COMPUTATIONAL FLUID DYNAMICS ANALYSIS AND OPTIMIZATION OFDIFFERENT PARAMETERS OF HEAT EXCHANGER USED IN AERO ENGINES

Kundan kumar[1], Prof. M.K. Chopra[2]
M.Tech scholar[1], Vice principal, dean academic& head [2],
R.K.D.F. INSTITUTE OF SCIENCE & TECHNOLOGY,
BHOPAL (MADHYA PRADESH, INDIA)

Abstract

In order to maintain the temperature of the fin-tube heat exchangers,this will show potential applications in aero engines for their high efficiency and security as compared with other types of heat exchangers used in different areas, heat exchangers used in aero engines are more compact and it is at much higher temperatures and larger temperaturedifferences. The change in temperature over the increase in heat exchanger depth and the temperature gradient in the near wall region are more conspicuous, it may have difference of several hundred degrees. After using the heat exchanger large temperature change means significant increase in fluid properties variation, which must be considered when evaluating the exchanger performance. Inside the aero engine air coming from compressor is often used for turbine airfoil cooling in aero engines. Here in this work tube type heat exchangers are selected for the presentresearch it has a plain finned tube configuration. Hot air coming from compressor flows across the tube bank while here in this analysis cold fluid that may be fuel oil flows inside the tubes. The two configurations have the same tube geometry which is a staggered tube arrangement.it numerically investigates the heat transferand pressure variation characteristics of such exchangers with consideration of the air property variability which caused the air temperature changes.

1. Introduction

A heat exchanger is a device used to transfer heat between a solid object and a fluid, or between two or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant.

1.1 Flow Arrangement

There are three primary classifications of heat exchangers according to their flow arrangement. In parallel-flow heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In counter-flow heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is the most efficient, in that it can transfer the most heat from the heat (transfer) medium per unit mass due to the fact that the average temperature difference along any unit length is higher. See countercurrent exchange. In a cross-flow heat exchanger, the fluids travel roughly perpendicular to one another through the exchanger. For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger. The exchanger's performance can also be affected by the addition of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or induce turbulence. The driving temperature across the heat transfer surface varies with position, but an appropriate mean temperature can be defined. In most simple systems this is the "log mean temperature difference" (LMTD). Sometimes direct knowledge of the LMTD is not available and the NTU method is used.In later past year, the changes in computing power have expanded the interest of engineers and researchers to simulate their issues with computational and numerical methods. A ton of computational apparatuses and strategies have been produced in the most recent decades to break down liquid flow, combustion, and distinctive methods of warmth exchange. Utilization of warmth exchangers in extensive variety of uses draws in the specialists and researchers to work in this field.

2. Material Used

For the initial analysis it has taken the material same as that of taken by Lingdong et.al [1]. So here is considering the GH2132 alloy (Fe-25Ni-15Cr) is chosen as the fin material, whose thermal conductivity is set as 14.2 W/ (m^2-K). The material properties of GH2132 is shown in the below table

Table.1 Properties of material GH2132

Properties	values
Density	7.99 g/cm^3
Specific heat	447 J/kg-k
Thermal conductivity	14.2 W/m-C

After analyzing the above material to increase the performance of heat exchanger here it consider the two different materials to increase the heat transfer rate. So here it considered two materials one is GH3044 and the other one is S66280. The material properties of these material is shown in the below table.

Table.2 Properties of material GH3044

Properties	**values**
Density	8.89 g/cm^3
Specific heat	440 J/kg-k
Thermal conductivity	11.7 W/m-C

Table.3 Properties of material S66280

Properties	**values**
Density	7.98 g/cm^3
Specific heat	460 J/kg-K
Thermal conductivity	12.2W/m-C

3. Development of CFD Model

In order to develop the CFD model of the heat exchanger analysis different sub method or steps have to be performed. Different steps required for the development of CFD model is perform in this section. To increase the performance of heat exchanger used in aero engines here in this work it find out the effect of different material used for the construction of tubes and fin used in heat exchanger. To increase the heat transfer rate here it has taken the three different material for tube and fins construction and analyzed the temperature distribution and pressure drop at four different velocities that is 5, 10, 15, 20 m/s. after finding out the optimum material for given boundary condition it has also calculate the effect of change in thickness of fins used for heat transfer.

3.1 Developing Solid Model

In order to achieve the above objective here first it has to develop the solid model of heat exchanger based on the geometry used given in Lingdong et.al [1] the geometric specification of heat exchanger used in the analysis is defining the tube bank configurations include the tube outside diameter (D), transverse tube pitch (P_t), longitudinal tube pitch (P_l), and number of tube rows (N). They are taken to be D = P_l = 3.0 mm, P_t = 6.0 mm and N = 12 in this research. The plain finned tube configuration involves additional parameters including the fin pitch (F_p) and fin thinkness (d_f), which are specified to be F_p = 1.1 mm and d_f = 0.1 mm. due to the periodicity and symmetry of the heat exchanger geometry for numerical analysis it considered the two dimensional that is 2D airflow passage as shown in the fig. here the solid model of the heat exchanger is prepared in the design modular of Ansys.

