International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 06 | June -2017 www.irjet.net p-ISSN: 2395-0072

CFD Analysis of Double Pass Foldable Cum Portable Solar Air Heater

Yash J Patil[1], Sanjeev D Suryawanshi[2], Eknath R Deore[3]

[1] ME (CAAD) Student, Mechanical Engineering Department, SSVPS's BSD college of Engineering, Dhule, Maharashtra, India

[2] Professor & HOD, Mechanical Engineering Department, SSVPS's BSD college of Engineering, Dhule, Maharashtra, India

[3] Professor, Mechanical Engineering Department, SSVPS's BSD college of Engineering, Dhule, Maharashtra, India

---***---

Abstract - *In this paper, double pass, foldable cum portable solar air heater is analyzed using Computational Fluid Dynamics (CFD) study. Also, results obtained by CFD study are compared with results obtained through experimental study. Polyester material is used for absorber, polyethylene is used for glazing and heatlon is used for insulation. Simulation is performed using Ansys Fluent tool. Ansys Fluent software was used to numerically solve the heat transfer, radiation and fluid flow equations. Some Important parameters like outlet air temperature, outlet air velocity and mass flow rate are evaluated along with absorber plate temperatures. Re-Normalization Group (RNG) k-ε turbulence model with enhanced wall treatment is used to simulate turbulence conditions for air flow through solar air heater duct. Material properties like density, thermal conductivity etc. are assigned for materials polyester, polyethylene and heatlon. DTRM solar load model is used to trace incoming solar radiations on absorber and glazing of solar air heater. Solar ray tracing is done by giving date and time of day on which experimentation was performed for result validation. Solar rays are traced for 11 April 2017 at the geographical location of Dhule (Maharashtra) by assigning latitude - 20^0 54' and longitude - 74^0 46' to solar load calculator, which is inbuilt in Ansys Fluent. During the CFD analysis outlet air temperature, temperature rise, outlet velocity and mass flow rate has been determined. Maximum temperature of 56.08^0C is observed at time 2:00 p.m. of day with average outlet velocity of 1.2 m/s.*

Key Words: Foldable, Portable, Solar Air Heater, CFD, Fluent, RNG k-ε model, DTRM model.

1. INTRODUCTION

Conventional solar air heater mostly consists of a flat plate collector with an absorber plate generally made of metal, transparent cover system of glass at the top and insulation at the bottom and on the sides. Whole assembly is enclosed in a sheet metal container. Working fluid is air and the passage for its flow varies according to the type of air heater. Materials for the construction of air heater are similar to the liquid flat plate collectors. Transmission of solar radiation through the glazing and its subsequent absorption in the absorber plate can be taken into account by expressions identical to those of liquid flat-plate collectors. Unlike liquid flat plate collectors, system is not pressurized and therefore, light gauge metal sheets can be used. In solar air heaters leakage is also not a big problem, unlike in liquid collectors.

The advantages of computational simulations are that they can produce extremely large volumes of results at virtually no added expense and it is very cheap to perform parametric studies to optimize performance of equipment. The second reason for such work on computational simulation is that some parameters like flow pattern and unutilized portion cannot be viewed experimentally and they are difficult for test. Also, experimental study is expensive as well as time consuming. The results obtained with the CFD study are of acceptable quality provided that good meshing and appropriate application of boundary conditions should have been done.

