Effect of Change in Momentum Ratio and Length of Pipe on the Mixing of Fluids at T-Junctions Using CFD

Navoday Borkar, Tanmay Chitnis

Department of Mechanical Engineering, Sinhgad College of Engineering (SCoE), Pune.

DOI: 10.29322/IJSRP.10.05.2020.p10106
http://dx.doi.org/10.29322/IJSRP.10.05.2020.p10106

Abstract- T-junctions are widely used to facilitate the mixture of two fluids at different temperatures and in most cases, different velocity also. Temperature of the mixture at the outlet of the main pipe undoubtedly plays a significant role in all the industries where T-junctions are implemented as their main intention is to get reduced temperature of the fluid. This paper deals with the three main cases which are expected to rather have a significant impact on the outlet temperature viz. short tee, increased velocity ratio and long tee respectively. Effects of mentioned cases have been investigated using CFD through ANSYS FLUENT to monitor temperature and velocity of the mixture. It was concluded that increasing the momentum ratio would bring down the temperature more effectively as compared to increase in the length of the pipe though the later resulted into better mixing of the fluids.

Index Terms- ANSYS, CFD, mixing efficiency, mixing of fluids, momentum ratio, T-junctions, etc.

I. INTRODUCTION

A tee is formed by two pipe sections joined at a right angle to each other. The hot fluid (water) is flowing through the main pipe while the cold fluid (water) enters the main pipe through the branched pipe. There have been traditionally developed empirical design equations to solve the mixing problems but it does not take into account the complexity of turbulent flow in applications like petrochemical products, waste water treatment, blending of oils, etc.

With due respect to all the other industries, nuclear industry is the one where efficiency of the mixing of fluids in the mixing chambers is very critical. There have been much of research regarding the thermal stresses arising in the T-junctions due to the large temperature difference and also due to frequent changes in the same [1].

A large temperature gradient is created along the length of the pipe due to mixing of fluids near the junction as well as to the downstream of the main pipe [2].

The temperature fluctuations cause cyclic thermal stresses and subsequent fatigue cracking of the pipe wall. Thus, anticipating the failure of the piping systems in nuclear power plants is very essential as its failure may lead to irreparable damage. In order to assess the structural strength, stability and life of such T-junctions, it is essential to know the following: (i) magnitude of the temperature fluctuations, (ii) regions of pipe wall that experience the temperature fluctuations, (iii) response of the material of the pipe to the thermal fatigue loading. [3]

Noguchi et al. (2003) carried out experimental studies of thermal striping with scale models of T-junctions using hot and cold water. They investigated the flow pattern and characteristics of the temperature fluctuations with the help of flow visualization and temperature fluctuation measurements. They concluded that velocity ratio determines the flow pattern and temperature fluctuation behaviour. Also, upstream elements like elbow and valve had little effect on temperature fluctuations.

Large Eddy Simulation (LES) and Direct Numerical are expected to give more accurate results since the research involves turbulent mixing of two fluids. Though Kuczaj et al. had studied the same using LES sub grid scale developed by the Vreman (2004). It was concluded that finer mesh was required to capture the temperature fluctuations near the centre line of the pipe of the mixing zone. [4], [5].

Zahid Khokhar et al. [7] have presented their research regarding mixing of fluids with side tees. They have performed a grid independence study for the centre line temperature and also validated the same using experimentation. Turbulence was modelled using the standard k-ε model. The pipe length required for achieving 95% mixing was found to be a function of the ratio of the velocities of the side and main streams.

II. METHODOLOGY

Mixing tee possess two inlets and one outlet. The momentum ratio for this type of tee is the ratio of velocity of cold fluid to velocity at mixed fluid at outlet.

Mixing of the fluids inside the pipe was simulated using ANSYS Fluent, a general purpose three dimensional CFD package. The turbulence model selected to setup the required case was k-Ɛ. Though Large Eddy Simulations (LES) and Direct Numerical Simulation (DNS) will give most accurate results in this case as it involves turbulent mixing, but many experiments have agreed to the fact that k-Ɛ model reasonably agrees with the experimental results.
As mentioned above, governing equations i.e. mass and momentum equations are solved using numerical methods as DNS would consume a lot more computational power and time.

A. Geometry

Two geometries of mixing tees were prepared for the analysis in CAD software SolidWorks. One with an outlet length of 1000mm and other with an outlet length of 1500 mm from T junction which can be observed from Fig. 1 and Fig. 2 with same inner and outer diameter of 242.93mm and 273.05 mm respectively.

