Computational Fluid dynamics analysis and validation of the effect of angle of inclination of the pipe on temperature of the fluid for Constant surface heat flux condition

Tharoon T
Project Assistant, Department of Mechanical Engineering, Coimbatore Institute of Engineering and Technology, Coimbatore-641109, Tamilnadu, India.
tharoonkvp@gmail.com

Abstract: *The main intent of this research work is to find out the contribution of angle of inclination on flow of fluid in a pipe and to study comparative analysis of various viscous model. A comparative analysis is done containing the data of test of a pipe with inclination such as 0^o, 30^o, 45^o, 60^o and 90^o to determine the competence of the laminar and various turbulent model such as Standard K-epsilon, RNG K-epsilon and K-Omega in CFD. Models of pipe with inclination 0^o, 30^o, 45^o, 60^o and 90^o are created in ANSYS SPACECLAIM, meshing is done with help of grid sensitivity analysis and the analysis is carried out in ANSYS FLUENT. In order to determine the validity of the models developed and simulation results and the outlet temperature of the fluid in a pipe with 0^o, 30^o, 45^o, 60^o and 90^o have been gathered and selected laminar and various pertinent turbulent model such as Standard K-epsilon, RNG K-epsilon and K-Omega which generated results close to data of test. The pipe is subjected to constant surface heat flux. It can be noted that the contribution of the angle of inclination of the pipe is not significant on the temperature of the fluid flow in the pipe and the experimental outlet temperature of the fluid in the pipe with 0^o, 30^o, 45^o, 60^o and 90^o is matched with K Omega model results. This research work confirms that the numerical analysis validation and created models useful in the design and performance of the pipe.*

Key Words: Pipe inclination, model, CFD, Constant surface heat flux, turbulent.

1. INTRODUCTION:

Pipe communications are very important in industries because of the fluids is transported from one place to another place through pipe networks. The pipe flow analysis is very essential in engineering field. Several engineering problems deals with flow and thermal analysis. The analysis of flow is very essential because of unpermissive engineering applications. The viscous effects and other effects are takes place in flow of fluid in a pipe. This effects are identified to be acceptable to laminar and turbulent flow conditions. The computational fluid dynamics (CFD) codes are FLUENT, CFX, STAR CFD, FIDAP, ADINA, CFD2000 and PHOENICS in the design and analysis of flow problems [23] that require huge amounts of computer power, memory and computational time [24, 25]. In order to become a summation of all process of design, the CFD developmental activities are progressing hastily. Due to lack of universally applicable turbulent models, there is a need to examine the adequacy of CFD simulations with test data [26]. Hence, validation through testing is essential for all kind of design and numerical simulations of flow and thermal analysis. [27-36]. In order to scrutinize the competence of the models of turbulent in computational fluid dynamics (CFD) codes, a contrastive study is done in this research paper containing the test data of a pipe with inclination 0^o, 30^o, 45^o, 60^o and 90^o. ANSYS (FLUENT) is used for done the flow analysis for the indicated boundary conditions. The outlet temperature of the fluid in the pipe with inclination 0^o, 30^o, 45^o, 60^o and 90^o have been obtained and compared with the measured temperature of the fluid to trace the pertinent turbulent model which generated results nearer to the measured values. Contour plots of temperature are obtainable for the specified boundary conditions. The evaluations of outlet temperature are matching well with measured values. The numerical analysis is validated by the experiment and accomplished models through comparison of data in the study will be useful in the design of pipe with 0^o, 30^o, 45^o, 60^o and 90^o. Literature surveys from various works explored that pipe network is very important in industry and the inclinations of the pipe was required because of ease of transportation of the fluid. CFD analysis predict the numerical value of the temperature of the fluid consideration of laminar and various turbulent model.

2. METHODOLOGY:

2.1 Problem Description

Pipe lines are used in industry and it plays an important role in flow of fluid. In this study a pipe is considered with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o. The main objective is to find out whether inclination affect the temperature of the fluid. The pipe is subjected to constant surface heat flux. Here, the nature of the flow is considered that means the problem can be divided into four cases. Case 1: laminar flow, Case 2: K Epsilon Standard, Case 3: K Epsilon RNG, Case 4: K Omega

2.2. Geometry

The geometry of pipe with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o was created on ANSYS SPACECLAIM to study the temperature of the fluid when flow enters a pipe with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o. The fluid domain is captured in the pipe with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o in SPACECLAIM. The diameter, length and thickness of the pipe is 50mm, 2m and 10mm respectively. The geometry models are shown from the figure 1 to 5.