A thermodynamic analysis of solar air heaters has been performed by Abhishek Saxena et al. [1]. This review article focus on the developments that has followed round the globe in various aspects of solar air heating systems since 1877 up to now, with a glimpse of some novel patents of solar air heaters. Vipin B. Gawande, A. S. Dhoble et al. [2] performed an experimental and two-dimensional computational fluid dynamics (CFD) analysis of a solar air heater has been carried out using right-angle triangular ribs as artificial roughness on the absorber plate as a convenient method for enhancement of thermal performance of solar air heater. The relative roughness pitch (P/e = 7.14–35.71), Reynolds number (Re = 3800–18000), and relative roughness height (e/D = 0.021–0.042) have been selected as design variables for both analysis. ANSYS FLUENT 14.5 with renormalization group k-ε turbulence model is selected for the analysis of computational domain of solar air heater during CFD analysis. An enhancement in Nusselt number and friction factor with a decrease in relative roughness pitch (P/e) and increase in relative roughness height are presented and discussed with reference to experimental investigation and CFD analysis. a transient 3-D mathematical model for double-pass solar collector with porous media in the lower channel has been developed by Abdel Illah Nabil Korti [3]. Numerical simulations model based on setting mass, momentum and energy balances on finite volumes method are carried out. The governing equations inside the two channels, together with the energy equation in the absorber, insulating and glass cover walls were solved iteratively in a segregated manner. Effects of porosity (70-90 %), mass flow (0.03-0.07 kg/s), solar intensity (514-714 W/m^2) and spacing glass absorber- insulation (7-10 cm) on the dynamic and thermal behaviors of the double-pass solar collector with and without porous media have been discussed. Tabish Alam, Man-Hoe Kim [4], presented numerical study on heat transfer and friction characteristics in rectangular solar air heater duct with semi elliptical shape obstacles. 3-D simulations have been conducted using Renormalization-group k-ε turbulence model. Obstacles are placed on the absorber plate in V-down shape at different angle of attack (α), ranging from 300 to 900. Two different arrangements of obstacles namely; inline and staggered arrangements have been investigated. Debayan Das, Tanmay Basak [5] studied role of distributed/discrete solar heaters during natural convection in the square and triangular cavities: CFD and heatline simulations. "Five different cases based on the heater location on the side walls of the square, triangular-type 1 and triangular-type 2 (inverted triangle) enclosures are studied. The mathematical tool of 'heatlines', which represents the trajectories of flow of heat is used to visualize the total energy flow in the domain. Galerkin finite element method with adaptive grids has been implemented for solving the governing equations of heat and fluid flow and Poisson-type of equations for solving the stream function and heat function. The local and average Nusselt numbers vs. average or cup mixing temperature have been evaluated. Based on the optimum thermal mixing and high temperature uniformity, the discrete solar heating strategy involving the positioning of the heaters near the bottom portion of the side walls and heaters along the central portion of the side walls were found to be energy efficient irrespective of any enclosure. The asymmetrically distributed heating strategy was also found to be suitable for the systems leading to the significant temperature uniformity over a larger region. Overall, the results displayed an increase in the mixing efficiency index for the square and triangular-type 2 (inverted triangle) enclosures in the presence of discrete solar heaters". Sanjay K. Sharma, Vilas R. Kalamkar [6], performed experimental and numerical investigation of forced convective heat transfer in solar air heater with thin ribs. "The rib placement on the absorber plate of solar air heater affects the heat transfer and flow characteristics. This work emphasis on the effects, due to different rib positions on the absorber plate of the solar air heater. Four different rib arrangements of thin transverse continuous and truncated rib position are considered under this study. The experimental and 3D numerical investigation has been carried out. The CFD investigation result is validated with the experimental results. The geometrical and flow parameters are kept constant for each rib configuration (Cases) under investigation. This study on the heat transfer and friction faction due to variations in rib position, on a single heated wall surface (absorber plate), gives several significant outcomes.

Major steps involved in CFD analysis procedure are creating model geometry, meshing the geometry, setting solver, performing calculations through iterative runs and last step is verification and validation of CFD model.

2. GEOMETRY CREATION

Geometry of model consists of an absorber of polyester material, transparent glazing or cover of polyethylene and insulation of heatlon at bottom and sides. For modelling of double pass solar air heater, CATIA V5 is used. Different parts of model like absorber, glazing, side walls and bottom insulation are modelled separately in part design and then assembled in assembly design module using appropriate constraints. The final 3D CAD geometry is made after applying materials and then it's volume is extracted using Ansys Fluent as shown in figure 1. Four slots are made at inlet section of double pass foldable cum portable solar air heater at the bottom inlet air side.

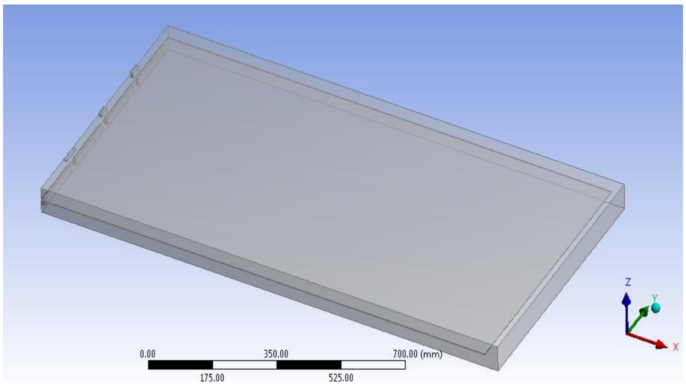

Fig -1: Extracted volume of double pass solar air heater geometry

3. CFD MODEL

CFD tools are used to carry out detail analysis of heat and mass transfer, fluid flow and several other parameters. They provide numerical solution of differential equations used in heat transfer and fluid flow analysis. Ansys Fluent is used in this study.