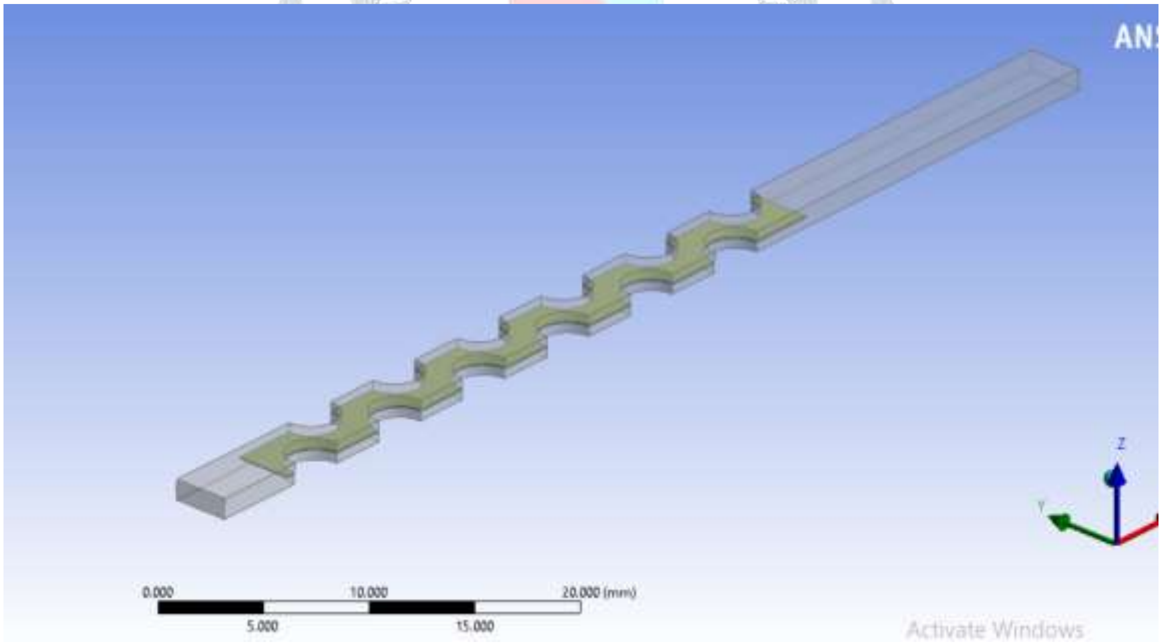

Fig.1 Solid model for computational analysis

3.2 Air Condition and Air Physical Properties

Air coming from the compressor and entering the heat exchanger in the aero engine operation assume to operate at altitude of 11 km and flies at mach 0.8 as given in Lingdong et.al [1].

During the analysis it considered the local atmospheric temperature and pressure that is calculated as 216.65 K and 22.63 kPa[17]. The inlet total pressure recovery coefficient is assume to be 0.97, the compressor compression ratio considered during the analysis is 25, the compressor efficiency is 0.90, and the air adiabatic index is 1.4, the air temperature and pressure at the inlet of heat exchanger is considered to be same as those at the compressor outlet, and it is considered as 653.99 K and 0.84 MPa[18]. The air velocities at the exchanger inlet (frontal air velocities) are set to range from 5 to 20 m/s, with the tube wall temperature being taken as 298 K. To perform the CFD analysis on the heat exchanger here it takes the FLUENT module of ANSYS. ANSYS (FLUENT) is basically used for the analysis related to fluid flow and heat transfer.

3.3 Meshing

After developing the solid model of given geometry, it is then discretizing in to number of elements and node because the numerical analysis is completely depends on the number of elements and number of nodes. During the numerical analysis the result were calculated at each node and element. So to discretize the complete solid model in number of small element here it used the different tool to enhance the mesh

