# PERFORMANCE ANALYSIS OF HEAT EXCHANGER USING DIFFERENT MATERIALS BY CFD AT DIFFERENT MASS FLOW RATE OF WATER AND AIR VELOCITY

**Ishwar Jagan Dhangar [1], Dr. 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: Heat exchanger is used as a radiator in the automobile industry. During the heat transfer from the radiator different parameters were responsible for heat transfer. To find out the effect of different parameters here in this work it develops the computational fluid dynamic model of radiator. In order to find out the effect of fins on the radiator tubes, here it has find out the temperature of water flowing inside the radiator for both with and without fins radiator (heat exchanger). Here in this work it also find out the effect of different materials on the heat transfer rate and also predict the effect of different mass flow rate of water inside the tube. To predict the effect of change in material here it considered the three different materials that is Aluminium, brass and copper and for mass flow rate of water it considered the five different mass flow rates that is 180, 260, 340, 420 and 500 kg/hr. to predict the effect of variation of seep of air flowing over the radiator which is responsible for heat transfer here it considered four different velocities that is 3, 4, 5 and 6 m/s. based on the different combination made through mass flow rate of water and velocity of air temperature of water at the exit of radiator for different material were calculate.***

## INTRODUCTION

A heat exchanger is a device used to transfer heat between a solid object and a fluid, or between two or more fluids or gases. It is device in which heat is transferred from one fluid to another fluid .the hot fluid gets cooled, and the cold fluid is heated. 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.

### Material Used

For the initial analysis it has taken the material same as that of taken by Chaudhari et.al [1]. So here is considering the steel for manufacturing of tube and aluminium is used for fins. The material properties of aluminium and steel is shown in the below table

Table 4.1 Properties of Aluminium

| Properties | values |
|---|---|
| Density | 2.7 g/cm$^3$ |
| Specific heat | 871 J/kg-k |
| Thermal conductivity | 202.4 W/m-C |

Table 4.2 Properties of Steel

| Properties | values |
|---|---|
| Density | 8.03 g/cm$^3$ |
| Specific heat | 502.48 J/kg-k |
| Thermal conductivity | 16.27 W/m-C |

After analyzing the heat transfer rate in steel and aluminium for different mass flow rate here it considered the two different materials for tube and fins that is brass and copper. Here the thermal conductivity of brass and copper is greater than the steel which is used in the base paper. The material properties of brass and copper is shown in the below table

Table 4.3 Properties of material copper

| Properties | values |
|---|---|
| Density | 8.9 g/cm$^3$ |
| Specific heat | 385 J/kg-k |
| Thermal conductivity | 401 W/m-C |

Table 4.4 Properties of material Brass

| Properties | values |
|---|---|
| Density | 8.4 g/cm$^3$ |
| Specific heat | 380 J/kg-k |
| Thermal conductivity | 109 W/m-C |

### 4.3 Development of CFD Model

In order to find out the performance of radiator here it develops the CFD model of radiator (Heat exchanger). Heat transfer from radiator depends on the flow rate of fluid inside the radiator. In the current case water is flowing inside the radiator, air flow velocity which is responsible to carry heat from radiator to atmosphere is flowing in the cross verse direction to radiator. In order to find out the effect of change in material used for the manufacturing of tube and fins, change in mass flow rate of fluid flowing inside the radiator and change in speed of air flowing over radiator. Also it considered three different material that is copper, brass and aluminium, it also considered five different mass flow rate of water that is 180 kg/hr, 260 kg/hr, 340 kg/hr, 420 kg/hr, 500 kg/hr. And four different velocity of air that is 3, 4, 5, 6 m/s. Using different combination of air velocity and water mass flow rate here it find out the water temperature at the exit of radiator based on which performance of radiator is analyzed.

