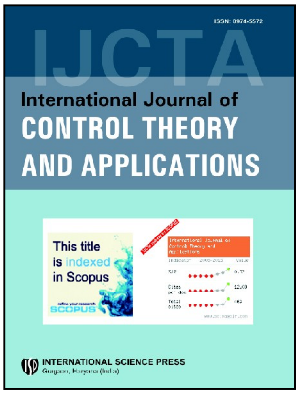

International Journal of Control Theory and Applications

ISSN : 0974-5572

Volume 10 • Number 28 • 2017

CFD Analysis of a Thermal-Mixer for Solar Concentrated Air Heater

S. Babu, T. Prabhu, V. S. Arjun, G. Sai Phanendra and Y. Eswar Chandra

Department of Mechanical Engineering, PSG College of Technology, Coimbatore-641004
E-mail: sjsham@gmail.com & eswar.yarlagadda95@gmail.com

***Abstract*:** A Computational Fluid Dynamics (CFD) analysis of a thermal-mixer to be used in combination with a solar concentrator air heater to vary the temperature of the outlet air. The purpose of the CFD analysis was to determine whether the outlet temperature and pressure of thermal-mixer possessed uniform conditions and minimal thermal induced stresses. CFD analysis was carried out on a scale 1:5 to minimize computational time and complexity. This work describes the analysis using ANSYS FLUENT Software. The effect of various velocities of the inlet air in mixing has been studied in this paper. The velocity plays a role in thermal mixing by influencing the turbulence of the fluid. The percentage of uniformity achieved is 90.9 percent to that of the average.

Index Terms: Thermal-mixer, CFD, Turbulence, Solar Concentrated air collector.

1. INTRODUCTION

The thermal mixer ensures constant input conditions to engine inlet by combining the hot and cold streams whenever needed as per the engine requirement. The available journals used for studying the various designs of thermal mixer, mixing of turbulent hot and cold fluids, temperature fluctuations in hot and cold fluid in a tee connector have been listed and their conclusions have been discussed from the references [1-2].

Mixing two different fluids in a micro mixer is one of the most basic processes in micro fluidics. Due to the small size of the device, pressure driven flows in simple channels are laminar and mostly uniaxial, so that confluence liquids tend to flow side by side. On the other hand, as the Schmidt number is typically very large, the Peclet number. [4] Accordingly, in the absence of any transverse convection, complete mixing is reached after the distance LE”d Pe along the channel that can be prohibitively long. The easiest way to overcome this difficulty and enhance mixing is to induce transverse flows through clever geometries. [6] The simplest one consists of a T-shaped micro mixer.

Micro mixer, when the two inlet fluids are either both water (W–W case) or water and ethanol (W–E case) as discussed in paper [5]. We showed that, at smaller Reynolds numbers (Re=100), mixing is controlled by transverse diffusion, and therefore by the residence time of the fluid mixture occupying the interfacial region. Accordingly, as ethanol is slightly more viscous, and therefore slower, than water, the degree of mixing in the

(W–E) case is slightly larger than that of the W–W case. On the other hand, at larger Reynolds numbers, mixing water and ethanol may take considerably longer as discussed in papers [4-6]

The effectiveness of thermal mixing is found better at lower Reynolds number. The measured temperature field in the T-shaped micro channel agrees well with the theoretical model.

The theoretical and experimental investigations for the thermal mixing in the T-shaped micro channel are discussed in papers [8-10]. The internal fluid temperature was measured by the fluorescence imaging technique. The temperature was captured by the thermal tracer camera as in the paper the temperature distributions under different flow rates are investigated. The results indicate that the Reynolds number is a crucial parameter to the heat transfer process in the micro channel. The temperature field changes with different flow rate combinations. The LFID is stratified compared with the mass mixing.

2. BACKGROUND

The current work deploys Computational Fluid Dynamics (CFD) tools in the development and analysis of the Thermal Mixer to ensure the input conditions of SCAH.

