Thermal Analysis of Heat Exchanger used in 30 kW CL-CSP plant with the help of CFD (Fluent)

Shubham Mishra[1, a], Akash Patel[1,b], Dr. Mukesh Pandey[2], Anurag Gaur[3]
[1]School of Energy and Environment Management Department, UTD RGPV, Bhopal- 462033.
[2]HOD, Energy Department, UTD RGPV, Bhopal- 462033.
[3]Assistant Professor, Energy Department, UTD RGPV, Bhopal-462033.

Abstract- Double pipe heat exchanger is simplest form of heat exchanger containing two coaxial pipes, one inside another. Hot fluid flows inside the pipe and cold fluid flows in the gap between the two pipes, the surface of the inner pipe is the heat transfer surface which is responsible for the heat transfer between the fluids. This analysis based on the computational fluid dynamics where air to air heat exchanger utilized in cross linear concentrated solar power system CL-CSP. The following research is under process at RGPV BHOPAL. The study is based on the analysis of temperature distribution of heat transfer fluid (air), comparison of effectiveness of heat exchanger with and without insulation, and flow behavior of HTF in both hot and cold side of pipes. CFD simulations have been performed on the Ansys 16.0. K-ε turbulence models have been assessed and model has shown to provide more accurate results for temperature prediction.

Keywords- CFD, Double Pipe Heat Exchanger, and Heat transfer fluid (HTF), CL-CSP.

I. Introduction

Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change [1]. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense is known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and pressure drop [2].
CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design[3].
CL-CSP system is the combination of two solar thermal technologies which is Linear Fresnel and Solar tower. When linear Fresnel LF and (high efficiency) of central tower is hybridized on a specific platform along with proper management of different parameters, a new system formed which is known CL-CSP system. The basic concept of CLCSP system is as follow [4].

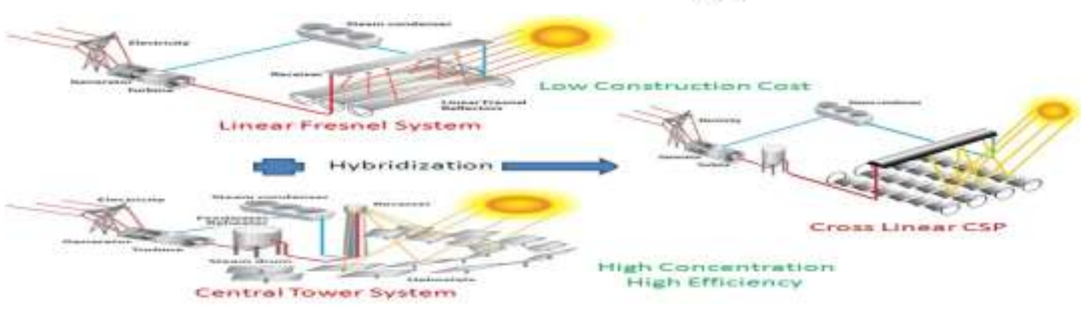

Figure 1: Hybridization of cross and linear type solar absorption system [5]

The CL-CSP project is a Technology based attempt by a puddle of University and Industries from India i.e. RGPV, MNRE GOI, BERGEN group and Japan's Toyo Engineering Corporation, Tokyo Institute of Technology & Solar flame Corporation. This projects aim at the proof on the principle of the cross linear concentration (CL) concept which is invented and formulated by Prof. Yutaka Tamura, Emeritus Professor of Tokyo Institute of Technology and Representing Director of Solar Flame Corporation (Tokyo Tech Venture). Research based on aim that a new advanced technology on the CSP can start the solar energy economy and enhances the employment in India with production of the CL-CSP components. Cross linear (CL) concentration system with a temperature range of 300-800 degree Celsius is hybridization of two existing solar thermal technologies i.e. solar tower and linear Fresnel system [4]. System situated between point and linear focusing concentration, Hence with the CL system we can get a higher temperature of around 650 degree Celsius by applying linear focusing method. Mirror lines aligned on north –south axis and receiver line, on east west axis. The both lines crossed each other at right angle. The component of CL-CSP plant for the solar field is small which can easily manufactured by Indian production factories. This project is collaborative project b/w India and Japan to help the solar energy development in India [6]. In this plant double pipe heat exchanger used and shown in figure 2.

