Numerical Analysis of Tube-Fin Heat Exchanger using Fluent

M. V. Ghori & R. K. Kirar

Patel college of Science and Technology, Rajiv Gandhi Proudyogiki Vishwavidhyalaya,
Ratibad, Bhopal- 462036, India
E-mail : mayurmech7053@gmail.com,ravindrakirar@gmail.com

Abstract - Three-dimensional CFD simulations are carried out to investigate heat transfer and fluid flow characteristics of two-row plain Tube and Fin heat exchanger using FLUENT software. Heat transfer and pressure drop characteristics of the heat exchanger are investigated for Reynolds numbers ranging from 330 to 7000. Model geometry is created and meshed by using GAMBIT software. Fluid flow and heat transfer are simulated and results compared using both laminar and turbulent flow models k-, and SST k-omega, with steady-state solvers to calculate pressure drop, flow, and temperature fields. Model validation is carried out by comparing the simulated value friction factor f and Colburn factor j to experimental results investigate by Wang. Reasonable agreement is found between the simulations and experimental data, and the fluent software has been sufficient for simulating the flow fields in tube-fin heat exchangers.

Keywords - Tube- fin, heat exchanger, fluent, gambit

I. INTRODUCTION

Vestas Aircoil A/S produces compact tube-and-fin heat exchangers for ship motors, as well as other types of heat exchangers and cooling towers .The heat exchanger cools heated, compressed air from the motor with cooling water. Fins are used to increase heat transfer area on the air side, since the air has the largest influence on the overall heat transfer resistance. This study involves building a model of a fin-and-tube heat exchanger geometry using open-source software, creating a suitable mesh, setting up the cases (choosing solvers, numerical solution methods etc.) making the CFD calculations with fluent and comparing results to known experimental data. The following subsections describe this study in more details. First, a summary of other research in the heat-transfer field related to tube-and-fin heat exchangers is presented to put this study into perspective with the other work available in the open literature.

II. PREVIOUS RESEARCH AND EXPERIMENTS

The experiments carried out by Wang were conducted using a forced draft wind tunnel [1]. An air straightener was used to keep flow moving in the x-direction, an 8-thermocouple mesh was placed in the inlet and a16-thermocouple mesh in the outlet locations of which determined by ASHRAE recommendations. All equipment for data acquisition thermocouples, pressure transducer, airflow measurement station, and flow meter were checked for accuracy prior to running the experiments [2]. Water at the inlet was held at 60ºC, airflow velocities were tested in the range from 0.3 m/s to 6.2 m/s. Energy balances were monitored during the experiment for both the hot- and cold-side and reported to be within 2 [3].The uncertainties for the primary measurements (mass flow rate for air and water, pressure drop, and temperature of the water and air) were very small and therefore these measurements can be assumed to be accurate [4]. For this experimental study, the geometrical parameters for a two-row heat exchanger based on experimental research are used to build a CFD model, and results read from the graphs (friction factor and Colburn j-factor against Reynolds number) in the article are used to validate the results of the CFD simulations [6]. The parameters of interest: friction factor f and Colburn j-factor are widely used in industry to characterize pressure drop and heat transfer, respectively, and thereby determine heat exchanger performance and suitability for specific duties [7]. Determining and using these parameters for performance prediction is part of the heat exchanger design process.

The two-row fin-and-tube heat exchanger studied has a staggered tube arrangement. Analyzing flow and heat transfer using CFD can make calculations to predict heat-exchanger performance [8]. However, it is not possible to perform CFD simulation on the entire heat exchanger, due to the large number of volumes and calculations required. Therefore, a small section of a heat exchanger consisting of one channel of air between two fins, with the air flowing by two tubes is modelled for this project. Simulations of the air flow through this passage are carried out, while relevant characteristics of the air flow are sampled and averaged at the inflow, minimum free-flow area (s), and outflow. The characteristics sampled are: flow velocity (in all three directions: x, y, and z), temperature, pressure, and turbulence model parameters k, epsilon, and omega. These measurements are then used for calculating relevant performance parameters such as pressure drop, friction and Colburn factors, heat transfer rate, Reynold's number, etc.

III. Computational Domain And Boundary Conditions

The GAMBIT Software is used to create and mesh the computational model. A diagram of the studied model is shown in Figure 1, and consists of the air flow area between two fins of plain fin geometry and around the surfaces of two rows of tubes, and a schematic of the model with dimensions is shown in Figure 1, with the geometrical dimensions listed in Table 1 [5].