### 4.3.1 Solid model

To develop CFD model here first it has to develop the solid model of radiator. The solid model of radiator is based on the geometry specification taken in the chaudhary et.al [1]. The geometric specification of radiator is shown in the table below

Table.4.5 showing the geometric specification of radiator

| Parameter | Value |
|---|---|
| Fin material | Aluminium |
| Tube material | Steel |
| Tube inner diameter | 0.013 m |
| Tube outer diameter | 0.0146 m |
| Fin outer diameter | 0.0343 m |
| Fin thickness | 0.001 m |
| Number of tube pass | 10 |
| Length of one pass of tube | 0.5 m |
| Fin Space | 0.03933 m |
| No. of Fins | 900 |

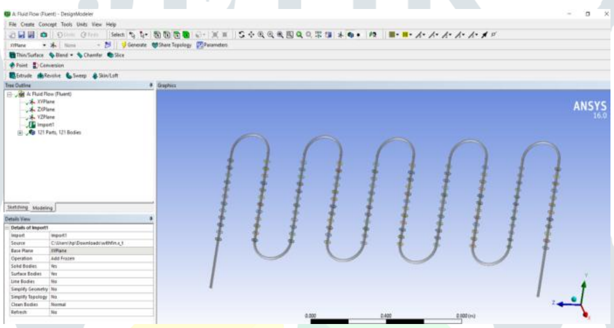

Fig.4.1 showing the Ansys design-modular with solid model of radiator

The geometry of the radiator is created in the design modular of Ansys. The radiator is having circular fins on the tube as shown in the figure.

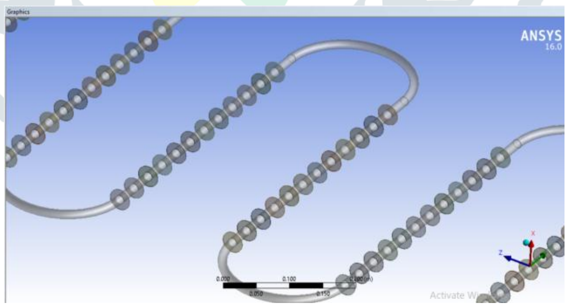


Fig.4.2 showing the solid model of radiator having circular fins

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. For performing the analysis first it open the ANSYS workbench and then FLUENT module as shown in the fig. The shows the solid model of radiator based on the geometry of the base paper it is constructed. It is used for the initial analysis of radiator as conducted in the base paper. chaudhari et.al [1] perform the experimental analysis on radiator.

### 4.3.2 Meshing

After developing the solid model of radiator as per given geometry taken from the experimental analysis in chaudhary et.al [1], it is then discretized 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. The meshing of the solid model is shown in the below figure below.

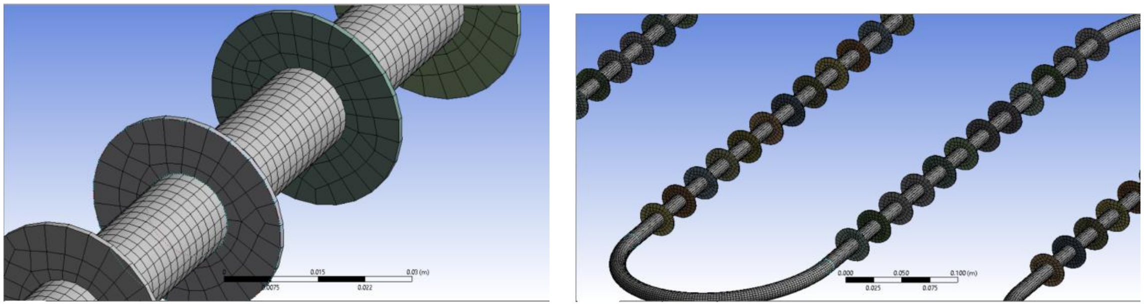

Fig.4.5 meshing of the given model

In order to improve the result of system meshing accuracy must be high. To improve the meshing of the given system here it uses the different tools to improve the mesh. For better connectivity in between the solid phase and liquid phase here it gives the inflation in between the two phases and applied some refinement to improve connectivity. It is necessary to perform the mesh independent test, it means that if it is increasing the number of element weather the property of system is increasing or not. Here in this analysis it used the temperature of water at the exit of tube in radiator as a parameter to check the mesh independency. So in order to check the mesh independency here it has perform mesh with different number of elements and find out the temperature of air at the exit. During the mesh independency test here initial boundary conditions were considered that is mass flow rate of water is 0.05 kg/s, velocity of air is 3 m/s

Table.4.6 Grid selection

| Number of elements | Temperature (K) of water at exit of radiator |
|---|---|
| 243254 | 668 |
| 152548 | 665.5 |
| 122259 | 665 |

After analyzing the mesh with different number of element and finding the temperature of air at the exit it is observed that the property of heat transfer is independent of number of element in mesh. There is a slide variation in the temperature of water which is not that much of concern.

