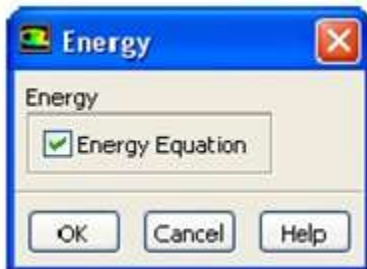
i.Enable the Energy equation option.

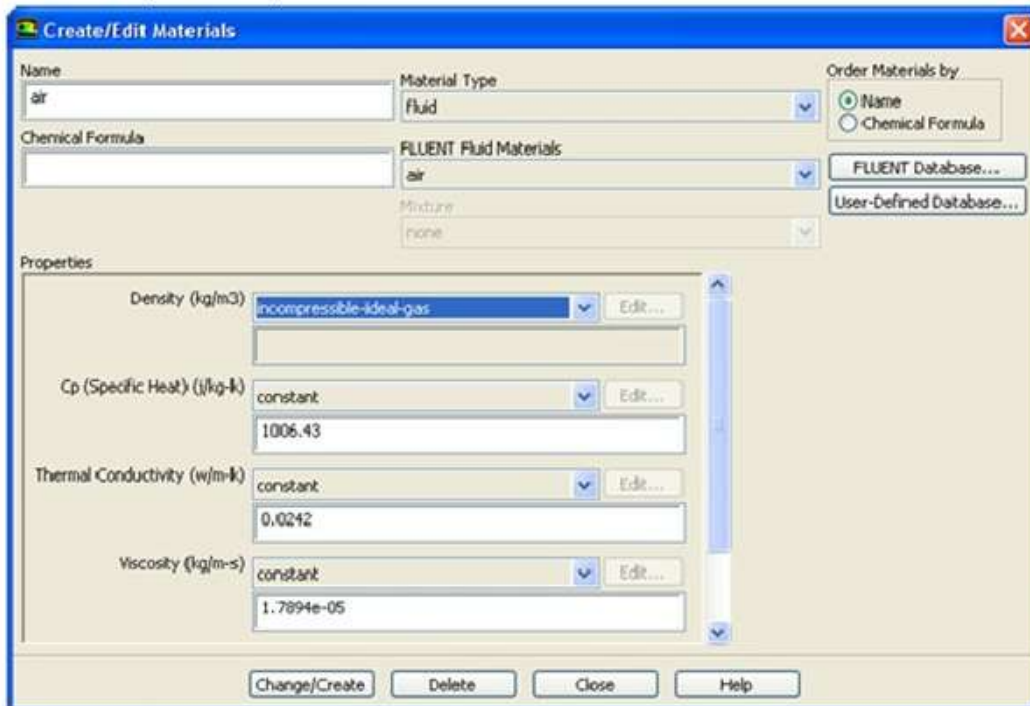**ii.**Click Ok to close the Energy dialog box.

b.Enable the k-epsilon turbulence model.

Models ➡ Edit

Again. Make sure to enable or disable any option according to your project. These selections has been done according to our project.

4.Set up your materials for the CFD simulation.

Note that in this figure Air is used instead of water. Unlike ours where we used water.

*Step3:*Define Input and Output Parameters in ANSYS Fluent and Running the Simulation

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define input parameters for the velocity inlet, set output parameters for the heat exchanger volumes, define heat source boundary conditions for the evaporator and heat exchanger volumes, define heat source boundary conditions for heat exchanger, then calculate a solution.

1. Define an input called in_velocity for the velocity at the inlet boundary

Boundary conditions ⟶ inlet-water ⟶ Edit

This displays the velocity inlet dialog box
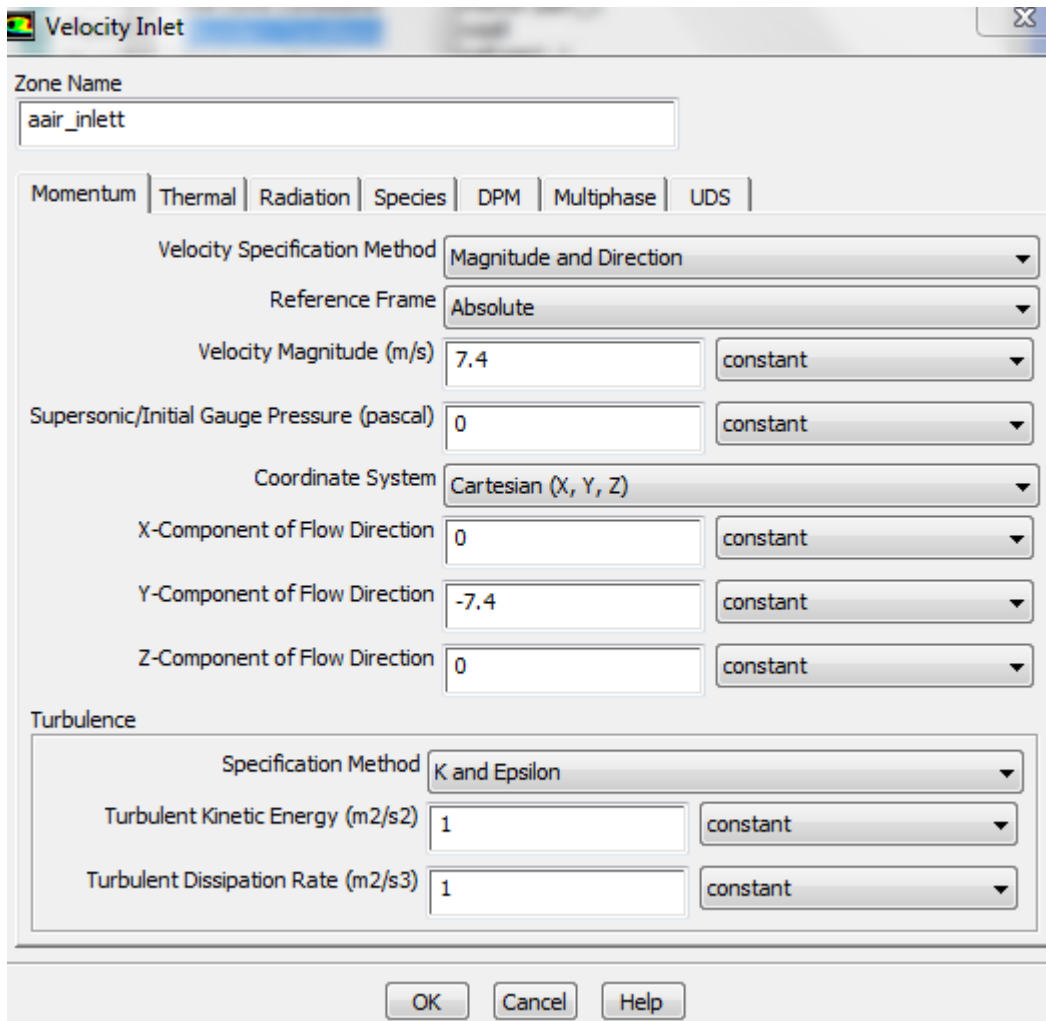
Note that this figure has been captured while performing a project that used air.