1. Geometric dimensions of heat exchanger model

Geometric Parameter		
Fin thickness	T	0.130 mm
Fin pitch	F_p	2.240 mm
Fin collar outside diameter	D_c	10.23 mm
Transverse pitch	P_t	25.40 mm
Longitudinal Pitch	P_l	22.00 mm
Tube wall thickness	Δ	0.336 mm

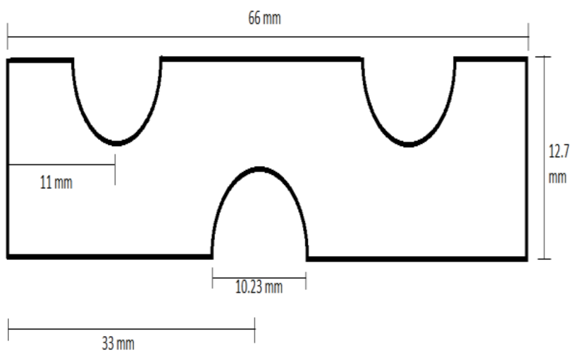

Fig. 1 : Illustration of the main computational domain and geometric parameters of the heat exchanger model studied (z-direction not shown).

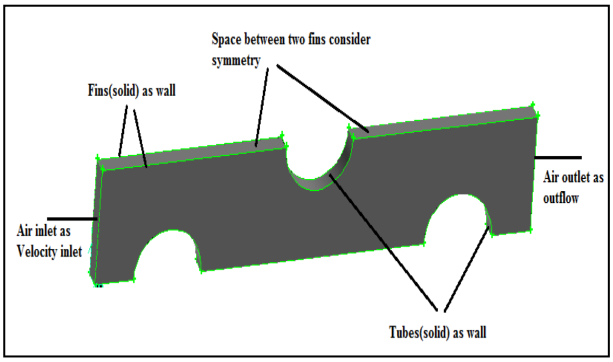

Fig. 2 : Computational domain, including boundary conditions (BC) and extended flow volumes.

The computational domain has contains boundary conditions as shown in Figure 2.with the following conditions:

Tube surfaces as a wall, Dirichlet Boundary condition

$T = T_w = 60°C = 333K$

Air velocity u = v = w = 0, that is no-slip condition at tube surfaces.

Fins as a wall, Dirichlet Boundary condition

$T = T_{fw} = 60°C = 333K$

Air velocity u = v = w = 0, that is no-slip condition at tube surfaces.

Inlet as a velocity inlet, Dirichlet Boundary condition

Uniform velocity $u = u_{in}$, u_{in} ranging from 0.3 m/s to 6.2 m/s.

v = w = 0

$T = 5°C = 278K$

Outlet as an outflow, Neuman Boundary condition

Zero gradients, u, v, w, pressure, and temperature.

Side planes as a symmetry, symmetry planes

$(\partial u/\partial y)=0$, $v = 0$, $(\partial w/\partial y) = 0$, $(\partial T/\partial y) = 0$

The entire computational domain was made up of approximate 70000 finite volumes, with a structured grid throughout most of the domain, while the areas around the tubes are more unstructured. The cell number was chosen based on the results of a grid independence test.

Performance parameters

This section describes how heat transfer and pressure drop are characterized. Included are dimensionless groups, equations for heat transfer and efficiency calculations, and equations for making pressure drop calculations. Following the descriptions of the performance parameters, the values as read from the graphs in the research done by Wang [1], for friction factor *f* and Colburn *j*-factor vs. Reynolds number are tabulated and graphed.

Meshing of Geometry

The geometry and computational domain were created using the geometry creation and meshing software GAMBIT. The Fluent 5/6 version 6.3.26 used for this project did not have sophisticated geometry and meshing capabilities, and therefore another meshing program was required. For this purpose, GAMBIT was chosen. The Mesh Volumes operation (volume mesh and volume modify commands) creates a mesh for one or more volumes in the model. When you mesh a volume, GAMBIT creates mesh nodes throughout the volume according to the currently specified meshing parameters.

Specifying the Volume

GAMBIT allows you to specify any volume for a meshing operation; however, the shape and topological characteristics of the volume, as well as the vertex types associated with its faces are decide to determine the types of mesh schemes that can be applied to the volume.

Specifying Scheme Elements

GAMBIT allows you to specify any of the following volume meshing Elements option.

2. *Meshing scheme elements*

Option	Description
Hex	Specifies that the mesh includes only hexahedral elements.
Hex/Wedge	Specifies that the mesh is composed primarily of hexahedral elements but includes wedge elements where appropriate.
Tet/Hybrid	Specifies that the mesh is composed primarily of tetrahedral elements but may include hexahedral, pyramidal, and wedge elements where appropriate.

Specifying Scheme Type

GAMBIT provides the following volume meshing Type options.

Hexahedral elements Meshes with cooper meshing scheme

In this project the geometry is meshed by hexahedral elements mesh was used. The details about this type of mesh are given below.

