CFD Analysis of Hot And Cold Fluid Mixing In T-Pipe By Placing Nozzle At Different Places

Mr. Narla Sandeep[1], Mr. V.Satyanarayana[2], Mr. A.V.Sridhar[3]
[1]Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[3]Associate professor, Dept of Mechanical Engineering
[1, 2, 3]Kakinada institute of technology and science, Divili

Abstract- *the computational fluid dynamics analysis of mixed flow of hot and cold water in T- pipes and it is mainly concentrated on analytical approach to the areas where Pipes (used for flow) are mostly susceptible to damage. The simulation is done on T-pipe by placing the nozzle at three different places, to know the pressure, temperature and velocity contours throughout the flow and comparison was made. These T- pipes are mostly used in nuclear reactor cooling system to reduce the heat in the nuclear reactors by mixing hot and cold water, where the mixing will takes place efficiently. The 2D model of the pipe is made by GAMBIT and analysis is to be carried out by using K-Epsilon in FLUENT software.*

Keywords- Mixing hot and cold fluid, Different types of T-pipes, Nozzles, Using K-Epsilon in FLUENT software.

I. INTRODUCTION

Pipe network are mainly used for transportation of fluids from one place to another in different components. All these components cause loss in pressure due to change in momentum of the flow. This means conversion of flow energy in to heat due to friction or energy lost due to turbulence. Flow analysis is very important in nuclear power plant cooling systems to know how the variations are taking place with respect to the cross section. These T- pipes are used in nuclear power plant cooling systems to reduce the heat in the nuclear reactors by mixing hot water with cold water in the T- pipes. Mixing fluid of different temperature in T-junction geometries became of significant importance in the field of nuclear reactor safety.

CFD

Computational fluid dynamics, usually abbreviated as **CFD**, is a branch of fluid mechanics that uses numerical methods and Algorithms to solve and analyze problems that involve fluid flows, Computers are used to perform the Calculations required simulating the interaction of liquids and gases with surfaces defined by boundary conditions with high-speed supercomputers better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

STUDIES AND FINDINGS

The current study used FLUENT software, to solve the balance equation using control volume approach. In the GAMBIT software, a fine meshing is done by using successive ratio and later given the boundary conditions for the geometry and for the media. The geometry was done in the GAMBIT with measurements; pipe diameter is 50mm, radius of the pipe 25mm and length of the pipe 500mm. Defining required boundaries like inlet, outlet and wall of the geometry and mesh under tetrahedron. Defining the boundary conditions for the water. The velocity at inlet of the cold fluid is 4m/sec and at outlet is 2m/sec and the gravitational acceleration of $9.81 m/s^2$ in downward flow direction was used.

T-PIPE MODELS

Fig: 1.1 first model of T- pipe

Fig: 1.2 second model of T- pipe

Fig: 1.3 Third model of T- pipe

Fig: 1.4 Fourth model of T- pipe

INPUT PARAMETERS

1. Pipe dia. =50mm
2. Length of the pipe =500mm
3. Velocity of cold fluid---- inlet =4m/sec
   Outlet =2m/sec
4. Acceleration =9.81m/s^2
5. Cold water temp. =299^{0}c
6. Hot water temp. =395^{0}c
7. Material used =Al

THEOREM

Unlike other turbulence models, k-ε model focuses on the mechanisms that affect the turbulent kinetic energy. The mixing length model lacks this kind of generality. The underlying assumption of this model is that the turbulent viscosity is isotropic, in other words, the ratio between Reynolds stress and mean rate of deformations is same in all directions.In other words

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho\epsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

Rate of change of k or ε + Transport of k or ε by convection = Transport of k or ε by diffusion + Rate of production of k or ε - Rate of destruction of k or ε.

Problem Solving Steps

Once you have determined the important features of the problem you want to solve, you will follow the basic procedural steps shown below.

