Parametric Analysis of Heat Exchanger Using Computational Fluid Dynamics

B.Swathi[1], K.V.J.P.Narayana[2]

[1]*M.Tech student, V.K.R, V.N.B & A.G.K College of Engineering, Gudivada*

[2]*Asst.Prof, V.K.R, V.N.B & A.G.K College of Engineering, Gudivada*

Abstract-Air to air tubular cross flow heat exchanger used for cooling of large electric motors has been taken for study. The computational fluid dynamics analysis was done by using ANSYS FLUENT 15.0 on heat exchanger and obtained results were compared with experimental results. Then, the actual model was simulated by varying the cold air inlet velocities from 14 to 20 m/s using CFD. A computer program was developed using MATLAB programming based on analytical formulation of heat exchanger using ε-NTU technique for actual design. The results obtained from the MATLAB program at different velocities have shown close agreement with the CFD simulation results. Two models were also modeled by changing the geometrical parameters such as baffle position and number of tubes. Simulation was performed and results were compared with the actual model. It has been seen that there is not much change in temperature distribution of hot and cold air on shifting of baffle locations. It is found that with increase in cold air inlet velocities, there is not significant change in outlet temperatures of both hot and cold air, whereas pressure drop has shown parabolic increasing trend. Also, on addition of three additional tubes the values of temperatures of hot and cold air has decreased, so an optimum model with three additional cold air tubes and 14 m/s cold air inlet velocity was proposed and it was seen that the thermal performance of heat exchanger has increased even with lower cold air flow rates and pressure drops.

I. INTRODUCTION AND OBJECTIVES

1.1 Introduction

Cross flow heat exchangers are widely used in thermal systems. Based on construction, cross flow heat exchanger are two types: Tube with fins and tubes heat without fins. Fins are used with cross flow heat exchangers when there is huge difference between the heat transfer coefficients of hot and cold mediums, such as water and air. In such cases, fins are required to increase the heat transfer area along the air side to augment the heat transfer rate, but at the expense of power requirement; because finned type heat exchangers have more pressure drop as compared to without fin heat exchangers (Toolthaisong and Kasayapanand, 2013). Hence, fin is not a necessary installation in case of heat exchangers, when the heat transfer is occurring between the mediums having almost same heat transfer coefficients. There are also several other arrangements to increase the rate of heat transfer such as shape and arrangement tubes, position of baffles etc. According to Gomez et al., 2009, the thermal effectiveness of a cross flow heat exchanger could be upgraded by a more uniform temperature difference field. This can be attained in two different ways: either redistributing the heat transfer area or rearranging the connections between tubes. Different methods have been adopted by Chang et at., 2010, which aims at enhancing the fan performance by changing the geometry, redesigning new heat exchanger with guide vanes and optimizing the distance between axial fans.

Cross flow heat exchangers have a wide range of applications such as in air pre-heater, economiser, super-heater in thermal power plants, automotive radiator and cooling of large electrical motors, etc (Ishak et al., 2013). Air to air tubular cross flow heat exchangers are used for cooling of large electric motors in various industries. There is generation of enormous amount of heat due to energy loses in the windings of electrical motors at various loads. So, air is circulated across motor windings is cooled by using air to air tubular cross flow heat exchanger. These types of heat exchangers require low operational as well as maintenance cost due to use of air and simple tubes.

Although, the electric motors are the critical component of all the industrial systems (Gomez et al., 2009), but yet very little work has been carried to improve the cooling of electric motors. The improper cooling of motors leads to overheating, which leads to decrease in the performance of motors and sometimes may damage the windings of motors. Hence, the effort should be made to reduce the temperature of motors by providing effective cooling techniques. Therefore, the further research is needed to improve design an effective heat exchanger that would enhance the cooling rate and result in lower temperatures across windings of electric motors. Hence, the present study has been carried out to study the pressure, temperature and velocity distribution of hot and cold air in tubular cross flow heat exchanger using commercial code, Fluent. A MATLAB program was also developed for the actual model and results were validated with CFD results. Air to air cross flow tubular heat exchanger has been taken for simulation using Computational Fluid Dynamics (Kumar et al., 2003).

1.2 Objectives

In the context of above limitations the following specific objectives have been undertaken in the present study:

1. To simulate the air to air tubular cross flow heat exchangers by using CFD and to obtain temperature as well as pressure distributions for same.

2. To perform the simulation by changing the geometrical and flow parameters of heat exchanger to maximize the heat transfer rates.

3. Develop a model using MATLAB programming to design similar type of heat exchanger with similar type of mission for varying capacity.

II. METHODOLOGY AND SIMULATION PROCEDURE

Heat exchangers have been used in many industries for several decades. The use of tubular cross flow heat exchangers has become popular in last few years for cooling of large electric motors. From the comprehensive literature review, it has been found that the adequate research has not been done in the field of cooling of electric motors by air cooled tubular cross flow heat exchangers. Hence, an extensive simulation procedure has been performed in order to provide some effective designs for such heat exchangers.

2.1 Methodology

Simulation on cross flow tubular heat exchanger by using commercial CFD (Computational Fluid Dynamics) code Fluent was performed using following methodology:

- Simulation has been performed on cross flow tubular heat exchanger and results are validated using experimental results for actual geometry.
- Grid independent test has been performed and optimum grid size has been found for the same geometry.
- Again, simulation has been done using different model with optimum grid size and accurate turbulence model has been obtained for the same heat exchanger.
- Then, by using the optimum grid size and turbulence model, CFD simulation is performed by changing the locations of baffles, increasing number of tubes and other geometrical parameters of heat exchangers.
- After doing the above step, best design of heat exchanger has been obtained that could maximize the rate of heat transfer at lower costs and leads to proper and uniform cooling of electric motors at lower pressure drops.
- A computer program using MATLAB programming was developed for actual geometry and results obtained from the program were compared with CFD results.

2.2 Simulation procedure

The case study done by Kumar et al., 2003 at Crompton Greves Ltd. on cross flow tubular heat exchanger has been taken for simulation using Computational Fluid Dynamics. Results obtained from CFD simulation are compared with experimental results. Further some other design changes (such as changing baffle positions, number of tubes, cold air inlet velocities, etc) have been simulated and compared with actual design. The major geometrical dimensions and simulation procedure is explained below.

Geometrical Details:

Geometry of heat exchanger was modelled in ANSYS Workbench 15.0, geometrical details given in table 2.1. Geometry consists of twenty seven number of tubes housed in a staggered arrangement. The bottom faces of second and third section of rectangular enclosure are inlet of hot air; whereas, the bottom faces of first and fourth section are outlets of hot air. The cold air passes through the tubes.

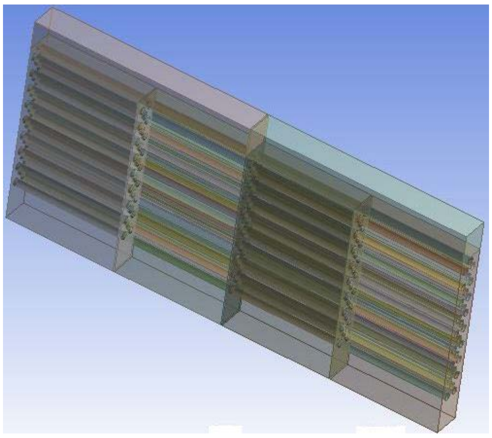

Figure 2.1: Geometry (3-D view)

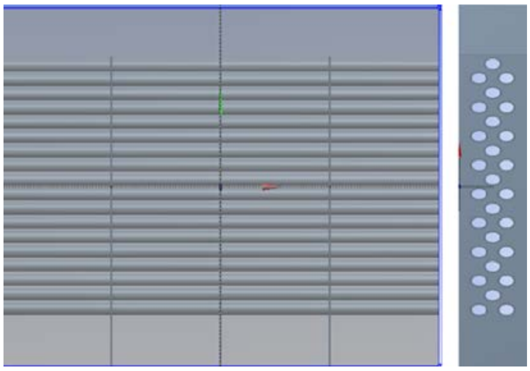

Figure 2.2: Front and side view of geometry

Table 2.1 Geometrical Details

SNO	DESCRIPTION	UNITS	DIMENSIONS
1	Overall dimensions	mm	1610 X 100 X 765
2	Inner tube diameter	mm	22
3	Outer tube	mm	26

	diameter		
4	Length of tube	mm	1610
5	Number of tubes		27
6	Transverse pitch	mm	61
7	Longitudinal pitch	mm	41

Modelling Assumptions:

To simulate the heat exchanger some assumptions were assumed and given below:

1. Heat leakage to surroundings and the effect of radiation losses are neglected.

2. Due to symmetry of geometrical construction of heat exchanger along its width, it is divided into two halves from the centre plane along the length. Only one half was taken for CFD simulation and symmetry boundary condition was applied along the sectioned plane.

Mesh Description:

Structured mesh was generated for the heat exchanger model. Non conformal mesh was generated for pipes and rectangular enclosure in order to decrease the total cell count which in turn effects the total computational time for the analysis. Inflation layers were generated near the walls in order to maintain the desired y+ value. Further details of the mesh are given in table 2.2.