The flow geometry is extracted from whole CAD model using Fluent and meshed using HyperMesh. Linear tetrahedron mesh is created with appropriate nodes and elements.

Numerical simulation is carried out using steady state, pressure based solver. RNG k-ε turbulence model with standard wall function is used. The RNG k-ε turbulence model of Fluent gives good results for internal flows. Discrete Transfer Radiation Model (DTRM) is used for simulating radiation heat transfer and solar load is applied to it through solar calculator for the location of Dhule for 11 April 2017 with constant direct and diffuse solar radiation in the region of Dhule where experimental work is carried out. All other required parameters are given according to weather conditions of that day.

4. BOUNDARY CONDITIONS

According to flow conditions, boundary conditions are applied on the computational domain. For the air inlet 'Inlet Velocity' boundary condition is specified and remains constant and inlet air temperature is varied according to the ambient temperature recorded by thermometer. For air outlet 'pressure outlet' condition is specified in accordance with the fan attached for drawing the air through solar air heater. All the material boundaries are considered opaque and side walls are considered as opaque adiabatic walls. Glazing is considered as semi-transparent with appropriate values of absorptivity and transmissivity. Default values of wall roughness and wall height are considered for this analysis.

The convergence criteria are achieved for continuity, x-velocity, y-velocity, z-velocity, k, ε and the energy equation at each time step. The convergence criteria for energy was 1e−6 and for all other parameters it was set at a level of 1e−3. Physical and thermal properties of different engineering materials used in the simulation are shown in Table 1. In this study air is assumed to be incompressible and other engineering materials used in the simulation study are treated as isotropic and homogeneous.

TABLE -1: Thermal and fluid properties of materials used in simulation study

Material	Density ($Kg\ m^{-3}$)	Specific heat capacity ($J\ Kg^{-1}\ K^{-1}$)	Thermal Conductivity ($W\ m^{-1}\ K^{-1}$)
Air	1.215	1006	0.0242
Polyester (Absorber)	1450	1320	0.3
Polyethylene (Glazing)	920	2300	0.4
Heatlon (Insulation)	80	2000	0.033

5. SIMULATION RESULTS

CFD model is studied for the location of Dhule for 11 April 2017 from 1:00 p.m. to 5:00 p.m. by giving ambient air temperature as inlet air temperature measured for 1 hour time interval. As suction fan is used to draw air at outlet of double pass solar air heater, outlet condition of pressure outlet has been applied to validate CFD model with experimental model. Figure 2 shows temperature contour on absorber surface showing maximum and minimum temperatures of absorber surface at time 1:00 p.m.

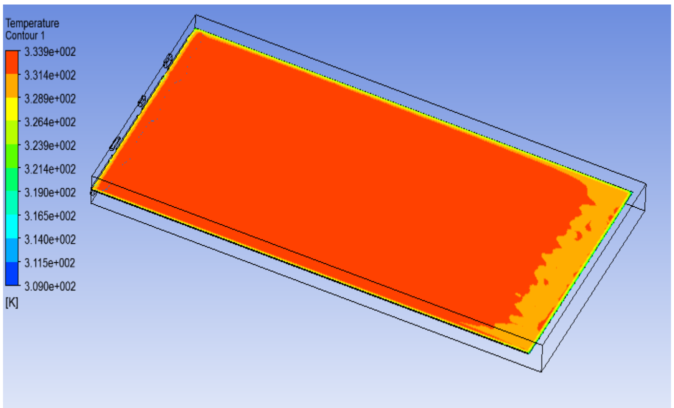

Fig -2: Temperature contours on absorber from top of double pass solar air heater at 1:00 p.m.