### 4.4 Value of heat transfer coefficient for Different velocity of air

- **For air Velocity 3 m/s**

Water inside the radiator tube is flowing at the rate of 180 kg/hr, whereas the velocity of air flowing over the radiator is near about 3 m/s For this case it calculates the Nusselt number, Prandtl number, Reynolds number and finally finds out the value of heat transfer coefficient. During the calculation following data were considered. The value of standard data for different properties was taken from the book that is Heat and Mass transfer by Nortan.

Density of air = 1.21 kg/m$^3$
Specific heat of air = 1.05 KJ/kg k
Dynamic viscosity of air = 0.000018 Pa-s
Velocity of air = 3 m/s
Specific heat of water = 4.185 KJ/kg-k
Density of water = 1000 kg/m$^3$
Thermal conductivity of air = 0.02534
Outer diameter of tube = 0.0146 m
Inner diameter of tube = 0.013 m
Reynolds number is calculated through the formula given below

$$Re = \frac{\rho V D}{\mu} \quad (1)$$

Where Re is the Reynolds number, $\rho$ is the density, V is the velocity; D is the outer diameter of tube and $\mu$ is the dynamic viscosity.
Re = 2944.33
Now it calculate the Prandtl number, to calculate the value of Prandtl number following formula is used

$$Pr = \frac{\mu C_p}{K} \quad (2)$$

Where $C_p$ is the specific heat of air at constant pressure, K is the thermal conductivity of air
Pr = 0.7458
If the value of Prandtl number is greater than or equal to the 0.6 than the formula used to calculate the Nusselt number is shown in the below

$$Nu = 0.023\,(Re)^{4/5}(Pr)^{1/3} \quad (3)$$

= 12.4296
It is also known that the Nusselt number is also finding out through the formula given below. From this formula it can calculate the value of heat transfer coefficient that is h

$$Nu = \frac{hD}{K} \quad (4)$$

h = 21.59 W/M$^2$ k

After calculating the heat transfer coefficient it is then applied on the surface of tube during simulation. Based on the above calculation here it has calculated the value of heat transfer coefficient for different velocity. The value of h is shown in the table below

Table.4.7 showing the value of h for different velocity of air

| Velocity of air (m/s) | Heat transfer coefficient (h) $W/m^2$ K |
|---|---|
| 3 | 21.59 |
| 4 | 27.15 |
| 5 | 32.45 |
| 6 | 37.55 |

For the simulation of this case, 3 m/s velocity of air is given in the simulation and also the mass flow rate of water that is 0.05 kg/s is given. The value of mass flow rate of water given in the software is shown in the below fig.4.9 below

**Result**

After calculating the heat transfer coefficient for different velocity of air. It is applied in the ANSYS fluent to calculate the value of temperature of water flowing inside the radiator at the inlet and exit. Through ANSYS it can also find out the value of temperature of air at the inlet and exit. In order to validate the CFD model of Radiator here it find out the value of overall heat transfer coefficient for different velocity and different mass flow rate of water and then it is compare with the Experimental value for overall heat transfer coefficient.