Table 2.2Mesh description

Number of nodes	591023
Number of elements	543464
Average skewness	0.1847

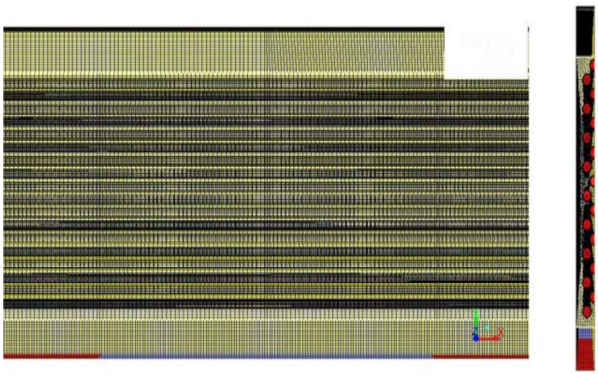

Figure 2.3: Meshed heat exchanger (Front view and side view)

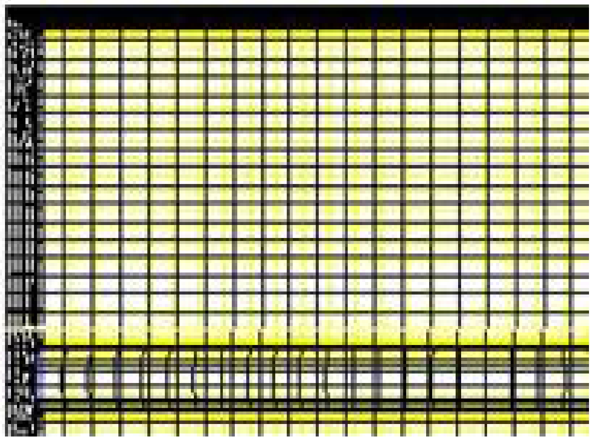

Figure 2.4: Magnified view of mesh

Properties of air and mild steel:

The hot air is coming from motor is cooled by the ambient air which is passed through the tubes of the heat exchanger. The heat exchanger is made of mild steel. Properties of hot, coldair and mild steel used for simulation are given in table 2.3. These properties are obtained at bulk mean temperatures of both hot and cold air.

Table 2.3 Properties of Air and Mild Steel

Property	**Unit**	**Hot air**	**Cold air**	**Mild steel**
Bulk mean temp	K	315.88	312.04	
Density	Kg/m^3	1.1086	1.121	7833
Specific heat	J/kg-k	1007.63	1007.48	465
Thermal conductivity	W/m-k	$27.475x10^{-3}$	$27.191x10^{-3}$	54
Dynamic viscosity	N-s/m2	$192.095x10^{-7}$	$190.287x10^{-7}$	N.A

Boundary conditions:

Velocity and temperature of hot and cold air were specified at the inlet. The turbulent intensity at the inlet was set as 2% because geometry was symmetrical, hence less turbulence occurs. The pressure value at the outlet of both cold as well as hot fluid was specified as zero because both hot and cold air coming out of heat exchanger is at ambient pressure. The thickness of tubes and baffles is neglected during geometry modelling. To account this, shell conduction boundary condition had been taken. The values of operati parameters were specified in Table 2.4.

Table 2.4 Boundary Conditions

Type of Fluid	Inlet Velocity (m/s)	Outlet Pressure, pa (guage)	Inlet Temperature (^{o}c)	Turbulence Intensity (%)	Hydraulic Diameter (m)
Cola Air	18	0	35	2	0.022
Hot Air	1.35	0	63	2	0.160

2.3 Description of different designs

In addition to simulation of actual design, two additional designs were obtained by changing the geometrical parameters such as location of baffles and number of tubes. All these designs were also simulated using CFD and their detailed description is given below.

Actual design:

Tubes are housed inside a rectangular enclosure, which is divided into four sections by using partition plates, internal baffles and end plates. End plates separate the hot air from the cold air. Partition plate divides the heat exchanger into two equal halves whereas internal baffles guide the path of hot air. The cold air flows through the tubes whereas the external surface of tubes is exposed to hot air. The partition plate distributes hot air in middle of two sections of heat exchanger and the cooled hot air leaves the heat exchanger from other two sections. This cooled hot air is re-circulated into the motor windings and the cold air coming out of tubes is discharged to atmosphere.

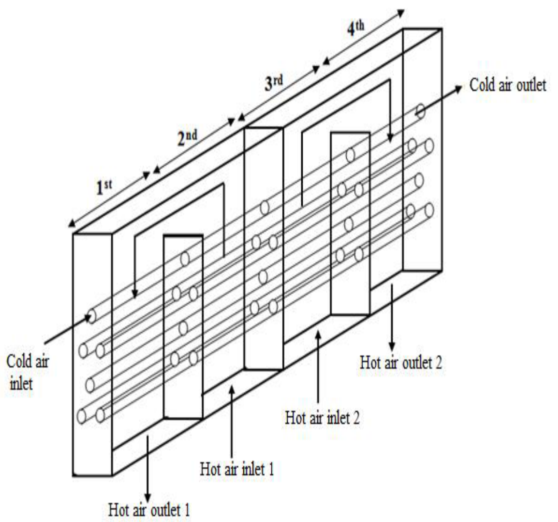

Figure 2.5: Schematic of heat exchanger

Change in baffle position:

As per the temperature drop obtained for first and second half of heat exchanger using CFD simulation, optimum ratios of heat transfer area was obtained using theoretical correlations.

Based on this, area and length of tubes were obtained for each half. Central baffle has been shifted towards right by 295 mm from the central plane. Further, each half of heat exchanger is divided into two parts of equal length. The amount of hot air passed through each section was also varied in the same proportion as that of area i.e. the first section of heat exchanger has more amount of air as compared to second section. The detailed geometrical description of this design is shown in Figure 2.6.

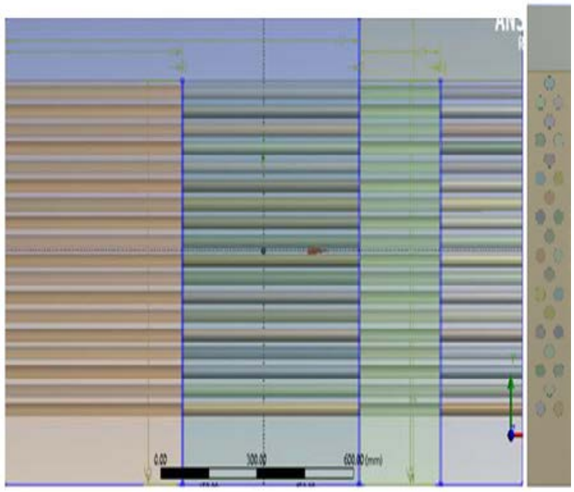

Figure 2.6: Change in baffle position design (Front view and side view)

Additional Three Tubes:

In this case, the cold air tube has been increased from 27 tubes to 30 tubes and simulation has been carried out. Due to increase in number of tubes, heat transfer area has increased which leads to higher heat transfer rate. Detailed view of Geometry is shown Figure 2.7.

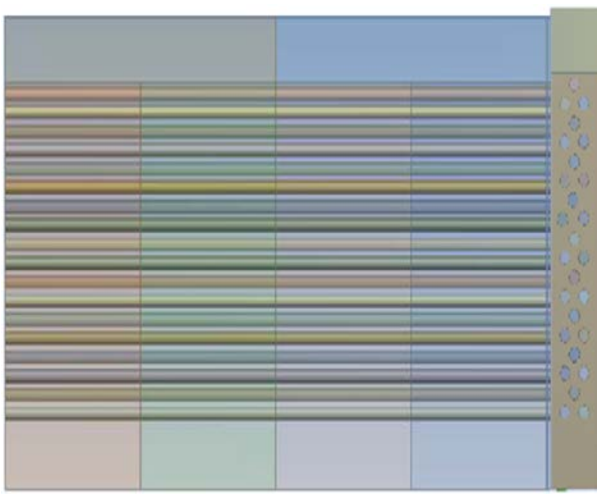

Figure 2.7: Additional three tubes design (Front view and side view)

Final design:

A final design based on the simulation results of actual model at different cold air inlet velocities and two models were suggested. From the results of velocity variation on actual model it has been found that with increase in the velocity there was not much drop in the temperature where as the pressure drop has increased drastically. Also by comparing the different designs it was observed that the addition of three tubes increases the area of heat transfer. Hence both of these effects were combined to obtain the final model which was operated at 14 m/s inlet cold air velocity with three additional tubes and the results obtained were compared with the actual model.

2.4 MATLAB programming

A program based on analytical formulation was developed for actual model. Results obtained from MATLAB program were compared with CFD simulation results at different cold air inlet velocities. A schematic of flow chart for the program is shown in Figure 2.8.

The software first evaluates the intermittent temperatures of the heat exchanger. Then it will consider the first leg of the heat exchanger and calculate the heat dissipation rate and outlet temperature of both the streams. Then taking the proper inlet temperatures, the heat dissipation rate and the outlet temperature of both the streams of second leg of the heat exchanger will be evaluated. Similarly the performance of other two legs of exchanger will be performed. Finally it totalises the heat dissipation rate of the exchanger (Gangacharyulu etal., 2004).

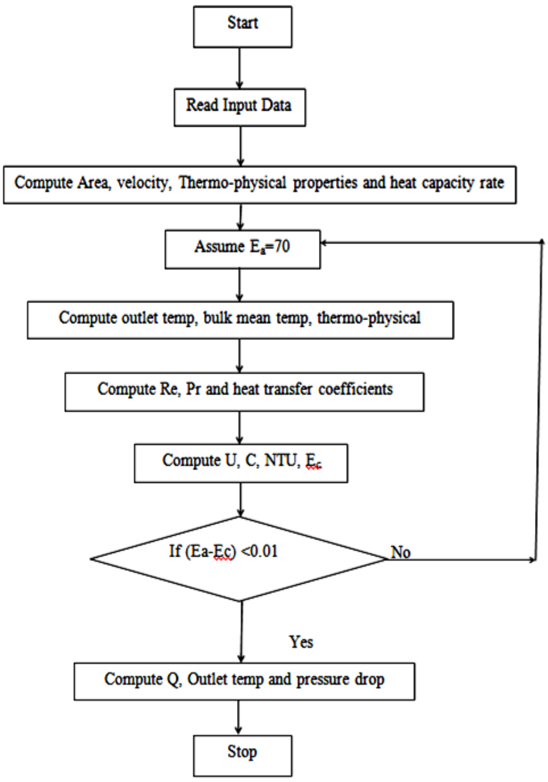

Figure 2.8: Flow chart of MATLAB program

III. CFD SIMULATION RESULTS AND COMPARISION OF DIFFERENT DESIGNS

The use of tubular cross flow heat exchangers for cooling of large electric motors has been increased in past few decades using air. In spite of its applications in almost all industrial applications, not much work has been reported to date on such type of heat exchangers. So, effort has been made here, to simulate such systems using CFD and results of simulations are presented. Comparison is also made among different designs by varying the geometrical parameters.