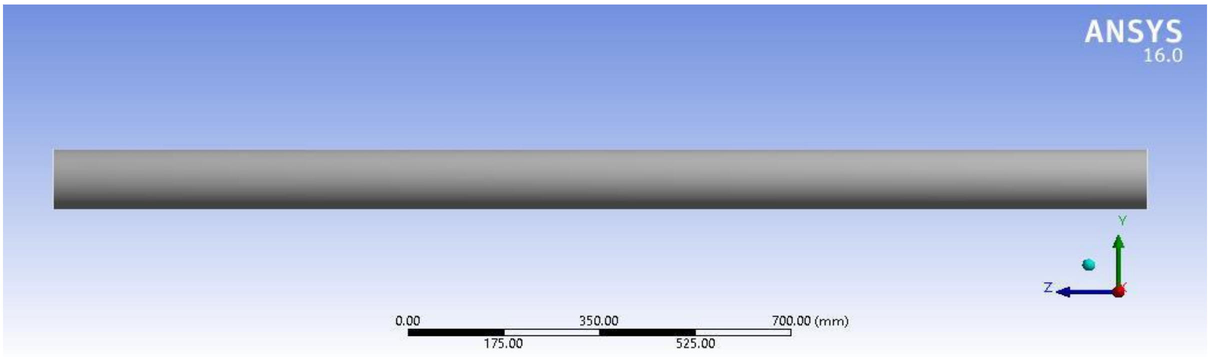

Fig 1. Geometry model of pipe with inclination 0^o

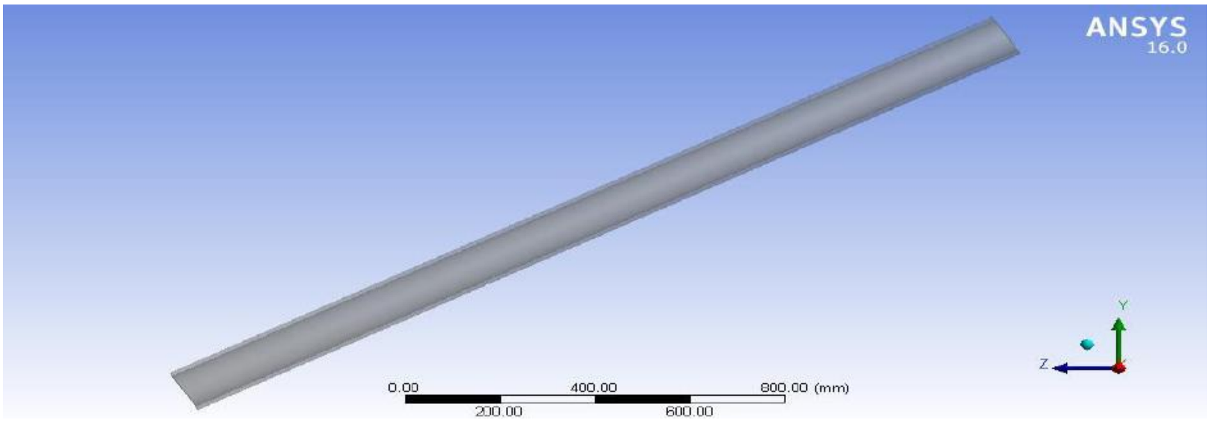

Fig 2. Geometry model of pipe with inclination 30^o

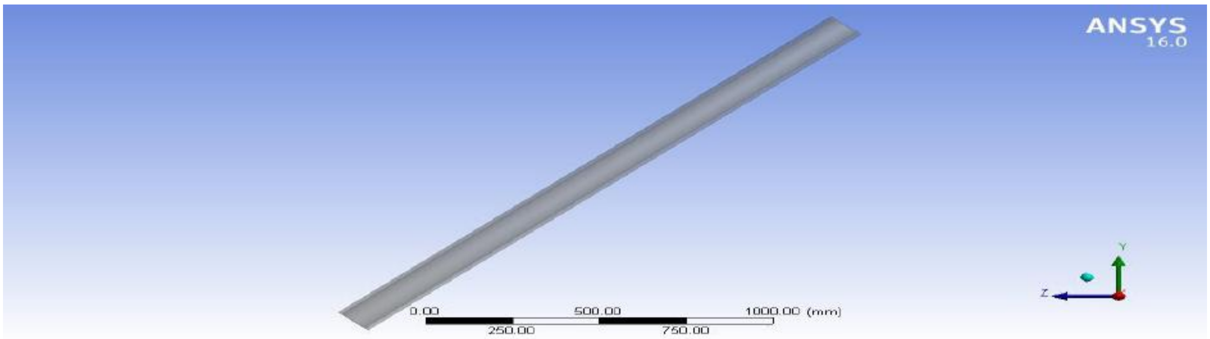

Fig 3. Geometry model of pipe with inclination 45^o

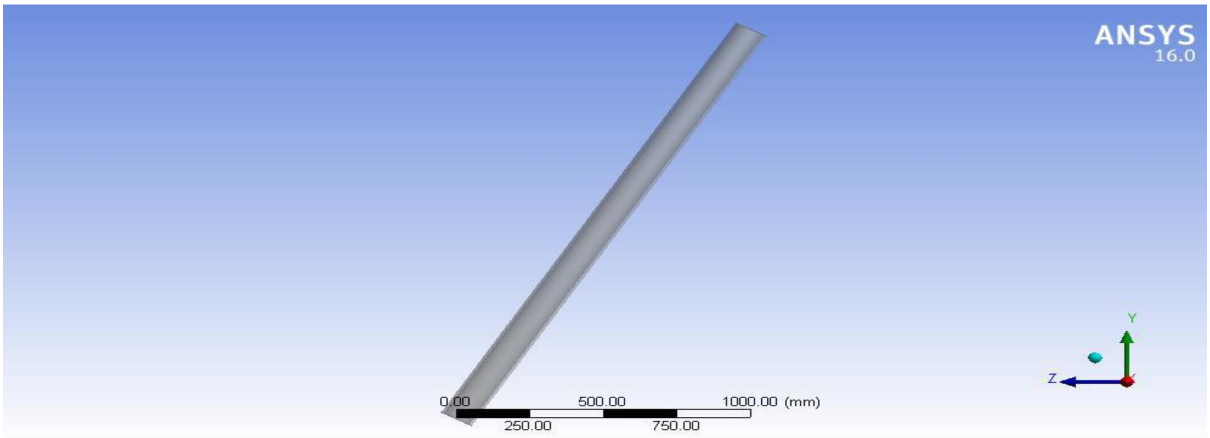

Fig 4. Geometry model of pipe with inclination 60^{o}

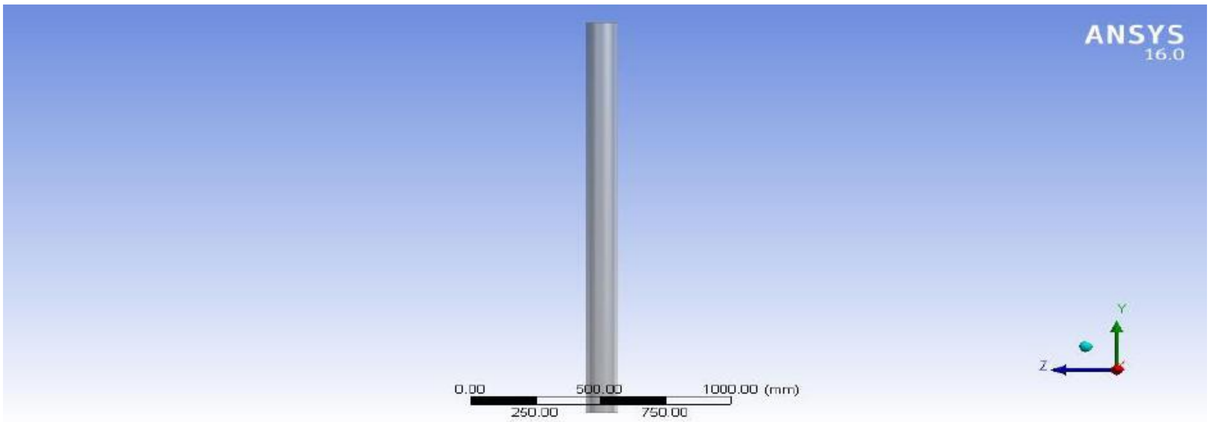

Fig 5. Geometry model of pipe with inclination 90^{o}

2.3. Meshing

The determination of a mesh for the CFD simulation is not a trivial task. The coarse mesh leads large errors. The fine mesh will take high computation time. To establish the impact of size of the mesh on the simulation results, any CFD simulation should be escorted by a grid sensitivity analysis. This kind of analysis is obtainable here. For the purposes of the grid sensitivity analysis, the temperature of the fluid is calculated and compared for different grid densities. The process was repeated for CWT case to compare the grid enslavement for same boundary conditions. Rectangular cells are used for mesh process in pipe wall and domain of the fluid. The three dimensional model consists of 1680 cells. Finite volume formulation is followed for the governing equation. This grid sensitivity analysis is made for all cases. One end of the pipe is named as inlet and the other end is called outlet. The meshing models are shown from the figure 6 to 10.