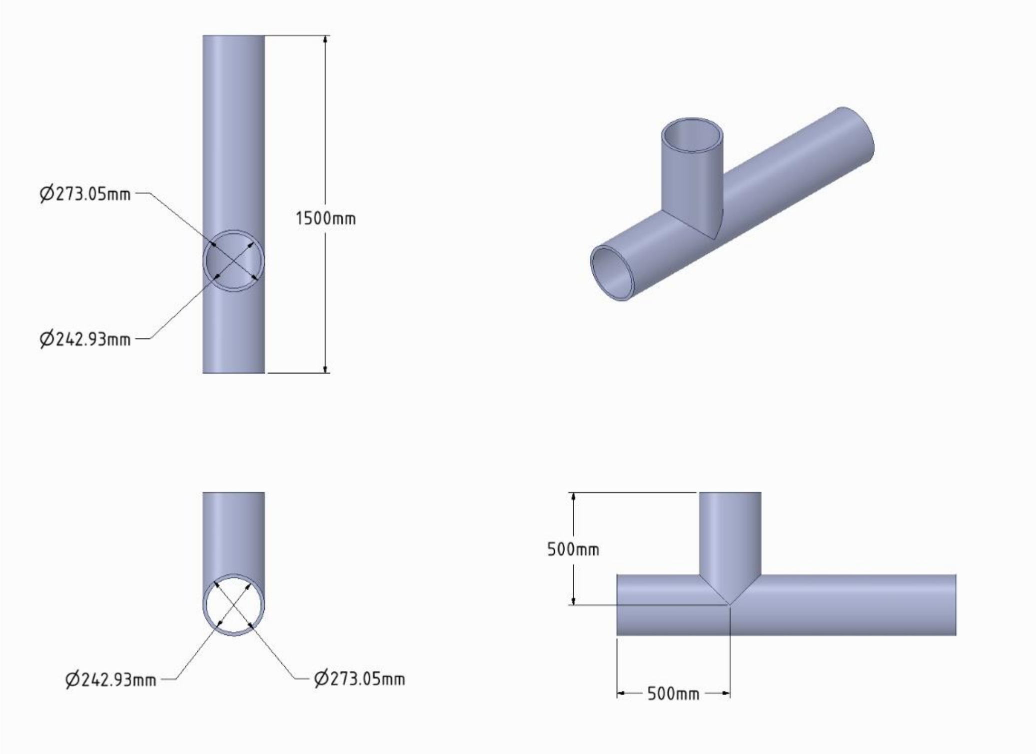

Fig. 1 Dimensions of Small Mixing Tee

http://dx.doi.org/10.29322/IJSRP.10.05.2020.p10106

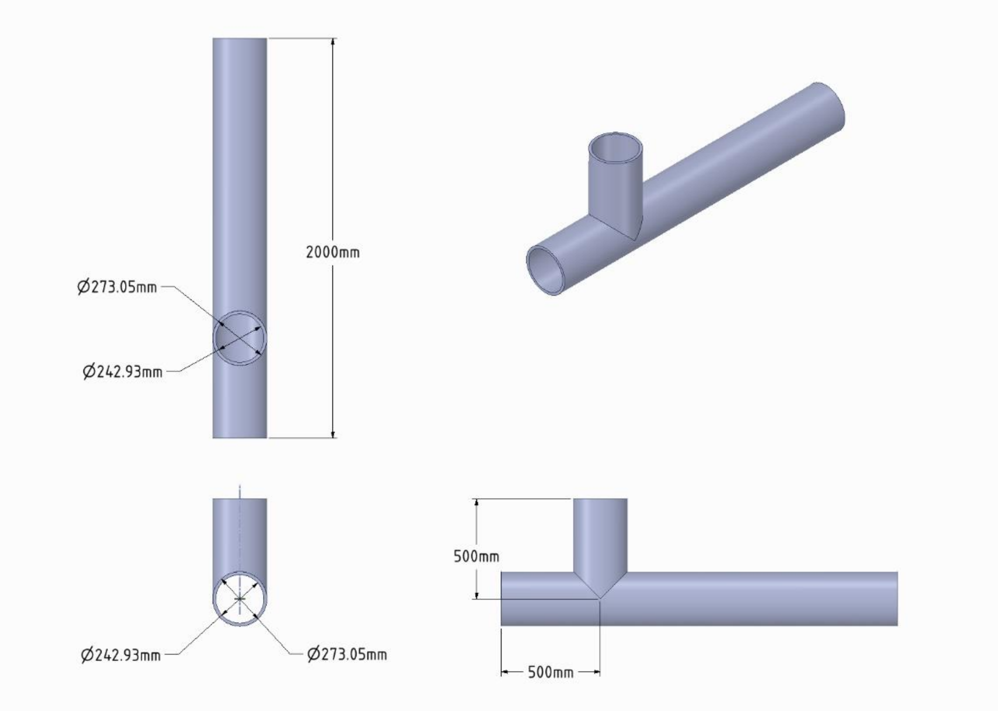

Fig. 2 Dimensions of Long Mixing Tee

Above prepared geometries are not numerically acceptable as the geometry of tee represents solid domain. Fluid domain was required for simulation which was obtained by volume extract feature.

B. *Meshing*

Once the numerically acceptable fluid domain was created, the geometry of tee is imported into ANSYS. Meshing of both tees is done by using Patch conforming method which can be observed in Fig. 3. Tetrahedral elements with size of 10mm are used. For Short tee, number of elements and nodes are 70111 and 121229 respectively. Similarly, for long tee, number of elements and nodes are 82254 and 422810 respectively.

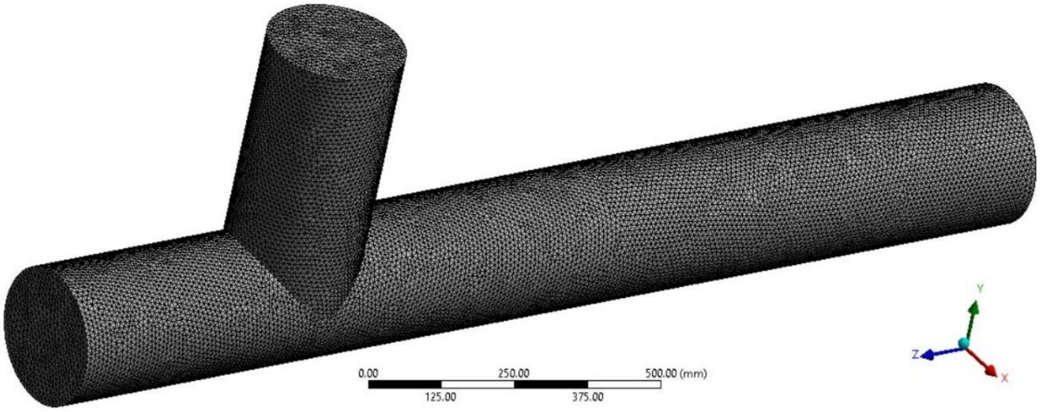

Fig. 3 Meshing of Tee

C. *Boundary Conditions*

Boundary condition for both the inlets is set to “Velocity-Inlet” and outlet is set as “Pressure-Outlet” as shown in Fig. 4. Water is selected as working fluid for fluid domain with density of 998.2 kg/m^3, specific heat of 4182 J/kg-K, thermal conductivity of 0.6 W/m-K and viscosity of 0.001003 kg/m-s. In Fluent, segregated three-dimensional, steady state solver is used for turbulent flow. Temperature at Inlet Z is 450 K and at Inlet Y is 333 K for all cases. Also, the gauge pressure at outlet is 0 Pa for all cases. Momentum ratio and the velocities with following three input cases are tabulated in Table 1.