3.1 Effect of cold air inlet velocity

Inlet velocity (mass flow rate) of cold air has been varied for actual design and its effect has been studied on outlet temperatures, heat transfer coefficient and pressure drop of hot and cold air. Inlet velocity of the cold air has been varied from 14 m/s to 20 m/s in order to study the effects on the outlet temperatures. The curves between the temperature and cold air velocity are shown in Figure 3.1 for cold and hot air outlets. It can be seen from the graph, as the inlet air velocity has increased, the temperature at all the outlets has decreased.

This is due to the fact that with increase in the velocity the heat transfer coefficient goes up which leads to increase in the heat transfer rate. A decrease of 1.83%, 2.598%, 4.78% has been found in outlet temperature of hot air outlet 1, hot air outlet 2 and cold air outlet, respectively, when velocity of cold air was varied from 14 to 20 m/s.

Table 3.1 CFD simulation results (temperature and pressures)

Velocity (m/s)	Temperature (^{o}C)		
	Cold outlet	Hot outlet 1	Hot outlet 2
14	43.47992	48.73804	51.77844
16	42.24738	48.4747	51.31119
18	41.9433	48.1651	50.85605
20	41.40912	47.85962	50.43283

Velocity (m/s)	Pressure (Pa)		
	Cold outlet	Hot outlet 1	Hot outlet 2
14	238.9795	44.39334	45.2314
16	2823258	44.39334	45.23142
18	342.9379	44.39335	45.23142
20	411.9749	44.39337	45.23141

Velocity (m/s)	Pressure (Pa)	
	Cold outlet (simulation results)	Cold outlet (Darcy equation)
14	238.9795	203.73
16	282.3258	259.83
18	342.9379	314.29
20	411.9749	384.58

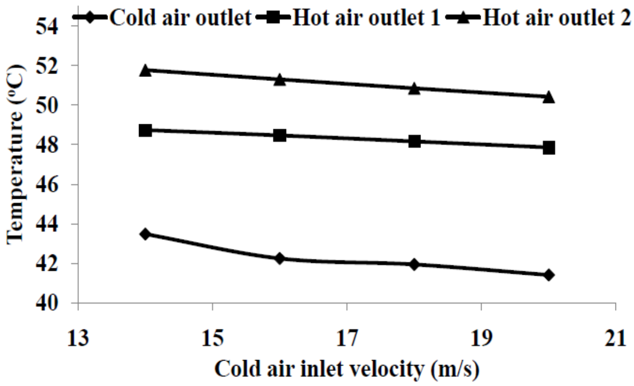

Figure 3.1: Temperature versus cold air inlet velocity

Figure 3.2, shows the variation for the pressure drop through the tubes for cold air with variation in inlet velocities.

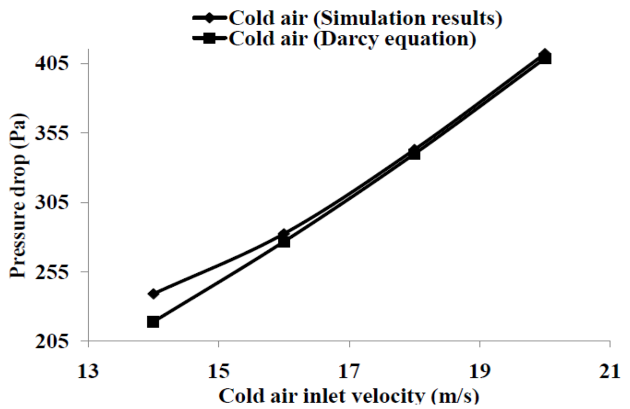

Figure 3.2: Pressure drop versus cold air inlet velocity

The pressure drop has shown a parabolic variation with increase in the inlet air velocities. This pressure drop obtained from CFD simulation has shown close agreement with theoretically obtained pressure drop. An over-prediction of 9.52% was obtained in CFD simulation results at cold air inlet velocity of 14 m/s. The pressure drop has increased by 72.39% when the inlet velocity was varied from 14 m/s to 20 m/s. It can be concluded from both the graphs that it is not worth to increase the cold air velocity because heat transfer rate is enhanced by small margin whereas huge increase in pressure drop was found which leads to increase in the requirement of pumping power and operational cost. Also, another reason for lesser temperature drop might be due to less time available for heat transfer due to increase in the velocity. The increased air velocities also lead to increase in the tube vibrations which increases the maintenance cost also.

Figure 3.3, shows the variation of heat transfer coefficient with change in velocity. It can be seen from the graph that with increase in velocity there is increase in heat transfer coefficient. Both the CFD simulation curve and theoretical curve shows the same trend. A decrease of 24.82% and 21.09% was obtained for theoretical and simulated heat transfer coefficient when air velocity was decreased from 20 m/s to 14 m/s.

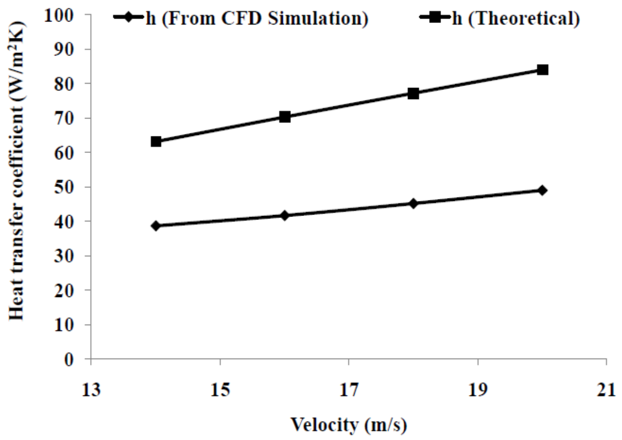

Figure 3.3: Heat transfer coefficient versus velocity

3.2 Comparison of different designs

Three designs were compared with the actual design by keeping all the parameters and boundary conditions fixed other than making geometrical changes. Figure 3.2 to 3.5 shows the outlet temperatures obtained for different designs.

Temperature variations:

It can be seen from Figure 3.4 that outlet temperatures at hot outlet 1 has decreased from the actual design when baffle positions has been shifted and three additional tubes were used.

Table3.2: MATLAB results (temperature and pressure)

Velocity (m/s)	Temperature (^{o}C)			Pressure (pa)
	Hot outlet 1	Hot outlet 2	Cold outlet	Cold inlet
14	53.12	55.64	45.87	203.73
16	52.97	55.24	44.66	259.83
17.83	52.92	54.98	43.73	314.29
20	52.93	54.77	42.78	384.58

Table 3.3: Comparison of different designs

Model	Hot Fluid Temperature (^{o}C)				Cold Fluid Temperature (^{o}C)		pressure Drop (pa)		
	Inlet 1	Outlet 1	Inlet 2	Outlet 2	Inlet	Outlet	Hot Inlet 1	Hot Inlet 2	Cold Fluid
Actual Design	63	48.16	63	50.86	34.4	41.94	44.39	45.23	342.93
Baffle Positi	63	46.34	63	51.23	34.3	42.05	57.14	28.98	340.83

on Change									
3 Additional Tubes	63	45.72	63	49.51	34.40	42.13	47.43	46.43	340.09

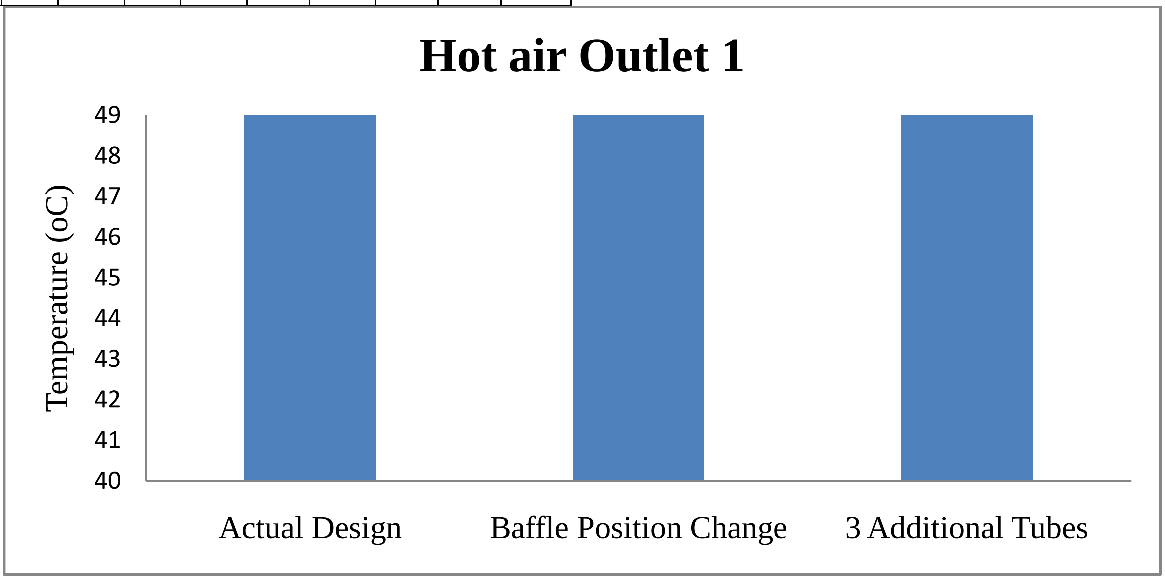

Figure 3.4: Hot air temperatures at outlet 1 for different designs

On shifting of the baffle positions the temperature has decreased due to increase in the area and time available for heat transfer. On addition of three additional tubes the area available for heat transfer has increased which causes decrease in outlet temperature of hot air.

