International Journal of Computer Aided Manufacturing
Vol. 1: Issue 2
www.journalspub.com

# A Computational Study of Solar Air Heater for Its Performance Enhancement by Numerical Approach Using Standard CFD Tools

*Tridib Kumar Mahata*
Department of Mechanical Engineering, Manipal Institute of Technology, Manipal (Karnataka), India

***Abstract***

*In the following report, the main objective is to calculate the efficiency of the solar air heater, mean exit temperature of air using CFD (Computational Fluid Dynamics) simulation. CFD software promises to be a futuristic field of application especially in the field of thermal engineering. The analysis is performed on a two-dimensional built solar air heater. The geometry modelling is done in "Gambit" software followed by its analysis for mean exit temperature of air in "Fluent" (ANSYS). The solar air heater is modelled using both rectangular and circular shaped fins placed in an alternate fashion to give high turbulence effect to air flow. Few elementary calculations are also done in order to find the net heat flux and hydraulic diameter of the duct.*

**Keyword:** Computational fluid dynamics (CFD), gambit, solar air heater

***Corresponding Author**
*Email ID: tridibkumarmahata@gmail.com*

## INTRODUCTION

A two-dimensional solar air heater is to be constructed as per the dimensions mentioned below in the figure. The heater consists of three main regions namely the entry region, test region and exit region. Air is made to pass through the heater at the inlet of the entry region. The air is heated by a constant heat flux acting over the boundary enclosed by the fins in the test region (Figures 1–6). [1–3]

Calculate the following:

1) The Heat Flux acting by analytical solution
2) For the six mass flow rates of air mentioned in the tables, find the exit mean temperature, heat content of exit air and efficiency of the heater through CFD simulation.

| Sr. No | Mass Flow Rate(kg/s) |
|---|---|
| 1. | 0.014 |
| 2. | 0.02 |
| 3. | 0.025 |
| 4. | 0.03 |

3) Also plot the graphs of the following: [2–4]

- Exit mean temperature vs mass flow rate.
- Heat content of exit air vs mass flow rate.
- Efficiency of the heater versus mass flow rate.