The increasing costs of facility operations and safety requirements often lead to an increase in the overall financial and schedule costs of new physics experiments. The ysimultaneous increase in the ubiquity, availability,

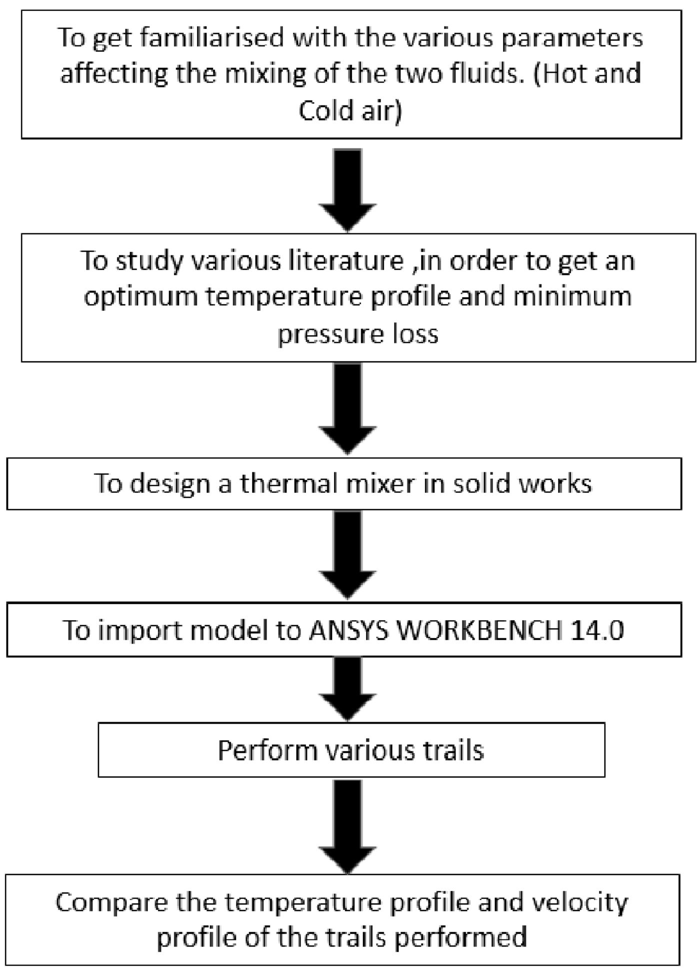

Figure 1: Pre-solver Layout

and maturity of computational tools allows for their relatively rapid and inexpensive deployment because the costs of computational facilities are typically distributed broadly across organizations. Furthermore, the maturity of physics models present in current computational tools, and repeated successes with their use over time, have raised confidence in the resulting data to levels where a subject matter expert utilizing numerical simulations can now deploy these tools to answer many lingering questions before any experimental apparatus is built. Despite this, both designers and simulation experts generally agree that to support the design processes for practical devices, experimentation is still required to provide both real-life data and validate computational models. In combination, the above factors result in computational tool splaying an increasingly important role in the early stages of experimental planning and post-test investigations with experimental data still required to fully vetting the results of the latter.

(A) Need of the Experimentation

The numerical analysis involves simple geometry, ideal boundary conditions to get an initial idea about any proposed concept. When any system has to be realized in real, it has to be tested in laboratory level first before scale up. With input from CFD such as mixer geometry, cold flow injectors, its orientation, and actual mixing time, response time etc., the specification of each components of an experimental set up has to be selected and developed.

The controller has to synchronize with sensor inputs and provide the signal output for the control valves to ensure the input conditions of Scramjet Engine testing. The preliminary trial with the experimental set up helps to test the complete functioning of a thermal mixer, which is realizable with a proper control mechanism. Based on data from preliminary trial, the experimental set up has to be fine-tuned for its performance. The experience gained from the test facility will help to develop the Scramjet Engine testing

This chapter deals with the objectives of the project and the methodology adopted to achieve them. The mixing of hot and cold fluid have been understood from the above journals and is designed and analysed.

- To design a thermal mixer in incorporating all the required parameters.
- To conduct an analysis on the designed architecture with appropriate boundary conditions.

Perform trials with various inlet velocities until we achieve a uniform temperature profile at the outlet with minimal pressure loss.

3. NUMERICAL ANALYSIS

Continuity Equation also called conservation of mass. The overall mass balance is

Input – output = accumulation

Assuming that there is no storage the Mass input = mass output. However, as long as the flow is steady (time-invariant), within this tube, since, mass cannot be created or destroyed then the above equation will be

$$m_1 = m_1 \tag{1}$$

$$\frac{dm_1}{dt} = \frac{dm_2}{dt} \tag{2}$$

$$\rho A_l u_l = \rho A_2 u_2 \tag{3}$$

$$A_1 v_1 = A_2 v_2 \tag{4}$$

It is also called equation of motion. According to Newton's 2nd law (the time rate of change of momentum of the fluid particles within this stream tube slice must equal to the forces acting on it).