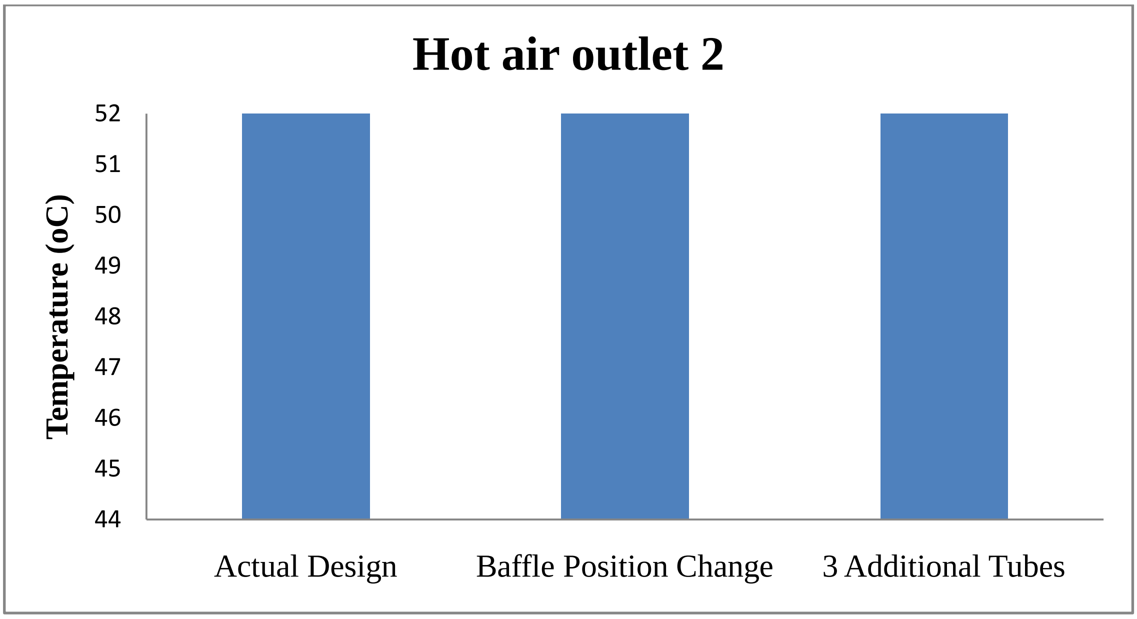

Figure 3.5: Hot air temperatures at outlet 2 for different designs

It can be seen from Figure 3.5 that, outlet temperature at hot air outlet 2 has increased due to change in the baffle position. A decrease in the hot air outlet 2 has been seen in case of three additional tubes. On shifting the baffle position the outlet temperature of hot outlet 2 has increased due to decrease in the area of heat transfer. On addition of three additional tubes, the area available for cold air heat transfer has increased because of which there is decrease in the outlet temperature at hot outlet 2.

It can be seen from Figure 3.6 that, cold outlet temperature has increased in case of three additional tubes because the heat transfer area has increased due to increase in the number of rows and hence increases rate of heat transfer.

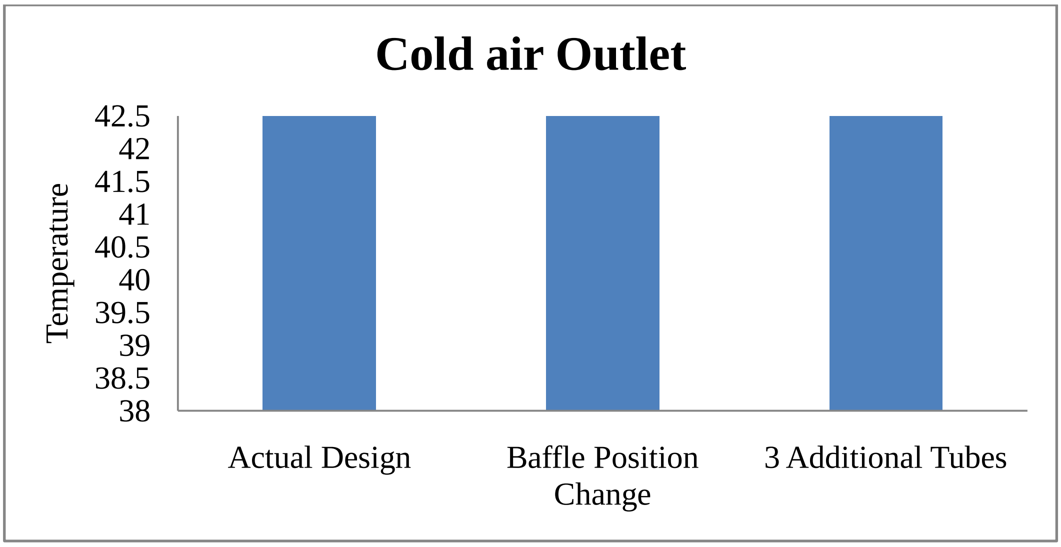

Figure 3.6: Cold air temperatures at outlet for different design

The outlet temperature of the cold air in case of change in the baffle position is almost same because the area of heat exchanger is constant irrespective of change in the baffle position.

Pressure drop variation:

Figures 3.7 to 3.9 represent the pressure drop for hot and cold air for all the designs. It can be seen from figure 3.7 that on shifting of central baffle position, the pressure drop has increased in comparison to actual design. This is because the section is handling moreamount of hot air in comparison to other designs. A marginal increase in pressure drop is also obtained on addition of three additional tubes.

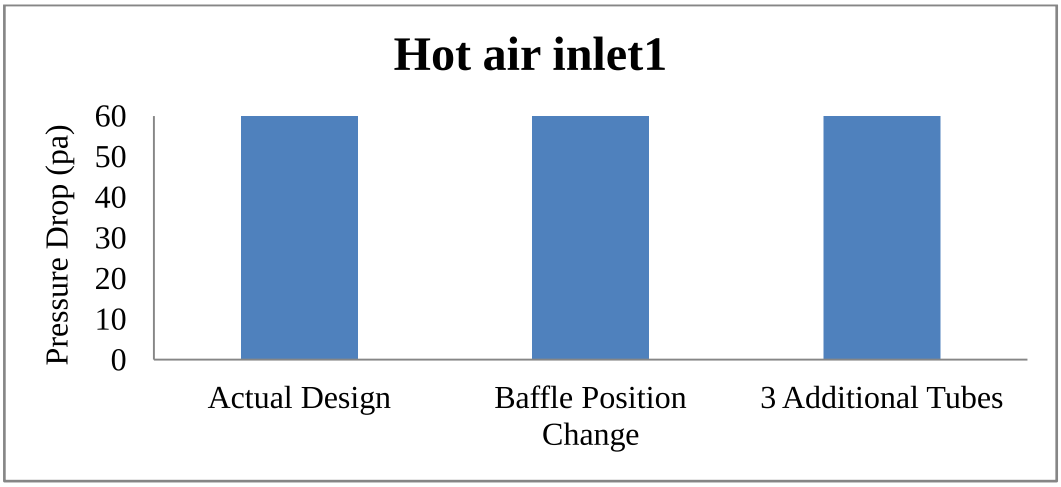

Figure 3.7: Pressure drop for first section of hot air

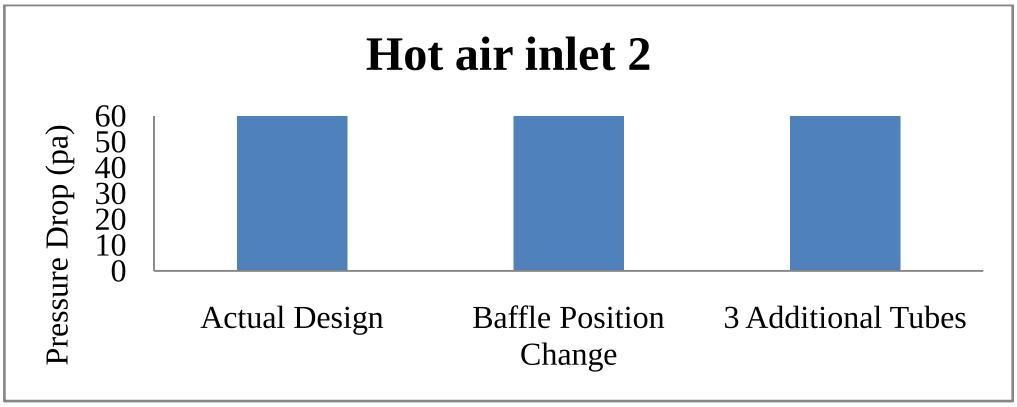

Figure 3.8: Pressure drop for second section of hot air

For second section of heat exchanger (Figure 3.8), baffle shifting model has shown minimum pressure drop because the amount of hot air for this section was very small in comparison to air handled by other models. A marginal increase has been observed in pressure drop on three additional tubes.

It is clear from Figure 3.9 that, there is not much variation in cold air pressure drop. Pressure drop has decreased for both the models. The decrease in pressure drop on addition of three tubes is obtained because, same amount of air is distributed in more number of tubes, which has decreased the air velocity in tubes and velocity has overcome the effect of additional tubes because pressure drop is directly proportional to square of velocity.