- Create the model geometry and grid.
- Start the appropriate solver for 2D or 3D modeling.
- Import the grid.
- Check the grid.
- Select the solver formulation.
- Choose the basic equations to be solved: laminar or turbulent (or inviscid), chemical species or reaction, heat transfer models, etc. Identify additional models needed: fans, heat exchangers, porous media, etc.
- Specify material properties.
- Specify the boundary conditions.
- Adjust the solution control parameters.
- Initialize the flow field.
- Calculate a solution.
- Examine the results.

II. RESULTS OBTAINED

FIRST MODEL

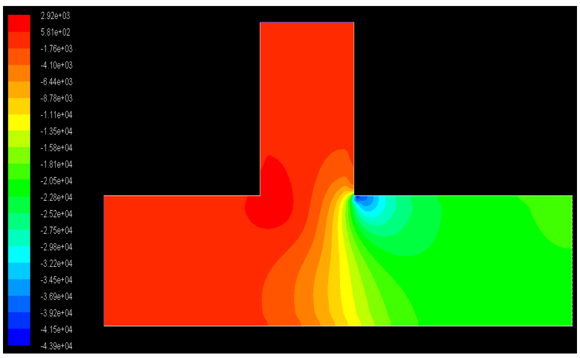


Fig2.1 pressure contours

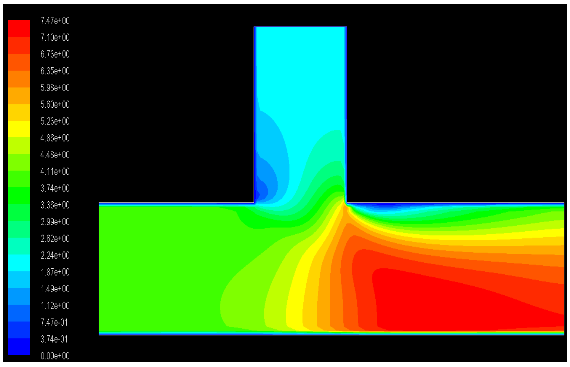


Fig: 2.2 Velocity contours

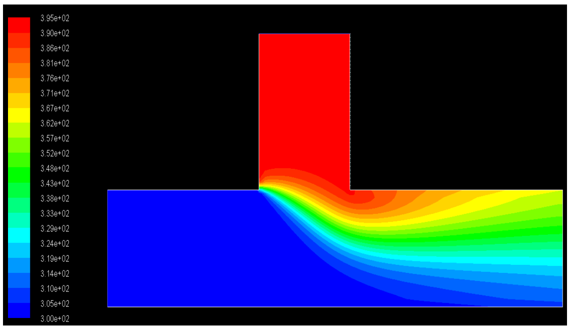


Fig: 2.3 Temperature contours

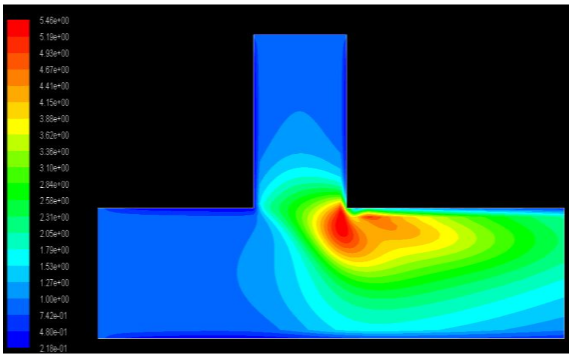


Fig: 2.4 Turbulence contours

Tab: 2.5 Results of flow analysis

S.NO	PARAMETER	MIN	MAX
1	Pressure(pascal)	-43875.04	2921.139
2	Velocity(m/s)	0	7.472435
3	Temperature(K)	300	395
4	Turbulent(m^2/s^2)	0.218194	5.456457

Tab: 2.6 Results of mass flow rate

Mass Flow Rate	(kg/s)
Interior	-21023.578
Inlet_C	119.78401
Inlet_H	59.892004
Outlet	-179.67601
Wall	0