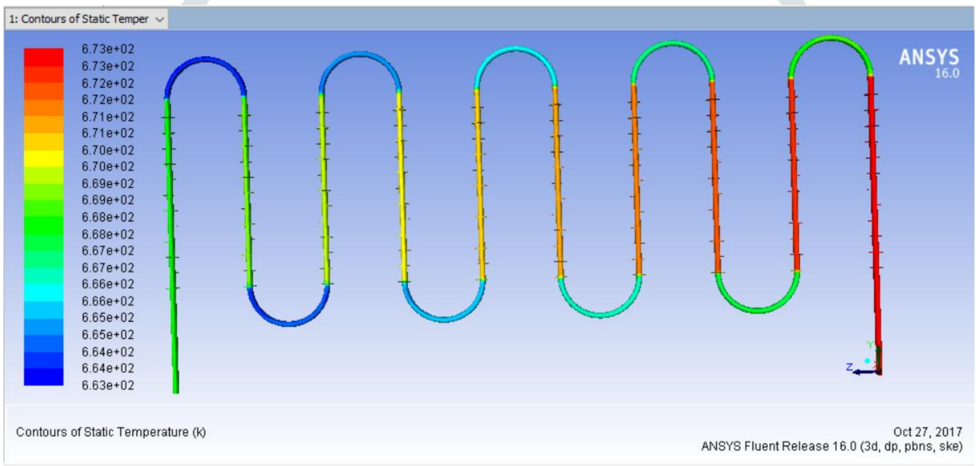


Fig.4.14 showing the contour of temperature for air velocity 3 m/s

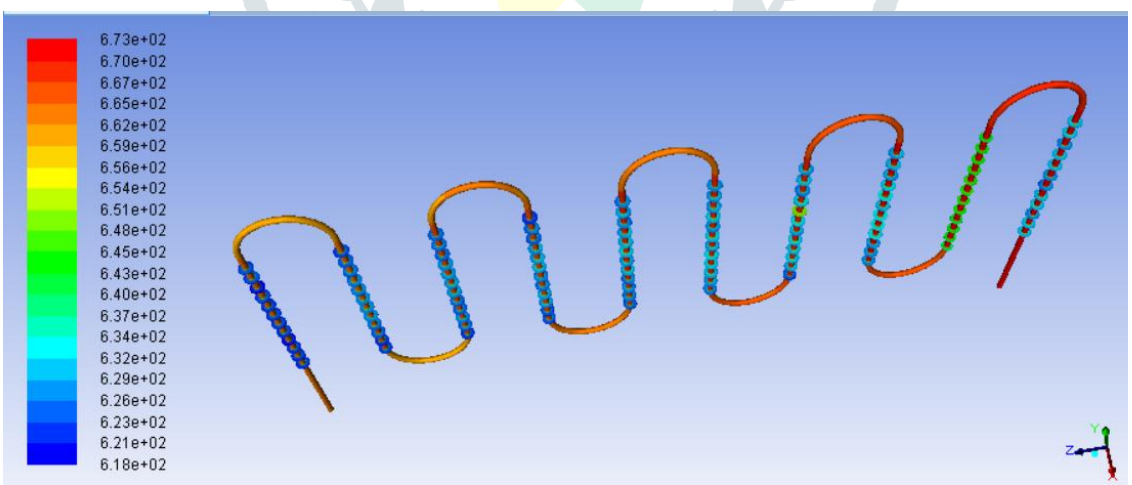


Fig.4.15 showing the contour of temperature for air velocity 4 m/s

The temperature value of water for 3 m/s air velocity and 180 kg/hr mass flow rate is calculated through software. The temperature distribution for this case is shown in the figure above. From the above analysis it is found that the temperature of water at the exit of radiator is near about 668 k. whereas the temperature of air at the exit of radiator is 410.13 K.

Using the above values of temperature of air and water at the exit of radiator, here it calculate the overall heat transfer coefficient for case having air velocity 3 m/s and water mass flow rate is 0.05 kg/s. to calculate the value of overall heat transfer coefficient following formula were used

$$q = UA(T_o - T_i)_{air} \quad (5)$$

Where q is the heat transfer rate, UA is the overall heat transfer coefficient, $T_o$ *and* $T_i$ is the temperature of air at the exit and inlet of radiator. To calculate the heat transfer rate, heat losses from the water during the mass flow rate of 0.05 kg/s is calculated because whatever the heat loss by the water is equal to the amount of heat gain by air. So to calculate the amount of heat loss from water following calculation were used

$$q = \dot{m}C(T_{wi} - T_{wo}) \quad (6)$$

Where q is the heat loss from water, $\dot{m}$ is the mass flow rate of water, $T_{wi}$ is the temperature of water at the inlet of radiator and $T_{wo}$ is the temperature of water at the exit of radiator.

$$q = 0.05 \times 4.185 \times 1000 \times (410.13 - 300)$$
$$q = 1046.25 \text{ W}$$

Now form eq. 5 it can find out the value of Overall heat transfer coefficient

1046.25 = UA (673-668)

UA = 9.5 $W/m^2$ K

Based on the above calculations value of the overall heat transfer coefficient for different velocity of air and for different mass flow rate of water. The value of overall heat transfer coefficient for different cases is shown in the table below

**4.7 Validation of the CFD model**

In order to check the correctness of CFD model of radiator here the values obtained from the CFD analysis is compared with the values obtained from the Experimental analysis performed in the base paper.