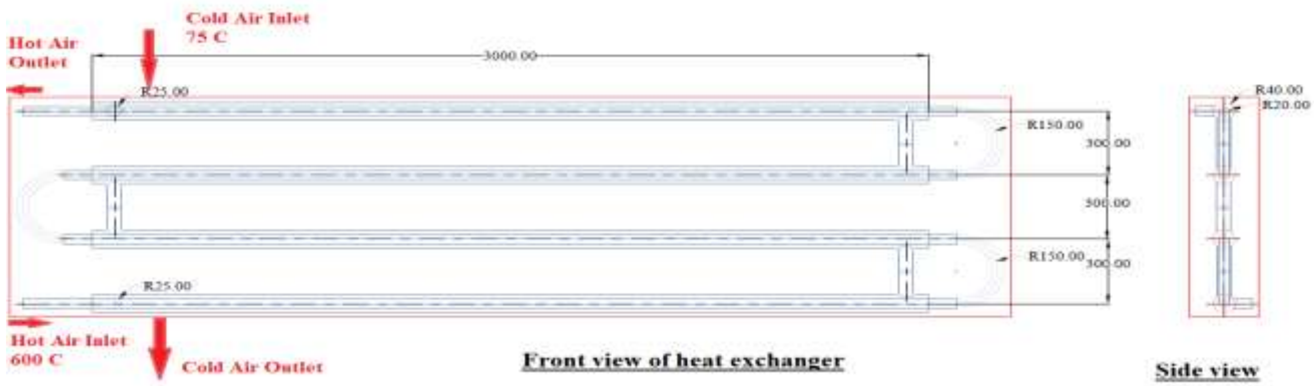

Figure 2: CAD model of double pipe heat exchanger

II. Design Parameters

Double pipe heat exchanger consists of a tube inside a hollow pipe. The HTF inside the pipe is cold air which is supplied by air drum at an operating temperature and pressure shown in table1 and the HTF in the tube is hot air coming from the cavity linear receiver (CLR) at a particular temperature and pressure as shown in table 1.

Table 1 Properties of HTF used in experimentation

Properties of HTF		Shell side	Tube side
Fluid Quantity	kg/hr	334	334
Temperature	^0C	75	600
Operating Pressure	Pa	654000	589000
Specific Heat	kJ/kg^0C	1.019	1.115
Thermal Conductivity	W/m^0C	0.0299	0.0587
Density	kg/m^3	1.2	1.2

The inner tube is of Stainless steel (SA 213 TP 304 H) and outer pipe is of Carbon Steel (SA 106 Gr B) and their properties used in the analysis are shown in table 2. The insulation used is of Rockwool fiber and its properties are also shown in table 2.

Table 2 Properties of material used in heat exchanger

Properties of Stainless Steel (SA 213 TP 304 H)		Value
Density	kg/m^3	8060
Specific Heat	kJ/kg^0C	530
Thermal Conductivity	W/m^0C	17
Properties of Carbon Steel (SA 106 Gr B)		**Value**
Density	kg/m^3	7850
Specific Heat	kJ/kg^0C	461
Thermal Conductivity	W/m^0C	51
Properties of Rockwool		**Value**
Density	kg/m^3	160
Specific Heat	kJ/kg^0C	2100
Thermal Conductivity	W/m^0C	0.038

III. Modeling and Meshing

Figure 2 (a) shows the 3D model of heat exchanger without insulation and Figure 2(b) shows model with insulation. The modeling is done using UNIGRAPHICS 8.5.