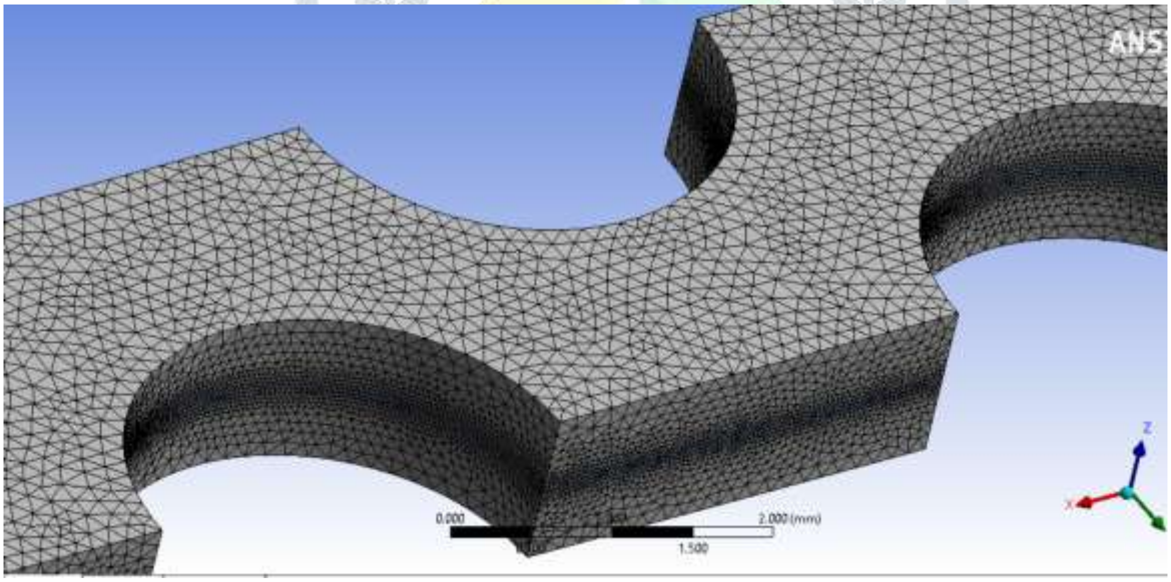

Fig.2 Top view of meshed solid model

3.4 Selection of Model

After performing the mesh, here it selects the model for doing the further analysis. To perform the numerical analysis it is necessary to select appropriate model according to the condition of problem. So in order to find out the appropriate model for the given problem, here it considered the six different model that is K-epsilon Standard, K-epsilon RNG, K-epsilon Realizable, K-omega Standard, K-omega BSL and K-omega SST. So to find out the appropriate model here it has taken the all six model and find out the temperature difference for each model and then with the help of numerical analysis it find out the coefficient of heat transfer for each case. The value of heat transfer coefficient for each case is shown in the table at four different velocities.

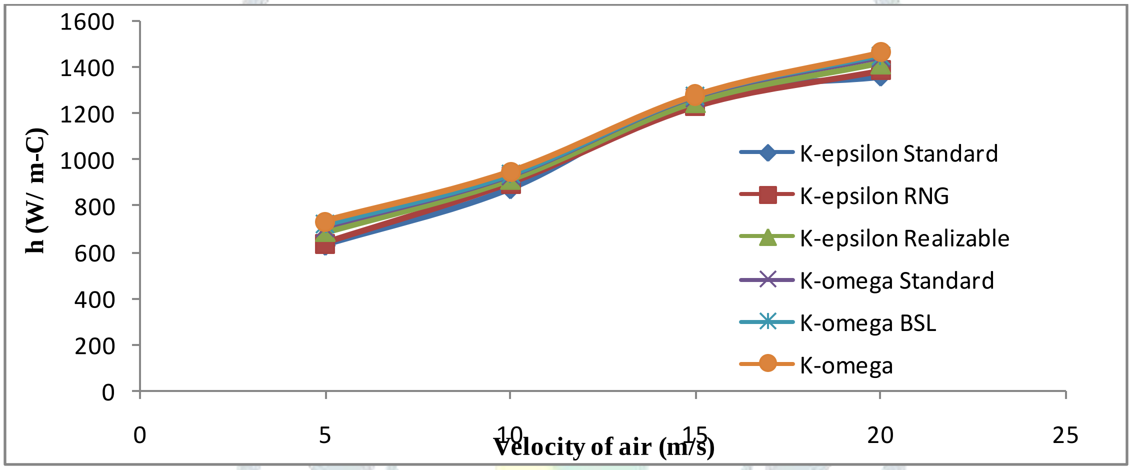

Fig.3 Comparison of heat transfer coefficient for different model

Based on the above analysis it is found that there is not much difference in between the different model. But in the entire six models K-epsilon Standard model is showing the average values as compared to other model, so it is better to prefer this model for further analysis. Here in this work it used the K-epsilon Standard model for the further analysis.

3.5 Boundary Condition

Here in this analysis the frontal air entering the heat exchanger is at different speed because it considered four different velocity of air that is 5, 10, 15 and 20 m/s. but for the initial analysis it is taken as 10 m/s. and the pressure is near about 0.84 MPa. The input of boundary condition in to the numerical analysis is shown in the fig; the temperature of air at the inlet of heat exchanger is 653.99 K. During the analysis the temperature of tube and fin of the tube is near about 298 K.

the temperature of the tube and fins are kept constant during each analysis that is at each different velocity. During each time the temperature of air entering the heat exchanger is at 653.99 K. the fig. showing the input of boundary condition at the inlet is heat exchanger.

4. Result

In order to validate the CFD model of heat exchanger used in aero engines, here it first find out the temperature of air at the exit of heat exchanger for different velocity. Here it considered the four different velocities that is 5, 10, 15, 20 m/s and in each case air exit temperature is get calculated and then it is compared with the values of air exit temperature calculated in base paper analysis performed by Lingdong et.al [1]. Here in this analysis it considered the material GH2132 for tube and fins. The thickness of fins for this analysis is 0.1 mm, whereas the gap in between the two fins is near about 1.1 mm.