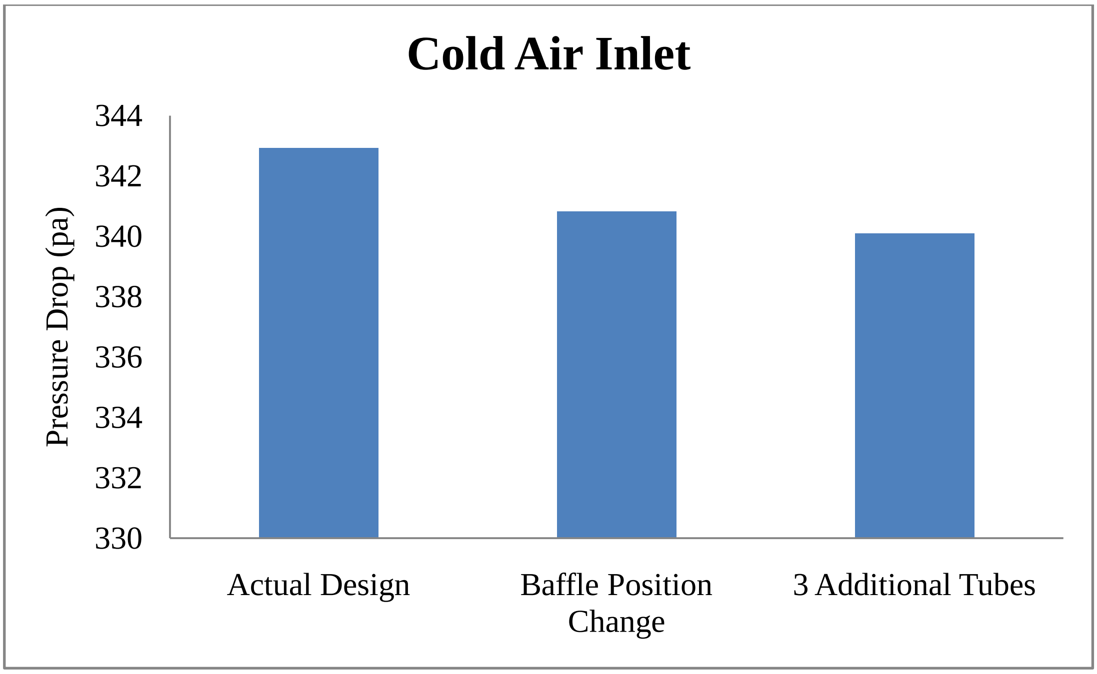

Figure 3.9: Pressure drop for cold air

3.3 Pressure and temperature distributions

Pressure distribution:

The pressure of cold fluid in the tube is higher at inlet and decreases as it moves towards outlet (Figure 3.10). The high pressure gradient along the tube length is due to friction between air and the tube wall. A small amount of negative pressure is observed nearby the outlet of cold fluid due to jet effect of cold fluid exiting from the tube.

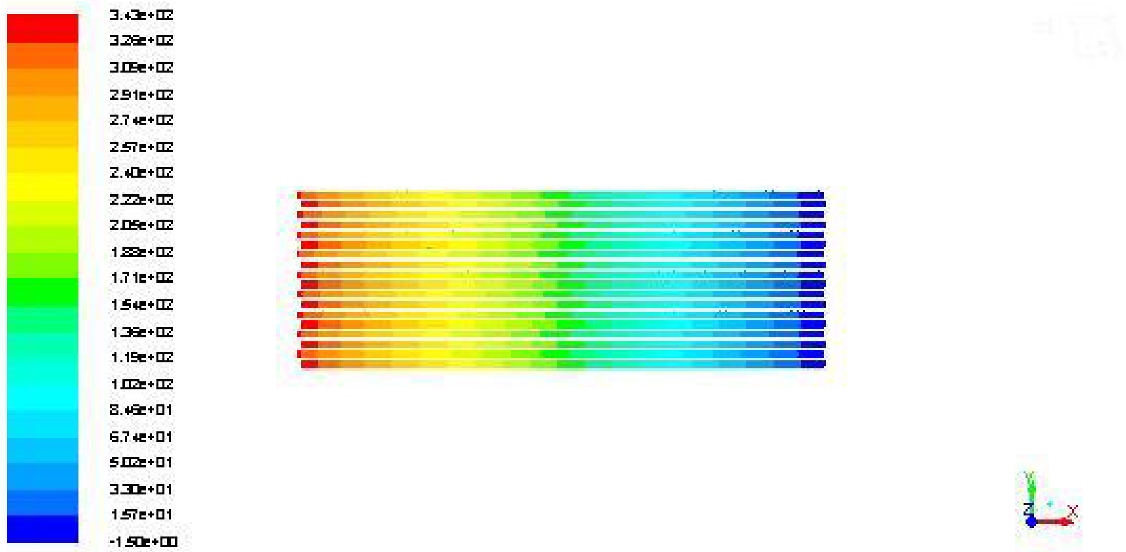

Figure 3.10: Pressure distribution of cold air

The pressure gradient is gradual in the hot stream (Figure 3.11). The pressure drop across tube bundle is about (45.30 Pa) as compared to the pressure drop in tube side along its length, which may be of the order of 342.93 Pa. At the inlet of hot air, more pressure is observed due to tube bundle and the pressure decreases, as the hot air moves up. In the second section, the pressure is low near the exit from second section to first section. The pressure decreasesfrom top to bottom in the first section. The similar behaviour is observed in the third and fourth section of the heat exchanger. There is appearance of high pressure zones near the top corners of first and fourth section due to stagnation of air.

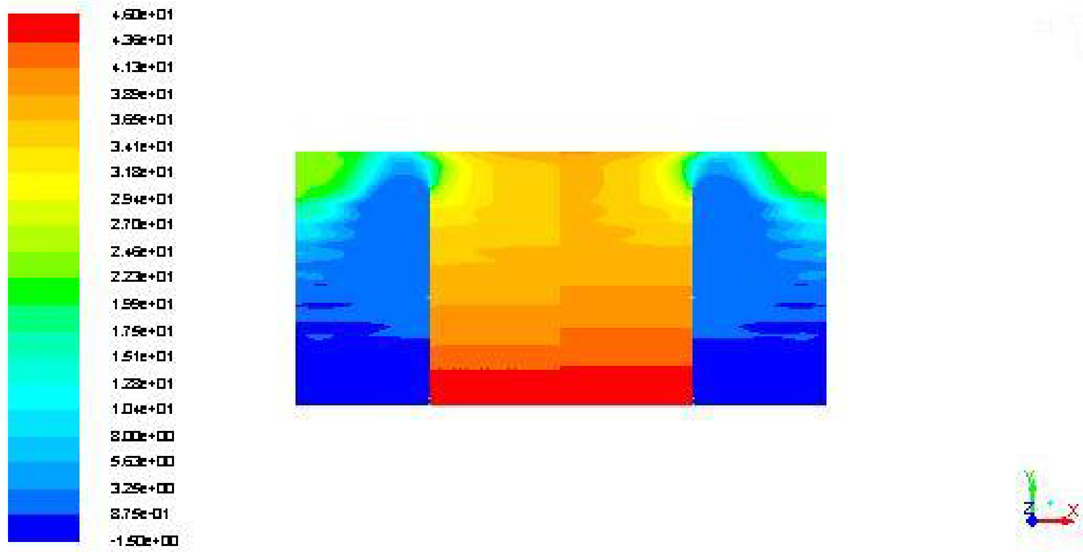

Figure 3.11: Pressure distribution of hot air

Temperature Distribution:

Fig. 3.12 shows the temperature distribution of heat exchanger in a vertical plane along its length. The temperature plot reveals that the hot air looses heat as it moves upward and the heat is gained by the cold fluid. Inside the tube, the temperature of cold air increases along its length as it picks up the heat from hot fluid while moving towards outlet. At the outlet, the exit air coming from bottom tubes are hotter than that of the exit air coming from top tubes, as the heat sources are located at the bottom.

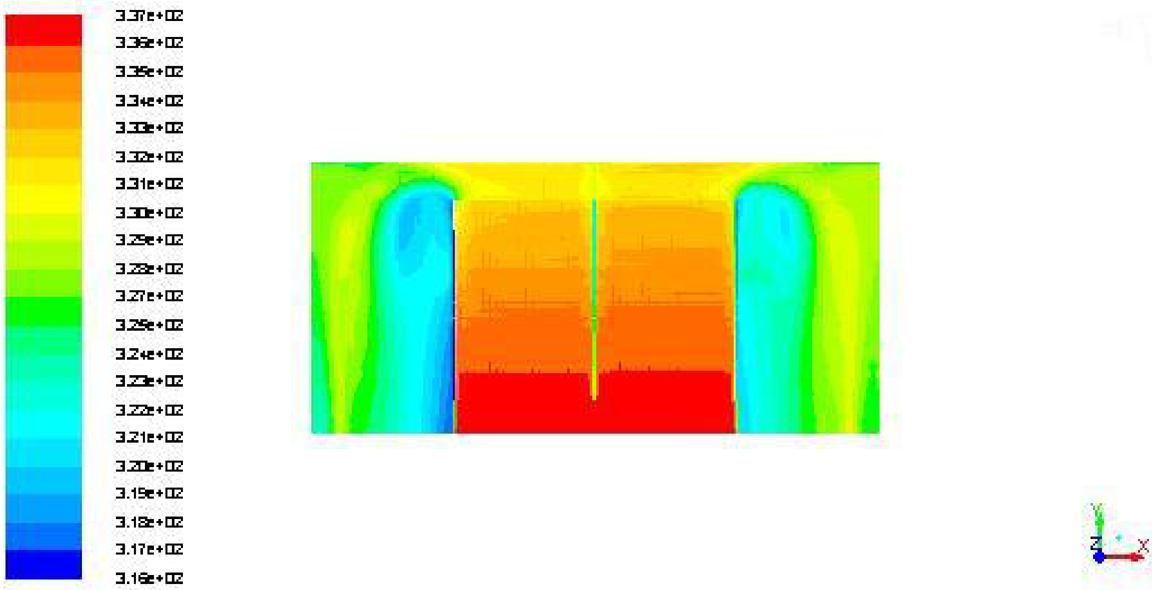

Figure 3.12: Temperature distribution of hot air

The temperature drop of hot air in the first section of heat exchanger is high (14.84 ^{o}C) because of high temperature difference between cold air and hot air (Figure 3.13).