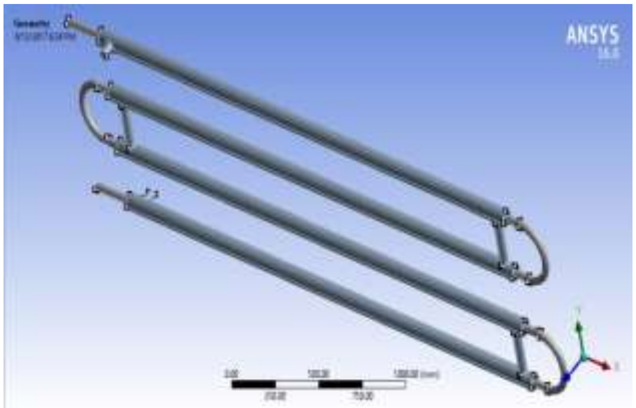

Figure 2(a): 3D model of heat exchanger without insulation

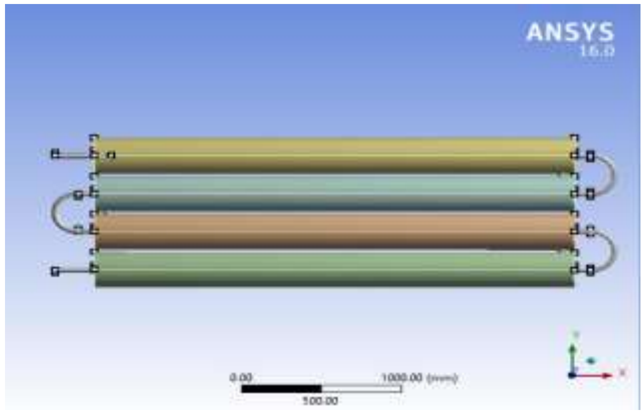

Figure 2(b): 3D model heat exchanger with insulation

Figure 3(a) shows the meshing of the model. Advanced size meshing is used with curvature feature on. Figure 3(b) shows the inflation given at the boundaries of inner and outer pipe which is used for better results. Maximum size element = 183.350 mm.
Minimum size element = 0.916770 mm.
The total numbers of nodes are 714132 and total numbers of elements are 2034529.

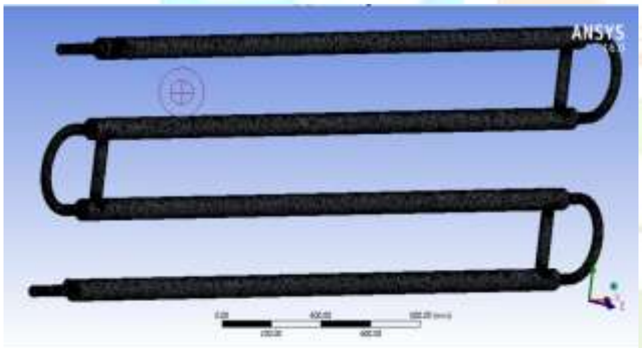

Figure 3(a): Meshing of the model without insulation

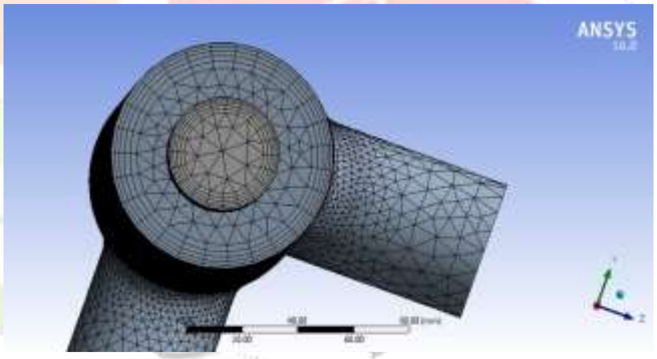

Figure 3(b): Inflation at the boundaries

Figure 4(a) shows the meshing of the model of heat exchanger with insulation. Advanced size meshing is used with curvature feature on.
Maximum size element = 373.180 mm.
Minimum size element = 1.86590 mm.
The total numbers of nodes are 846024 and total numbers of elements are 2176259.