- Case 1 velocity 5 m/s

Here in this case velocity of frontal air is 5 m/s and the temperature of air at the inlet is 653.98 K. After applying the boundary condition it find out the air exit temperature. The contour plot of air temperature distribution for this case is shown in fig.

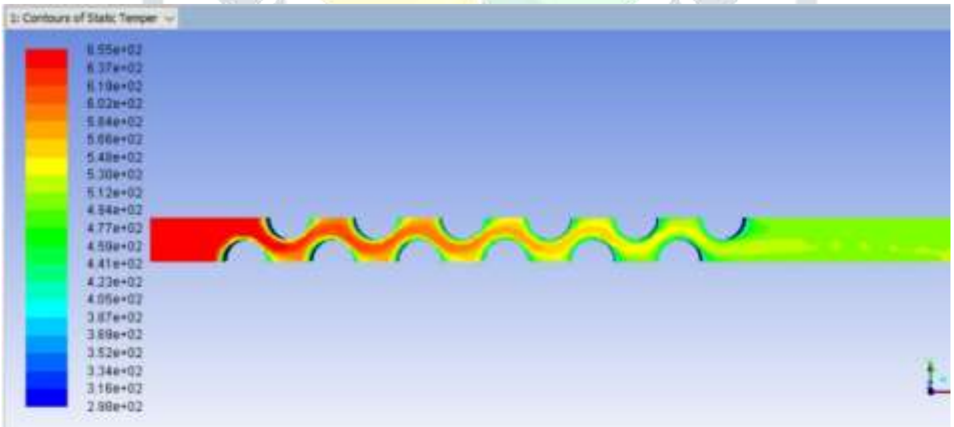

Fig.4. Temperature contour of heat exchanger at 5 m/s velocity- case 1

From the above analysis it observe that the temperature of air at the exit of heat exchanger is 509.8937 K. from the numerical analysis it has also analyzed the gradient of pressure changes inside the domain and also find out the change in velocities and velocity vectors.