Table.4.8 showing the value of overall heat transfer coefficient calculated through CFD

| Mass flow rate of water ($kg.s^{-1}$) | Overall heat transfer coefficient at velocity (3 $ms^{-1}$) ($W/m^2K$) through CFD |
|---|---|
| 0.05 | 9.5 |
| 0.0722 | 19.20 |
| 0.0944 | 21.6 |
| 0.116 | 46 |
| 0.138 | 52 |

Table.4.9 showing the value of overall heat transfer coefficient calculated through experiment perform Choudary et al [1]

| Mass flow rate of water ($kg.s^{-1}$) | Overall heat transfer coefficient at velocity (3 $ms^{-1}$) ($W/m^2K$) through Experiment |
|---|---|
| 0.05 | 14.07 |
| 0.0722 | 20.65 |
| 0.0944 | 22.89 |
| 0.116 | 47.98 |
| 0.138 | 53.57 |

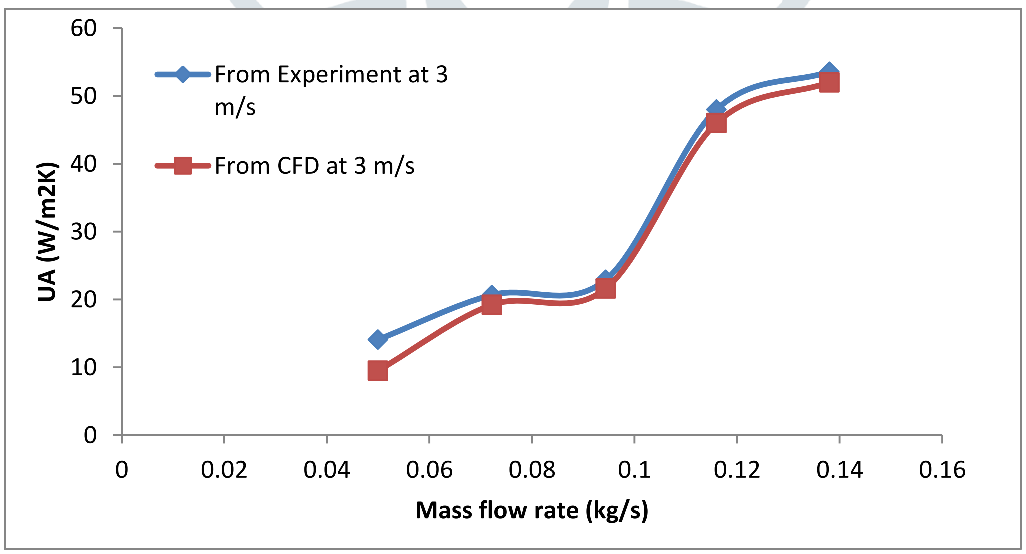


Fig.4.16 showing the comparison of value of overall heat transfer coefficient for experimental and CFD values

From the above comparison graph it is found that the value of overall heat transfer coefficient obtained from the CFD analysis is quite closed to the value obtained from the experimental analysis performed by chaudhary et.al [1] for both velocity of air that is at 3 m/s and 4 m/s. hence the CFD model of the radiator analysis is correct.

The comparison of value of temperature of water at the exit of radiator for with and without fins model is shown in the table below

Table.5.1 Showing the Comparison of temperature for with and without fins radiator

| Mass flow rate of water (kg/s) | Temperature of water at the exit (K) for with fins radiator | Temperature of water at the exit (K) for without fins radiator |
|---|---|---|
| 0.05 | 668 | 668.6 |
| 0.0722 | 668.3 | 669.2 |
| 0.0944 | 669.8 | 700 |
| 0.116 | 700.3 | 700.8 |
| 0.138 | 700.5 | 701 |

Form the above comparison it is found that value of temperature of water at the exit in without fins radiator is more as compared to the temperature of water in radiator having fins. So it is observed that as the surface area of radiator get increased the rate of heat transfer get increased and the temperature of hot liquid get decrease.