III. SECOND MODEL

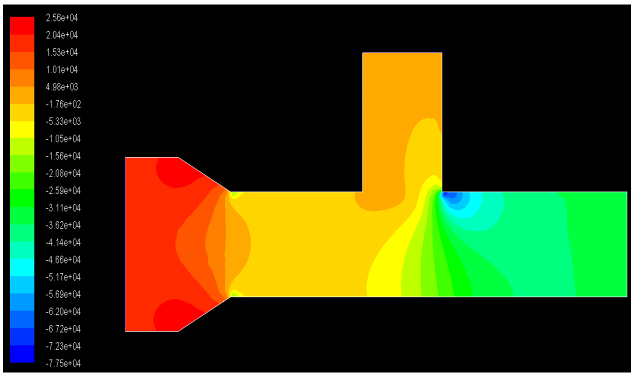


Fig: 3.1pressure contours

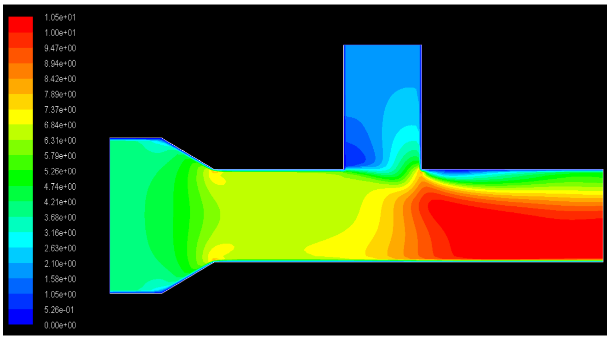


Fig:3.2.velocity contours

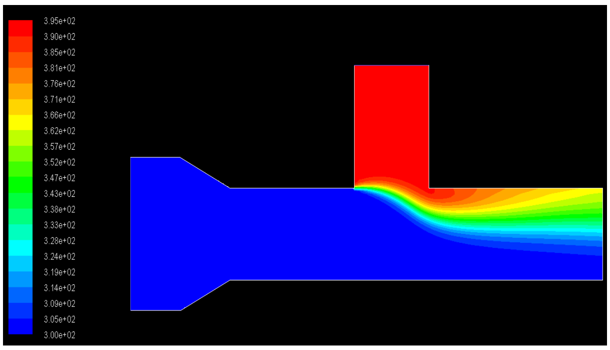


Fig: 3.3.Temperature contours

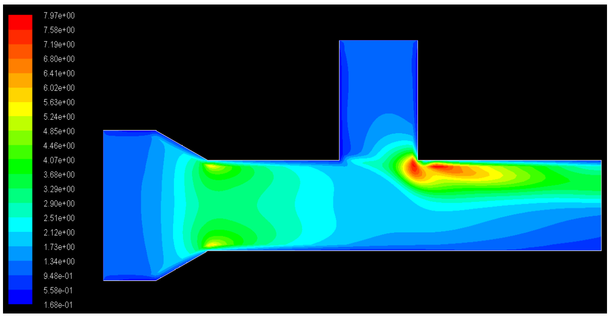


Fig: 3.4.Turbulence contours

S.NO	PARAMETER	MIN	MAX
1	PRESSURE(Pascal)	-77467.77	2588.5
2	VELOCITY(m/s)	0	10.52352
3	TEMPERATURE(K)	300	395
4	TURBULANCE(m^2/s^2)	0.1677681	7.973526

Mass Flow Rate	Interior	Inlet_C	Inlet_H	Outlet	Wall
(kg/s)	-4658.3701	199.6400	59.892004	-259.53202	0