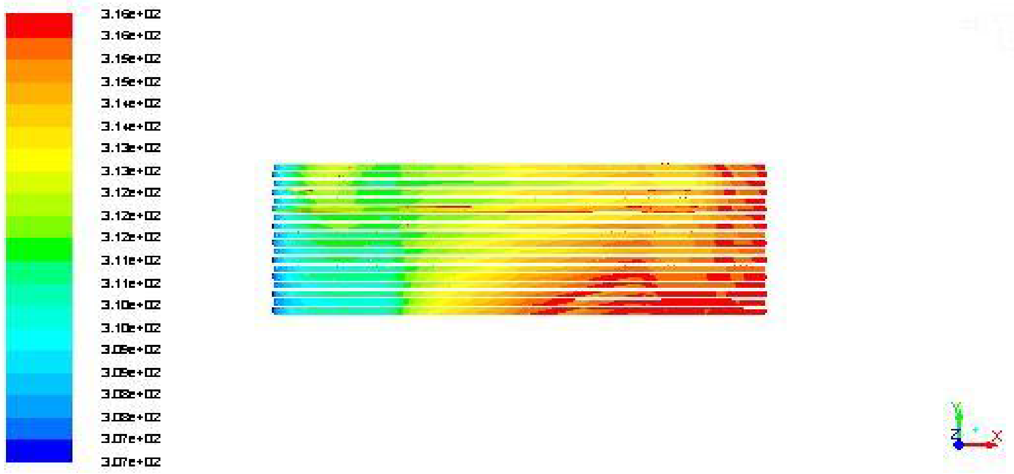

Fig. 3.13: Temperature distribution of cold air

In the fourth section, the temperature drop of hot air is low (12.15 ^{oC}). The reason behind this observation is that the cold air has already gained heat from second section;therefore its capacity to extract further heat from hot air in third section has been reduced. This non uniform temperature distribution pattern of air limits the performance of heat exchanger and in turn the performance of electrical motor.

Velocity distribution of hot air:

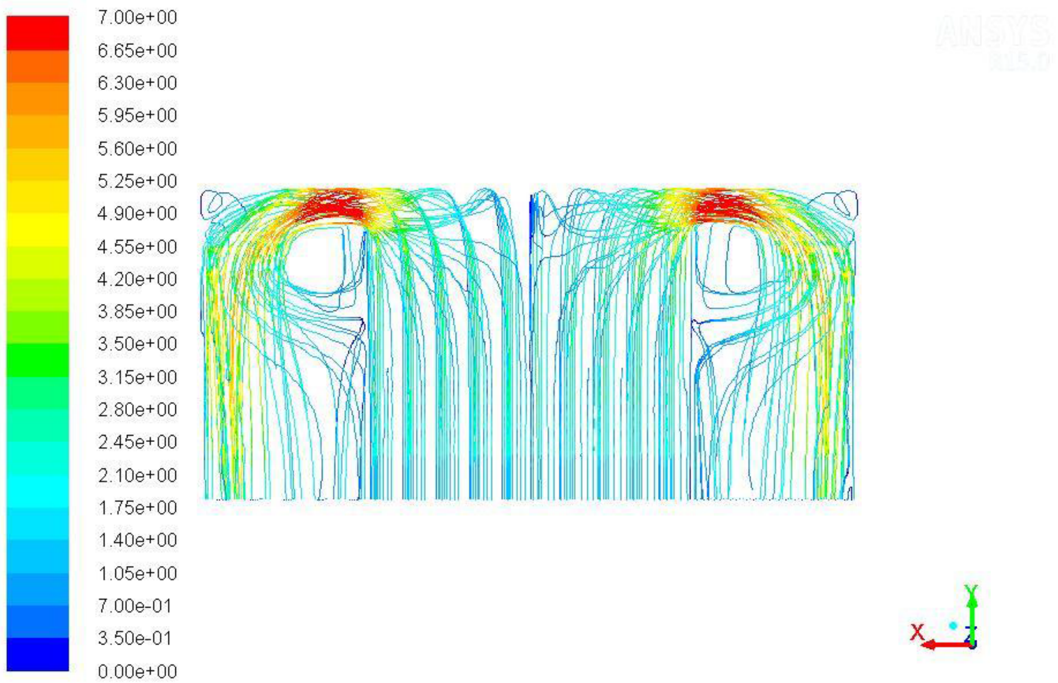

Figure 3.14: Velocity distribution for hot air for actual design

Figure 3.14, shows the velocity distribution of hot air. It can be clearly seen that air is moving upwards at constant velocity and as it crosses the tubes, the velocity of air increases drastically as it moves from one section to another. This increase in the velocity is due to less area for circulation for air at the top of heat exchanger. Hot fluid recirculation has been observed at the top corner of 1st and 4th section. There is some vacant area near the baffles where no air has passed. This is due to high momentum of air which causes air to move outwards. As air passes further through the section some of the air moves towards the vacantarea near the baffle, this is because there is low pressure in the vacant region which pulls the air towards it.

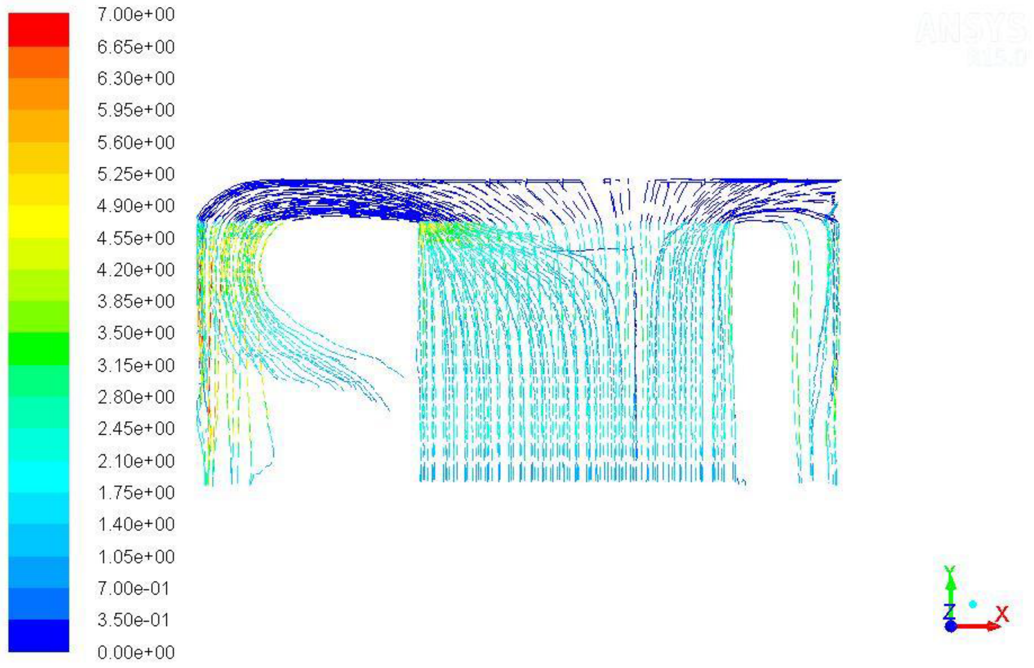

Figure 3.15:Velocity distribution of hot air for baffle position change model

Figure 3.15, shows the velocity distribution of hot air for change in baffle position. It has been seen that because of change in baffle position more air has remained stagnant in the top portion of the heat exchanger. It can be observed that air is taking turn near the outer baffles which causes more area to be vacant near the vicinity of the internal baffles. This causes improper utilization of area of heat exchanger.

3.4 Comparison of actual and final design

From the results of velocity variation on actual model it has been found that with increase in the velocity there is not much drop in the temperature whereas the pressure drop has increased drastically. Also by comparing the different designs it is observed that the addition of three tubes increases the area of heat transfer which leads to increase in heat transfer rate. Hence both of these effects are combined to obtain the final model which was operated at 14 m/s inlet cold air velocity with three additional tubes and the results obtained are compared with the actual model.

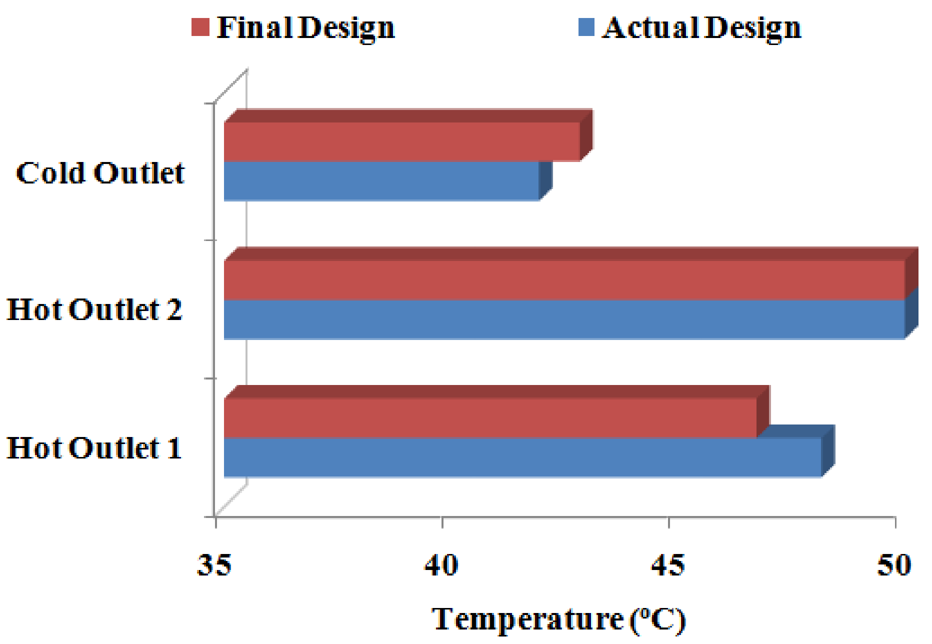

Figure 3.16: Comparison of outlet temperatures between actual and final design

Figure 3.16, shows the comparison of outlet temperatures of cold and hot air of actual design with the final design. It can be seen from the graph that the temperature of the hot outlet 1 has decreased when compared to the actual model whereas the temperature of hot outlet 2 is almost same for both the actual as well as final design. The temperature of the cold air outlethas increased for final design in comparison to the actual design because of more amount of area and time available for heat transfer.

Figure 3.17, shows the comparison of pressure drop between actual and final design for hot air. It can be seen from the graph that there is slight increase in pressure drop of hot air at both the inlets in final design as compared to actual model. This is because of increase in number of cold air tubes from 27 to30 which causes hindrance in flow of hot air which further leads to increase pressure drop.