In the velocity inlet dialog box, select new input parameter from the drop down list for the velocity magnitude. This displays the input parameter properties dialog box. And from there, enter in velocity for the name, and enter your current value, based on your project.

Click OK to close the input Parameter properties dialog box.

In the inlet velocity inlet dialog box, select intensity and hydraulic diameter from the specification method drop down list in the turbulence group box. Retain the value of the turbulence intensity according to your project and enter the hydraulic diameter.

In the thermal tab of the velocity inlet dialog box, select New input Parameter… from the drop-down list for the temperature.
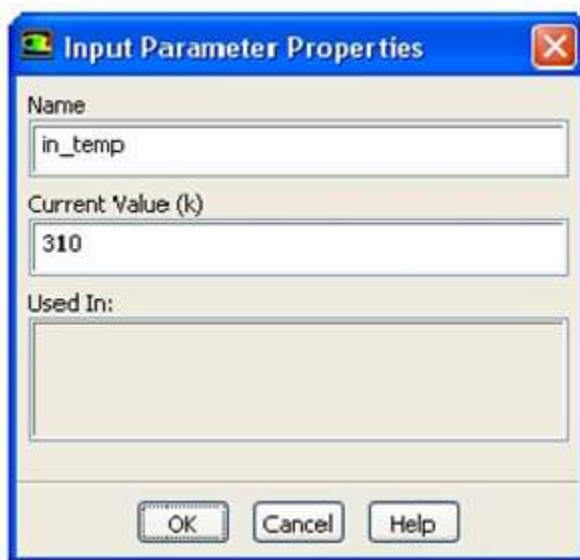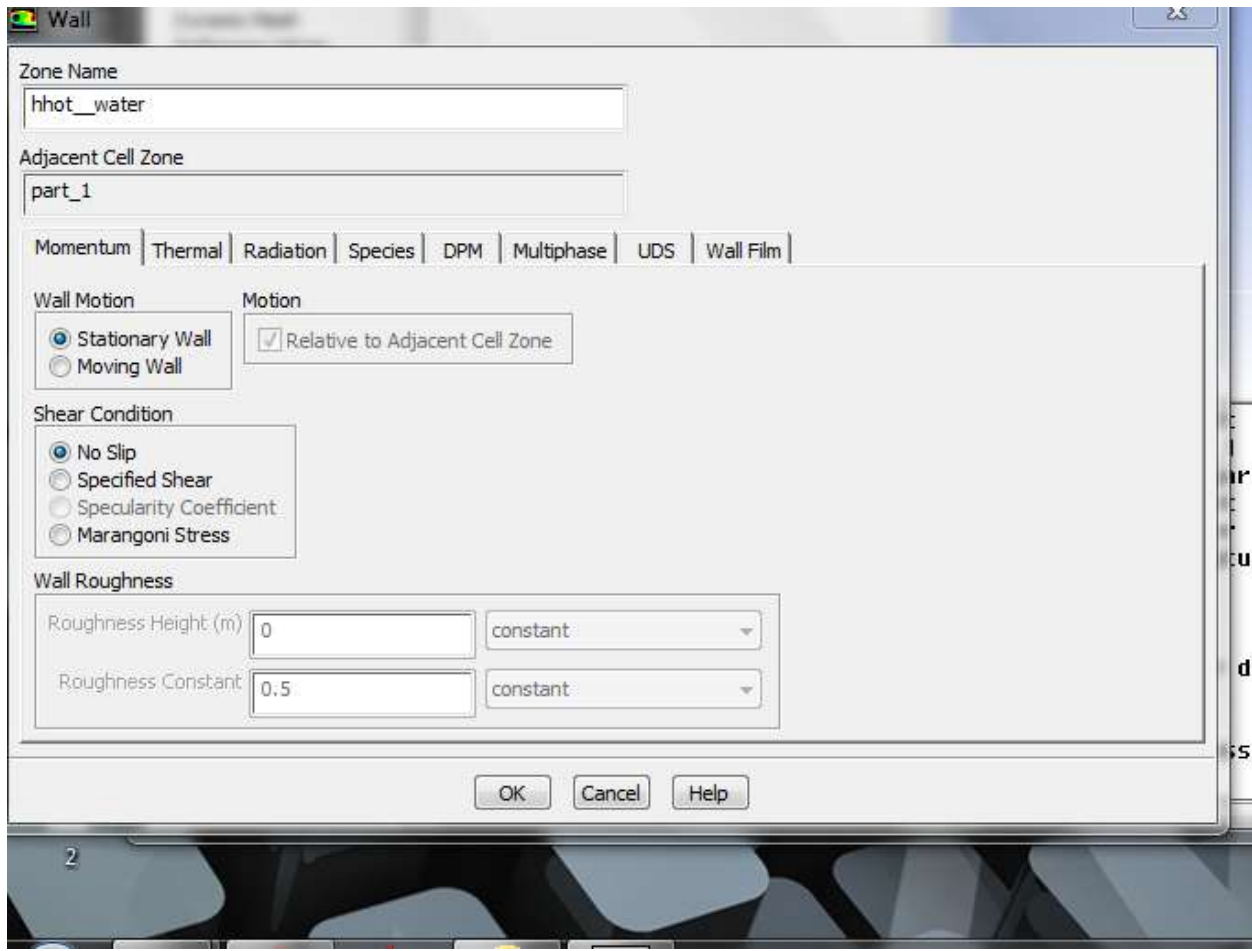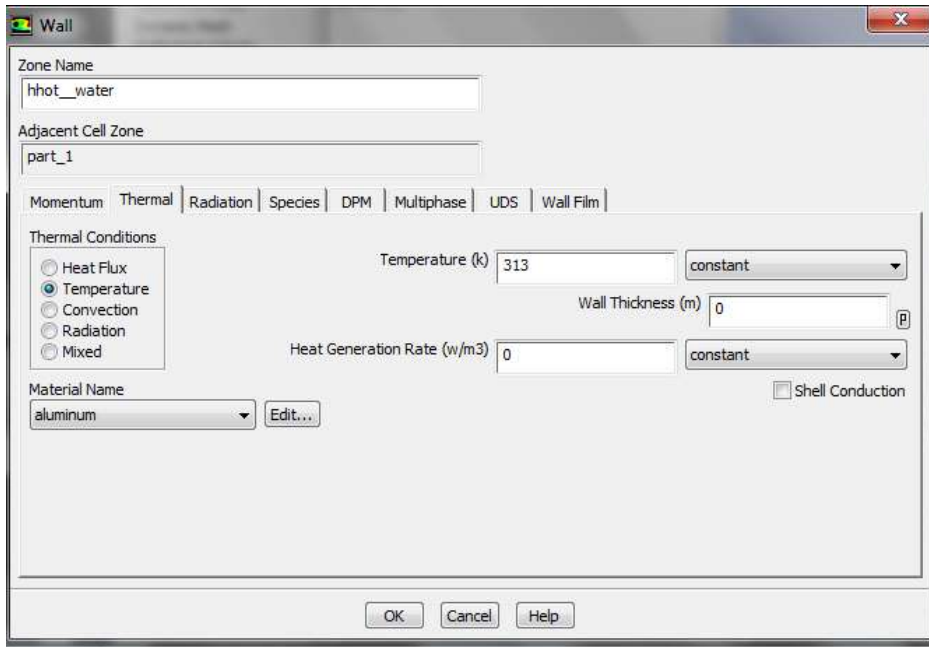

You know can assign your desired inlet temp. Keep the name in_temp and then click OK to close the input parameter properties dialog box. Again click OK to close the velocity inlet dialog box.