Table 1 Study of Grid sensitivity

					Experimental
Number of cells in Y direction	15	20	25	10	-
Number of cells in Z direction	112	120	130	100	-
Total Number of cells	1680	2400	3250	1000	-
Temperature of k Omega (K)	300	293	278	256	301.5

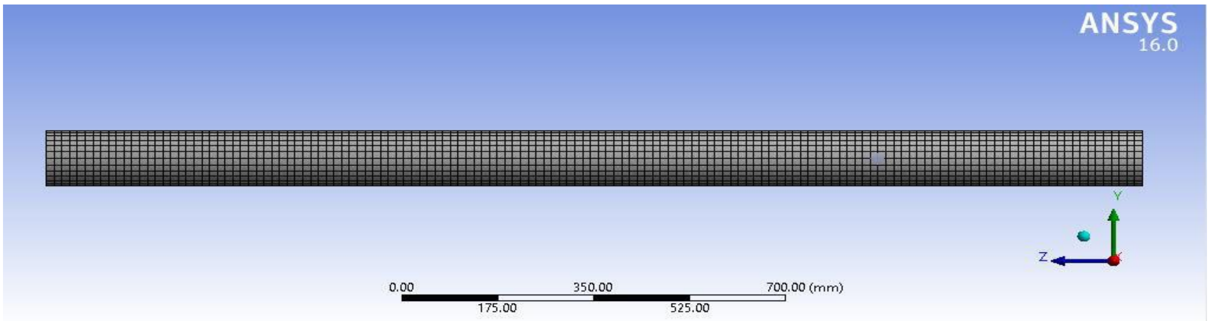

Fig 6. Meshing model of pipe with inclination 0^o

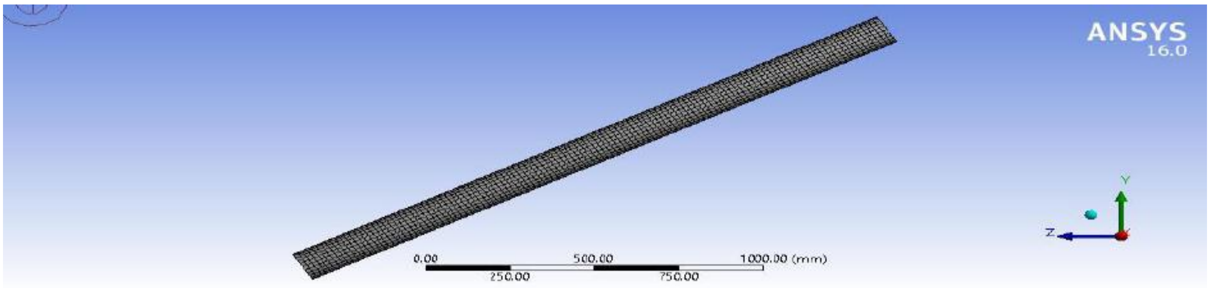

Fig 7. Meshing model of pipe with inclination 30^o

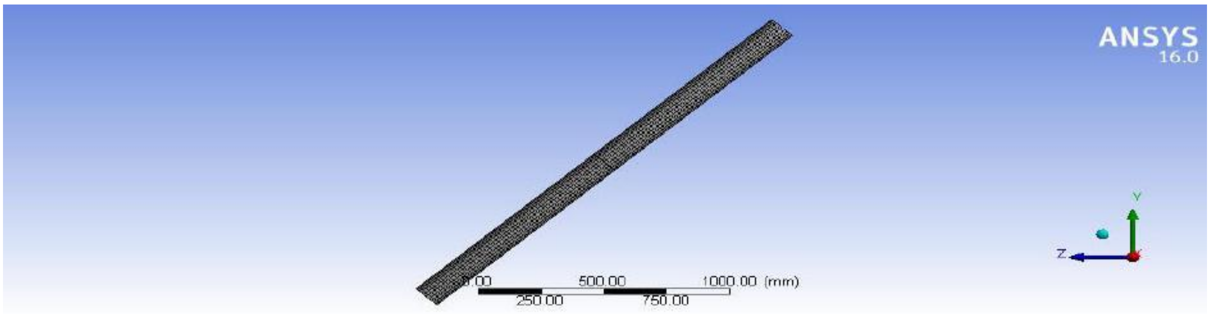

Fig 8. Meshing model of pipe with inclination 45^o

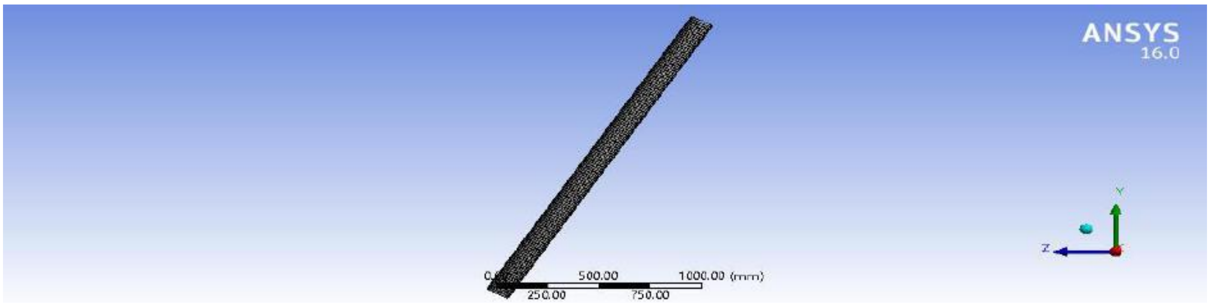

Fig 9. Meshing model of pipe with inclination 60^o

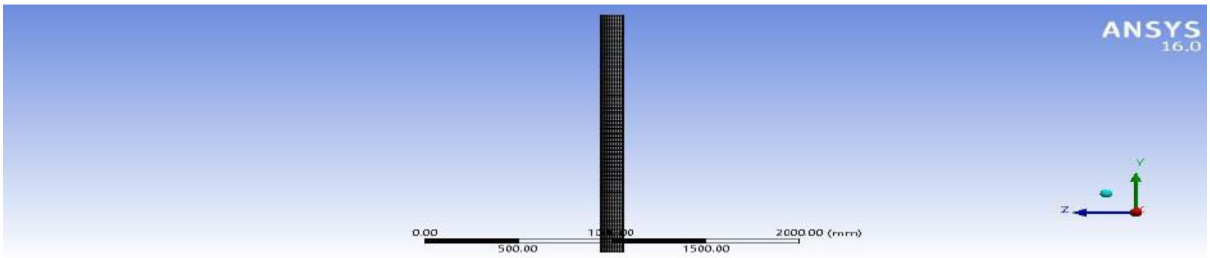

Fig 10. Meshing model of pipe with inclination 90^o

2.4. Boundary Conditions

For the inlet the velocities of water must be given. The velocities of water at inlet was given as 0.1m/s. At the outlet since the value of temperature was unknown, outflow boundary condition was given for the outlet. The pipe is subjected to constant surface heat flux which means the surface heat flux was 1016 w/m^2. The same boundary condition is used for all cases.

2.5. Simulation

Simulation was performed on the pipe with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o generated and applying mentioned boundary conditions. Simulation was performed for fluid domain on ANSYS FLUENT. A comparative work is done on laminar model and various turbulent models such as Standard K-epsilon, RNG K-epsilon and K-Omega to determine the deviation of temperature of the fluid along the length of pipe and angle of inclination. The temperature velocity coupling affirms SIMPLE algorithm is used as method of solution. The solution convergence is limited by fitting the residual of relative is 10^{-6} for all cases.