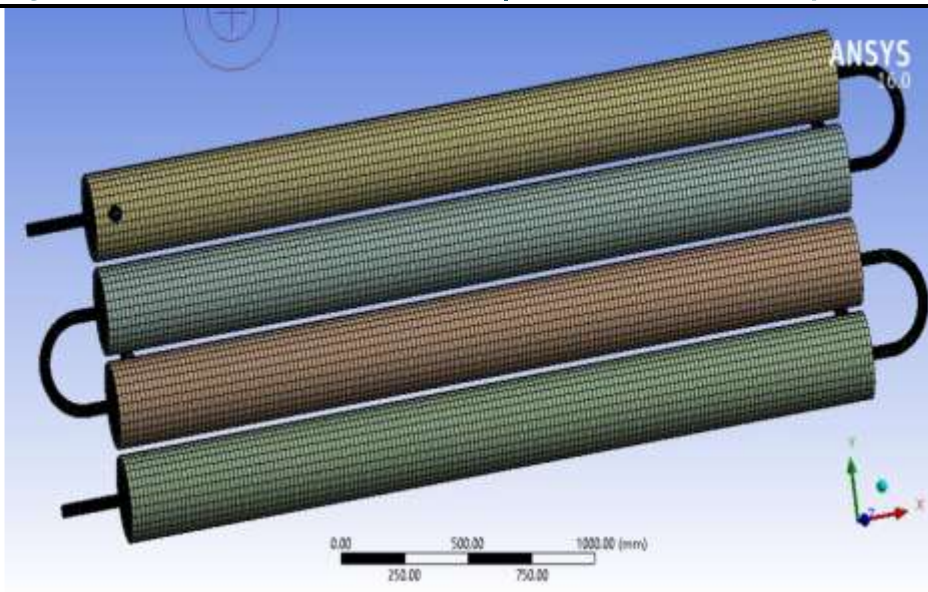

Figure 4(a): Meshing of the model of heat exchanger with insulation

IV. Heat Transfer Model

For turbulent flow analysis k-ε model is being used.

1. The turbulent kinetic energy k, and its rate of dissipation Ɛ following equation [7].

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\epsilon-Y_m+S_k$$

2. Rate of dissipation Ɛ following equation [7].

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho\epsilon\mu_i)}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\mu+\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}(G_k+C_{3\epsilon}G_b)-C_{2\epsilon}\rho\frac{\epsilon^2}{k}+S_\epsilon$$

The Energy equations used in this CFD analysis can be written as [8]:

$$\nabla\cdot\left(\vec{V}(\rho E+p)\right)=\nabla\cdot(k\nabla T-\textstyle\sum_i h_i \vec{j}_i)$$

Where k is the thermal conductivity, T is the moist air temperature and J_i is the diffusion flux of species i. The two terms on the right hand side of above Eq. represent the energy transfer due to conduction and species diffusion, respectively.

The following assumption has been made for the following analysis.

a) Steady state heat transfer, include conduction and convection, effect of radiation can be neglected
b) The specific heat of the fluid both hot and cold is remains constant and does not vary with temperature.
c) The pressure at the pipe and tube remains constant i.e. constant pressure heat transfer.
d) During the flow the kinetic energy of the largest scale motions is transferred to successively smaller scale motions without loss. It is only at the small scales that energy dissipation occurs for the effective model application of k-ε model [9].

V. Result and Discussion

A. Temperature Variation with respect to Tube length:

Figure: 5(a) shows the graph of temperature variation of hot air inside each tube with respect to length of pipe. Figure: 5(b) shows the volume rendering of temperature variation of hot fluid.

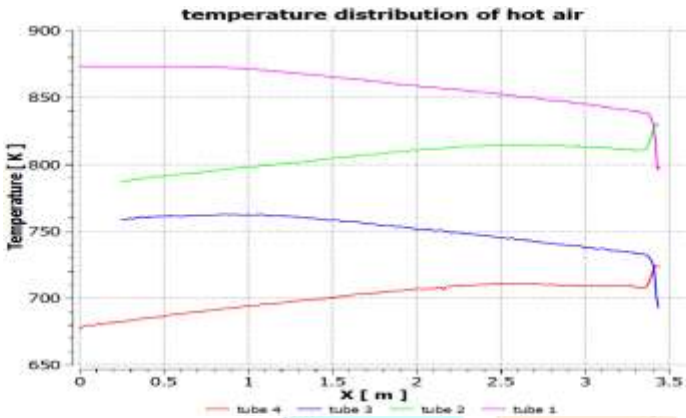

Figure: 5(a)

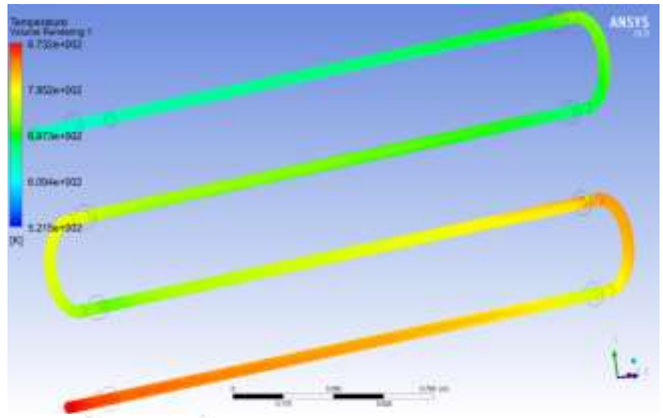

Figure: 5(b)