The following figure gives a glance about our boundary and inlet condition according to our project,
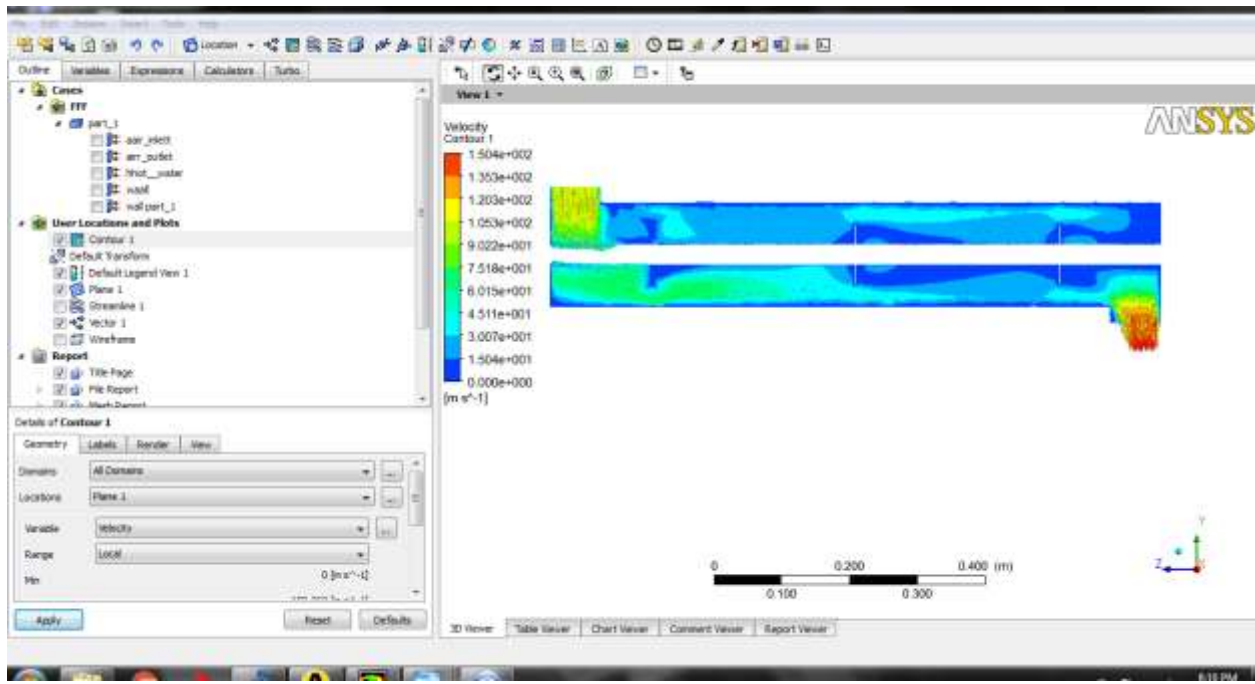