 *© JournalsPub 2016. All Rights Reserved* 

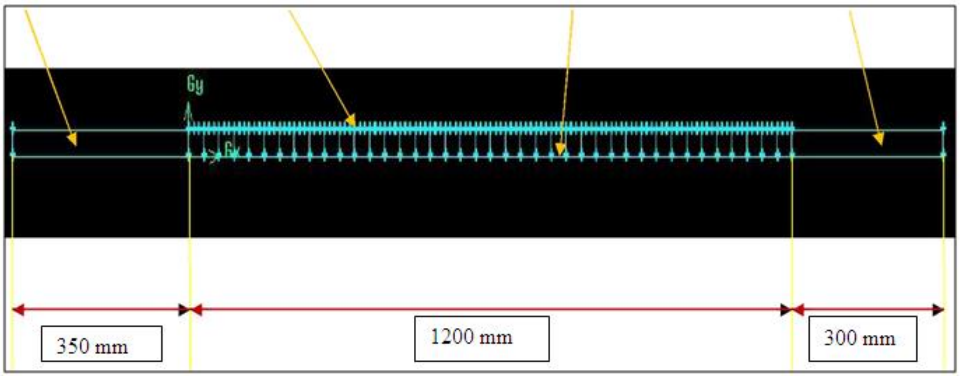



***Fig. 1:*** *Uniform UDL Meshing of Top Layer of Plate.*

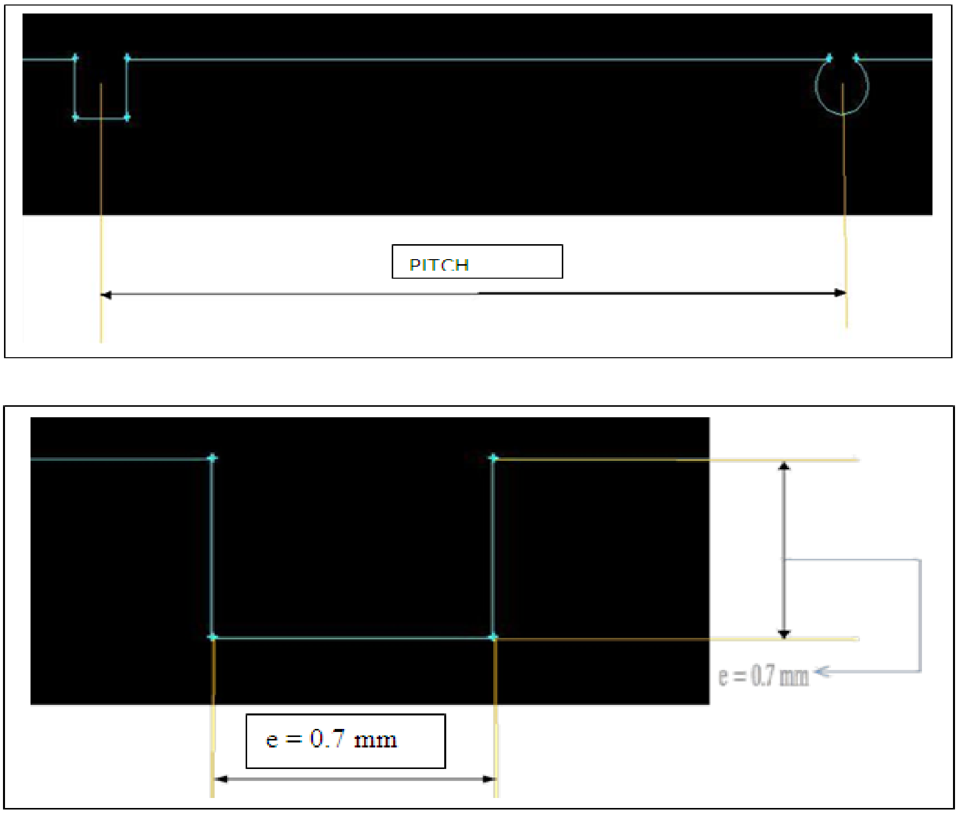



***Fig. 2:*** *Internal Layer of Solar Heater with Counter to Increase the Turbulence.*

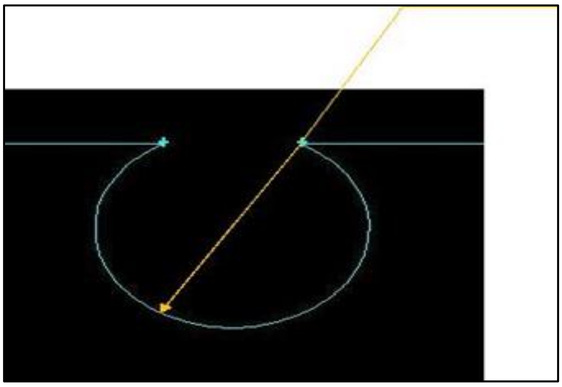


***Fig. 3:*** *Internal Layer of Solar Heater with Circular Counter to Increase the Turbulence.*

## MODELING IN GAMBIT SOFTWARE

### General Commands Used [4–6]

Geometry Command Button
Vertex Command Button
Edge Command Button
Face Command Button
Connect vertices/edges/faces
Delete vertices/edges/faces/meshes