The GAMBIT 2.4.6 version was used for creating the hexahedral meshes with cooper meshing scheme for this tube-fin heat exchanger computation domain. It could make the more structured orthogonal mesh desired. Hexagonal mesh cell numbers approximately 70000 cells. This section presents some preliminary results and observations from the simulations carried out in this project. The simulations included ten different Reynolds numbers based on ten inlet airflow velocities ranging from 0.3 m/s to 6.2 m/s, all simulated in three flow models laminar, k-epsilon turbulence model, and SST k-omega turbulence model. They are describing as a Characteristics of Flow and Characteristics of Heat Transfer.

IV. Results And Discussions

A. *Characteristics of Flow*

A characteristic of flow describes the initial observations found using Contour Display after running the CFD simulations in Fluent. The characteristics of low-Reynolds flow and high Reynolds flow are compared with contour plots of velocity and contour plot of turbulent kinetic energy k. This gives behaviour of air flow velocity inside the computational domain.

B. *Velocity Observations*

The flow patterns of the two cases at low and high-Reynolds numbers (inlet air velocity 0.3 m/s vs. 6.2 m/s) are similar. The air enters at the inlet on the left and flows in the direction of the arrows, and exits at the outlet on the right-hand side as shown in figure 3 and figures 4the boundary layer growth in the tube as air flow with 0.3m/s.

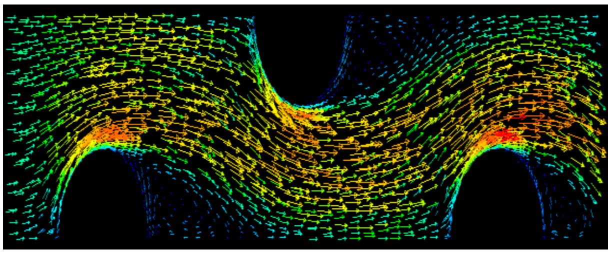

Fig. 3 : Vector plot for U velocity field, SST k-omega flow model, inlet air flow 0.3 m/s.

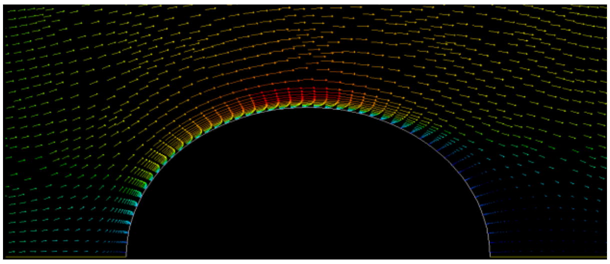

Fig. 4 : Vector plot for U velocity field, laminar flow model, inlet air flow 0.3 m/s with boundary layer on tube.

Below three flow models laminar, k-epsilon turbulence and k-omega turbulence models contour are presented with low Reynolds number and high Reynolds number in the figure 5 to figure 10 now we can easily understand the flow characteristics of air in heat exchanger between two fins.

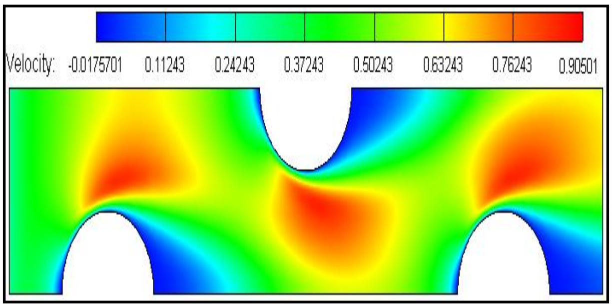

Fig. 5 : Contours plot for U velocity field, Laminar flow model, inlet air flow 0.3 m/s.

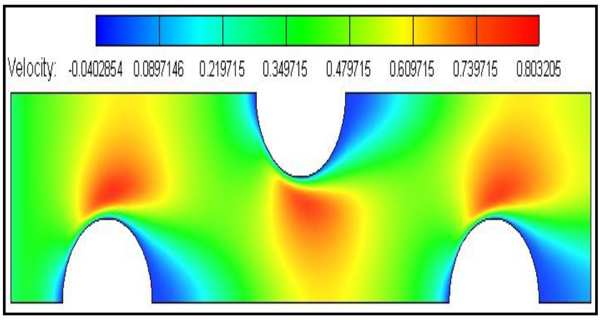

Fig. 6.: Contours plot for U velocity field, k-epsilon flow model, inlet air flow 0.3 m/s.

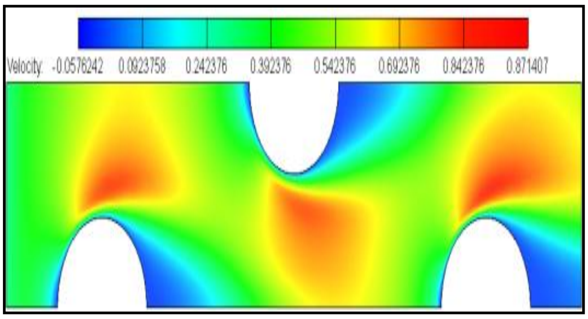

Fig. 7 : Contours plot for U velocity field, SST k-omega flow model, inlet air flow 0.3 m/s.