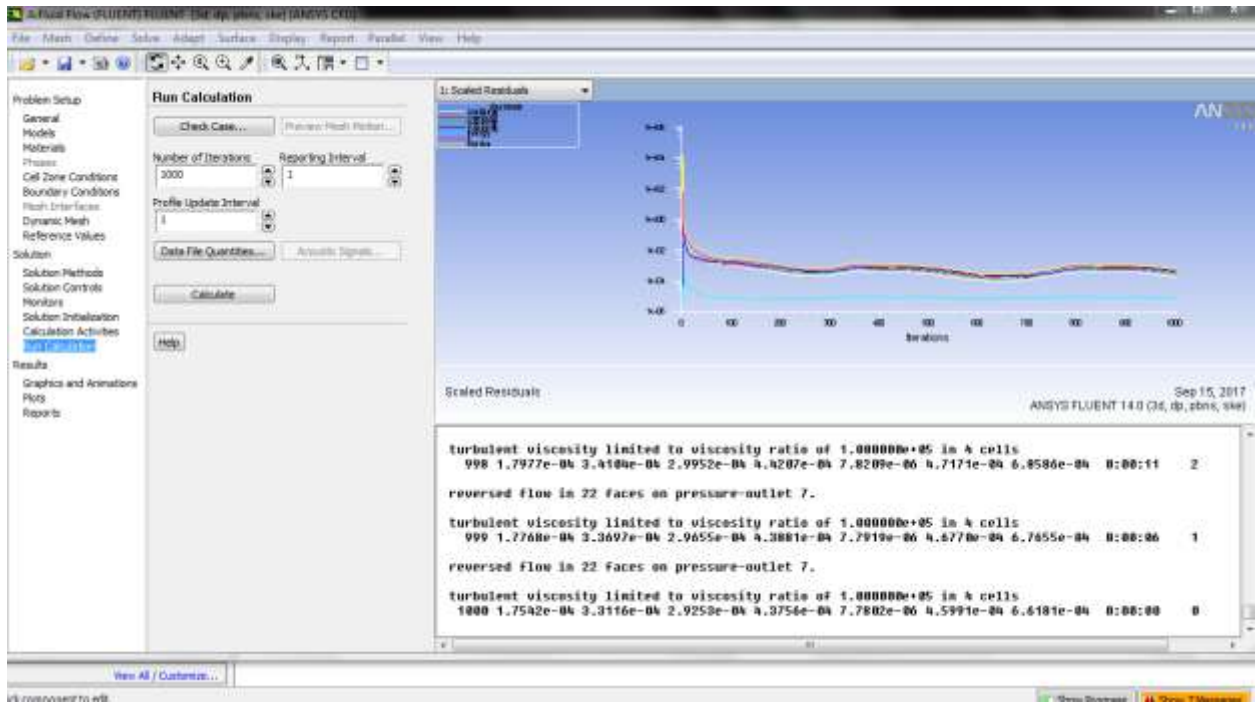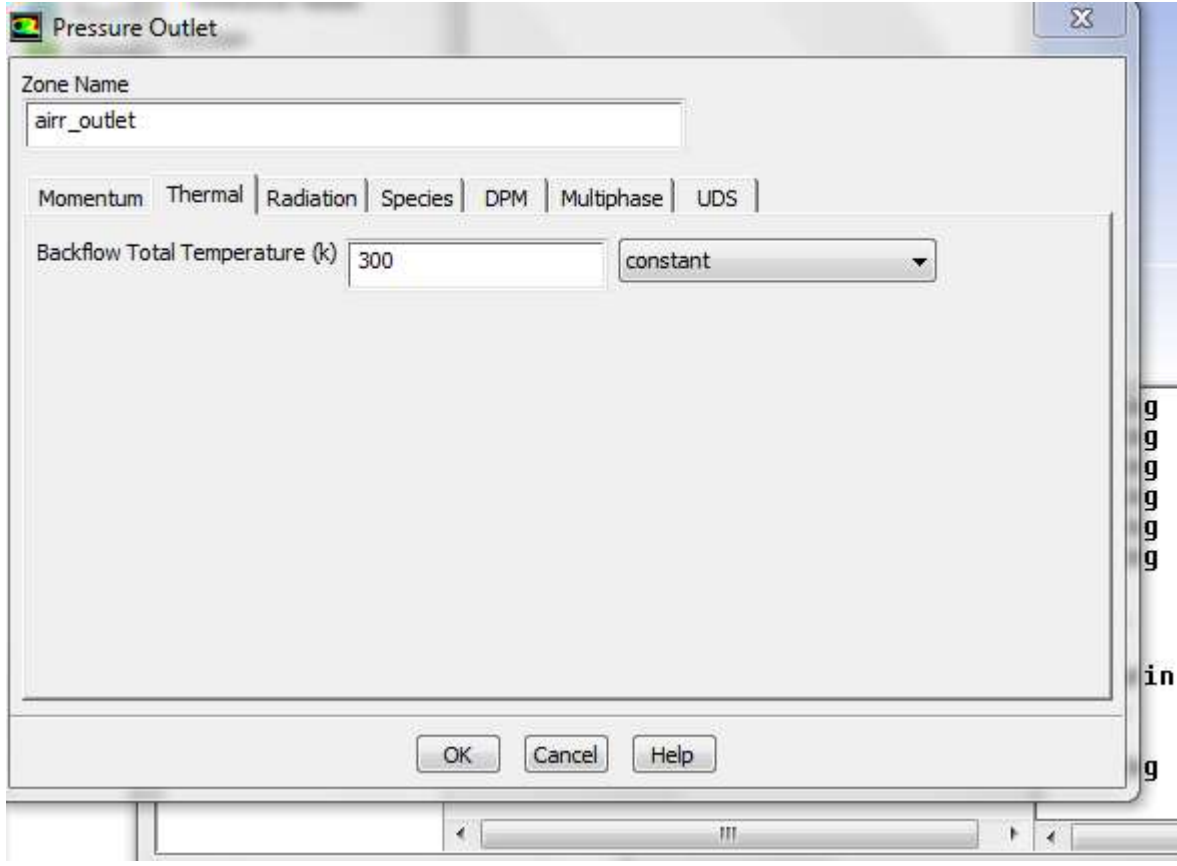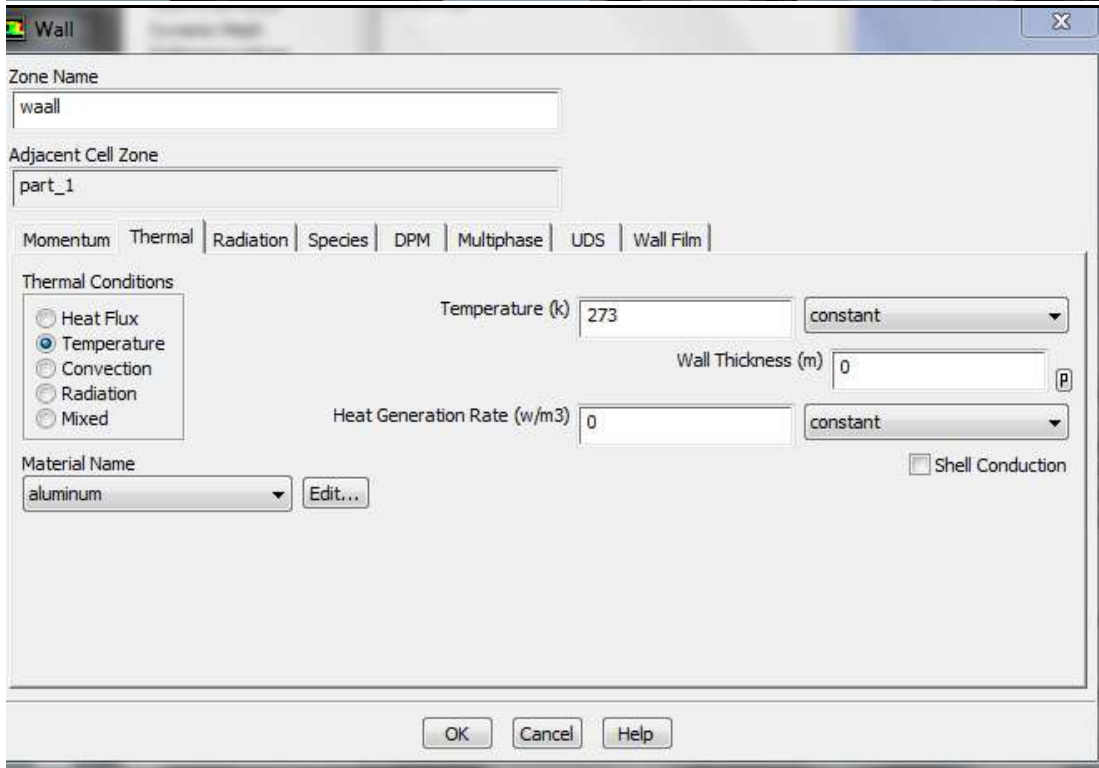
5.Solution Initialization.