***Mesh Command Button***

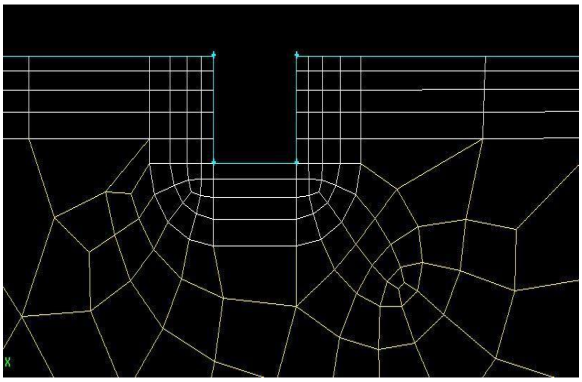


***Fig. 4:*** *Mesh Around Square Fin.*

*Boundary Layer Command button*
*First Row, a = 0.1*
*Growth Factor, b/a = 1.2*
*Rows = 4*

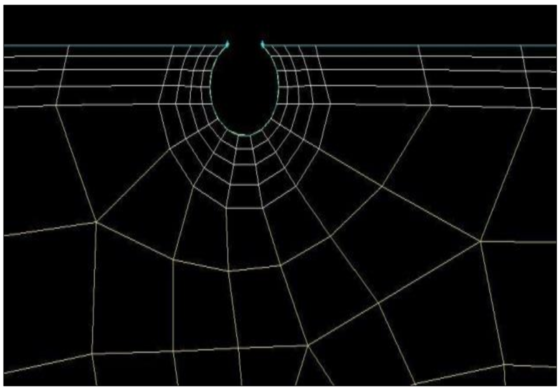


***Fig. 5:*** *Mesh Around Circular Fin.*

*Mesh Edges*
*Ratio = 1*
*Interval size = 15*

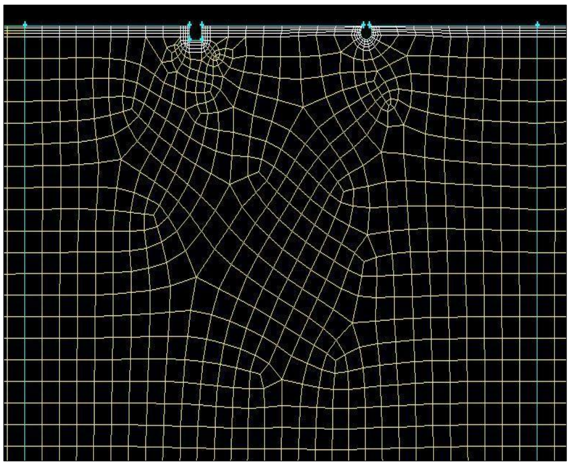


***Fig. 6:*** *Meshing the Whole Plane.*

*Mesh Faces*
*Elements: Quad*
*Type: Pave*
*Interval size: 1*

**Zones Command Button**
Specify boundary types

| Name | Type |
|---|---|
| Heat In | Wall |
| Mass In | Mass Flow Inlet |
| Mass Out | Pressure Outlet |

Specify Continuum Types

| Region | Type |
|---|---|
| Inlet | Fluid |
| Exit | Fluid |
| Test | Fluid |
| Absorber plate | Solid |

**ANALYSING IN FLUENT**
The following analysis is done on the case having a mass flow rate of 0.014 kg/s.
**Models**
Energy – ON
Viscous Model- k-omega, SST, low Recorrections

**Materials**
Fluid (air)
Solid (Aluminium)

**Boundary Conditions**
**Zone: Heat in (Type-Wall)**
Heat Flux = 1000 W

**Zone: Mass in (Type-Mass flow inlet)**
Mass flow rate = 0.0933 kg/s
Direction Specification Method: Normal to Boundary
Turbulence Specification Method: Intensity & Hydraulic diameter Turbulent Intensity = 3%,
Hydraulic diameter = 0.05 m
Total temperature = 307 K