Tab: 3.5.Results of flow analysis and mass flow rate

IV. THIRD MODEL

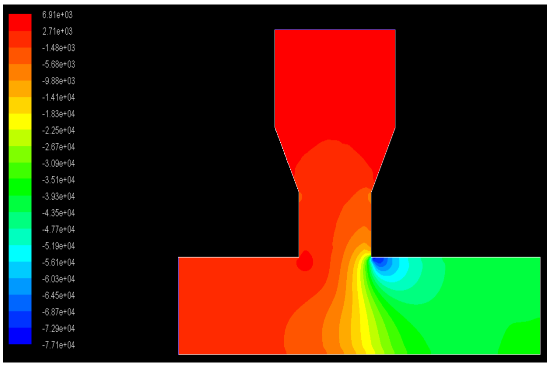


Fig: 4.1.pressure contours

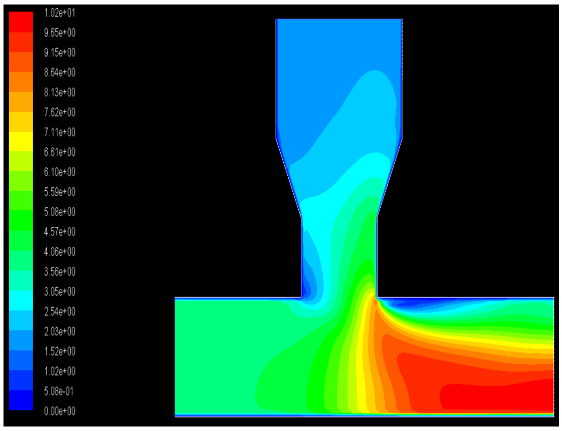


Fig: 4.2.velocity contours

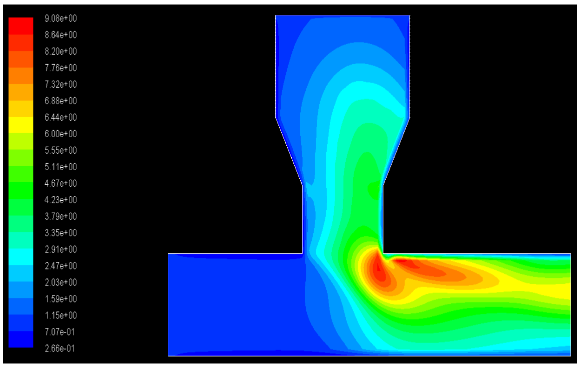


Fig: 4.3.Temperature contours

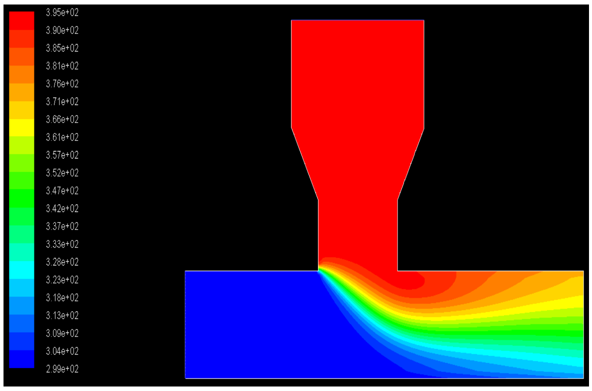


Fig: 4.4.Turbulence contours

S.NO	PARAMETER	MIN	MAX
1	PRESSURE(pascal)	-77065.7	6913.483
2	VELOCITY(m/s)	0	10.16165
3	TEMPERATURE(K)	300	395
4	TURBULANCE(m^2/s^2)	0.2659736	9.080444

Mass Flow Rate	(kg/s)
Interior	-4485.8018
Inlet_C	119.78401
Inlet_H	99.820006
Outlet	-219.60401
Wall	0