Retain the default selection of Hybrid initialization and click the initialize button.

After the initialization, you can directly set the number of iterations, run the simulation and wait for the result to converge. The following figure shows the wall conditions according to our project and the number of iteration at which the solution has converged.
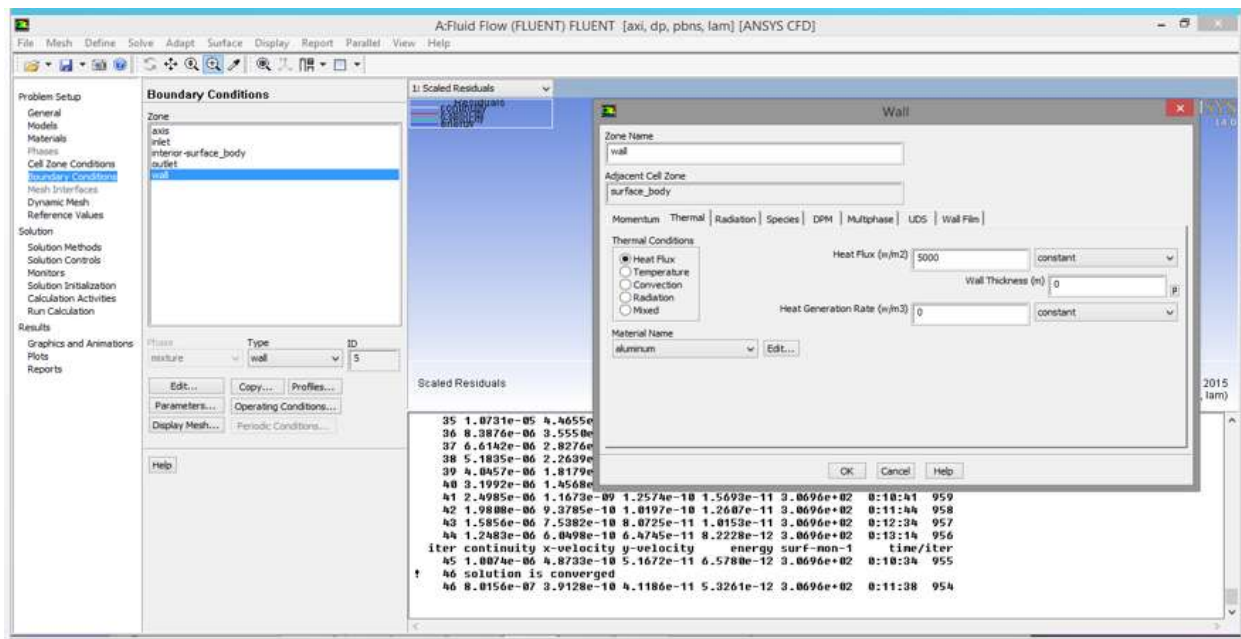
Note that, in our project, it converged at 1000 iteration. However it depends from one project to project. It can take fewer or more time to have a converged solution.

CHAPTER 5:

RESULTS AND APPLICATIONS.

We flowed the heated air through the spaces between the fin and the body. As the section area changes in the direction of the flow, the air decelerates and accelerates. The air also flows to the heating tube in the direction of the flow. First it flows parallel, then vertical and then parallel again because of the deflectors used in the system. Therefore, the movement of the air on the heating tube changes during the flow in such a way that both the flow section and the flow direction are continually repeated. However, the mass flow remains the same in all the flow and section changes. Considering this, the Reynolds number that characterizes the air should be defined according to the mass velocity $G=\frac{m_{air}}{A_s}$ .Here m_{air} is the mass flow of the air passing through the heater at a unit time and As is the area of reference section perpendicular to the flow direction between two fins ($A_s = D._s$). However, Now that heat production of the finned heating tube, the fin, the heating surface and the air temperatures are known for a certain geometric position and study condition, the values of radiative heat transfer and convective heat transfer can be determined. As convective heat transfer is dependent on movement of the air, the Red number (which characterizes the flow) should take into account the convective heat transfer calculated for unit surface ($q_c = Q_c/A_o$). As a result, the Reynolds number should be described according to ($q_c = Q_c/A_o$). After the required adjustments are made, the modified Reynolds number (which is obtained in such a way as to take this into account), can be stated as follows, depending on the measured values:

$$R_{ed} = R_{ed} \ Nu_d = \frac{q_c \ Gd^2}{\mu\lambda \ (T_w - T_0)}$$

Here, physical properties of the air

$$T_{film} = \frac{T_w + T_0}{2}$$

are obtained with the use of the film temperature. Red values were determined for the cross flows and for air flows at 0.04, 0.06 and 0.08 kg/s according to the imperforate state and according to the 0, 15, 30, 45, 60 and 90°angular positions of the holes opened on the fins of finned heating tubes. The heating tube's diameter was 29 mm, the fin diameter was 87 mm, and the distance between the fins was 10 mm. The tube length was 900 mm. According to different water input temperatures, the change of the modified Reynolds number (Re_d) is shown in the diagrams between Figures 4 . According to the experimental results for the required air flow (0.04 kg/s), the R_{ed} values for the imperforate position at water input temperatures from 40 - 60°C in the cross flow are higher than the R_{ed} values at the other positions. Above 60°C, however, the R_{ed} values at = 90°are higher than the R_{ed} values of the other positions. This case is shown in Figure. This result show that the flow characteristics of the air in the heater change. This change is also influenced by the angular positioning. The temperature gradient that causes