F = mass* acceleration

Consider a small element of the flowing fluid as shown below, Let dA: cross-sectional area of the fluid element, dL : Length of the fluid element, dW: Weight of the fluid element, u: Velocity of the fluid element, P: Pressure of the fluid element. Assuming that the fluid is steady, non-viscous (the frictional losses are zero) and incompressible (the density of fluid is constant).The forces on the cylindrical fluid element are, Pressure force acting on the direction of flow (PdA). Pressure force acting on the opposite direction of flow [(P+dP)dA]. A component of gravity force acting on the opposite direction of flow (dW sin θ).

Hence,

Total force = gravity force + pressure force

The pressure force in the direction of low

$$Fp = PdA - (P + dP)\, dA$$

$$= - dPdA \tag{5}$$

The gravity force in the direction of flow

$$Fg = - \mathrm{d}W \sin\theta \; \{W = m\, g = \rho\, dA\, dL\, g\}.$$

$$= - \rho\, g\, dA\, dL \sin\theta \; \{\sin\theta = dz / dL\}.$$

$$= -\rho g dA dz \tag{6}$$

The net force in the direction of flow

$$F = m\, a \; \{m = \rho\, dA\, dL\}$$

$$= \rho\, dA\, dL\, a.$$

$$= \rho dA u du. \tag{7}$$

We have,

$$\rho\, dA\, u\, du = - dP\, dA - \rho\, g\, dA\, dz \;\; \{\div \rho\, dA\}$$

$$dP/ \rho + udu + dz\, g = 0$$

Euler's equation of motion, Bernoulli's equation could be obtained by integration the Euler's equation.

$$\int dP/ \rho + \int udu + \int dz\, g = \text{constant.}$$

$$P/ \rho + u2/2 + \mathrm{z}\, g = \text{constant.}$$

$$\Delta P/ \rho + \Delta \mathrm{u}2/2 + \Delta \mathrm{z}\, \mathrm{g} = 0 \quad \text{(Bernoulli's equation.)}$$

The pressure difference actually applied is about 30 bar where both the hot and cold streams are sent into the mixer about a speed of Mach number 2.Considering the cost for experimental setup we actually scale down the speed and the mixer inlet and outlet diameter so that the cost of analysis is made less. We use equation of continuity and Bernoulli's equation to determine the flow. We are scaling down the model to 1:10 ratio so the the hot fluid velocity is about 20m/s and a diameter about 60mm,cold fluid diameter is about 60mm and the velocity of 8m/s.

4. CFD ANALYSIS OF THE PROPOSED MODELS

The above models were analyzed using ANSYS FLUENT 14.0. Analysis of two different geometries had been carried out, performing two trials with varying boundary conditions in the second model. The geometries were subjected to the fluid flow under the different boundary conditions and the results were discussed.

(A) Model-1

In this chapter the sequence of events that is the modeling of the two designs which we proposed and the analysis of the two models is shown and also we would be discussing why we have chosen the second model over the first model. The first step is the modeling of the first proposed model as shown in Figure 2: In order to get familiarized with analysis software and see how hot and cold fluids mix we have designed a simple faucet. Thereby the meshing was done in ANSYS Workbench 14.0. After meshing the model, boundary conditions are mentioned. The velocity and temperature profiles where obtained.

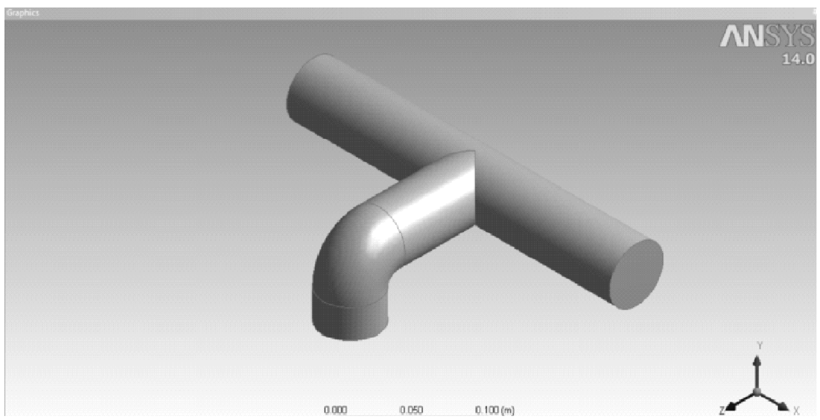

Figure 2: Model 1

(B) Model-2

The design for the second model as shown in figure 3 was inspired from a patent [3]. The design was carried out in the solid works software.