Temperature contours on absorber from top of double pass solar air heater are as shown in figure 2, shows absorber temperature in second pass of double pass solar air heater ranging from 322^0K to 339^0K. On this side of absorber solar radiations are directly falling through transparent polyethylene glazing.

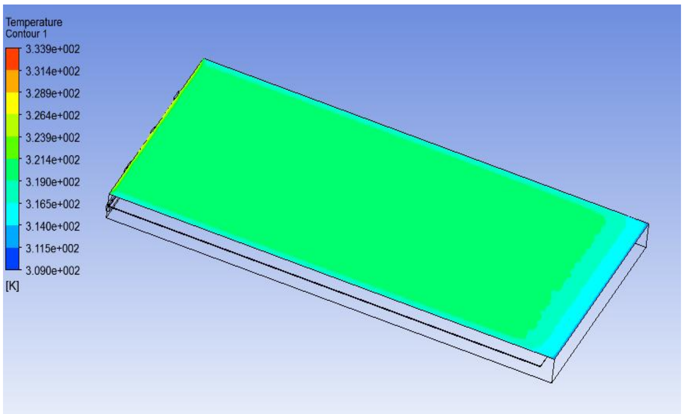

Fig -3: Temperature contours at glazing of double pass solar air heater

Temperature contours on polyethylene glazing are shown in figure 3. It shows temperature variation from 315^0K to 322^0K from starting of second pass of double pass solar air heater to outlet slots of air respectively.

Figure 4 shows temperature contours at mid plane of double pass solar air heater fluid zone i.e. at plane parallel to length of solar air heater. Temperature variation from lower left side from where ambient air is entering solar air heater traveling from left to right and coming in second pass and traveling from right to left towards air outlet can be easily visualized with the help of figure 4.

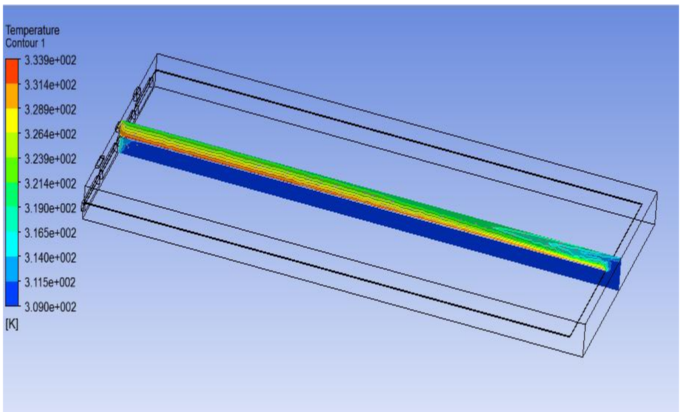

Fig -4: Temperature contours at mid plane of double pass solar air heater fluid zone

Report of average outlet temperature, average absorber temperature was observed for five timings of day as listed in Table 2. Along with these temperature values, outlet air velocity is also observed through contours of velocity and velocity streamlines.

Table -2: Simulated average outlet temperatures, temperature rise

Time in Hours	Average Temperature (^{0}C)			Temperature Rise (^{0}C)
	Absorber	Inlet	Outlet	
13:00	49	39	53.71	15.71
14:00	50	40	56.08	16.08
15:00	50	38	54.25	16.25
16:00	48	37	50.58	13.58
17:00	44	35	45.31	10.31

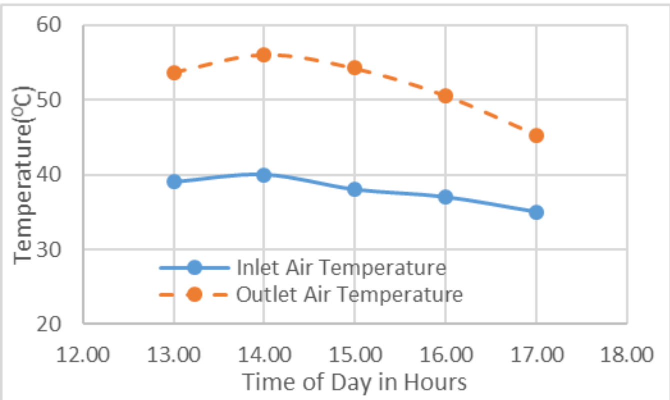

Chart -1: Graph showing temperature rise in double pass foldable cum portable solar air heater

Table 2 shows Temperature readings obtained through CFD simulation study. Maximum temperature is observed at 2:00 p.m. of day and maximum temperature rise is observed at 3:00 p.m. These readings with comparison of inlet and outlet air temperature are shown in chart 1.