the heat transfer between the fluids in cross flow is different from each other. It can be considered that Sci. Res. Essay
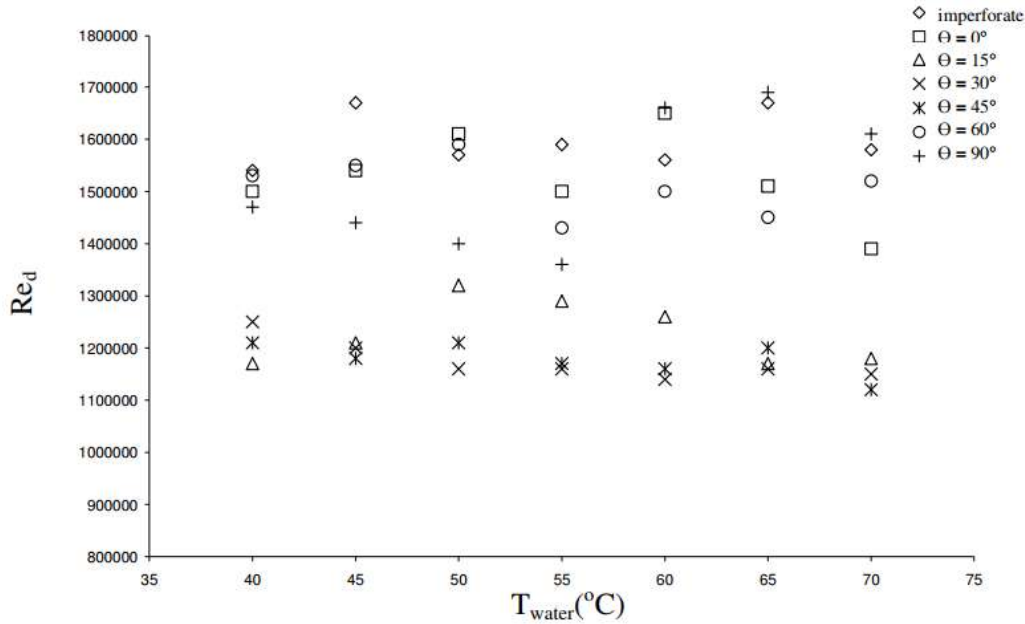

Figure . Change of $\overline{Re_d}$ value according to water input temperature $\mathbf{T_{wateri}}$ in cross flow at 0°, 15°, 30°, 45°, 60°, 90° angular positions without holes for s/d = 0.345 $\mathbf{m_{air} = 0.04}$ kg/s and $\mathbf{m_{water} = 1.1}$ kg/s.

This will cause the change in Re_d number. Because the physical properties of the fluid changes. And boundary layer thickness is related to flow speed and flow speed is related to the Reynolds number, the heat convection coefficient (h) in the convective heat transfer is dependent on the distribution of temperature within the thermal boundary layer. Because Pr=1 in gases, the Nusselt number that expresses the convective heat transfer must be a function of the Reynolds number. The change of Nu_d values according to the Red for the imperforate and perforated position in the interval $0° \leq \theta \leq 90°$ of the finned heating tube is shown in the diagrams between Figure 5. At air flow values of 0.04 kg/s, the Re_d and Nu_d values at the = 90° imperforate angular position in cross flow appear to be greater than the Re_d and Nu_d values at the other positions. This can be seen in Figures . Because the environmental temperature of the angular position at which Nu_d values are greater than the other angular positions is observed to be greater than the environmental temperatures of the other positions, it is thought that this case is influenced by the temperature of the environment, the mean temperature value appears to be greater than the mean temperature value at the lower surrounding air temperature, whereas the volume of the same mass flow increases because density decreases as temperature increases. As the increasing volume passes through the same flow section, its average speed increases. The currents created by the increase in average speed and acceleration increase the Reynolds number. As a result, both the Reynolds number and the Nusselt number increase.

23

APPLICATIONS:

It can be used to improve heat transfer and energy efficiency in various fields of thermal systems. Recently, the number of companies that see the potential and are in active development work for specific industrial applications is increasing project.

CONCLUSIONS:

The heater whose fins did not have any holes and the heater whose fins had holes with an angle = 0°and an s/d = 0.345 were studied. When Nu_d values were considered for the same air flow rate and the same Re_d , the Nu_d value of the heater with holes was found to be 2.43% higher than the Nu_d value of the heater without holes. However, there was a 3.84% decrease in pressure. For the imperforate state and with = 90° in the s/d = 0.345 state, when the Nu_d Number Were examined for the cross flow with the same Red numbers, it was found that the Nu_d number in the perforated state was 2.94%
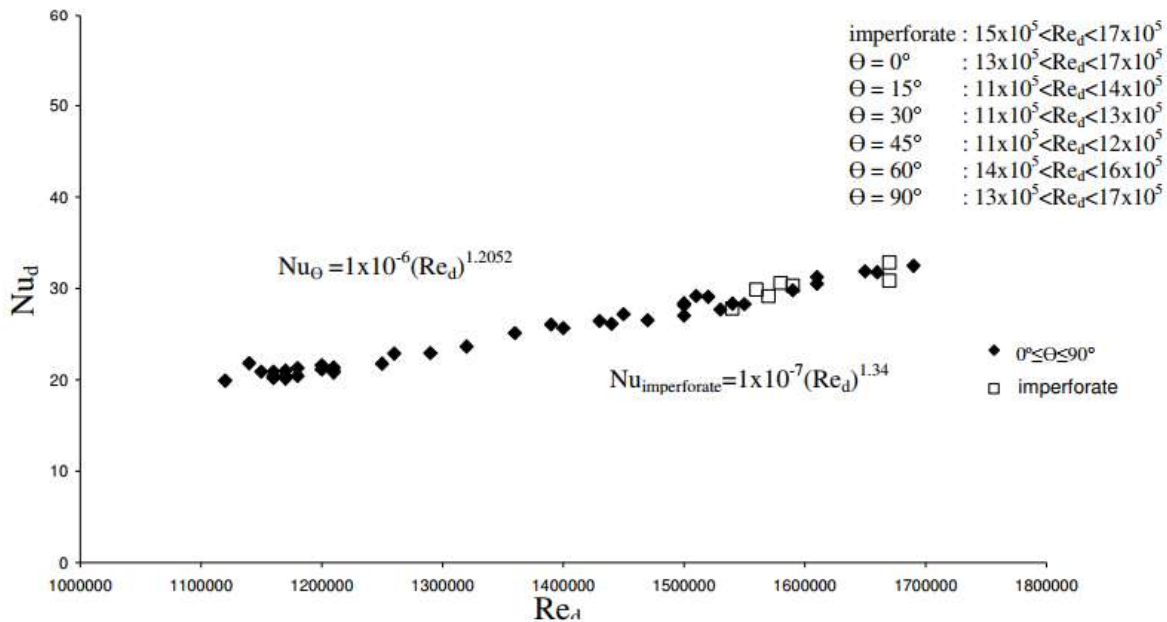

Figure . Change of Nu_d number according to Red for s/d=0.345, **m_{air} =0.04** kg/s and **m_{water} =1.1** kg/s in the interval **$0^0 \leq \theta \leq 90^0$** and in imperforate position for cross flow.