In air heat exchanger the inlet hot air temperature in inner tube is 600^{0}C (873K) which is taken from the designed value. The analysis shows for first tube in graph, temperature is decreased from 873K to 840K as length of tube increased up to X=3400mm just before pipe bend . In the portion of pipe bend the temperature is suddenly increases because there is no contact between shell and tube for heat transfer. Now in pipe two the temperature is 840K at the pipe length of X=3400mm and it decreases up to 790K at distance X=300mm. same process occurs in pipe three and four. Finally the temperature at outlet of the pipe i.e. pipe four is 680K.

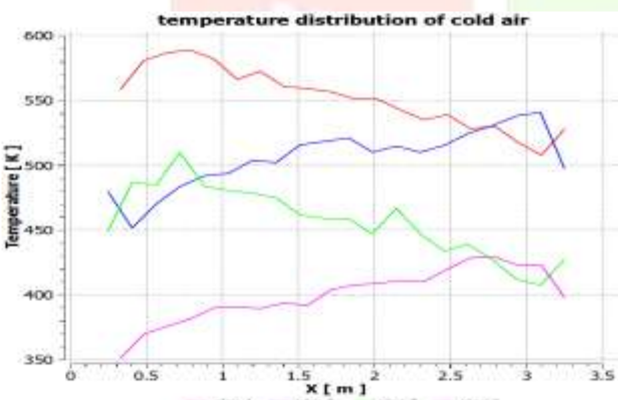

Figure: 6(a)

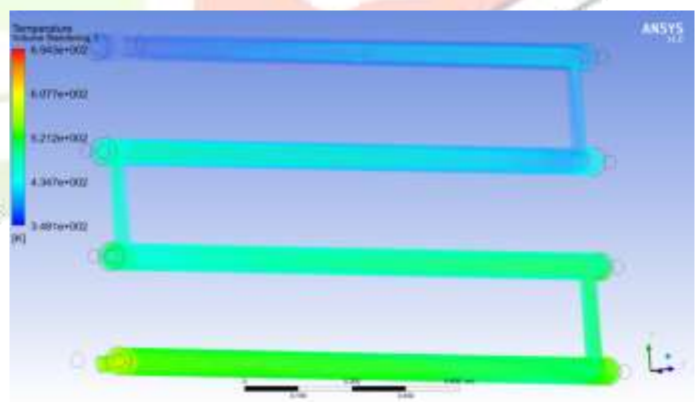

Figure: 6(b)

Figure: 6(a) shows graph of temperature variation of cold air inside each tube with respect to length of pipe. Figure: 6(b) shows the volume rendering of temperature variation of cold fluid. In pipe 4 cold air enters at 348K and increases 433K at distance 2550mm (2.55m) and then decreases to 400K. The decrease in temperature is because of flow behavior of air which can be seen in figure: 7(b) at the position X=2560. Now cold air which attains 400K enters in pipe 3 and due to high turbulence temperature increases and maximum temperature reached 510K and again decreases up to 450K due to sudden change in area of pipe. Similarly this process happens in pipe 2 and 1. Finally maximum temperature attained at outlet of shell is 588K.

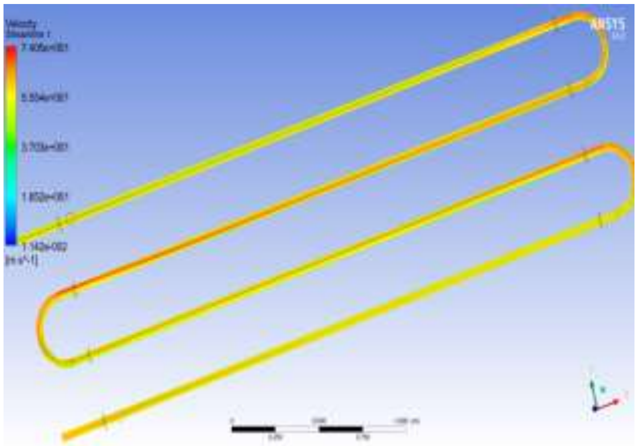

Figure: 7(a)