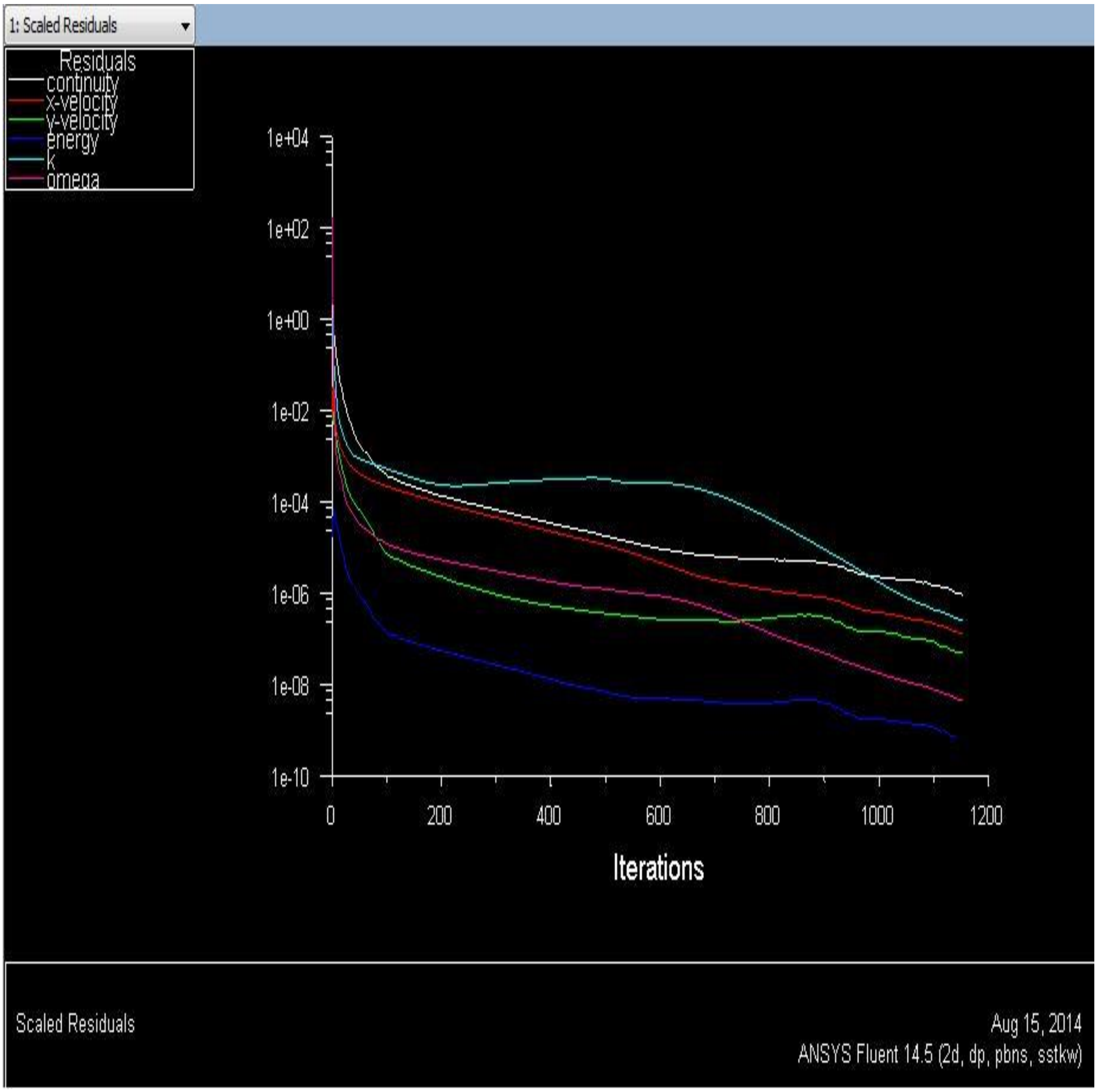
**Solution Methods**
Scheme-Simple
Gradient-Least squares cell based

Pressure-Standard
Momentum-Second order upwind
Turbulent Kinetic Energy – Second order upwind
Specific Dissipation Rate – Second order upwind (Figures 7, 8).