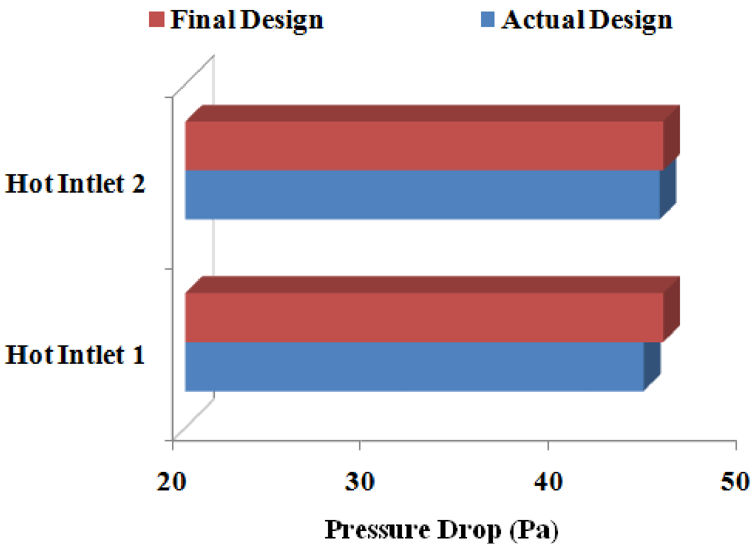

Figure 3.17: Comparison of pressure drop between actual and final design for hot air

Figure 3.18, shows the pressure drop of actual and final design for cold air. It can be seen that there is huge decrease in pressure drop in final design as compared to actual design. The reason for drop in pressure is due increase in number of cold air tubes from 27 to 30 and also due to decrease in cold air velocity from 18 m/s to 14 m/s. The increase in number of cold airtubes decreases the mass flow rate of cold air passing per tube in the final design when compared with the actual design. The decrease in mass flow rate also decreases the velocity of the cold air passing through the tube. Hence, both the changes directly results in drop in pressure. There is some increase in the pressure in the final design due to increase in number of tubes but its effect was overcome by effect of decrease in velocity as pressure drop is directly proportional to square of velocity (from Darcy Equation).

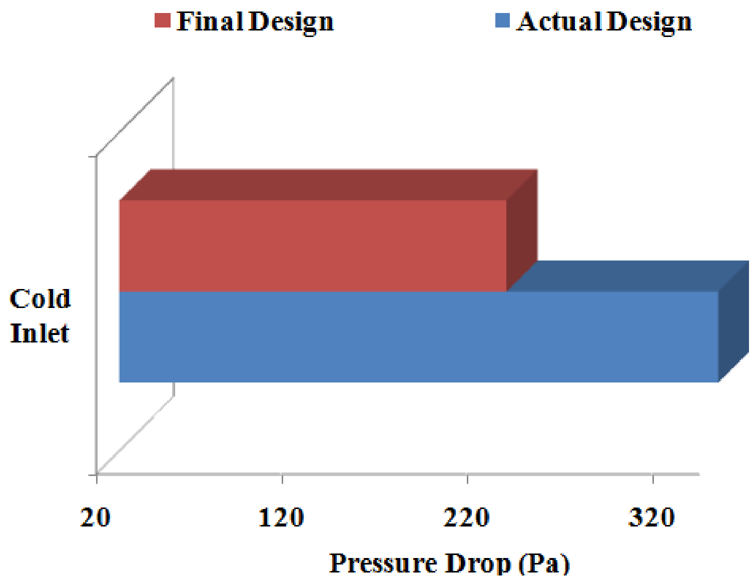

Figure 3.18: Comparison of pressure drop between actual and final design for cold air

With the final proposed design, more effective cooling of hot air was done with 13.52% lesser amount of cold air. Although, there is slight increase in pressure drop of hot air but the decrease in pressure drop of cold air is much significant. So, the effective cooling of the motor could be done at lower pumping power, which in turn decreases the operational cost.

3.5 Comparison of CFD and MATLAB results

Computer Program was developed based on the analytical formation using MATLAB programming for actual design. Geometrical and flow parameters at inlet are supplied as input to the program and the values of thermo-physical parameters of heat exchanger are obtained as output from the program.

Figure 3.19, shows the comparison of outlet temperature between CFD and MATLAB results between hot outlet 1, hot outlet 2 and cold outlet.

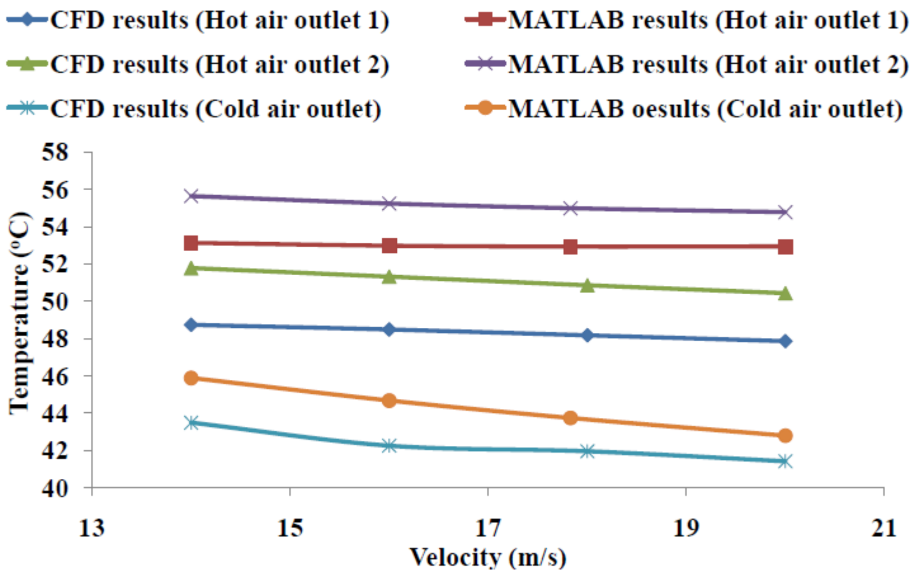

Figure 3.19: Comparison of outlet temperatures between CFD and MATLAB results

Results shown by CFD and MATLAB are in close context with each other for cold air outlet. A maximum error of 5.2% was observed for cold air outlet in between CFD and MATLAB results. An error of 9% has been seen in the results of CFD and MATLAB for hot outlet 1and hot outlet 2, this is due to neglecting of convective heat transfer coefficient at the top of heat exchanger during MATLAB programming.

Figure 3.20, shows the comparison of pressure drop between CFD and MATLAB results for cold air. Both CFD and MATLAB results show the same trend i.e. with increase in velocity of air pressure drop inside the tube increases.

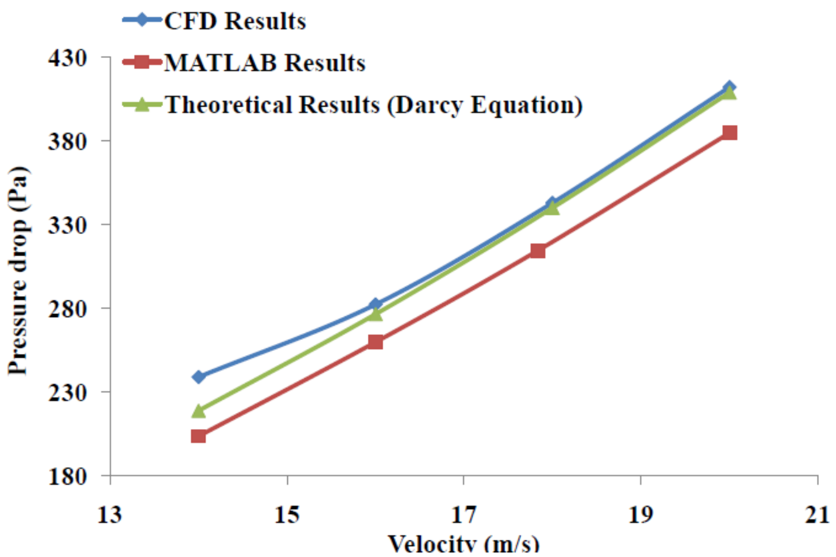

Figure 3.20: Comparison of pressure drop between CFD and MATLAB results

Table 3.4: MATLAB results (heat transfer through different sections)

Velocity (m/s)	Heat transfer (W)				
	Section 1	Section 2	Section 3	Section 4	Total
14	542.78	584.88	504.64	334.84	1967.1
16	544.53	599.4	527.18	357.69	2028.8
17.83	543.27	606.45	540.86	373.61	2064.2
20	539.31	609.4	551.01	388.1	2087.8