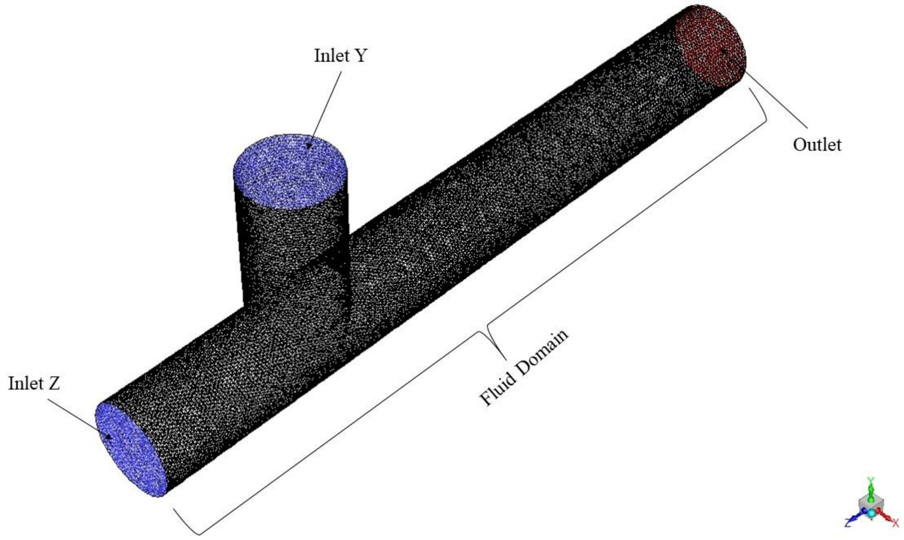

Fig. 4 Boundary Conditions

Table I: Input Cases

Parameters	**Input Cases for Boundary**		
	Case1	**Case 2**	**Case 3**
Geometry	Short Mixing Tee	Short Mixing Tee	Long Mixing Tee
Velocity at Inlet Z	1.7 m/s	1.7 m/s	1.7 m/s
Velocity at Inlet Y	3.4 m/s	6.8 m/s	3.4 m/s
Momentum Ratio	2	4	2

D. Simulation Results

For each case of simulation 300 iterations were performed as it proved sufficient enough to satisfy the convergence criteria. There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore, area weighted average of temperature and standard deviation of temperature is also monitored at outlet. For most problems, the default criterion in ANSYS Fluent is sufficient. This criterion requires scaled residuals decrease to 10^{-3} or less for all equations except energy equation, for which the criterion is 10^{-6}. But it can be observed from the following graphs of residuals that the energy equation has not reached the convergence criterion, this is because the temperature difference between input values and output is very less. From Fig. 7 it can be observed that the residuals haven't reached the convergence criterion but still the solution is converged as for more than 50 iterations the graph shows same behaviour at the end of graph. Also, the graph of area-weighted average of temperature and standard deviation of temperature at outlet show same behaviour for more than 50 iterations. Hence, we can confirm the convergence of solution by observation. Graph of residuals can be observed from Fig. 5, Fig. 6, Fig. 7 for Case 1, Case 2, Case 3 respectively.