**Residual Monitors**

| Residual | Absolute Convergence Criteria |
|---|---|
| Continuity | 1e-06 |
| x-velocity | 1e-06 |
| y-velocity | 1e-06 |
| Energy | 1e-06 |

**Number of Iterations**: 100000



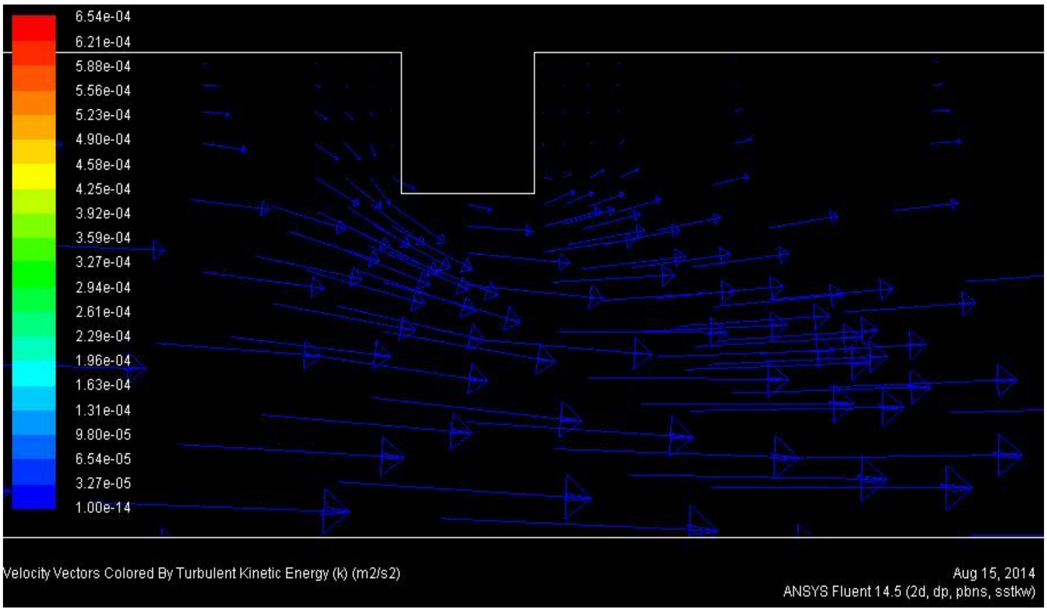



***Fig. 7:*** *Laminar Compressed Flow across the Counter.*

**CALCULATIONS:**
Given data:
Inlet temperature = 307 K
$C_p$ = 1005 J/kgK
m = 0.0933 kg/s
$Q_{in}$ = 1000 f or 1.2
1.0 = 1200 W

Exit heat energy, $Q_u = m \times C_p \times (T_o - T_i)$
$= 0.014 \times 1005 \times (319.96 - 307)$ W
= 182.34 W