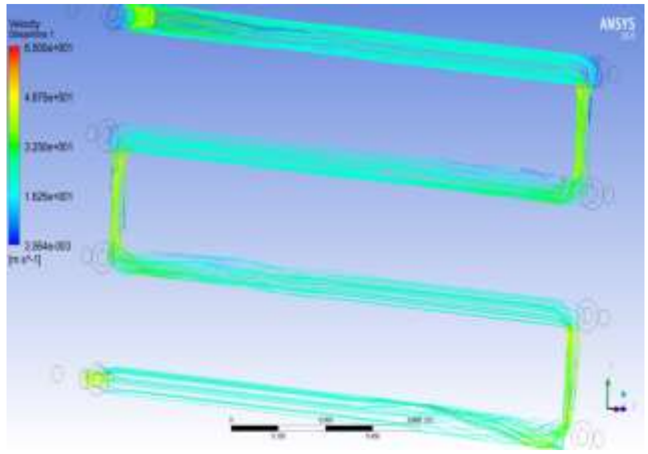

Figure: 7(b)

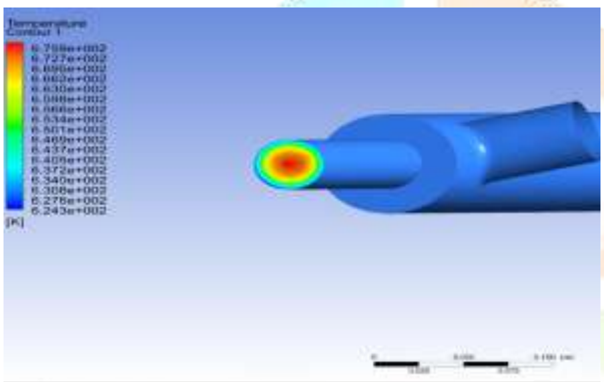

Figure: 8(a)

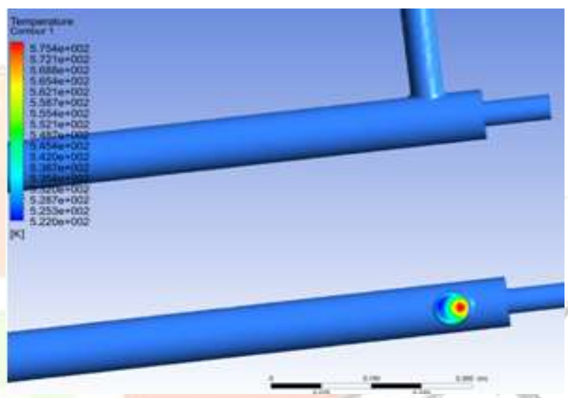

Figure: 8(b)

Figure: 8(a) shows the temperature contours of hot air at outlet and Figure: 8(b) shows the temperature contours of cold air at outlet.

B. Velocity distribution of hot air inside the pipe:

Figure: 9(a) shows the graph between velocity of hot air inside the tube and length of the tube. The graph shows the velocity variation of hot air inside the tube. Initially the velocity is 61.5 m/s and enters in pipe 1. This velocity continuously increases as length of the tube increase. The maximum velocity attains up to 71.7 m/s at distance of 3.3 m from starting point. Now hot air goes in pipe two and initially this velocity decreases due to bend and then increases from X=3.4m and attains maximum velocity 72.1 m/s at a distance X=2.3 m. Similarly the variation of velocity in tube 3 and 4 with respect to distance is shown in graph.

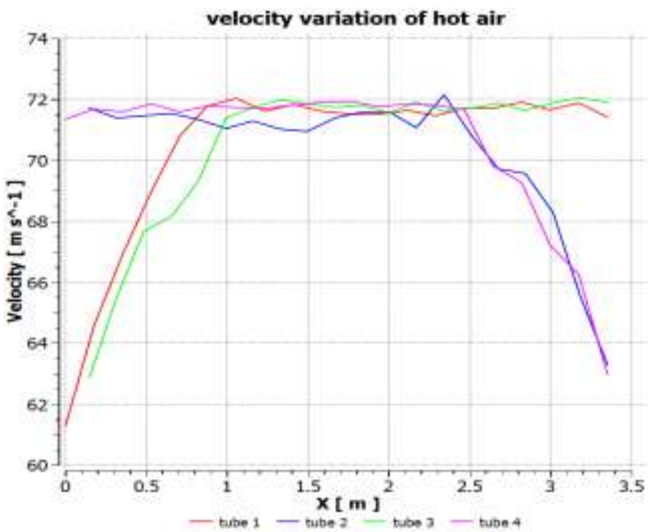

Figure: 9(a)