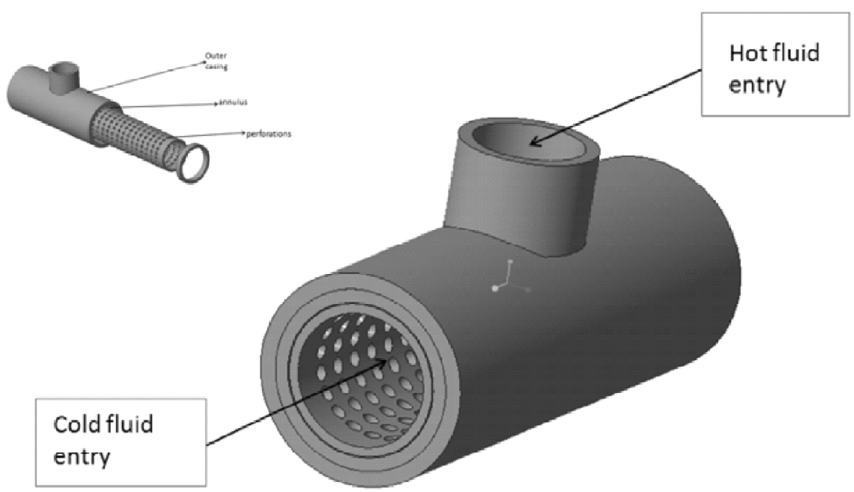

Figure 3: CAD model of the Mixer

By means of the present invention there is provided mixing apparatus constructed in the form of a simple T type fluid connector having a straight portion and inter seating stern that can be installed in the juncture of the tubes conducting the two fluids to be mixed. Within the apparatus a liner is disposed in concentric relation to the straight portion of the T and is spaced from the wall thereof so as to form a substantially closed, annular chamber into which fluid flowing from the second fluid inlet defined by the stem is conducted. Openings in the Wall of the liner effect mixing of the two fluids. The body of liquid retained in the annular chamber provides a thermal barrier effective to prevent the material forming the apparatus from being subjected to undue thermal stressing because of the substantial temperature differential imposed by the two fluids. Utilization of the apparatus, which is of substantially conventional form and therefore simple and inexpensive to construct, permits the mixing of high temperature, high pressure fluids at a fraction of the cost heretofore required.

5. RESULTS AND DISCUSSION

The architecture was modeled using Solid Works software and was converted into STEP format.Then the created model was meshed finely using ANSYS®MeshingTM.The number of elements was taken to be 2601657.Curvature for the body was turned ON. Finer meshes were generated in the mixing area by the INFLATION option in ANSYS.The inlets and outlets were named selection.Then the meshed file in .msh format was imported to ANSYS Fluent for analysis. In ANSYS, energy equation was turned ON and type of flow was taken as turbulent i.e. k-î model with 2 equations. The material was selected to be air.Then the boundary conditions were specified. The calculations were initialized. Number of iterations was taken to be 200 and the calculation was made to run

(A) Model I

The fluent analysis of the model 1 was carried out by using ansys workbench 14.0 software. The boundary conditions were specified as follows:

- Inlet 1 temperature: 700K
- Inlet 2 temperature: 200K
- Inlet 1 velocity : 10m/s
- Inlet 2 velocity : 10m/s

The results were obtained in the form of temperature and velocity profiles. These results were analyzed and reported.