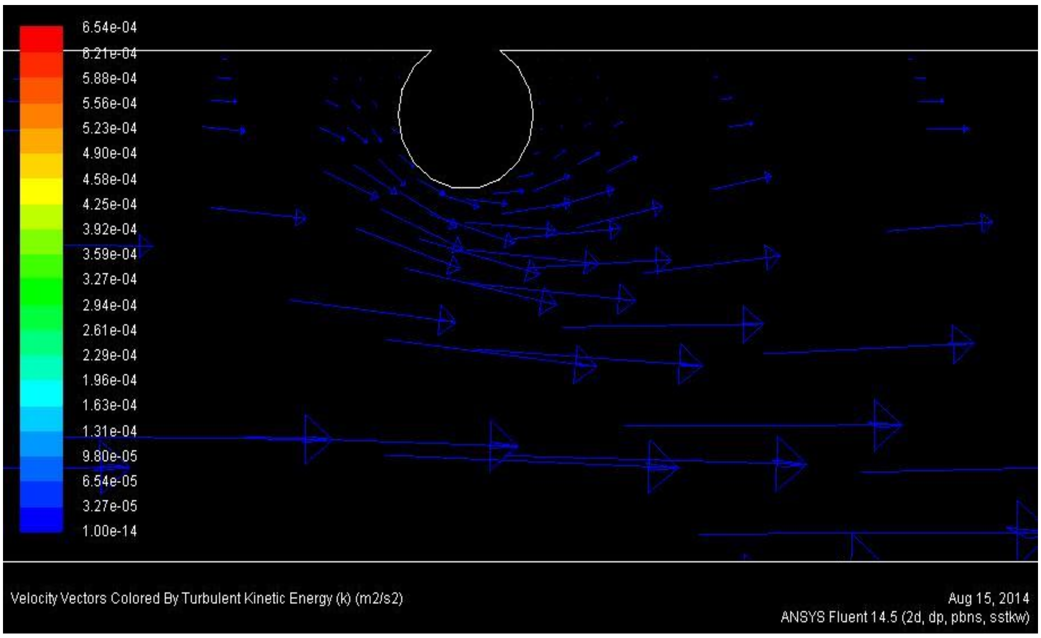
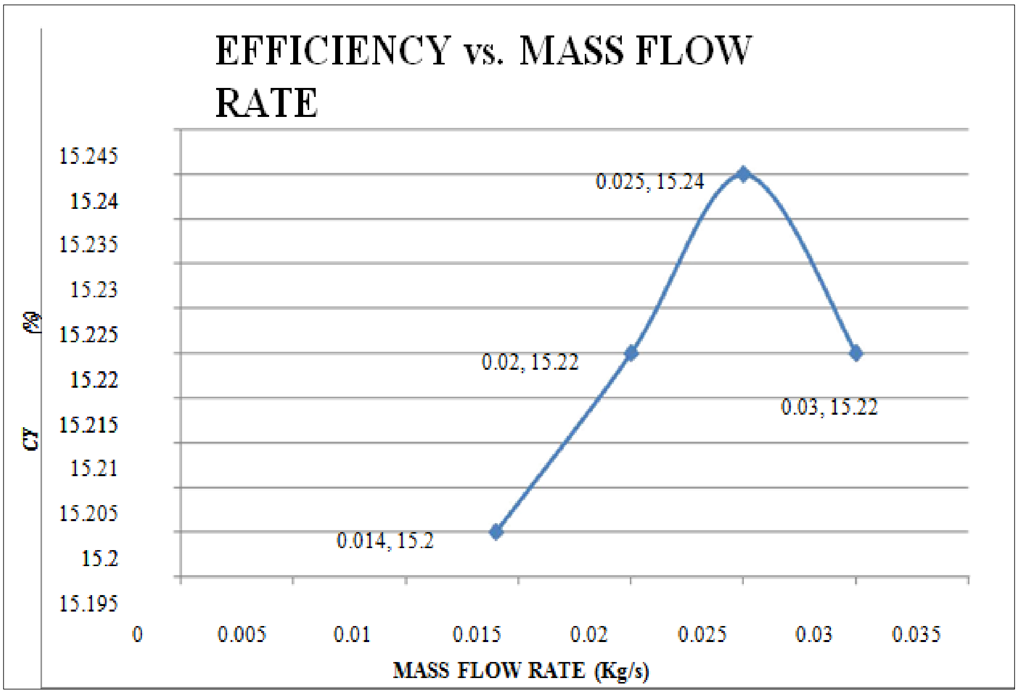
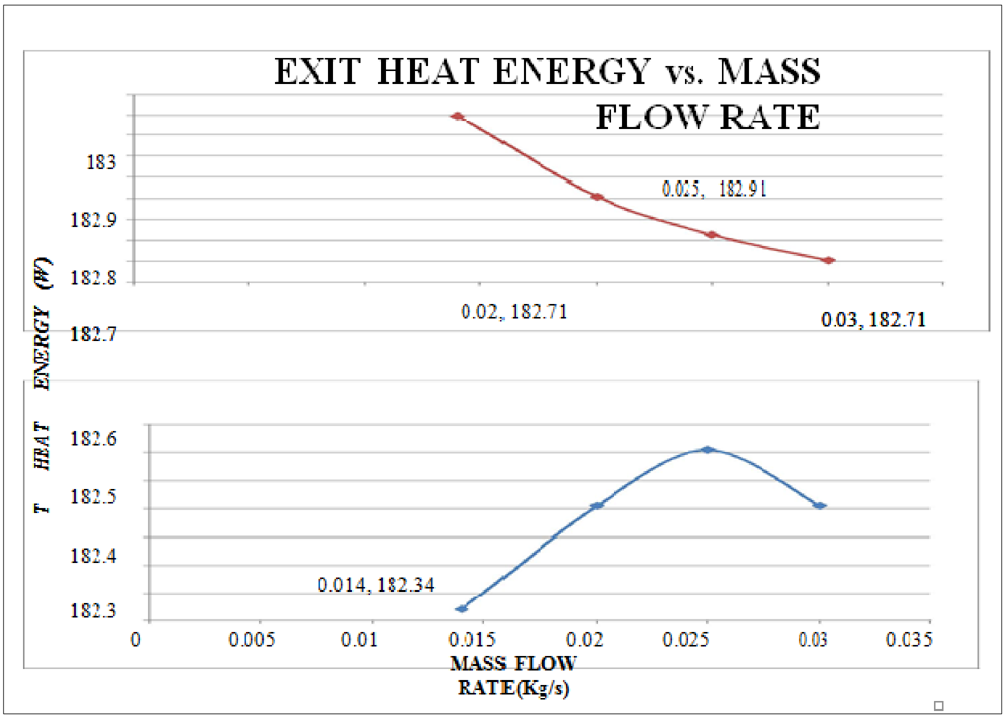