Tab: 4.5.Results of flow analysis and mass flow rate

V. FOURTH MODEL

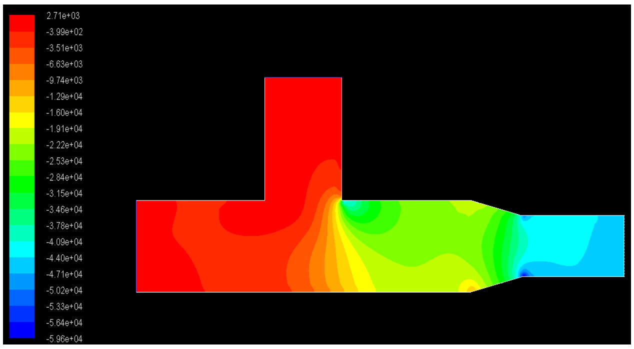


Fig: 5.1.pressure contours

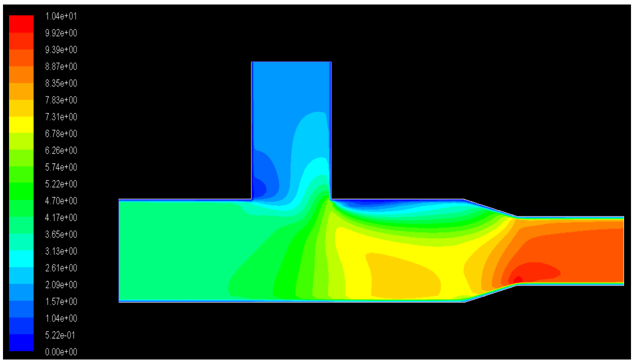


Fig: 5.2.velocity contours

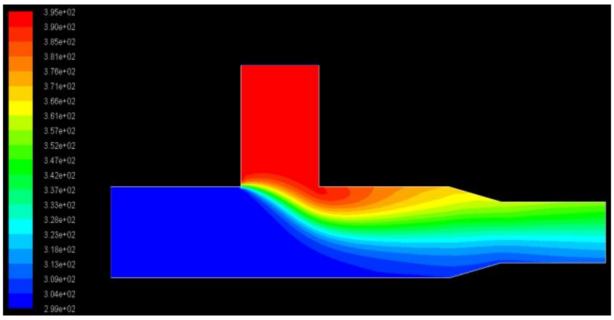


Fig: 5.3.Temperature contours

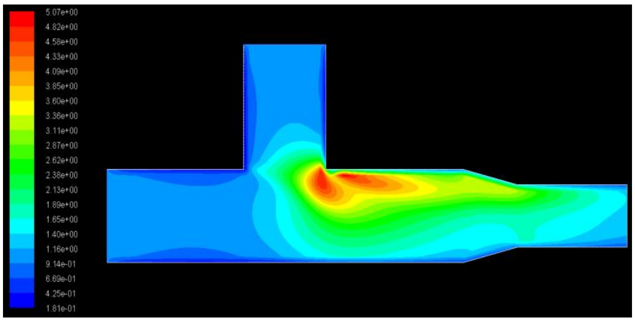


Fig: 5.4.Turbulence contours

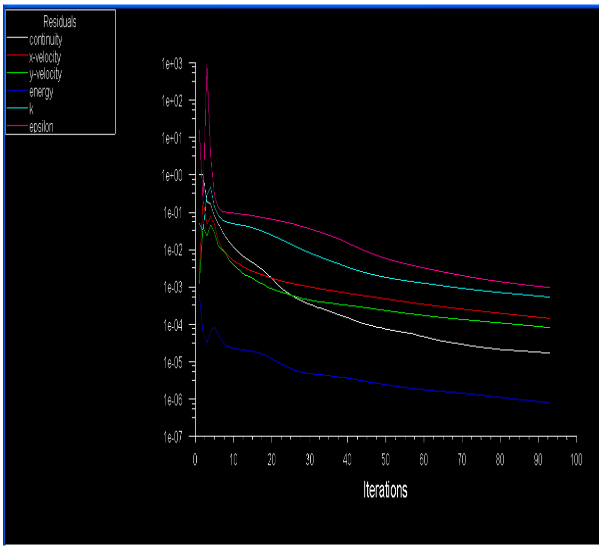
S.NO	PARAMETERS	MIN	MAX
1	PRESSURE(pascal)	-59552.35	2714.012
2	VELOCITY(m/s)	0	10.43727
3	TEMPARATURE(K)	300	395
4	TURBULANCE(m^2/s^2)	0.1806357	5.066483

Mass Flow Rate	(kg/s)
Interior	-3245.6271
Inlet_C	119.78401
Inlet_H	59.892004
Outlet	-179.67601
Wall	0

Tab: 5.5.Results of flow analysis and mass flow rate

SOLUTION STRATEGY

The simulation is done in the FLUENT based upon the governing equations. The steps followed in the fluent are define Model, define Material, define cell zone, boundary condition, solve, iterate, and analyze results. The convergent of the solution is shown in below fig.