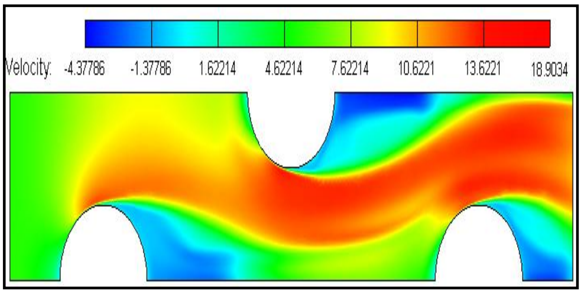

Fig. 8 : Contours plot for U velocity field, Laminar flow model, inlet air flow 6.2 m/s.

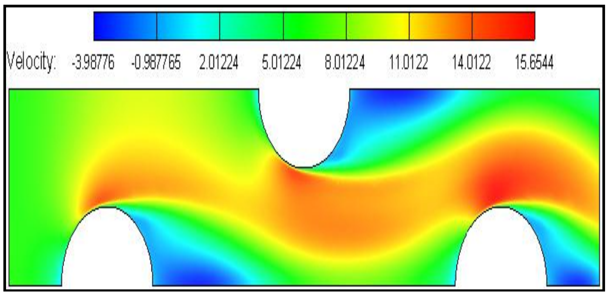

Fig. 9 : Contours plot for U velocity field, k-epsilon flow model, inlet air flow 6.2 m/s.

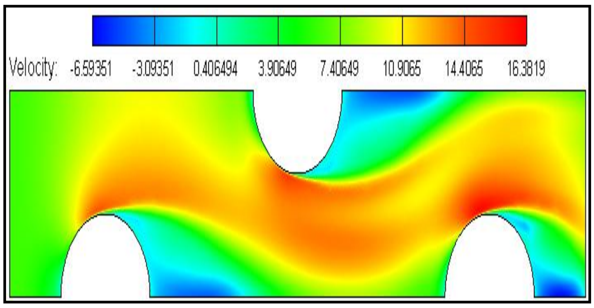

Fig. 10 : Contours plot for U velocity field, SST k-omega flow model, inlet air flow 6.2 m/s.

In above cases, as the air flows around the first tube, it begins to speed up and then the air velocity increases again as it goes around the second tube. This is because the free-flow area of air decreases, which showed that the velocity going around the second tube is faster than that going around the first tube. The minimum free-flow area is the area of the heat exchanger between two transverse tubes, so the area just above tube one or just below the second tube are the minimum free-flow areas. The flow is forced to speed up, as the tubes act as a type of pipe contraction in the air flow channel. As the velocity increases along the flow the Pressure of air flow is decreases accordingly.

It is observed that the size of the tubes impact the Reynolds number of the air flowing around them, since with larger tubes (at the same distance from each other), there would be an even smaller minimum free-flow area if the transverse pitch remained the same. In this study, the characteristic length for the Reynolds number is the tube collar diameter, and it can be seen here, that increases in this parameter (while keeping transverse pitch the same) can induce higher velocities and with it a higher turbulence and Reynolds number.

In the case of higher air flow velocity, the recirculation zones behind each tube contain small backflow areas. The second recirculation zone appears larger. In the case with 0.3 m/s inlet velocity, however, a recirculation zone was not noticeable as it was for the case with inlet velocity of 6.2 m/s.

C. Kinetic Energy k distribution

The kinetic energy contour plots can be seen to verify previous observations made regarding flow. It is seen in Figure 11, which illustrates the kinetic energy k distribution for the low Reynolds number case. There is no kinetic energy increase in the areas behind the tubes for this case. The kinetic energy increases (slightly) in a different area corresponding to the increase in velocity as the air flows around the second tube. The other area of figure 12 the plot of lower Reynolds number, which is exhibiting higher kinetic energy is in the area of higher temperatures, which can be seen from the contour. However, this illustrates that even at very low flow rates some turbulent kinetic energy could still be present.

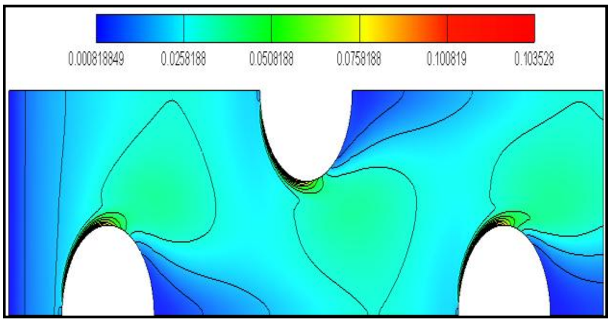

Fig. 11 : Contours of turbulent kinetic energy k distribution, SST k-omega model, inlet air velocity 0.3 m/s.