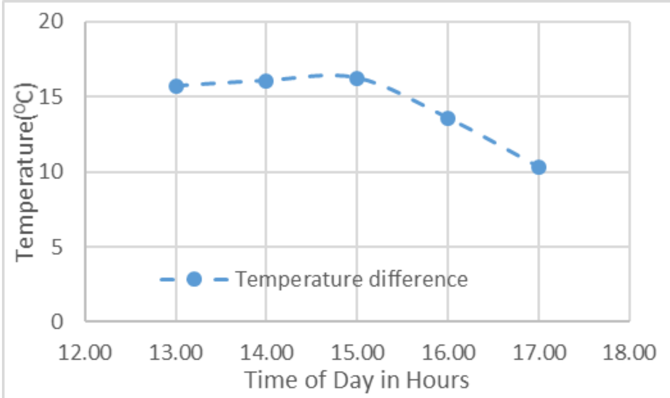

Chart -2: Graph showing variation of temperature difference with respect to time

Graph 2 shows temperature rise of air with respect to time.

6. CFD MODEL VALIDATION

Experimental study of double pass foldable cum portable solar air heater is performed on 11 April 2017. Results obtained through experimental study are compared with results obtained through CFD simulation.

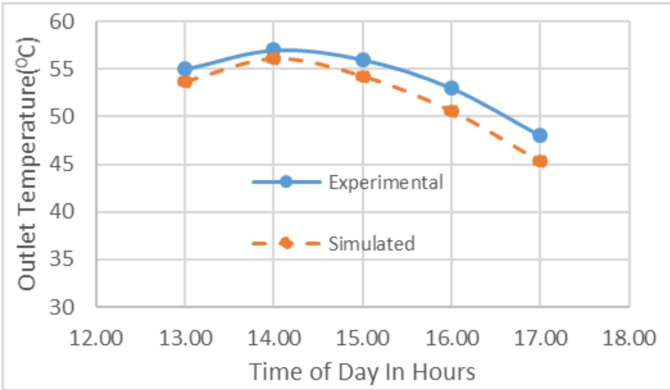

Chart -3: Comparison between Experimental and Simulated result values of outlet temperatures

Chart 3 shows comparison between outlet air temperatures of CFD simulation study and experimental study. Variation of readings is within acceptable limit.

7. CONCLUSIONS

Computational Fluid Dynamics study of novel metal less foldable cum portable double pass solar air heater is carried out with the help of Ansys Fluent. Results obtained through simulation shows good agreement with experimental results. Thus, it can be concluded that RNG k-ε turbulence model coupled with DTRM solar load model yields good results. Maximum temperature of 56.08^{0}C is observed at time 2:00 p.m. of day. Similar procedure of CFD analysis can be used for further performance optimization of solar air heater.

REFERENCES

Abhishek Saxena, Varun and A.a. El-Sebaii, ”A thermodynamic review of solar air heaters”, Renewable and Sustainable Energy Reviews 43,863–890.

Vipin P Gawande, A.S. Dhoble et al. “Experimental and CFD based thermal performance prediction of solar air heater provided with right-angle triangular rib as artificial roughness ,” J Braz. Soc. Mech. Sci. Eng. DOI 10.1007/s40430-015-0391.

Ablel Illah Nabil Korti, “Numerical 3D heat flow simulations on double-pass solar collector with and without porous media”, Journal of Thermal Engineering Yildiz Technical University Press, Istanbul, Turkey, Vol. 1, Issue No. 1, pp. 10-23, January, 2015.

Tabish Alam, Man-Hoe Kim, “Numerical study on heat transfer and friction characteristics in rectangular solar air heater duct with semi elliptical shape obstacles”, Energy 112 (2016) 588-598,2016.

Debayan Das, Tanmay Basak “Role of distributed/discrete solar heaters during natural convection in the square and triangular cavities: CFD and heatline simulations”, Solar Energy 135 (2016) 130–153, 2016.

Sanjay K. Sharma, Vilas R. Kalamkar, “Experimental and numerical investigation of forced convective heat transfer in solar air heater with thin ribs”, Solar Energy 147 (2017) 277–291.