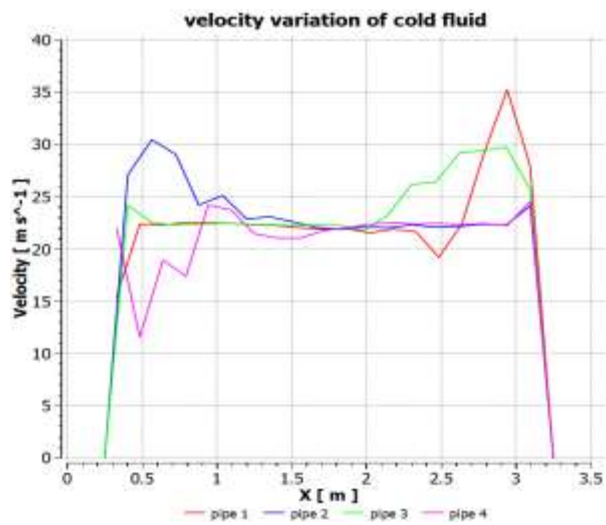

Figure: 9(b)

Figure: 9(b) shows the graph between variations of velocity of cold fluid with respect to length of the pipe. Initially cold air enters in pipe 4 and velocity of cold air is 22 m/s and due to sudden expansion the velocity decreases up to 11.7 m/s and again increases, this fluctuation of velocity can be seen in figure: 9(b). The velocity of cold air is at peak in the joint between pipe 1 and 2 due to sudden decrease in area of pipe. This maximum velocity goes up to 35 m/s in joining pipe.

C. Determination of Effectiveness of Heat Exchanger:

As we know the hot inlet temperature is 600^0C (873K) and outlet temperature is 407^0C (680K) and for cold air the inlet temperature is 75^0C (348K). The final temperature of cold air at outlet is determined by two methods; one is conventional method and second is software analysis.

In conventional method heat balance equation is used.

Heat balance equation for heat exchanger is given as [10]:

$$q = \dot{m}_c c_{pc}(T_{co} - T_{ci}) = \dot{m}_h c_{ph}(T_{ho} - T_{hi})$$

$$\eta = \frac{(T_{co} - T_{ci})}{(T_{hi} - T_{ci})} \text{for } \dot{m}_c c_{pc} < \dot{m}_h c_{ph}$$

$$\eta = \frac{(T_{hi} - T_{ho})}{(T_{hi} - T_{ci})} \text{for } \dot{m}_c c_{pc} > \dot{m}_h c_{ph}$$

Where

$\dot{m}_h$ = mass flow rate of hot air.

c_{ph} = specific heat of hot fluid at constant pressure.

$\dot{m}_c$ = mass flow rate of cold air.

c_{pc} = specific heat of cold fluid at constant pressure.

T_{hi} = temperature of hot fluid at inlet, T_{ho} = temperature of hot fluid at outlet.

T_{co} = temperature of cold fluid at outlet, T_{ci} = temperature of cold fluid at inlet.

η = effectiveness of heat exchanger

$T_{hi} = 600^0$C, $T_{he} = 407^0$C, c_{ph} = 1.115 kJ/kg^{0}C

$T_{ci} = 75^0$C, $T_{ce} = 286.18^0$C, c_{pc} = 1.019 kJ/kg^{0}C

The above equation the outlet temperature of cold air from the pipe is calculate and this value is 286.18^0C (559.18K). Now from the equation of effectiveness, the actual effectiveness can be calculated and after calculation Effectiveness is equal to 0.4022.

After actual calculation software analysis was done and this analysis gives outlet temperature of cold fluid equal to 315^0C (588K) and this gives higher effectiveness 0.4571.

From these analyses effectiveness is increased by the software analysis.