### 5.3 For copper material

Here in this Copper is used to manufacture tube and fins of the radiator. With copper here it considered the different mass flow rate of water that is 0.05, 0.0722, 0.0944, 0.1166, 0.1388 kg/s and it also considered the different velocity of air that is 3, 4, 5 and 6 m/s. the selection of copper material is shown in the below fig. During the analysis boundary conditions required for the analysis is same as that of considered in Aluminium material case. The value of temperature of water at the exit of radiator for different combination of mass flow rate and velocity of air is shown in the figure given below

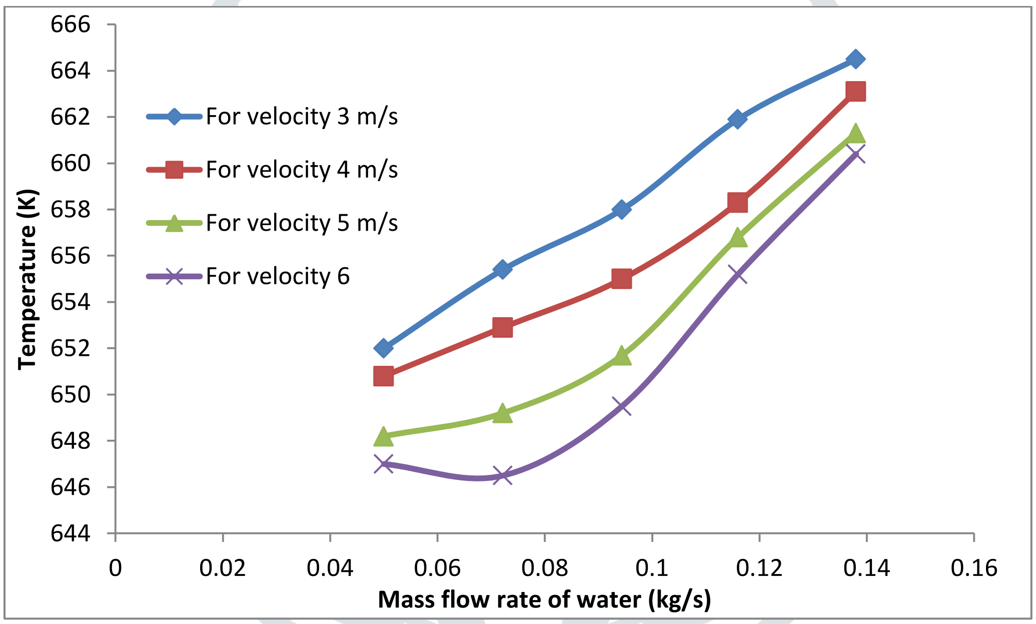


Fig.5.7 showing the value of temperature of water at the exit of radiator for copper material

From the above fig it is conclude that as the mass flow rate of water inside the radiator get increased the heat transfer from the hot water get decrease, due to this the temperature of water at the exit of radiator get increased as the mass flow rate increased. From the above analysis it is also found that as the as the velocity of air flowing over the radiator get increased the heat transfer rate get increased and the value of temperature of water at the exit of radiator get reduced with the increase in velocity of air. Here in this analysis it is also found that the heat transfer from the radiator at a particular mass flow rate of water and at particular velocity of air is greater than the heat transfer from the aluminium tub radiator. So it is conclude that the copper material is batter for heat transfer from the radiator as compared to the aluminium.

### 5.4 For Brass Material

Here in this brass is used to manufacture tube and fins of the radiator. With brass here it considered the different mass flow rate of water that is 0.05, 0.0722, 0.0944, 0.116, 0.138 kg/s and it also considered the different velocity of air that is 3, 4, 5 and 6 m/s. the selection of brass material is shown in the below fig. During the analysis boundary conditions required for the analysis is same as that of considered in copper and aluminium material case. The value of temperature of water at the exit of radiator for different combination of mass flow rate and velocity of air is shown in the figure given below