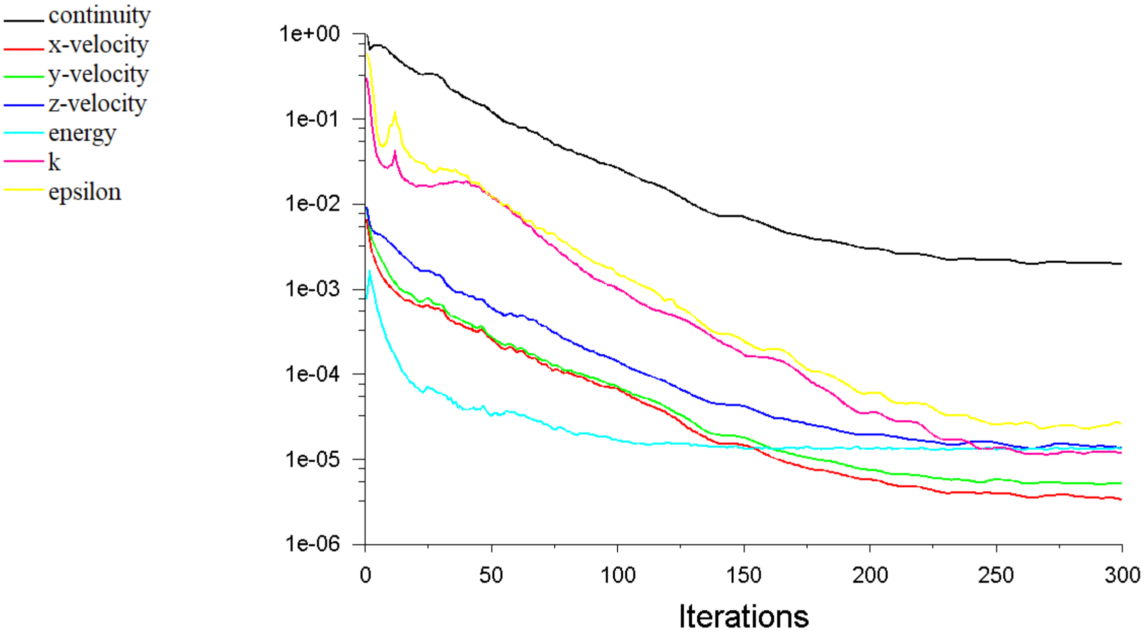

Fig. 5 Graph of Residuals for Case 1

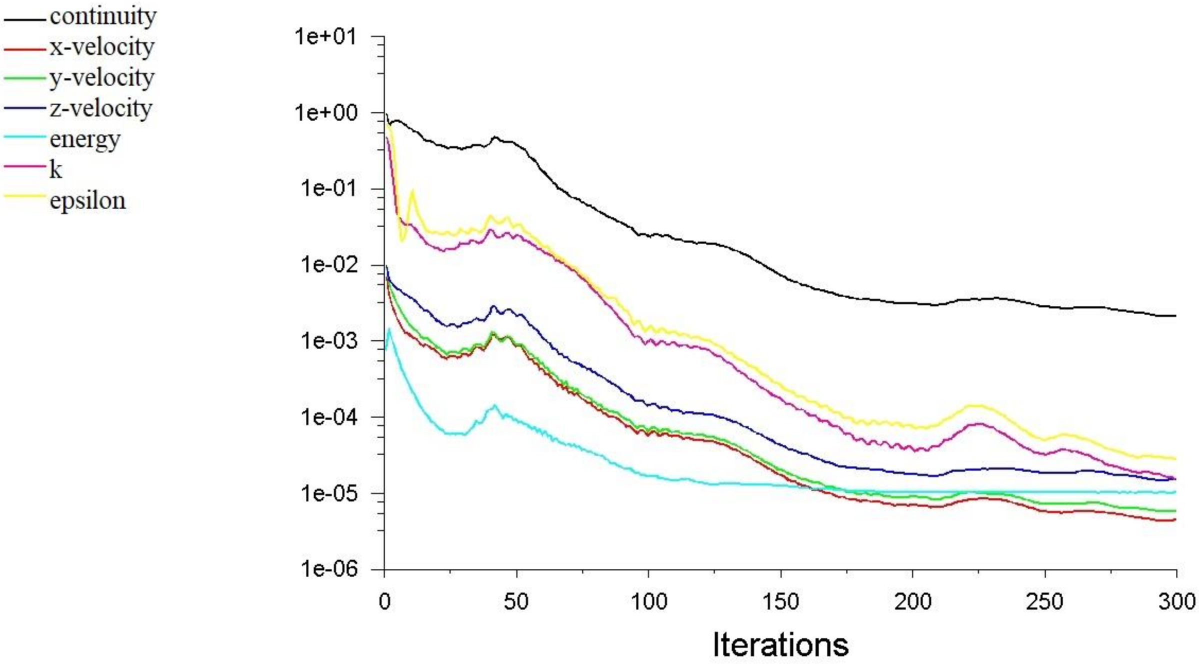

Fig. 6 Graph of Residuals for Case 2

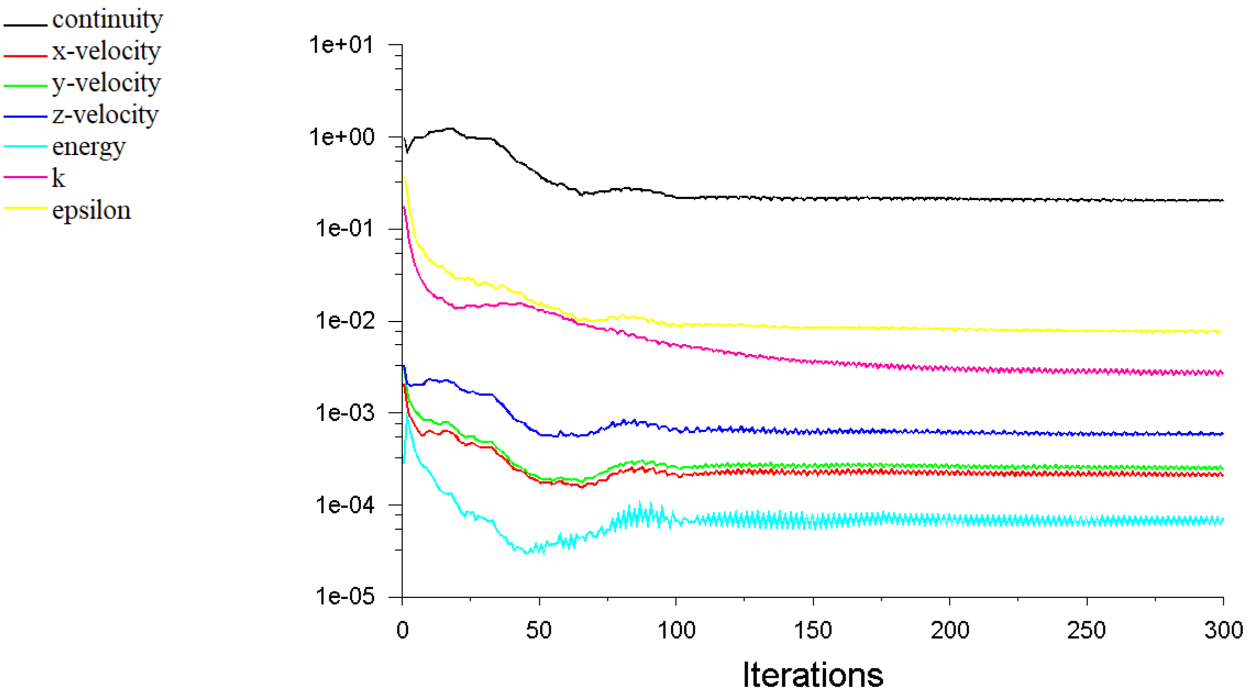

Figure 7 Graph of Residuals for Case 3

Area-weighted average of temperature shows the average value of temperature at outlet. Average temperature for Case 1, Case 2 and Case 3 is 367.45 K, 352.86 K and 366.48 K respectively. Graph of average temperature can be observed from Fig. 8, Fig. 9, Fig. 10 for Case 1, Case 2, Case 3 respectively.