For kinetic energy in the higher-Reynolds number case Figure 12 and 13 an increase in kinetic energy is found clearly after the first tube, in the same area as the close to recirculation zones observed in the higher Reynolds values. According to this plot, the second recirculation zone is not as turbulent as the first recirculation zone. This makes sense because the direction of flow has changed as the air moves between the two tubes, and is directed more 'downward' at an angle. Turbulence kinetic energy k at behind the third tube is high in the recirculation zone which shows the higher Reynolds number. Turbulence kinetic energy k is most use full to determines the Reynolds stresses.

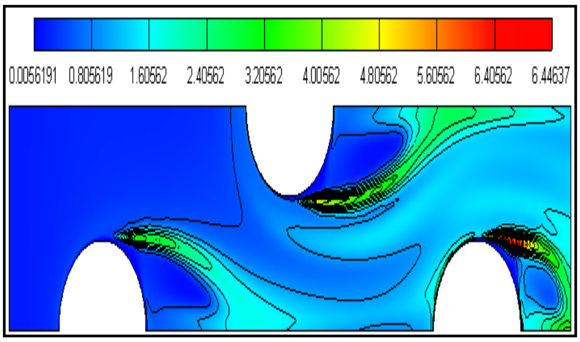

Fig. 12 : Contours of turbulent kinetic energy k distribution, SST k-omega model, inlet air velocity 6.2 m/s.

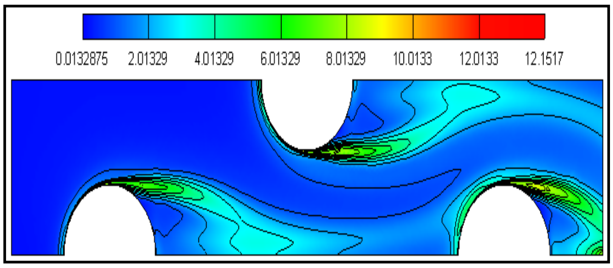

Fig. 13: Contours of turbulent kinetic energy k distribution, k-epsilon model, inlet air velocity 6.2 m/s.

Flow and kinetic energy plots have been compared and illustrated the effect flow has on kinetic energy. The next section describes the heat flow characteristics, as visualised in contours for the same cases as for flow and kinetic energy. It is found that the temperature changes, flow, and kinetic energy can be shown to be connected.

D. Characteristics of Heat Transfer

The first most noticeable difference between the two Reynolds number heat transfer characteristics is that once steady-state is reached, the slower-moving air (0.3 m/s) is heated up much more in the first two rows than in the case of higher Reynolds number flows. This must be due to the fact that the air flows so slowly, that there is much more time to absorb the heat (longer 'residence time'). It is seen that the temperature streamlines run practically perpendicular to the velocity streamlines in the beginning of the airflow channel, with the isothermal streamlines running vertical and the velocity of the flow horizontal. This acts as a cross-flow heat exchange, with the flow directly bringing the heat with it. It can then be seen that after the air has flowed through the initial section of the heat exchanger, this synergy between flow and heat transfer is no longer as effective. This means that the heat transfer coefficient is changing according to the streamline the flow is in at the time.

As shown in Figures 14 to 16 for low Reynolds the heat transfer coefficient is high for k-epsilon turbulence model so the air is heated at a temperature as a tube and fin temperature after flowing around on the first tube. For laminar and SST k-omega flow model almost same heat transfer take place.

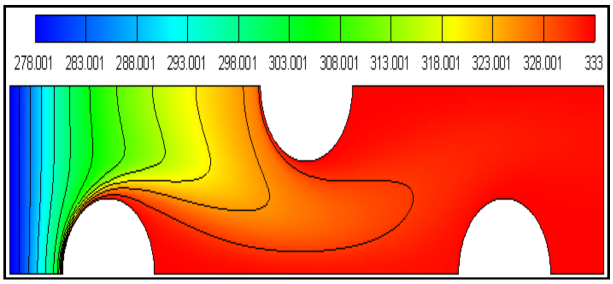

Fig. 14 : Contours of temperature field, laminar flow model, 0.3 m/s inlet air velocity.

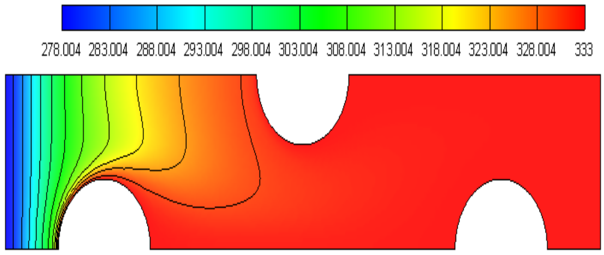

Fig. 15 : Contours of temperature field, k-epsilon flow model, 0.3 m/s inlet air velocity.