***Fig. 8:*** *Flow Across Circular Fin.*

*© JournalsPub 2016. All Rights Reserved* 




***Graph 1:** Representation of Efficiency Versus Mass Flow Rate.*




***Graph 2:** Representation of Exit Heat Energy Versus Mass Flow Rate.*

**Tabulation of Results**

| **Mass Flow Rate** (kg/s) | **Outlet Temp. (K)** | **Exit Heat Energy (W)** | **Efficiency of Heater (%)** |
|---|---|---|---|
| 0.014 | 319.96 | 182.34 | 15.20 |
| 0.02 | 316.09 | 182.71 | 15.22 |
| 0.025 | 314.28 | 182.91 | 15.24 |
| 0.03 | 313.06 | 182.71 | 15.22 |

## CONCLUSION

As per the given problem statement that is how to increase the heat transfer in a two dimensional solar air heater under different mass flow rate ranging from 0.014 to 0.03 kg/s and also by varying the material properties to get these solution listed as below:

1. The Exit temperature decreases with increase in mass flow rate ranging from (0.014 to 0.03 kg/s) and also with varying material Properties.
2. Exit Heat Energy of air increases with increase in mass flow rate and with the more Surface area Contact between the Air and the heated surface
3. Hence the efficiency of the solar air heater increases with increase in mass flow rate and also with varying material with different attributes.

## REFERENCE

1. Adreson J. *Computational Fluid Mechanics*. 1995.
2. Kalita N., Misra R.D. *CFD Analysis of Room Air Distribution*. LAP Lambert Academic Publishing; 2012.
3. Kumar S., Saini R.P. CFD based performance analysis of a solar air heater duct provided with artificial roughness. *Renew Energy*. 2009; 34(5): 1285–91p.
4. CFD lectures notes by Manjunath Sir (MIT Manipal)
5. Gambit and Ansys Fluent Tutorial
6. Chaudhari S., Makwana M., Choksi R., *et al*. CFD Analysis of Solar Air Heater. *Int J Eng Res Appl*. 2014; 4(6): 47–50p.