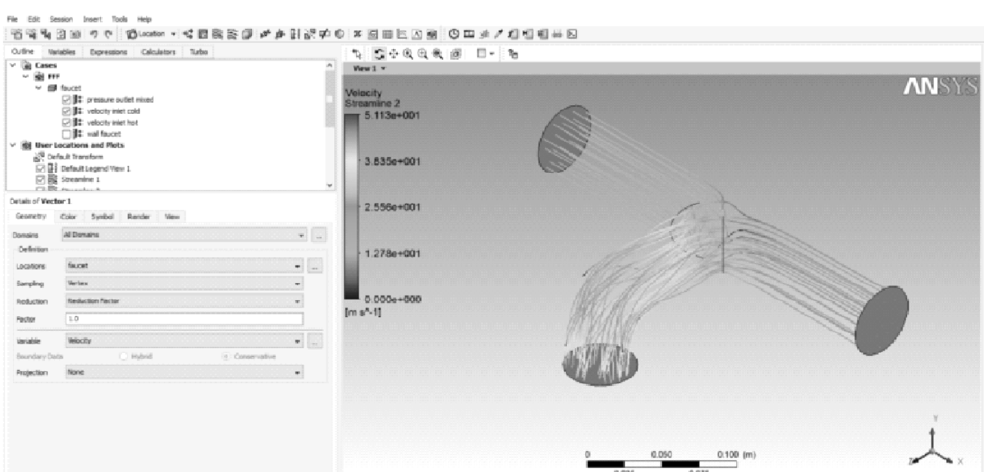

Figure 4: Velocity profile of streamline

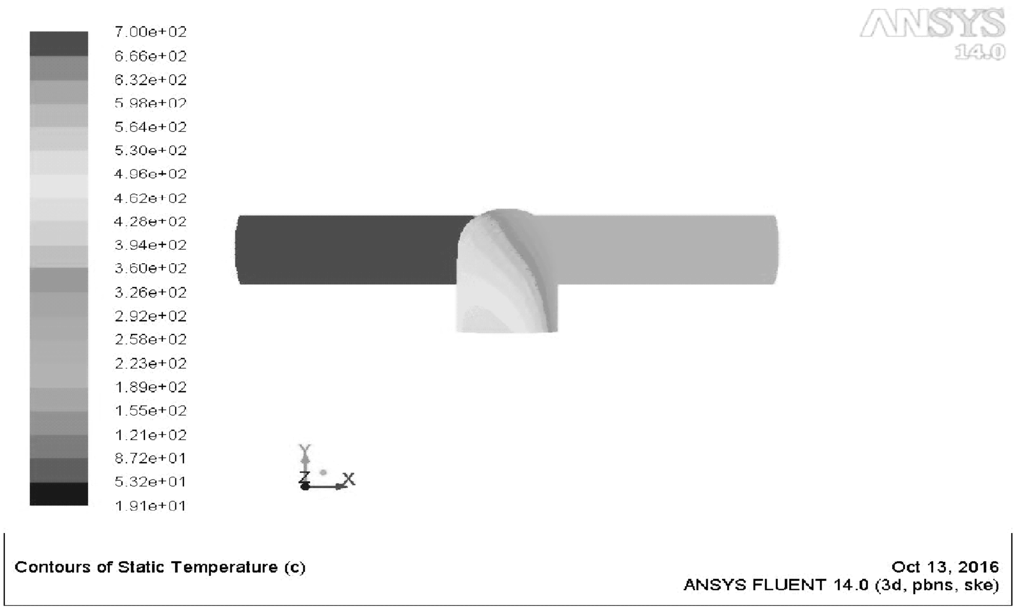

Figure 5: Temperature contour of the model

(B) Model II (Trial-1)

In the first trial of the second model low velocity of about 15m/s (hot fluid) was used to carry out the analysis. In the result obtained they were many cold spots. Therefore to understand the impact of velocity of the fluid in mixing, the velocity of the hot fluid was increased to 20 m/s (hot fluid) and the results obtained were convincing with a uniform distribution of temperature at a constant pressure at the outlet.

- Inlet 1 temperature: 373K
- Inlet 2 temperature: 300K
- Inlet 1 velocity :10m/s
- Inlet 2 velocity : 5m/s