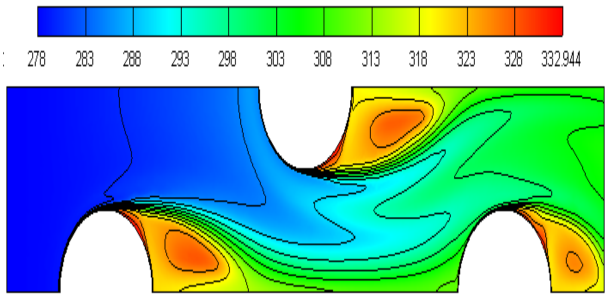

Fig. 16 : Contours of temperature field, SST k-omega flow model, 0.3 m/s inlet air velocity.

For high Reynolds number as in case of air flow with 6.2m/s shown in Figure 17 to 19 the air has not sufficient time to absorb the heat from the tube and fin. So the air temperature at the outlet is lower than as in low Reynolds number flow. Laminar flow attends the more velocity for air flow so it has absorbed less heat from the heat exchanger.

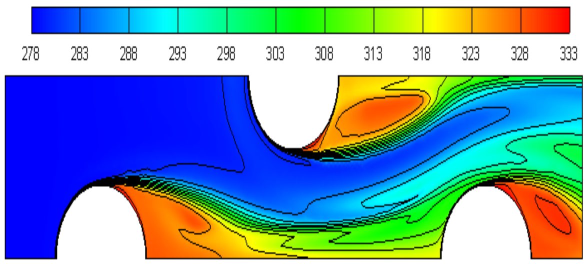

Fig. 17: Contours of temperature field, laminar flow model, 6.2 m/s inlet air velocity.

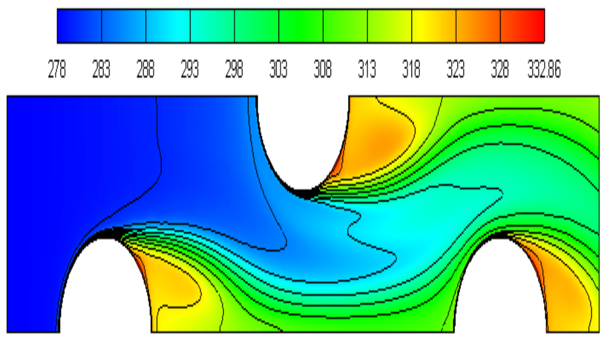

Fig. 18 : Contours of temperature field, k-epsilon flow model, 6.2 m/s inlet air velocity.

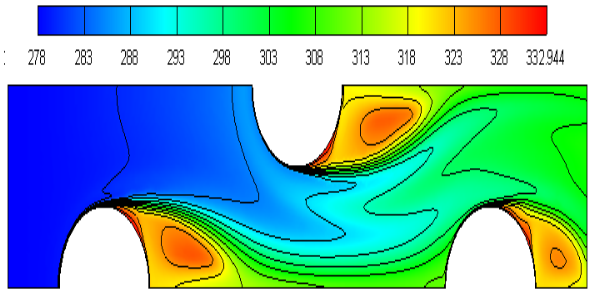

Fig. 19 : Contours of temperature field, SST k-omega flow model, 6.2 m/s inlet air velocity.

The largest temperature changes for the cases are occurring in the recirculation and slow velocity zones just after each of the tubes. As in the slow-moving flow in the case with 0.3 m/s velocity, the slow-moving areas of the heat exchanger are also better able to absorb heat. The staggered tube arrangement is designed to have these slower-moving and recirculation areas to keep the heat flowing to the air, but at the same time, not allowing recirculation zones to 'stagnate' as can occur in inline arrangements where these zones do not keep flowing.

Fluent software gives the value of pressure drop, maximum velocity, air outlet temperature and density of air at outlet. After getting these values friction factor f is determined for all models i.e. laminar and two turbulence model (standard k-epsilon and SST k-omega model). Values of friction factor are compared below in table 4 to 6.