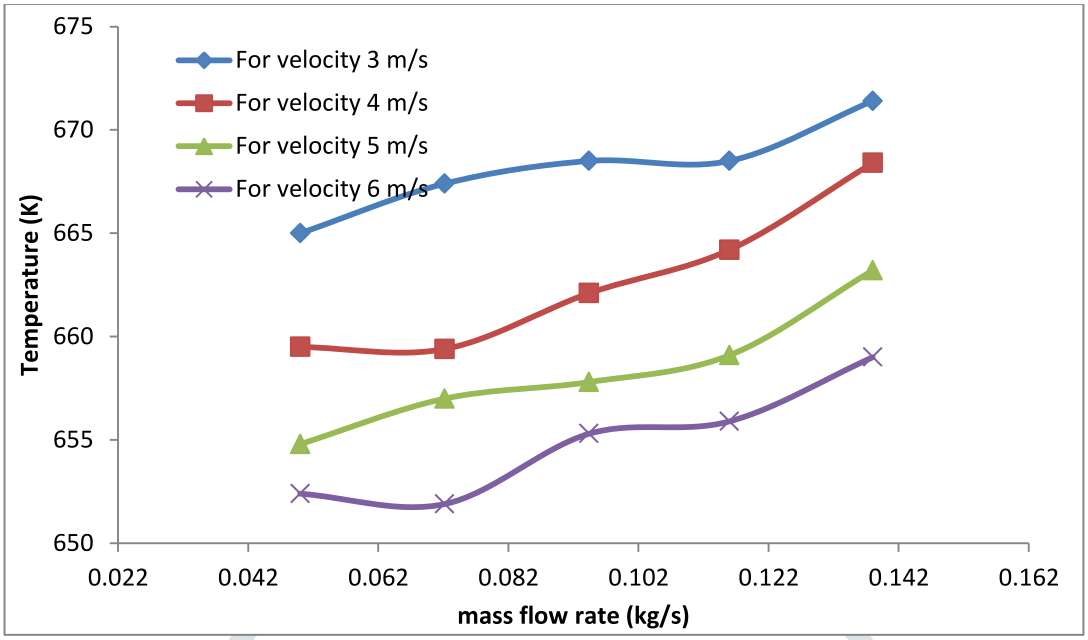


Fig.5.9 showing the value of temperature of water at the exit of radiator for brass material

From the above fig it is conclude that as the mass flow rate of water inside the radiator get increased the heat transfer from the hot water get decrease, due to this the temperature of water at the exit of radiator get increased as the mass flow rate increased. From the above analysis it is also found that as the as the velocity of air flowing over the radiator get increased the heat transfer rate get increased and the value of temperature of water at the exit of radiator get reduced with the increase in velocity of air. Here in this analysis it is also found that the heat transfer from the radiator at a particular mass flow rate of water and at particular velocity of air is greater than the heat transfer from the aluminium tube radiator. So it is conclude that the copper material is better for heat transfer from the radiator as compared to the aluminium and brass.

## CONCLUSION

- Computational Fluid dynamic model of Radiator type heat exchanger was developed based on the experimental analysis of radiator performed by Chaudhary.
- Through the numerical analysis one can find the effect of any parameters on the heat transfer rate.
- After analyzing the heat transfer for with fin and without fins radiator it is found that heat transfer in fins tube type radiator is more as compared to without fins radiator.
- It is also observed that as the mass flow rate of hot water flowing inside the radiator increases, the rate of heat transfer from hot water get decreases for a constant velocity of air.
- It is also found that as the velocity of air flowing over the radiator in the cross direction is increases the rate of heat transfer from hot water gets increases and water will cool rapidly.
- After analyzing three different materials it is concluded that heat transfer is maximum in copper tube and fins type radiator as compared to the brass and aluminium material used for tube and fins.

## REFERENCES


**[1]** Adina T.G, Alexandru D, Luminit G.P, Mihai C, Bogdan M.D. Entropy generation assessment criterion for compact heat transfer surfaces. Applied Thermal Engineering 2015; 280 87: 137-149.

**[2]** Errol B. Arkilic, Martin A. Schmidt & Kenneth S. Breuer, Slip flow in MicroChannels, Dynamics Symposium Oxford UK, July 1994.

**[3]** H. Abbassi, Entropy generation analysis in a uniformly heated microchannel heat sink, Energy 32 (10) (2007) 1932–1947.

**[4]** Abuzaid, M. Al-Nimr, Entropy generation due to laminar incompressible forced convection flow through parallel-plates microchannel, Entropy 6 (5) (2004) 413–426.

**[5]** Baghdar H, Haghighi K, Javadi M. Experimental and numerical investigation on particle deposition in a compact heat exchanger. Applied Thermal Engineering 2017; 115: 406-417.