VI. CONCLUSION

From the above analysis it is found out that flow will change greatly in t-pipe due to different cross section and also because of hot and cold water mixing process. Temperature, pressure and velocity variations occur at the outlet from the cold and hot water inlets. Hot water will possess a high average kinetic energy and cold water will possess low average kinetic energy, the mixture will have an average kinetic energy of an intermediate value. Initially hot water and cold water mixes up to some extent as the cross section goes on increasing mixing hot and cold water takes place completely then the temperature of the mixed flow reduces.
As the hot water and cold water enters at different velocities of mixed flow will be increased as pressure decreases. Pressure for mixed flow at the outlet will be reduced because of the cross section variation at the joining point of hot and cold water and also high velocities will cause more pressure drop. Turbulence is created at the inter section point of the T-pipe because the cross section at that point has changed suddenly and also the velocity at that point has increased suddenly due to the mixing of hot and cold water at that point.

From above results third model having low pressure, so it is having high velocity and second model is having low temperature. According to our requirement we have to choose better model of designed T-pipe and solvent, design parameters like length, material etc.

REFERENCES

[1] Simulation of heat transfer phenomenon in furnace using fluent-gambit, dr. swarup kumar mahapatra national institute of technology Rourkela, 769008.

[2] Fluid flow in t-junction of pipes, paritosh r. vasava, lappeenranta university of technology department of information technology laboratory of applied mathematics.

[3] Mohammed Abdulwahhab et al. / International Journal of Engineering Science and Technology (IJEST) ISSN : 0975-5462 Vol. 4 No.07 July 2012

[4] Large eddy simulation of hot and cold fluids mixing in a T-junction for predicting thermal fluctuations, Wie-yu Zhu, Tao Lu, Pie-xue Jiang Zhi-jun Guo Kuishing Wang, November 2009,volume 30, Issue 11, pp 1379-1392.

[5] Simulation of turbulent and thermal mixing in T-junctions using urans and scale-resolving turbulence models in Ansys cfx, Th. Frank, M.Adlakha, C. Lifante, H.M. Prasser, F.Menter, ANSYS Germany GmbH, Staudenfeldweg 12, D-83624 Otterfing, Germany ETH Zürich, Dept. Energy Technology, Zürich, Switzerland.
coupled CFD-FEM strategy to predict thermal fatigue in mixing tees of nuclear reactors, M.H.C. Hannink, A.K. Kuczaj, F.J. Blom, J.M. Church and E.M.J. Komen.

[6] Thermal mixing of two miscible fluids in a T- shaped micro channel, Bin Xu, Teck Neng Wong, Nam-Trung Nguyen, Zhizhao Che, John Chee Kiong Chai, Dec 2010; 4(4): 044102. Published online Oct 1, 2010.

[7] Assessment of thermal fatigue in mixing tee by fsi analysis, myung jo jhung, Research Management Department, Korea Institute of Nuclear Safety 62 Gwahak-ro, Yuseong-gu, Daejeon, 305-338, Korea, Received April 12, 2012 , Accepted for Publication June 27, 2012.

[8] Assessment of thermal fatigue in mixing tee by fsi analysis, myung jo jhung, Research Management Department, Korea Institute of Nuclear Safety 62 Gwahak-ro, Yuseong-gu, Daejeon, 305-338, Korea, Received April 12, 2012 , Accepted for Publication June 27, 2012.

[9] Large-eddy simulation of fluid flow and heat transfer in a mixing tee junction, Tao Lu, Yongwei Wang, Kuisheng Wang, Chinese Journal of Mechanical Engineering November 2012, Volume 25, Issue 6, pp 1144-1150.

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

[11] ANSYS, Inc., 2010, ANSYS CFX-Solver Theory Guide and ANSYS CFX-Solver Manager User’s Guide, Canonsburg, PA.

[12] ANSYS, Inc., 2010, Theory Reference for ANSYS and ANSYS Workbench Release 13.0, Canonsburg, PA.

[13] Bannantine, J.A., Comer, J.J., Hand rock, J.L., 1989, Fundamentals of Metal Fatigue Analysis, Prentice Hall, New Jersey.