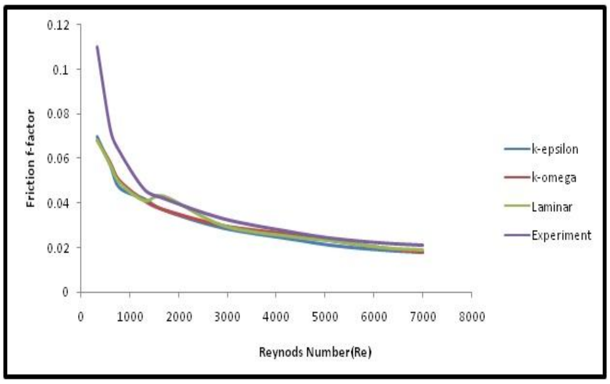

Fig. 20 : Comparing the graph of Friction factor f against Reynolds number Re

Friction factor *f* against Reynolds number *Re* for different inlet airflow velocities and flow models (laminar, and turbulence models k-epsilon and k-omega) with the same geometrical parameter are plotted in figure 20.It can be seen from figure 21, that in all models the friction factor was decreasing with increasing Reynolds number. All of the models are estimated the friction factor. At the lower laminar flows of Reynolds number from 330 to 1300, all the models found nearly identical results. As the flow moved into transition, it appears the laminar flow model came much closer to the experimental values. At the transition point from transition to turbulent, which appears to have a critical Reynold's value of between 1700 and 2900, once again, none of the models were better than another. After moving into turbulent flow, however, the k-omega SST had the best accuracy, and the laminar flow model able to model the friction factor compared to the k-epsilon model. We have seen that there was error in all models with experiment so plot a graph. Figure 22 which shows the all three model percentage error with the experiment values. Here the error in the model k-omega and laminar flow model is the least with respect to the experiment data but in k-epsilon large error occurred this is because both k-omega and k-epsilon are created for calculate turbulence, turbulent kinetic energy and dissipation rate, but in laminar flow do not increase the accuracy since there is no turbulence in laminar flow. The k-omega is the most accurate for friction factor because it model kinetic energy and specific dissipation rate at the wall or boundary. The k-epsilon model was the least accurate flow model because it calculated the equations to model kinetic energy and dissipation rate for free-flowing fluids, and therefore the friction factor against the wall is not capable to calculate accurately.

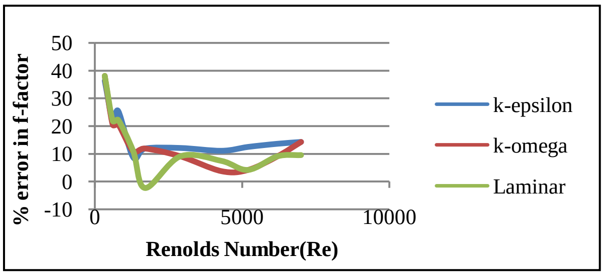

Fig. 21: Fanning friction factor f Error by flow model vs. Re.

A graph is plotted between Colburn-j factor and Reynolds number in the Figure 23 and an error graph is also shown in Figure 24the flow models showed very clear differences in abilities to simulate heat transfer at the different Reynolds numbers.

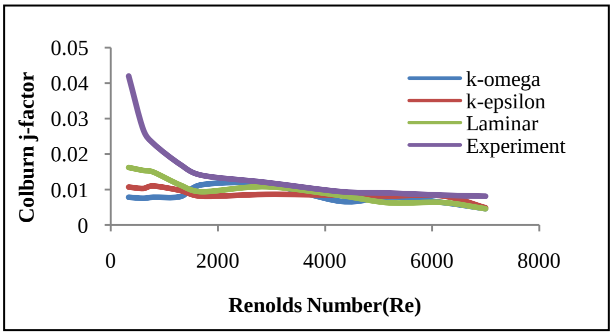

Fig. 22 : Colburn j-factor against Reynolds number Re for different inlet airflow velocities and flow models

In laminar flow, the laminar flow model was the best for predicting the j-factor, as would be expected. The transition heat flow was the best characterized with the k-omega turbulence model, while turbulent heat flow was best calculated using the k-epsilon model, although at the very highest Reynolds number 7000, none of the models were accurate.

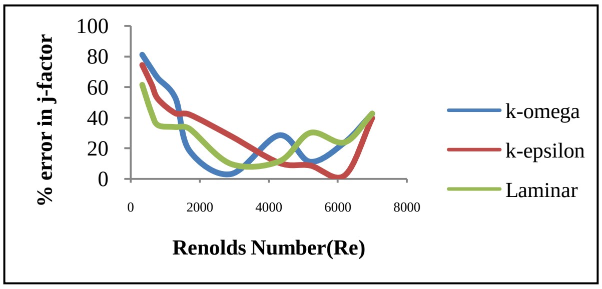

Fig. 23 : Colburn j-factor error: Error by flow model vsRe.

Here we seen that there is good agreement in data produced by experiment and by computer simulation. The results found that different flow models performed best according to Reynolds number and whether accurate solutions were desired for pressure drop or heat transfer.