VI. Conclusion:

The temperature distribution and velocity distribution of hot and cold fluid have been simulated using computational fluid dynamics. The CFD model has produced reasonably accurate results when compared with an already published experimental work. Some important conclusion drawn from this analysis given below:

- It was found from the analysis that the variation of velocity not followed the same pattern in all the four tube, it may be because variation of pressure of hot and cold fluid in the tube and pipe.
- The analysis shows that temperature at the bend of pipe is suddenly increases in every pipe; this can be understood by the pressure variation in the pipe which increases in the bend as there is loss of some velocity due to the loss due to bend.
- From the analysis the effectiveness is increased by 5.49% with the use of Ansys software as compare to the actual calculation.
- The effect of insulation in the pipe can be studied further and its effects on effectiveness can be analyzed.
- The variation of heat transfer coefficient with respect the length during the flow for both hot cold can be analyzed will help to find out the heat transfer along the length the pipe of heat exchanger and it require basic knowledge of steady flow energy equation and flow through pipes.

Nomenclature	
ρ	density (kg/m^3)
Cp	specific heat, (j/kg^{0}C)
$C1\varepsilon$, $C2\varepsilon$, $C\mu$	constant that experimentally determined.
Gk	generation of turbulent kinetic energy the arises due to mean velocity.
Gb	generation of turbulent kinetic energy that arises due to buoyancy.
*Y*m	fluctuating dilation in compressible
Sε,SK	source terms defined by the user.
k , *ε*	prandtl numbers turbulent kinetic energy and dissipation rate.
μ_t	turbulent viscosity.
k	turbulent kinetic energy
ε	rate of dissipation
i,j	indexes
^{0}C,K	temperature in degree Celsius and Kelvin.
C1,C2,Cmu	model constant
Abbreviations	
CFD	computational fluid dynamics
CL-CSP	cross- linear-concentrator solar power
HTF	heat transfer fluid
Subscript	
1, 2	number indexes

Acknowledgement

The project is sponsored by ministry of New and Renewable Energy, government of India and assisted by Japan's Toyo Engineering Corporation, Tokyo Institute of Technology & Solar flame Corporation. I would extend my sincere thanks to Prof. Yutaka Tamura, Professor of Tokyo Institute of Technology and Representing Director of Solar Flame Corporation (Tokyo Tech Venture) for sharing his rich experience and kind support. I would like to extend my gratitude to Mr. Himanshu Bora for his kind support.

References

[1] Bharmal Juzar et.al."Design modification for the improvement of heat exchanger efficiency" International Journal of Advance Research in Engineering Science of Technology Vol. 4, Issue 04,2017, e-ISSN: 2393-9877, p-ISSN: 2394-2444.

[2] http://www.cpdlr.com/notes-articles-engineering/302-cfd-methodology-for-heat-exchanger.html.

[3] Www.cfd-online.com. 2017. Transport equations for standard k-epsilon model. [ONLINE] Available at: https://www.cfd-online.com/Wiki/Standard_k-epsilon_model. [Accessed 5 june 2017]

[4] Y. Tamura et.al. (2014)," Cross linear solar concentration system for CSP and CPV" Elsevier science Ltd Energy Procedia 49 (2014) 249 – 256.

[5] Mr. Ravi Kumar Tiwari et.al. (2016)," Analytical Design for Heliostats System and Review of Its Working for A 30 kW CL-CSP System at RGPV Bhopal" IJSRD - International Journal for Scientific Research & Development| Vol. 4, Issue 03, 2016 | ISSN (online): 2321-0613.

[6] Cross linear solar concentration system for CSP and CPV, Y. Tamaura, S. Shigeta, Q.-L.Meng, T. Aiba and H. Kikura.

[7] Cable, Matt.An Evaluation of Turbulence Models for th Numerical Study of Forced and Natural Convective Flow in Atria. s.l. : Pearson Education, 2009.

[8] Rafat Al-Waked et.al. (2013),"CFD simulation of air to air enthalpy heat exchanger" Elsevier science Ltd Energy Conversion and Management volume 74 (2013) pp 377–385.

[9] http://www.leb.eei.unierlangen.de/winterakademie/2011/report/content/course01/pdf/0112.pdf

[10] R.C. Sachdeva," Fundamental of Engineering Heat and Mass Transfer". Fourth edition, New Age International Publishers, ISBN: 978-81-224-2785-1.