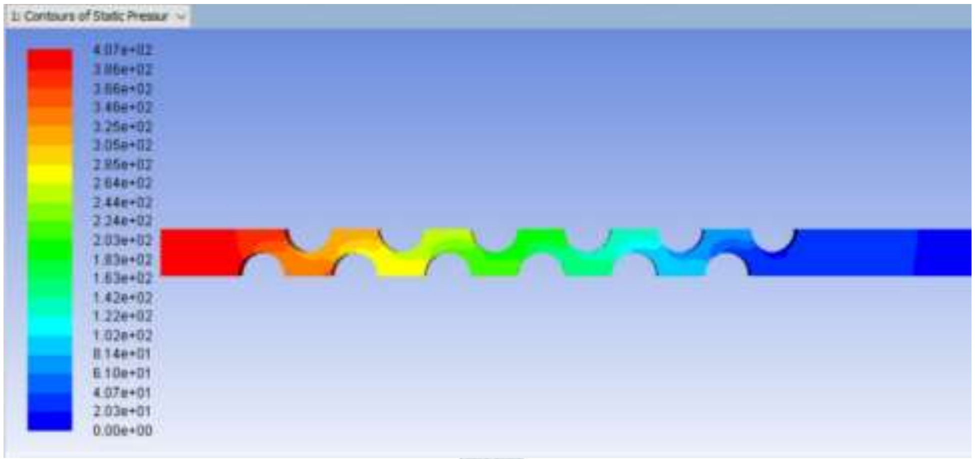

Fig.5 Contours of pressure distribution for case 1

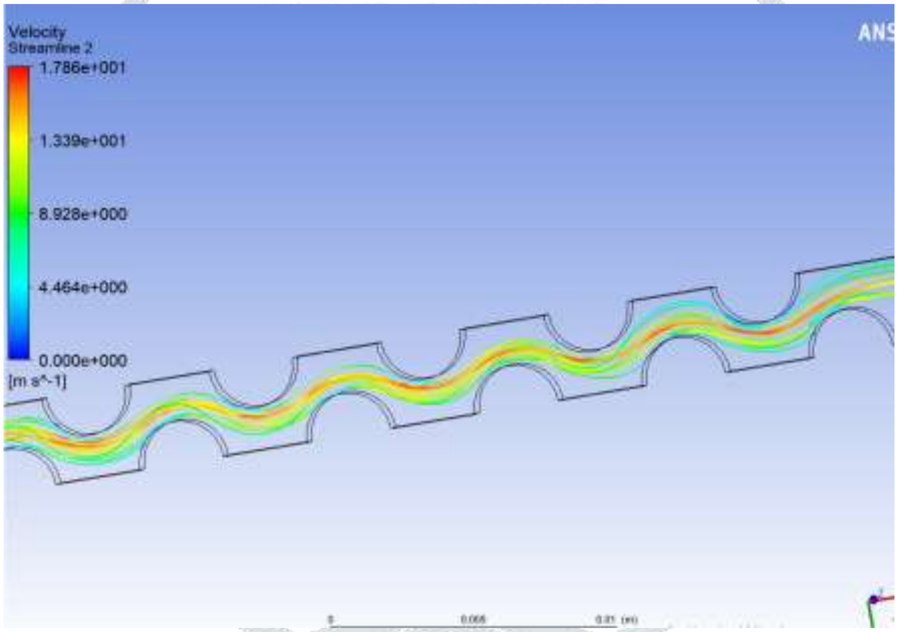

Fig.6 Velocity contours for case 1

- Case 2 velocity at 10 m/s

Here in this analysis the velocity of frontal air coming to heat exchanger is 10 m/s and the temperature of air at the inlet is same as that of case 1, other boundary conditions will also remain same as that of case 1. The temperature distribution profile for this case is shown in fig.

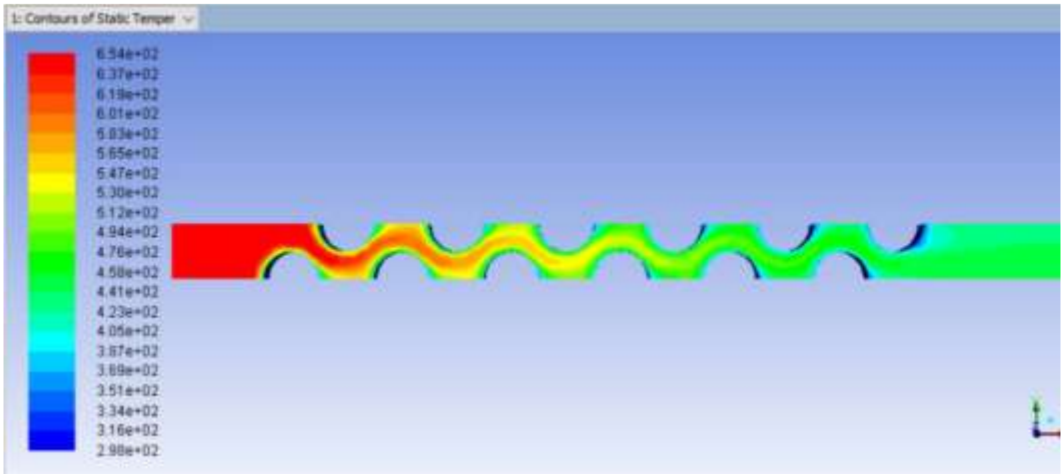

Fig.7 Contours of temperature for velocity 10 m/s – case 2

From the above fig. it is find that the temperature of air at the exit of heat exchanger is near about 443.8584 K and it also find out the pressure distribution throughout the computational domain, also explaining the velocity distribution throughout the domain.

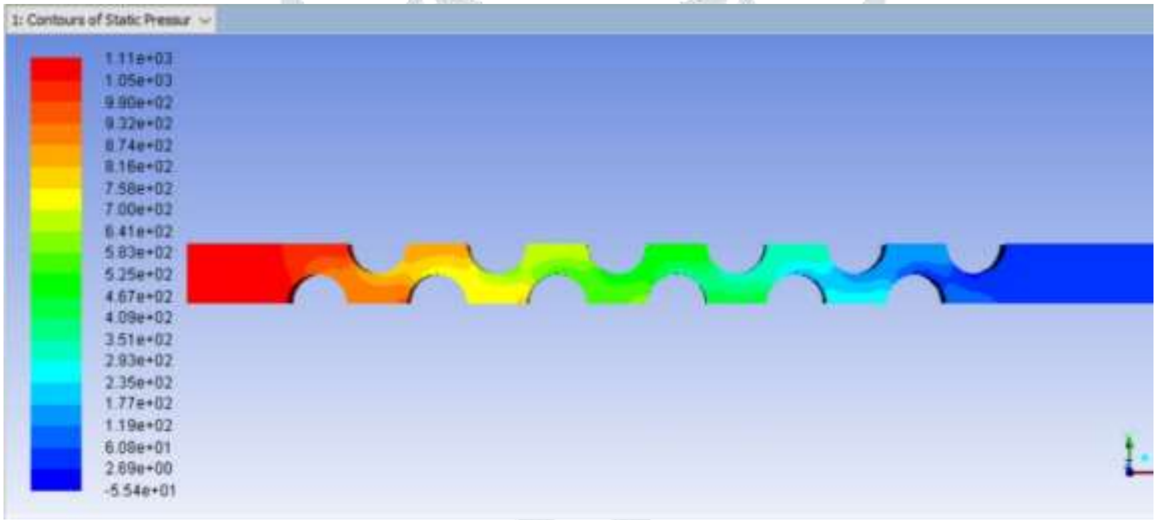

Fig.8 Contours of pressure distribution –case 2

Table 4.4 showing the value of air exit temperature of different velocity and the value of heat transfer coefficient for different velocities

Table 4. Air exit Temperature at different velocities

S.N	**Velocity (m/s)**	**Temperature of air at the exit (K)**	**Heat transfer coefficient (W/m-k)**
1	5	509.8937	664
2	10	443.8584	1024

3	15	374.2267	1155
4	20	370.65	1605

Comparison of value of temperature of air at the exit and heat transfer coefficient calculated through numerical analysis with the value of temperature and heat transfer coefficient given in the base paper.