V. Conclusion

The objective of this project was to make CFD simulations using fluent software, and validate the results against experimental data. The system to study was a Tube and Fin heat exchanger. The purpose of the work was to investigate the possibilities of eventually using CFD calculations for design of heat exchangers instead of expensive experimental testing and prototype production. To analyse the flow and heat transfer characteristics of the heat exchanger, a model of a two-row fin and tube heat exchanger was created using GAMBIT software to create the geometry and mesh. The resulting mesh was used for running a variety of simulations using a laminar flow model and two turbulence models for comparison of results. Ten different inlet flow velocities ranging from 0.3 m/s to 6.2 m/s and corresponding to Reynolds numbers ranging from 330 to 7000 were simulated in the three different flow models laminar, k-epsilon turbulence model, and SST k-omega turbulence model. Using the simulation results and some specific non-dimensional numbers, calculations related to heat flow and pressure loss can be carried out to determine the Fanning friction factor and Colburn *j*-factor for comparison with the experiment. It was found that the flow model accuracy depended on the flow regime and whether the friction factor *f* or *j*-factor was being determined. From the experimental values the laminar flow region for geometry of heat exchanger transfer to transitional at Reynolds number 1300, and remain in transitional stage up to Reynolds number 2900. The Reynolds number is calculated against of the tube outside diameter. For friction factor determination, little difference is found between the flow models simulating laminar flow, while in transitional flow, the laminar flow model produced the most accurate results for friction factor and the SST k-omega turbulence model was more accurate in turbulent flow regimes. For heat transfer, the laminar flow model calculated the most accurate j-factor, while for transitional flow the SST k-omega turbulence model was more accurate and the k-epsilon turbulence model was best for heat transfer simulations of turbulent flow.

Future prospective

Simulations for this study was carried out following as closely as possible the same operating conditions and geometrical configurations of the two-row tube-fin heat exchanger, with tube collar diameter of 10.23 mm and fin pitch 2.23 mm, as presented in the paper by Wang [1]. The Reynolds number ranges from 330 to 7000, which correspond to the frontal air velocity at the inlet ranging from 0.3 to 6.2 m/s. The work done for this project has shown that it is possible to make practical simulations of heat flow and pressure drop for a tube-and-fin heat exchanger using fluent software, and validate the results against experimental data. Data resulting from the simulations should be as accurate as possible, and therefore some considerations can be taken in future work to attempt to further improve the simulation conditions or calculations and the accuracy of the results. In this dissertation only standard k-epsilon and SST k-omega turbulence model is used but in future realizable k-epsilon turbulence model and Reynolds stress model can be used for better result and time variant solver can also be used for a validation of simulation result to experiment data.

VI. REFERENCES

[1] CC. Wang An experimental study of heat transfer and friction characteristics of typical louver fin-and-tube heat exchangers International Journal of Heat and Mass Transfer Volume 41, Issues 4–5, February–March 1998, Pages 817-822

[2] CC. Wang, Chi-Chuan; Chang, Yu-Juei; Hsieh, Yi-Chung; Lin, Yur-Tsai (1996). “Sensible heat and friction characteristics of plate fin-tube heat exchangers having plane fins”, International Journal of Refrigeration, Vol. 19, No. 4: pp. 223-230.

[3] Chi-Chuan Wang, Kuan-Yu Chi (2000). Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, part I: new experimental data, International Journal of Heat and Mass Transfer 43: 2681–2691

[4] Erek, Aytunc; Õzerdem, Baris; Bilir, Levent; Ilken, Zafer(2005). ”Effect of geometrical parameters on heat transfer and pressure drop characteristics of plate fin and tube heat exchangers”, Applied Thermal Engineering, Vol. 25: pp. 2421-2431.

[5] Fluent Incorporated, FLUENT User’s guide version 6.1, (2003).

[6] Fornasieri, E.; Mattarolo, L(1991). “Air-side heat transfer and pressure loss in finned tube heat exchangers: state of art”, Proceedings of the European Conference on Finned Tube Heat Exchangers, Lyon, France.

[7] Gongnan Xie, Qiuwang Wang, Bengt Sunden(2009). Parametric study and multiple correlations on air-side heat transferand friction characteristics of fin-and-tube heat exchangers with large number of large-diameter tube rows Applied Thermal Engineering 29: 1–16

[8] Gray, D. L.; Webb, R.L.(1986). “Heat transfer and friction correlations for plate fin-and-tube heat exchangers having plain fins”, Proceedings of the Ninth International Heat Transfer Conference, San Francisco .

[9] Jiong Li, Shuangfeng Wang, Wang Cai , Weijun Zhang(2010) Numerical study on air-side performance of an integrated fin and micro-channel heat exchanger” Applied Thermal Engineering 30: pages 2738-2745

[10] Kayansayan, N.(1994). “Heat transfer characterization of plate fin-tube heat exchangers”, International Journal of Refrigeration, Vol. 17, No. 1: pp. 49-57.