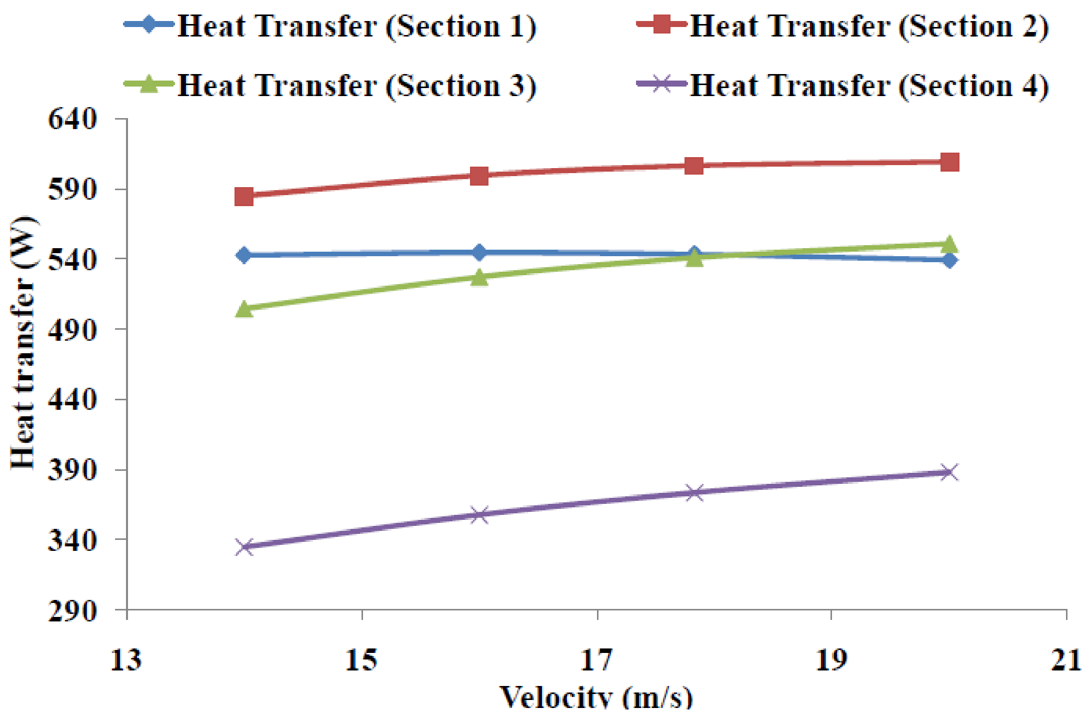

Figure 3.21: Comparison of heat transfer between different sections

Figure 3.21, shows the comparison of heat transfer between the four sections of the heat exchanger. It can clearly seen from the graph that the maximum heat transfer is occurring in section 2 of the heat exchanger. This is because of the fact that in section 2, inlet of hot air takes place, hence there is maximum temperature difference between hot and the cold fluid. Although, section 3 was also the inlet for the hot air but heat transfer in section 3 is less as compared to section 2. This is because the temperature of the cold air has increased while it reaches the third section; hence the temperature difference between the hot and the cold air is less as compared to second section and so is the heat transfer. Heat transfer in case of section 1 and section 3 is almost same. Section 4, shows the least heat transfer because the cold air has almost reached its maximum temperature and the temperature difference between the hot and cold air is very less and hence very less heat transfer.

IV. CONCLUSIONS

The CFD simulation of the air to air tubular cross flow heat exchanger has been performed and following conclusions were made.

It is found that with increase in velocity of cold air, the temperatures of the hot air decreases due to increase in the heat transfer coefficient.

The maximum decrease of 4.78% has been obtained in outlet temperature, whereas an increase of 72.39% is observed in pressure drop, when velocity was increased from 14 m/s to 20 m/s. So, 14 m/s cold air inlet velocity was found optimum for operation of heat exchanger.

Results obtained from MATLAB program has shown close agreement with the CFD simulation results for different cold air inlet velocities.

It is found the change in location of baffles does not affect much to the outlet temperatures of hot and cold air.

The addition of three more tubes has resulted in decrease of outlet temperatures of hot air as well as in pressure drop cold air without having much effect on hot air pressure drop. This is due to increase in area and time available for heat transfer.

From the velocity distribution, it has been seen that hot air moves from one section to another section of heat exchanger, due to which some area of heat exchanger faces starvation of air near the vicinity of internal baffles. This vacant region is maximum in case of baffle change design.

It has been observed from the temperature distribution that heat transfer is more effective in the first half of the heat exchanger in comparison to the second half. This is due to the availability of more cold air in first half, because cold air gains some heat before it enters to the second half. Hence, the temperature of hot air at outlet 2 is always more as compared to temperature of hot air outlet 1.

A final design three additional tubes and cold air velocity of 14 m/s was found to give better results than actual design with even lesser cold air pressure drop. A decrease of 2.98%is observed in temperature at hot outlet 1 and decrease of 33.29% in cold air pressure drop with proposed design in comparison to actual design

V. FUTURE SCOPE OF WORK

The use of air to air tubular cross flow heat exchanger is a growing technology and there are many areas in which further work can be carried out:

(1) Modification in other geometrical parameters such as longitudinal and transverse pitch, diameter of tubes can be varied to further improve the design.

(2) Additional baffles or arrangements can be used to make the flow distribution uniform in first and fourth section of heat exchanger.

(3) Analytical techniques can also be employed to investigate pressure and temperature distribution.

REFERENCES

[1]. Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., Studiyo, R. and Wachem, B. V. (2012).Computational Fluid Dynamics for Engineers.Publ. Cambridge University Press.1st Ed.
[2]. Aquaro, D. and Pieve, M. (2007). High Temperature Heat Exchangers for Power plants: Performance of Advanced Metallic Recuperators. Applied Thermal Engineering. 27: 389-400.
[3]. Brandt, H. (1934). Pressure Drop and Heat Transfer in Tube Heat Exchanger.Dissertation Hannover Polytechnic.
[4]. Buckinx, G, Rogiers, F. and Baelmans, M. (2013).Thermal Design and Optimization of Small Scale High Effectiveness Cross-Flow Heat Exchangers.Heat and Mass Transfer. 60: 210-220.
[5]. Chilton, T. H., and Generaux, R. P. (1933).Pressure Drop across Tube Bank.America Institute Chemical Engg. 29: 161-173.
[6]. Chumpia, A. and Hooman, K. (2014).Performance Evaluation of Single Tubular Aluminium Foam Heat Exchangers.Applied Thermal Engineering. 66: 266-273.
[7]. Chang, C. C., Kuo, Y.F., Wang, J. C. and Chen, S. L. (2010).Air Cooling for Large Scale Motors.Applied Thermal Engineering. 30: 1360-1368.
[8]. Dixit, T. and Ghosh, I. (2013). Two Stream Cross Flow Heat Exchangers in Thermal Communication with the Surroundings – A Generalized Analysis. Heat and Mass Transfer. 66: 1-9.
[9]. Du, X.P., Zeng, M., Dong, Z.Y. and Wang, Q.W. (2014). Experimental Study of the Effect of Air Inlet Angle on the Air-Side Performance for Cross-Flow Finned Oval-Tube Heat Exchangers. Experimental Thermal and Fluid Science. 52: 146-155.
[10]. Farsane, K., Desevaux, P. and Panday, P. K. (2000). Experimental Study of Cooling of Closed Type Electric Motor.Applied Thermal Engineering. 20: 1321-1334.
[11]. Gangacharyulu, D., Sharma, R. K., Dora, K. B. and Kumar, A. (2004). Thermal Performance Evaluation of Closed Air Circuit Aircooled (CACA) Heat Exchangers for HighRated Electrical Motors- A case study. International Journal of Heat Exchangers.International Journal of Heat Transfer. 5: 221-238.
[12]. Gnielinski, V. (1976).New International chemical Engineering. 16: 359-368.
[13]. Gomez, L. C., Navarro, H. A., Godoy, S. M., Campo, A. and Saiz-Jabardo, J. M. (2009).Thermal Characterization of a Cross-Flow Heat Exchanger with a New Flow Arrangement.Thermal Sciences. 48: 2165-2170.
[14]. Holman, J.P. (1996). Heat Transfer. Publ. McGraw-Hill. 8th Ed.
[15]. Incropera, F.P. and Dewitt, D.P. (2002).Fundamentals of Heat and Mass Transfer.Publ. John Wiley & Sons.5th Ed.
[16]. Ishak, M., Tahseen, A. and Rahman, M.M. (2013).Experimental Investigations on Heat Transfer and Pressure Drop Characteristics of Air Flow over a Staggered Flat Tube Bank in Cross Flow.Automotive and Mechanical Engineering. 7: 900-911.
[17]. Kast, W. (1974). Pressure Drop in cross flow across Tube Bundles. VDI- warmeattas, Section Ld, 2nd edn.
[18]. Kumar, V., Gangacharyulu, D., Rao, M.S., and Barve, R.S. (2003-2004). CFD Analysis of Cross Flow Air to Air Tube Type Heat Exchanger.Phoneics Journal of Computational Fluid Dynamics and its Applications.16-17 UK.
[19]. Navarro, H. A. and Gomez, L. C. (2005).A New Approach for Thermal Performance Calculation of Cross Flow Heat Exchangers.Heat and Mass Transfer. 48: 3880-3888.
[20]. Petukhov, B. S., Harnett, J. P., and Irvine T. F. (1970). Heat Transfer and friction in turbulent pipe flow with variable physical properties, advance in Heat Transfer. Academic, New York.6: 1970.
[21]. Petukhov, B. S., and Popov, V. M. (1963).Theoretical calculation of Heat Recovery and frictional resistance in turbulent flow in tubes of an incompressible fluid with variable physical properties, High Temperature. 1: 69-83.
[22]. Pongsoi, P., Pikulkajorn,S., Wang, C.C. and Wongwises, C. (2012). Effect of Number of Tube Rows on the Air-Side Performance of Crimped Spiral Fin-and-Tube Heat Exchanger with a Multipass Parallel and Counter Cross-Flow Configuration. Heat and Mass Transfer. 55: 1403-1411.
[23]. Sano, Y., Kuwahara, F., Mobedi, M. and Nakayama, A. (2012).Effects of Thermal Dispertion on Heat Transfer in Cross Flow Tubular Heat Exchangers.Heat and Mass Transfer. 48: 183-189.
[24]. Sieder, E. N. and Tate, G. E. (1936). Heat Transfer and Pressure Drop of liquids in tubes. Ind. Engg. Chem. 28:1429.

[25]. Staton, D. and Cavagnino, A. (2008).Convection Heat Transfer and Flow Calculations suitable for Electric Machines Thermal Models.IEEE Transactions on Industrial Electronics. 55(10): 3509-3516.

[26]. Toolthaisong, S. and Kasayapanand, N. (2013). Effect of Attack Angles on Air Side Thermal and Pressure Drop of the Cross Flow Heat Exchangers with Staggered Tube Arrangement. Energy Procedia. 34: 417-429.

[27]. Versteeg, H. K., and Malalasekera, W. (1995).An Introduction to Computational Fluid Dynamics.Publ. Longman Scientific and Technical.1st Ed.

[28]. Webb, R. L. (1971). A critical evaluation of analytical solutions and Reynolds analogy equations for Heat and Mass Transfer in smooth tubes. 4: 197-204.

[29]. Zukauskas, A. (1968). Heat Transfer in banks of tubes in cross flow of fluid Thermophysics1. Academy of Science of the Lithuanian SSR Inst.Of physical and Technical problems of enerrgies, Mintiritnius.