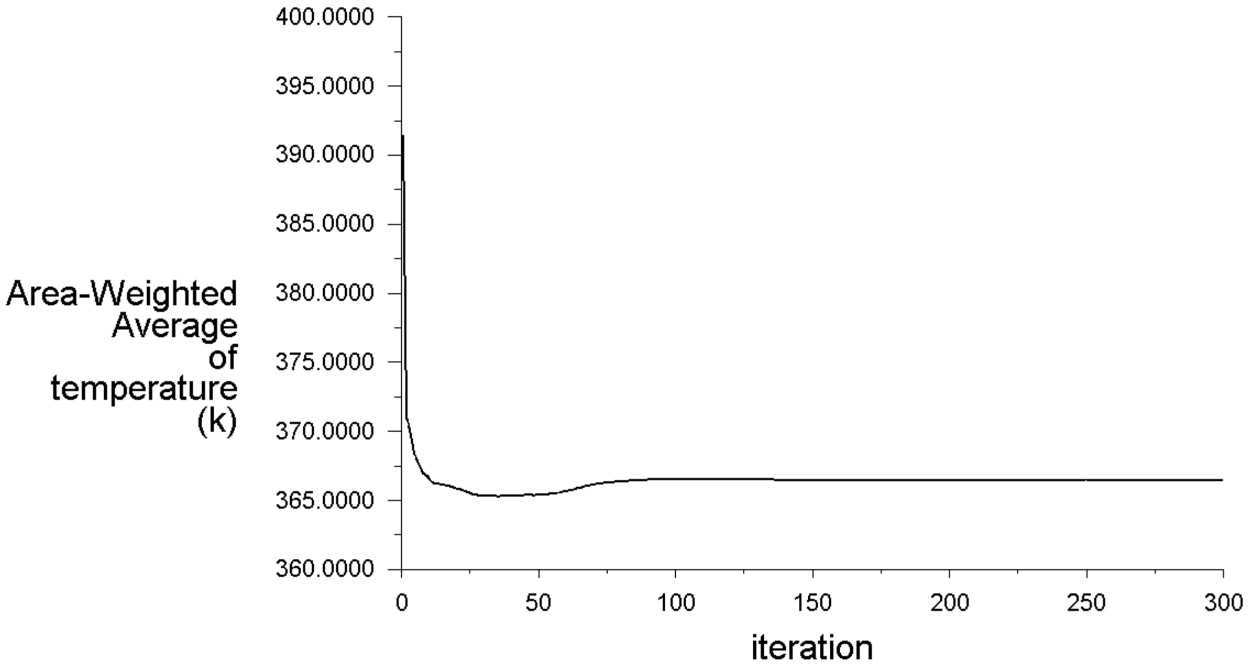

Fig. 8 Average temperature at outlet for case 1

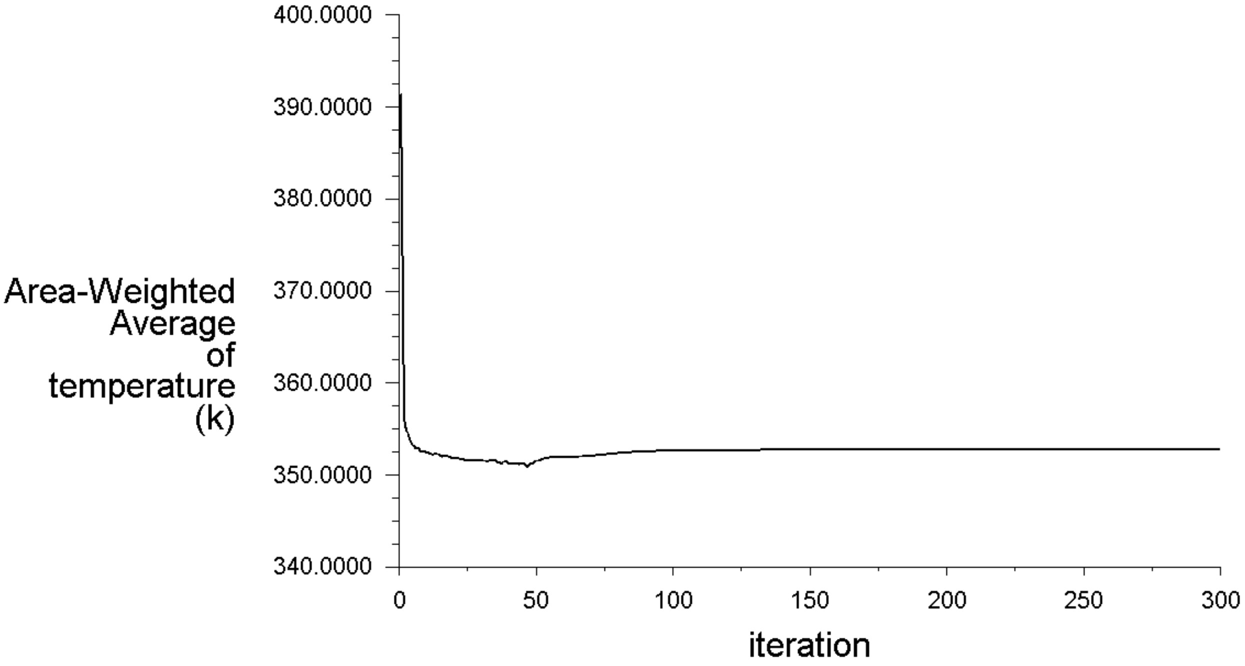

Fig. 9 Average temperature at outlet for case 2

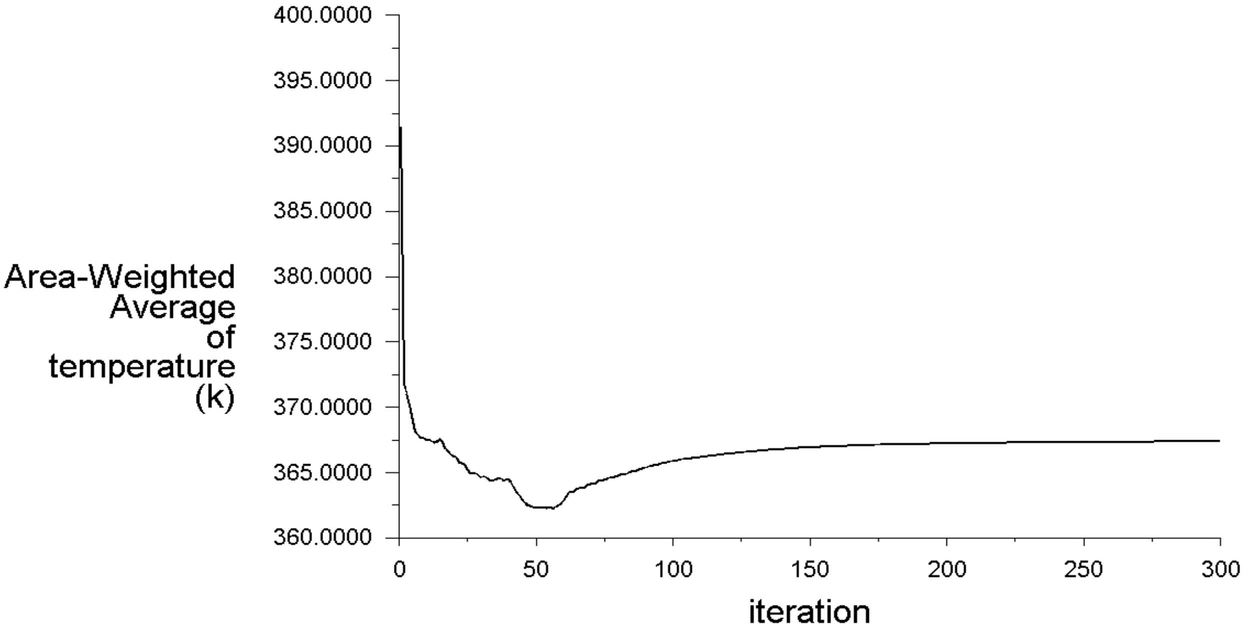

Fig. 10 Average temperature at outlet for case 3

Standard deviation shows the deviation of temperature at outlet. Standard deviation of temperature for Case 1, Case 2 and Case 3 is 35.15 K, 22.96 K and 34.86 K respectively. Graph of standard deviation can be observed from Fig. 11, Fig. 12, Fig. 13 for Case 1, Case 2, Case 3 respectively.