Table.5Comparison of temperature and heat transfer coefficient

S.N	Velocity (m/s)	Temperature (K) calculated through Numerical analysis	Temperature (K) Values obtained from base paper	Heat transfer coefficient (W/m^2-k) calculated form numerical analysis	Heat transfer coefficient (W/m^2-k) values from base paper
1	5	509.8937	502	664	670
2	10	443.8584	441	1024	1044
3	15	374.2267	370	1155	1172
4	20	370.65	367	1605	1641

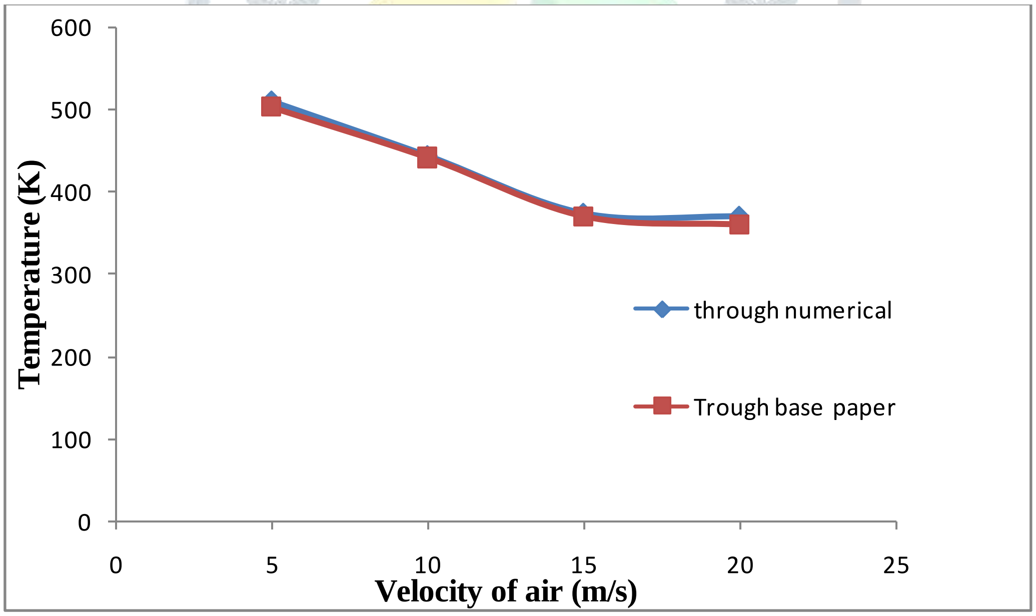

Fig.9 Comparison of temperature values

Table.6 Comparison of Temperature for Different Material

Case	Velocity of air (m/s)	Temperature (K) For material GH2132	Temperature (K) for material GH3044	Temperature (K) for material S66280
1	5	509.8937	493.436	510
2	10	443.8584	443.6	485
3	15	374.2267	368.74	376
4	20	370.65	355.84	362