**[6]** J.J. Brandner, E. Anurjew, L. Buhn, E. Hansjosten, T. Henning, U. Schygulla, A. Wenka, K. Schubert, Concept and realization of microstructure heat exchangers for enhanced heat transfer, Experimental Thermal and Fluid Science 30 (2006) 801–809.

**[7]** Jignesh M. Chaudhari, Dattatraya Subhedar, Nikul Patel “Experimental investigation of finned tube heat exchanger” international journal of innovative research in advanced engineering (ijirae) issn: 2349-2163 volume 1 issue 5 (june 2014)

**[8]** Chunxin Y Xiangdong X, Xingjuan Z, Peng K, Chao W, Han Y,. Study on the heat transfer characteristic of compact heat exchanger based on experimental data. Procedia 255 Engineering 2015; 121: 293-299.

**[9]** Camilleri R, Howey D.A, McCulloch M.D. Predicting the flow distribution in compact parallel flow heat exchangers. Applied Thermal Engineering 2015; 90: 551-558.

**[10]** K. Chen, Second-law analysis and optimization of microchannel flows subjected to different thermal boundary conditions, Int. J. Energy Res. 29 (3) (2005) 249–263.

**[11]** J.Y. Desmons Numerical and experimental study of a solar equipped with offset rectangular plate fin absorber plate Volume 31, Issue 13, Pages 2025-2206 RenewableznergyanInternationalJournal,http://www.sciencedirect.com/science/journal/0960148
**[12]** L.B. Erbay, M.M. Yalcln, M.S. Ercan, Entropy generation in parallel plate microchannels, Heat Mass Transfer 43 (2007) 729–739.
**[13]** Mushtaq I. Hasan, A.A. Rageb, M. Yaghoubi, Homayon Homayoni, Influence of channel geometry on the performance of a counter flow microchannel heat exchanger, International Journal of Thermal Sciences 48 (2009) 1607–1618
**[14]** Min-Hwan, Kyoungmin Kim, Kim, Dong Rip Kim, Kwan-Soo Lee, Thermal performance of microchannel heat exchangers according to the design parameters under the frosting conditions, International Journal of Heat and Mass Transfer 71 (2014) 626–632
**[15]** Bari S, Hossain S.N. Design and optimization of compact heat exchangers to be retrofitted into a vehicle for heat recovery from a diesel engine. Procedia Engineering 2015; 105: 472-266 479.
**[16]** Hassan H, Zahra H. Investigating the effect of properties variation in optimum design of compact heat exchanger using segmented method. Chemical Engineering Research and Design 2016; 112: 46-55.
**[17]** K.S. Lee, S.D. Park, J.S. Kwak, Measurement of the heat transfer coefficient in the dimpled channel: effects of dimple arrangement and channel height, J. Mech. Sci. Technol. 23 (2009) 624–630.
**[18]** Mirkovic, Z., Heat Transfer and Flow Resistance Correlation for Helically Finned and Staggered Tube Banks in Cross Flow, Heat Exchangers: Design and Theory Source Book, (edited by N. H. Afgan and E. U. Schlünder), Hemisphere, Washington, D. C, pp. 559-584, 1974.
**[19]** G.P. Peterson, Convective heat transfer and flow friction for water flow in micro-channel structures, Int. J. Heat Mass Transfer 39 (1996) 2599–2608.
**[20]** K. Sopian, M.H. Yazdi, S. Abdullah, I. Hashim, A. Zaharim, Entropy generation analysis of liquid fluid past embedded open parallel microchannels within the surface, Eur. J. Sci. Res. 28 (3) (2009) 462–470.
**[21]** Suzuki, K., Hiral, E., Miyake, T., Numerical and Experimental studies on a two Dimensional Model of an Offset-Strip-Fin type Compact Heat Exchanger used at low Reynolds Number. International Journal of Heat and Mass Transfer 1985 28(4) 823-836
**[22]** S.Y. Won, P.M. Ligrani, Flow characteristics along and above dimpled surfaces with three different dimple depths within a channel, J. Mech. Sci. Technol. 21 (2007) 1901–1909.
**[23]** Zhang L. W., Balachandar S., Tafti D. K. and Najjar F. M. 1997.Heat Transfer enhancement Mechanisms in Inline and Staggered Parallel Plate Fin Heat Exchanger. International Journal of Heat and Mass Transfer 40(10):2307-2325