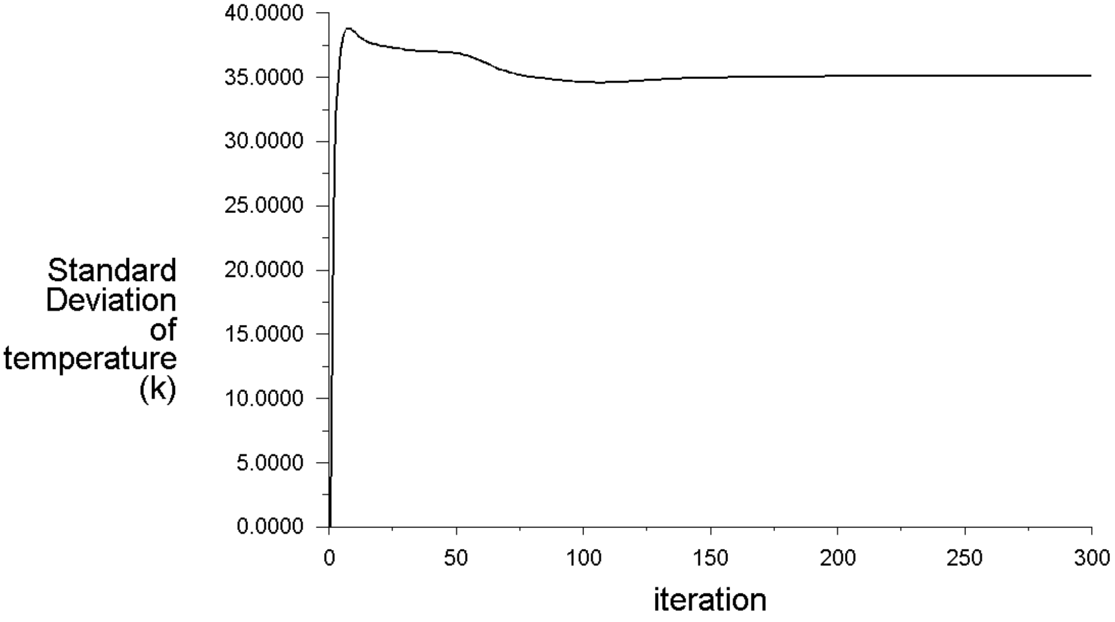

Fig. 11 Graph of standard deviation of temperature for case 1

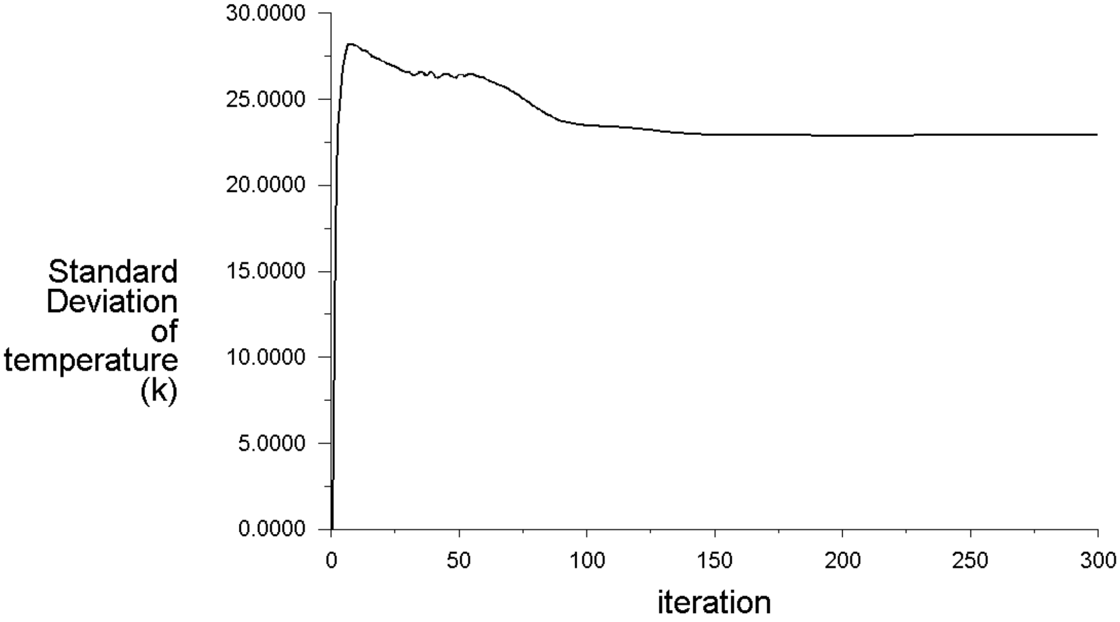

Fig. 12 Graph of standard deviation of temperature for case 2

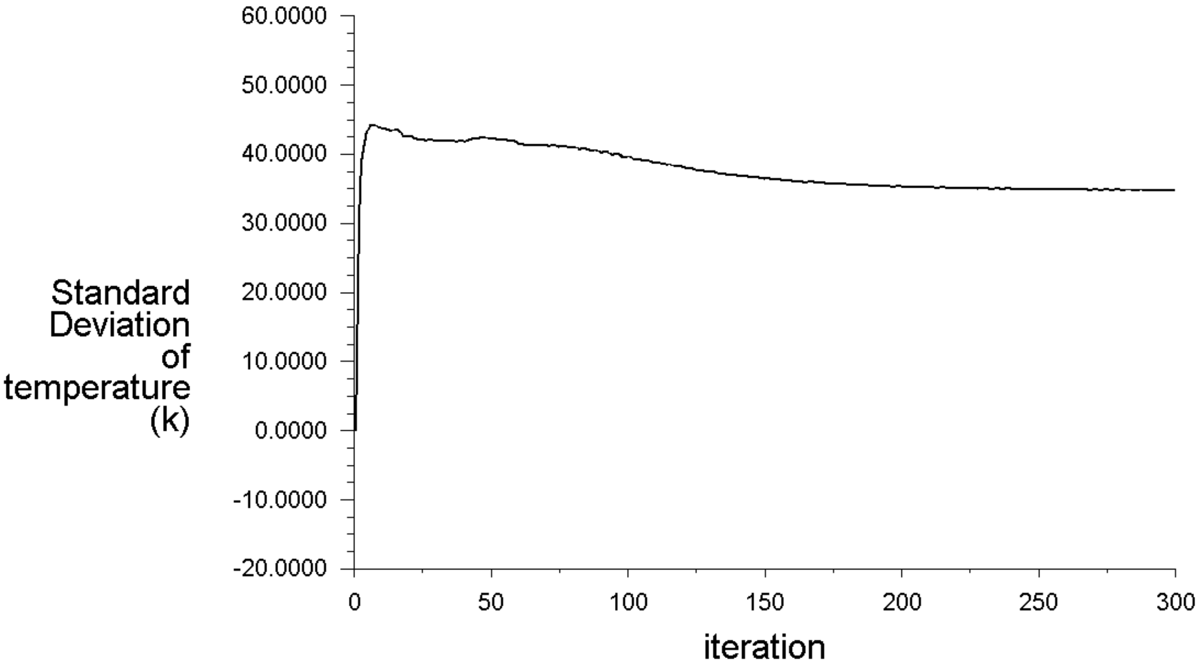

Fig. 13 Graph of standard deviation of temperature for case 3

Post processing is necessary for creating and investigating the behaviour of output results throughout the body. After the simulation, the results are examined and analyzed through CFD-Post to verify the results and conclusions drawn on the obtained results. Contour plots for temperature and velocity are observed for all the cases which provide a visual of actual process. It can be observed in the temperature plot the contours of high temperature represented by colour red are present in both case 1 and case 3 represented by Fig. 14 and Fig. 16 respectively. This indicates that there is high temperature at outlet, but it is not for case 2. It can be observed from Fig. 15 that the temperature at outlet is relatively less as the contours of high temperature diminished before reaching the outlet for case 2. Also, it can be observed from the contour plot of velocity that velocity at outlet in case 2 is relatively greater than that of other two cases. Hence, the simulation results can be confirmed on the basis of visuals obtained by post processing.