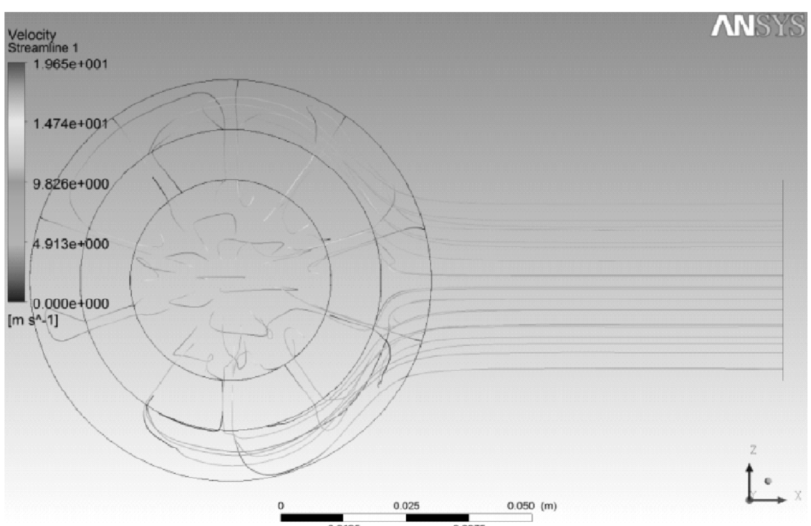

Figure 6: Velocity Profile of Streamline

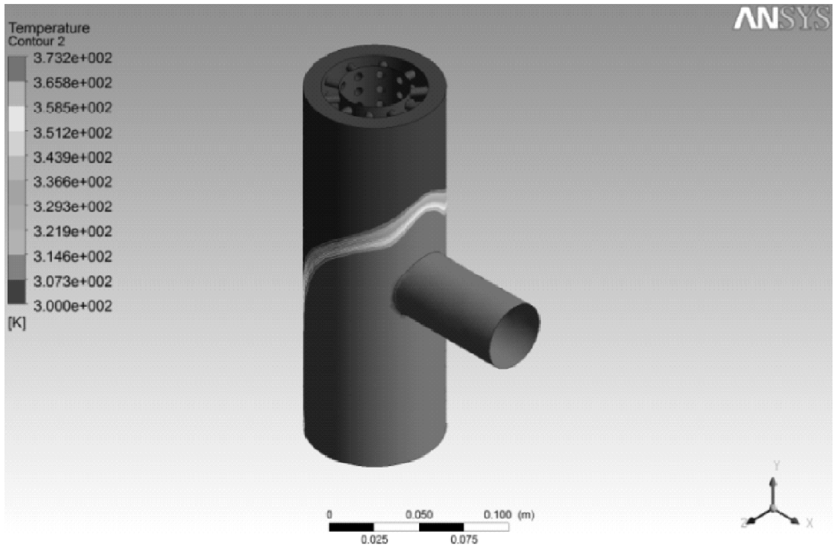

Figure 7: Temperature Contour 1

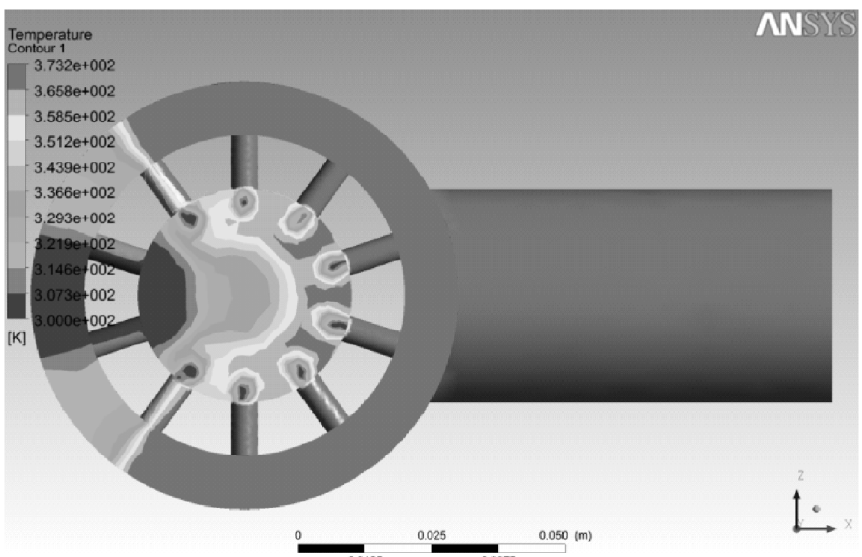

Figure 8: Temperature Contour (Trail-1)

The stream lines as shown in figure 6 and velocity and the temperature profile as shown in Figures 7 & 8 gives a clear image of the mixing of the two fluids. It has been ascertained that the perforations in the baffle plate play an important role in the efficient mixing of the two fluids.

(C) TRIAL-2

In the trial-2 the velocities of the hot and cold fluid have been changed to meet the requirements of the dimensions and for effective mixing of both the fluids. This has been done with help of the numerical analysis. The velocity of the hot fluid had been increased to increase the turbulence thereby increase the effectiveness of the mixing.

- Inlet 1 temperature: 373K
- Inlet 2 temperature: 300K
- Inlet 1 velocity : 15m/s
- Inlet 2 velocity : 5m/s