Nomenclature:

Distance between fins (m); **L,** Length of heating tube (m); **D**, Fin diameter (m); **d,** Outer diameter of heating tube (m); **t,** Fin thickness (m); **wR,**Uncertainty;e_{bi}, Black body radiation energy (W/m^2); **q,** Surface heat transfer per unit (W/m^2); **Qt**, Heat energy supplied from hot fluid (W); **Qc**, Convective heat transfer (W); **ΣQR**, Total radiative heat transfer (W); **Qw,** Amount of the heat transferred by air over heating tube (W); **Qf**, Amount of the heat transferred by air over fins (W); F_{ij} Shape factor between i surface and j surface; **h$_f$**, Heat convection coefficient around fin (W/m^2°C); **h$_w$,** Heat convection coefficient around body (W/m^2C); h$_0$, Visible heat convection coefficient (W/m^2); **A$_0$,** Total heat transfer area (m^2); **c$_{water}$**, Specific heat capacity of water (kJ/kg°C); **c$_{air}$,** Specific heat capacity of air (kJ/kg°C); **A$_w$,** Heating tube surface area except total fin surface area (m^2); **A$_f$,** Total area of the fins on the tube (m^2); **u$_m$,** Average speed of air (m/s); **T$_w$,** Temperature of heating tube surface (°C); To, Temperature of the heated air (°C); **T$_f$,** Fin temperature (°C); end **T$_f$** , Fin end temperature (°C);

$T_{water\,0}$, Input temperature of water to tube (°C); T_{water_0}, Output temperature of water from tube (°C);ΔT_{water} , Difference between the input and output temperature of water in heating tube (°C); ΔT_{air} , Difference between the input and output temperature of air from heat exchanger (°C); T_{air_i}Input temperature of air into heat exchanger (°C); T_{air_0}, Output temperature of air from heat exchanger (°C); m_{water} , Mass flow rate of water (kg/s); m$_{air}$, Mass flow rate of air (kg/s); V , Volumetric flow rate of air ($\frac{m^3}{h}$); G , Mass flow speed (kg/m 2 s); **Nu$_d$,** Nusselt number; **Re$_d$,** Reynolds number; $\overline{Re_d}$, Modified Reynolds number.

Greek symbols: $\mathcal{E}_i$, Emissivity coefficient for surface; λ, Heat conduction coefficient (W/m°C); λ_f, Fin material heat conduction coefficient (W/m°C); σ , Stefan-Boltzmann constant ($\frac{W}{m^2\,K^4}$); **η$_0$,** Finned-surface efficiency**; η$_f$,** Fin efficiency; , Kinematic viscosity (m^2 /s); μ, Dynamic viscosity (kg/ms). Subscripts: **t**, Total; **c,** Convection; **R**, Radiation; **w**, Wall; **f**, Fin.

REFERENCES:

- Karabacak R (1992). The effects of fin parameters on the radiation and free convection heat transfer from a finned horizontal cylindrical heater. Energy Conversion Manage.., 33: 997-1005.
- Karabacak R (1996). Experimental relationships for heat flux, Nusselts number and temperature difference in a finned heater. Energ. Conversion and Manage., 37: 591-597.
- Kayansayan N (1993). Thermal characteristics of fin – and – tube heat exchanger cooled by natural convection. Exp.erimental Thermal and Fluid Science. 7: 177-188.
- Leu JS, Liu MS, Liaw JS, Wang CC (2001). A numerical investigation of louvered fin and tube heat exchangers having circular and oval tube configurations. Int.ernational J.ournal of Heat and Mass Transfer. 44: 4235-4243.
- Souidi N, Bontemps A (2001). Countercurrent gas – liquid flow in plate – fin heat exchangers with plain and perforated fins. Int.ernational J.al of Heat and Fluid Flow. 22: 450-459.
- Wang CC, Hwang YM, Lin YT (2001). Empirical correlations for heat transfer and flow friction characteristic of harringbone wavy fin and tube heat exchangers. Int. J. Refrigeration. 25: 673-680.
- Wierzbowski M, Stasiek J (2002). Liquid crystal technique application for heat transfer investigation in a fin – tube heat exchanger element. Exp. Thermal and Fluid Sci., 26: 319-323.
- Xiao Q, Tao WQ (1990). Effect of fin spacing on heat transfer and pressure drop of two – row corrugated – fin and tube heat exchangers. Int. Commun. Heat Mass Transfer. 17: 577-586.
- Yakar G (2007). The effect of turbulence created in fin-tube type heat exchangers with perforated fin on heat transfer and pressure drop. PhD dissertation, Pamukkale University, Denizli, Turkey.
- Yakut K, Sahin B (2004). Flow – induced vibration analysis of conical rings Used for heat transfer enhancement in heat exchangers. Appl. Energy. 78: 273-288.
- Yan WM, Sheen PJ (2000). Heat transfer and friction characteristics of fin – and – tube heat exchangers. Heat Mass Transfer. 43: 1651- 1659.