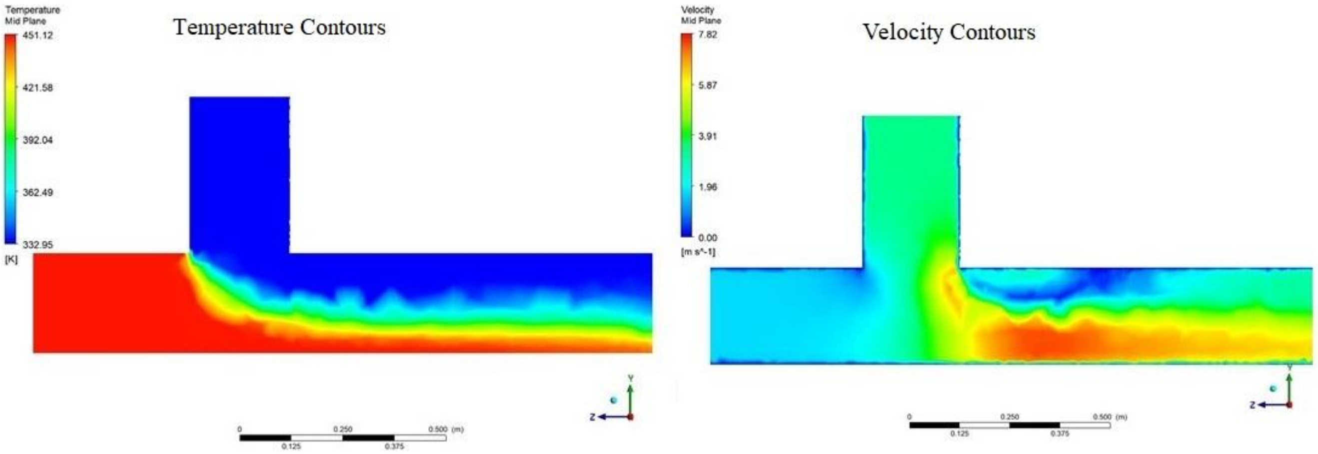

Fig. 14 Contours for Case 1

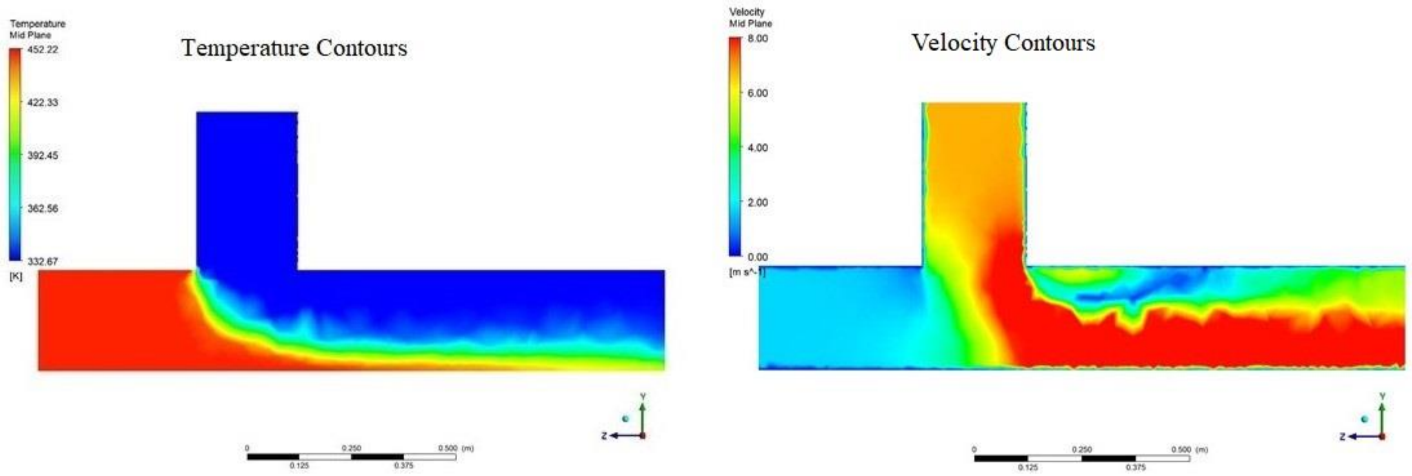

Fig. 15 Contours for Case 2

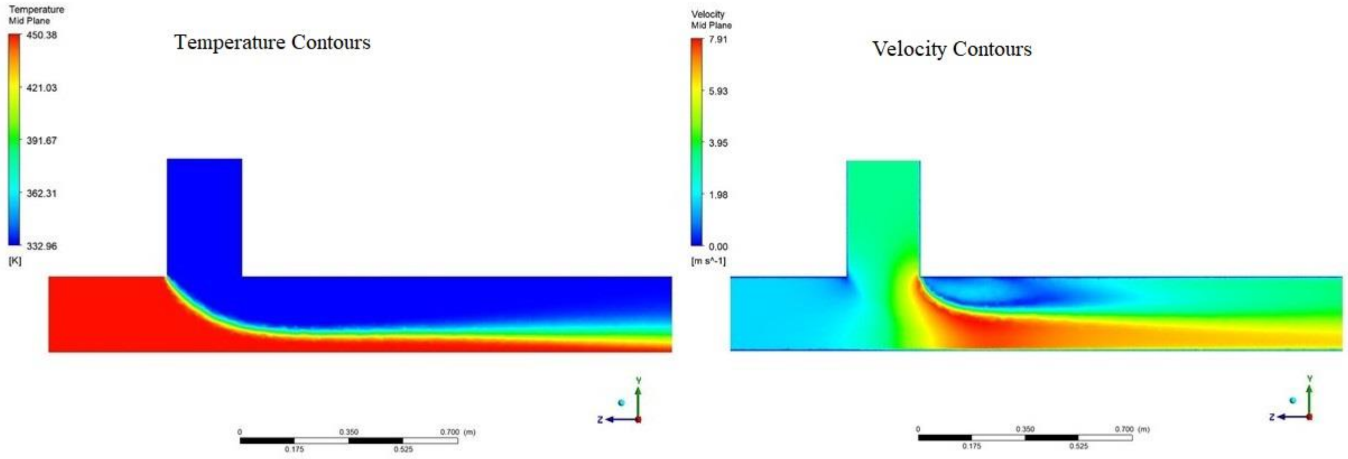

Figure 16 Contours for Case 3

III. CONCLUSION

From the result of area weighted-average of temperature at outlet it can observed that the temperature at outlet shows no significant change when the velocity is same even though the length of outlet is increase from the junction. But it has been observed that the outlet temperature shows significant decrease when momentum ratio is increased resulting in increased mixing efficiency.

The above-mentioned behavior is also same for the results of standard deviation of temperature at outlet. Hence, on the basis of obtained results and observations made, case 2 yields more mixing efficiency relative to other cases. Hence mixing tee efficiency momentum ratio. Also, the increase in length of tee does not play any significant role as compared to increase in momentum ratio on the efficiency of Mixing tee.

REFERENCES

[1] Andrew R. Gow, and Salim M. Salim, “Analysis of Thermal Mixing in T-Junctions Using Fluid-Structure Interaction,” Proceedings of the International MultiConference of Engineers and Computer Scientists 2018 Vol II IMECS 2018, March 14-16, 2018, Hong Kong.
[2] M. Lakshmiraju and J. Cui, “Numerical Modelling of Transient Thermal Mixing,” International Journal of Latest Research in Science and Technology, vol. 3, pp. 80-89, 2014.