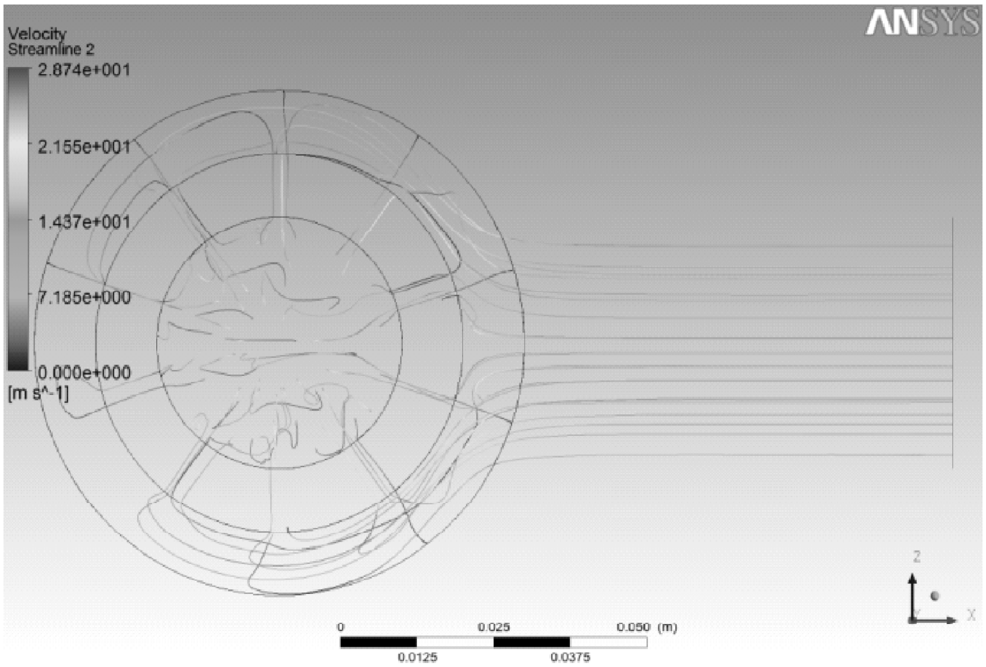

Figure 9: Velocity Profile of Streamline-2

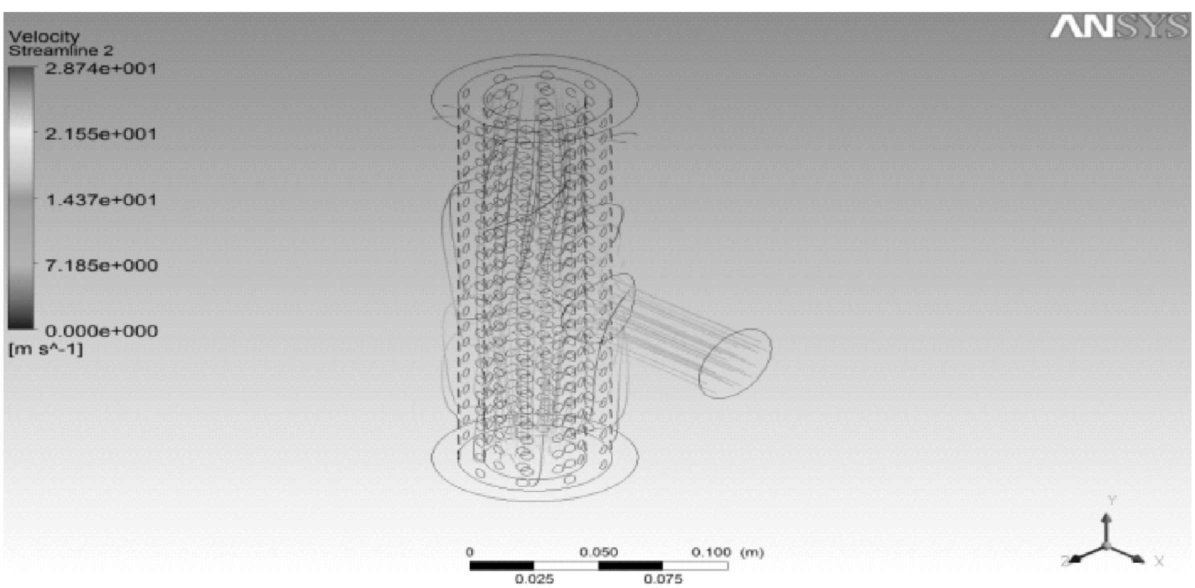

Figure 10: Side View of Velocity Profile-2

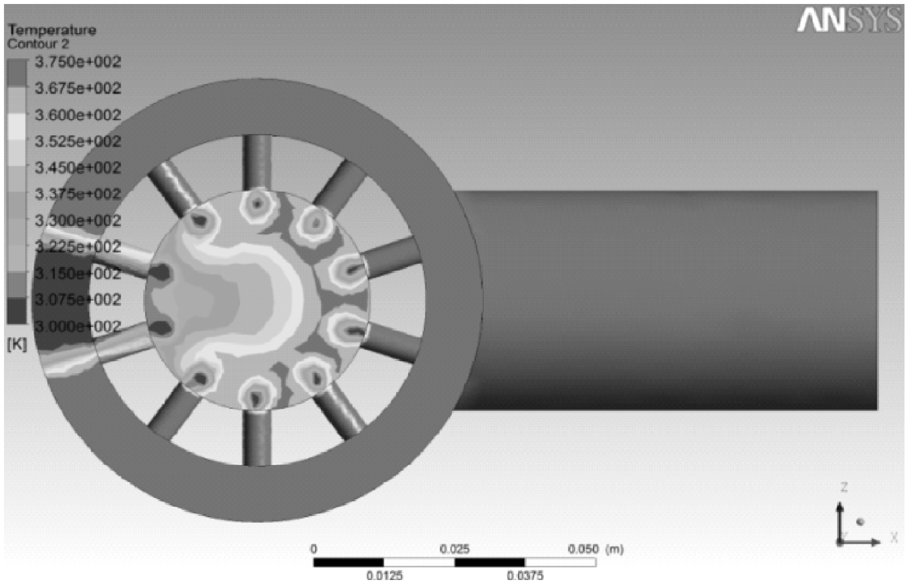

Figure 11: Front view of temperature contour (Trail-2)