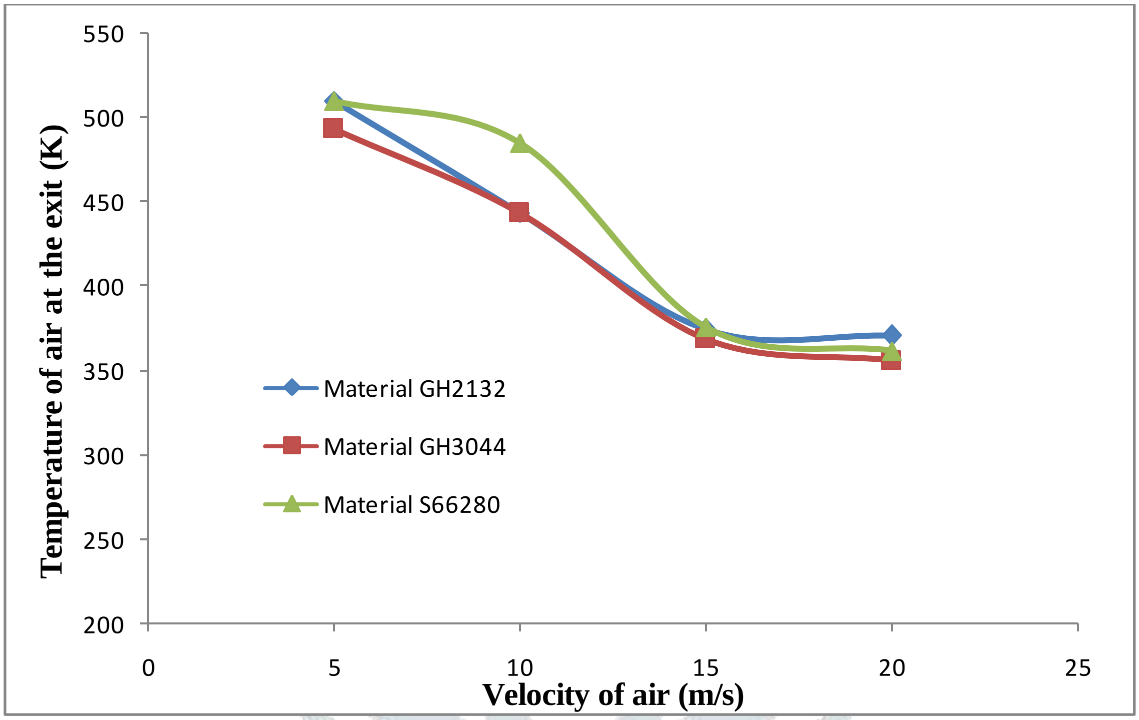

Fig.10 Comparison of Air Exit Temperature for Different Material

From the above comparison graph it is shown that the heat transfer rate is maximum for material GH3044, so as the heat transfer rate is maximum the temperature of air at the exit is minimum for this material. Through this analysis it is analyzed that as the material density, specific heat and thermal conductivity changer the heat transfer capacity of the material also changes. From the analysis it is also observed that though the thermal conductivity of material GH3044 is low as compared to the material S66280. Heat transfer is more in material GH3044 because the density and specific heat is also playing some role in heat transfer.

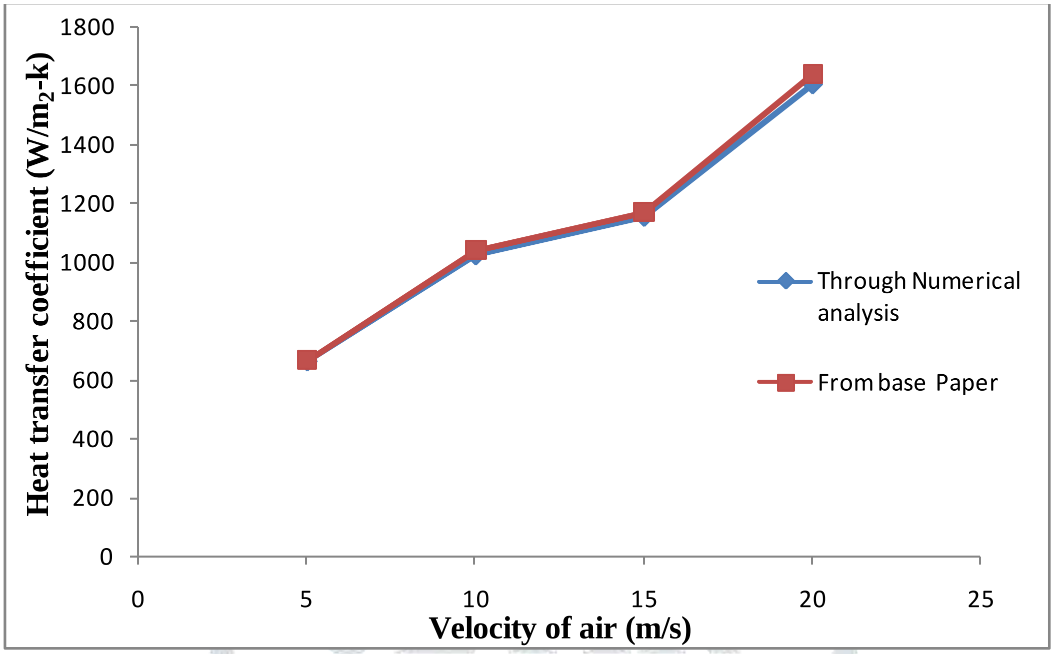

Fig.11 Comparison of heat transfer coefficient

From the above comparison it is shown that the value of temperature at the exit of heat exchanger obtained from the CFD analysis is closer to the value of temperature obtained from the base paper. It also analyzed that the value of heat transfer coefficient value obtained from the numerical analysis is close to value obtained from the base paper. So after analyzing the graph it shows that the CFD model of heat exchanger that is developing in this work is correct.

5. Conclusion

- The airside heat transfer and pressure variation characteristics of plain finned tube heat exchangers are numerically predicted with consideration of the air property variations caused by change in air velocity.
- Here it also find out the effect of material on the temperature of air at the exit, for analyzing the effect it consider the different steel alloy which is GH2132, GH3044 and S66820.
- After analyzing all the three material it is find out that GH3044 is showing the highest rate of heat transfer.
- Through this analysis it is also observed that rate of heat transfer also depends on the specific heat and density of the material though it is mainly depends on the thermal conductivity of material, but density and specific heat also play some role in it.

References

[1] LingdongGu[a], Jingchun Min[a*], XiaominWu[b], LijunYang[c]; "Airside heat transfer and pressure loss characteristics of bare and finned tube heat exchangers used for aero engine cooling considering variable air properties" ; International Journal of Heat and Mass Transfer, Elsevier 108 (2017) 1839 - 1849.