[3] Naik-Nimbalkar, V.S., et al., Thermal mixing in T-junctions. Chemical Engineering Science (2010), doi: 10.1016/j.ces.2010.08.017
[4] Hannink, M. & Kuczaj, Arkadiusz & Blom, F. & Church, J. & Komen, E.M.J.. (2008). A coupled CFD-FEM strategy to predict thermal fatigue in mixing tees of nuclear reactors. Eurosafe Forum.
[5] A. W. Vreman,” An eddy-viscosity subgrid-scale model for turbulent shear flow: Algebraic theory and applications,” Vreman Research, Godfried Bomansstraat 46, 7552 NT Hengelo, The Netherlands

[6] Rzezonka, B., Kastl, H., 1984. Stress and fatigue analysis of SNR-300 mixing devices, including optimizations and thermal shock tests on internal structures. Nuclear Engineering and Design 78, 69–78.
[7] Khokhar, Zahid Hafeez & Zughbi, Habib & Sharma, Rajendra. (2002). MIXING IN PIPELINES WITH SIDE-TEES.
[8] Anderson CFD book: the basic with application, McGraw-Hill series in aeronautical and aerospace.

AUTHORS

First Author – Navoday Borkar, B.E Mechanical (Final Year-Present), Sinhgad College of Engineering,Pune, ndvborkar@gmail.com .
Second Author – Tanmay Chitnis, B.E Mechanical (Final Year-Present), Sinhgad College of Engineering,Pune, tanuchitnis9@gmail.com.

Correspondence Author – Navoday Borkar, ndvborkar@gmail.com., 8552064775.

http://dx.doi.org/10.29322/IJSRP.10.05.2020.p10106