Thus the after carrying out the analysis on the model-2 trial-2 we have observed that there is appropriate mixing of hot and cold fluids as shown in the contours.

The following ANALYSIS gave us a temperature profile which is uniform at the center and the inlet temperature of the hot fluid is 373K and the temperature of the cold fluid is 300K .The temperature of the mixer obtained at the end of the outlet was around 337-330K at the center and the temperature profile was increasing when the fluid mixture is away from the center of the pipe. The fluid was cold at the opening of the pores due to sudden expansion. The introduction of the baffle plate had a tremendous impact by mixing the two fluids effectively.

Before trial-2 the velocity of the hot was fluid was 10m/s and the cold fluid velocity was 5m/s for which there was a cold spot present in the mixture as the velocity difference is less and the cold fluid being denser was not able to mix effectively at the end of the outlet. In trial 2 by increasing the velocity difference of the two streams i.e. making the hot fluid velocity to 15 m/s and cold fluid temperature to 5m/s we could see that the mixing was effective than the previous trail by having uniform temperature at the core and the cold spots we reduced and were only present at the opening of the pores so the velocity difference had a impact on mixing of two fluids by having a greater turbulence so that the two fluids were effectively mixing as anticipated earlier.

6. CONCLUSION

The analysis of the designed model was carried out using ANSYS FLUENT 14.0, from this study, to understand about the various parameter influencing the outlet temperature and pressure of the resultant mixture and to know how fast the mixing action is occurring,. Numerous iterations had been carried out varying the inlet velocity of the two fluids to get convergence for the initially provided boundary conditions. During the analysis reverse flow had been witnessed in several faces of the geometry. Despite reverse flow, it evident from the various contours that outlet mixture obtained had uniform temperature and pressure profiles in most places and the desired result (temperature equaling to the average temperature taking both hot and cold fluid into account) has been achieved.

7. ACKNOWLEDGEMENTS

We express our deep sense of gratitude to Dr. R. Rudramoorthy, Principal and P.R. Thyla, Head and Professor, Department of Mechanical Engineering, PSG College of Technology for providing us the opportunity to take up this research work.

REFERNCES

[1] Gianni Orsi, Mina Roudgar, Elisabetta Brunazzi, Chiara Galletti, Roberto Mauri n. Water–ethanol mixing in T-shaped microdevices.

[2] Bin Xu, Teck Neng Wong, Nam-Trung Nguyen, Zhizhao Che.Thermal mixing of two miscible fluids in a T-shaped micro channel.

[3] C.W Lawton, Patent No. 3,409,274 Mixing of High Pressure Fluids.

[4] M.H.C. Hannink, A.K. Kuczaj, F.J. Blom, J.M. Church and E.M.J. Komen, A coupled CFD-FEM strategy to predict thermal fatigue in mixing tees of nuclear reactors.

[5] Th. Franka,", C. Lifantea, H.-M. Prasserb, F. Mentera Simulation of turbulent and thermal mixing in T-junctions using URANS and scale-resolving turbulence models in ANSYS CFX.

[6] Wei-yu ZHU, Tao LU 1, Pei-xue JIANG , Zhi-jun GUO, Kui-sheng WANG, Large eddy simulation of hot and cold ûuids mixing in a T-junction for predicting thermal fluctuations.

[7] P. Jandik , B.H. Weig , N. Kessler , J. Cheng , C.J. Morris , T. Schulte, Initial study of using a laminar ûuid diffusion interface for sample preparation in high-performance liquid chromatography.

[8] Lin-Wen Hu *, Mujid S. Kazimi, LES benchmark study of high cycle temperature fluctuations caused by thermal striping in a mixing tee.

[9] B. Wegner *, Y. Huai, A. Sadiki, Comparative study of turbulent mixing in jet in cross-flow configurations using LES.

[10] F. A. L. Dullien, New network permeability model of porous media. AIChEJ. 21, 299-307(1975).