[2] S.S. Pawar[*], Vivek K. Sunnapwar, "Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger" ; Chemical Engineering Research and Design 1475 (2014) Article in press.

[3] Xing. Lu, Xueping Du, Min Zeng, Sen. Zhang, Qiuwang. Wang[*], "Shell-side thermal-hydraulic performances of multilayer spiral-wound heat exchangers under different wall thermal boundary conditions"; Applied Thermal Engineering, Elsevier 70 (2014) 1216 – 1227.

[4] V. Pandiyarajan[a], M. ChinnaPandian[b], E. Malan[a], R. Velraj[a*], R.V. Seeniraj[a] "Experimental investigation on heat recovery from diesel engine exhaust using finned shell and tube heat exchanger and thermal storage system" Applied Energy, Elsevier 88 (2011) 77-87.

[5] PaisarnNaphon, "Study on the heat transfer and flow characteristics in a spiral-coil tube" International Communications in Heat and Mass Transfer, Elsevier 38 (2011) 69-74.

[6] Tzong-Shing Lee., Wu-Chieh Wu, Yew-KhoyChuah, Sheng-Kai Wang, "An improvement of airflow and heat transfer performance of multi-coil condensers by different coil configurations"; International Journal of Refrigeration, Elsevier 33 (2010) 1370-1376.

[7] Rahul Kharat, NitinBhardwaj[*], R.S.Jha,"Development of heat transfer coefficient correlation for concentric helical coil heat exchanger" International Journal of Thermal Sciences, Elsevier 48 (2009) 2300–2308.

[8] C. Albanakis[a], K. Yakinthos[a*], K. Kritikos[a], D. Missirlis[a], A. Goulas[a], P. Storm[b], "The effect of heat transfer on the pressure drop through a heat exchanger for aero engine applications" ; Applied Thermal Engineering, Elsevier 29 (2009) 634–644.

[9] J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, RohidasBhoi, "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers" Elsevier, International journal of Chemical Engineering Research and Design 86 (2008):221-232.

[10] Timothy Rennie J., Vijaya G.S Raghavan, "Numerical studies of a double-pipe helical heat exchanger", Elsevier, Applied Thermal Engineering 26 (2006) 1266–1273.

[11] Vimal Kumar, SupreetSaini, Manish Sharma, K.D.P. Nigam, "Pressure drop and heat transfer in tube in tube helical heat exchanger" Elsevier, Chemical Engineering Science 61 (2006) 4403–4416.

[12] PaisarnNaphon, SomchaiWongwises, "A study of the heat transfer characteristics of a compact spiral coil heat exchanger under wet-surface conditions", Elsevier, Experimental Thermal and Fluid Science 29 (2005) 511–521.

[13] Yonghan Kim, Yongchan Kim, "Heat transfer characteristics of flat plate finned-tube heat exchangers with large fin pitch", Elsevier, International Journal of Refrigeration: 28 (2005) 851–858.

[14] He Huang, Louis J. Spadaccini and David R. Sobel., "FUEL-COOLED THERMAL MANAGEMENT FOR ADVANCED AERO ENGINES", Proceedings of ASME TURBO EXPO 2002, GT- 2002- 30070 .

[15] Herchang Ay., JiinYuh Jang., Jer-Nan Yeh., "Local heat transfer measurements of plate finned-tube heat exchangers by infrared thermography", International Journal of Heat and Mass Transfer 45 (2002) 4069–4078.

[16] Chi-Chuan Wang, Kuan-Yu Chi, Chu-Jung Chang, "Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, part II: Correlation", Elsevier, International Journal of Heat and Mass Transfer 43 (2000) 2693 – 2700.

[17] Greg B. Bruening, Won S. Chang, "COOLED COOLING AIR SYSTEMS FOR TURBINETHERMAL MANAGEMENT", Presented at the International Gas Turbine &Aeroengine Congress & Exhibition Indianapolis, Indiana — June 7-June 10, 1999.

[18] C. C. Wang, W. L. Fu, C. T. Chang, "Heat Transfer and Friction Characteristics of Typical Wavy Fin-and-Tube Heat Exchangers", Elsevier, Experimental Thermal and Fluid Science, 14 (1997) 174-186.

[19] D. Lakehal, W. Rodi, "Calculation of the flow past a surface-mounted cube with two-layer turbulence models", Elsevier, Journal of Wind Engineering and Industrial Aerodynamics, (1997) 65- 78.

[20] Chi-chuan Wang, Yi-chung Hsieh, Yur-tsai Lin, "Performance of Plate Finned Tube Heat Exchangers under Dehumidifying Conditions", Journal of Heat Transfer, ASME Vol.- 119/109 FEBRUARY 1997.

[21] Jiin-Yuh Jang, Mu-Cheng Wu, Wen - Jeng Chang, "Numerical and experimental studies of three dimensional plate-fin and tube heat exchangers", Pergamon, International Journal of Heat and Mass Transfer, Vol. -39 (1996) 3057- 3066.