3. RESULTS AND DISCUSSION:

3.1. Effect of angle of inclination on temperature

The pipe with inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o is analysed with the help of four models such as laminar model Standard K-epsilon, RNG K-epsilon and K-Omega. The temperature is calculated for each case and it is tabulated in the table 2. From the table we can understand that the influence of angle of inclination of the pipe on temperature is not significant. The angle of inclination of the pipe is contributed on temperature of the pipe ranges from 1% to 2% deviated among all angle of inclinations such as 0^o, 30^o, 45^o, 60^o and 90^o

Table 2. Temperature for various angle of inclination and turbulence model

Model / Angle of Inclination	Θ = 0^o	Θ = 30^o	Θ = 45^o	Θ = 60^o	Θ = 90^o
Laminar	294.5563	294.4292	294.4694	294.4614	294.5848
Standard K-Epsilon	300.3483	300.2727	300.2787	300.2815	300.3601
RNG K-Epsilon	300.2621	300.1962	300.2036	300.2047	300.2734
K-Omega	301.4096	301.2525	301.2589	301.2686	301.4298

3.2 Comparative study

In this study for all cases the temperature varies with respect angle of inclination and laminar model and various turbulent model such as Standard K-epsilon, RNG K-epsilon and K-Omega. The temperature profile for various angle of inclination (0^o, 30^o, 45^o, 60^o and 90^o) is as shown in from figure 11 to 15. By observing the temperature profile for various angle of inclination we can see that among four models such as laminar, Standard K-epsilon, RNG K-epsilon and K-Omega, the experimental outlet temperature of the fluid in the pipe with inclinations 0^o, 30^o, 45^o, 60^o and 90^o is matched with K Omega model results. So the temperature contour plot for K Omega model is shown in the figure 16 to 20.

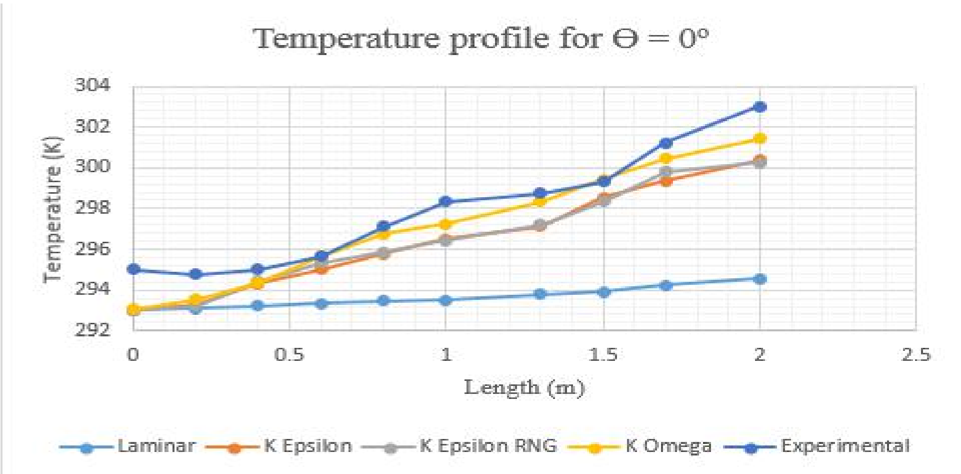

Fig 11. Temperature profile for 0^o

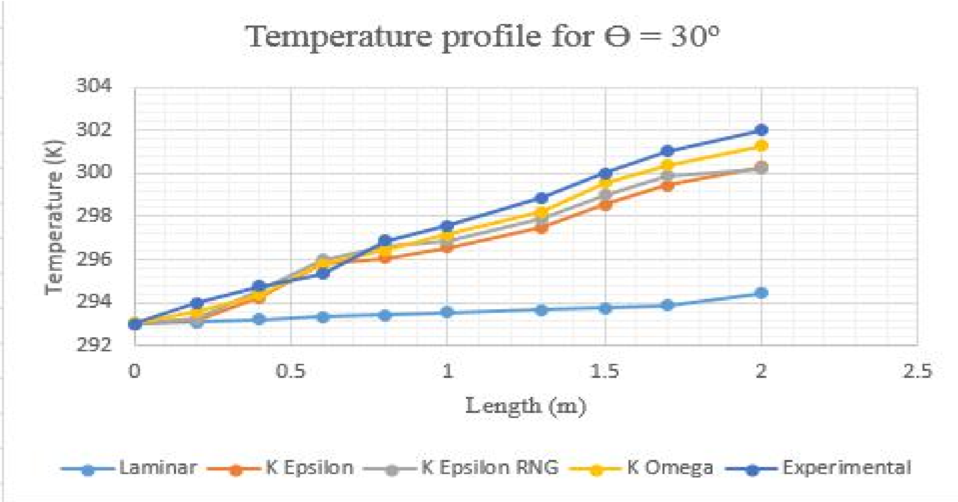

Fig 12. Temperature profile for 30°

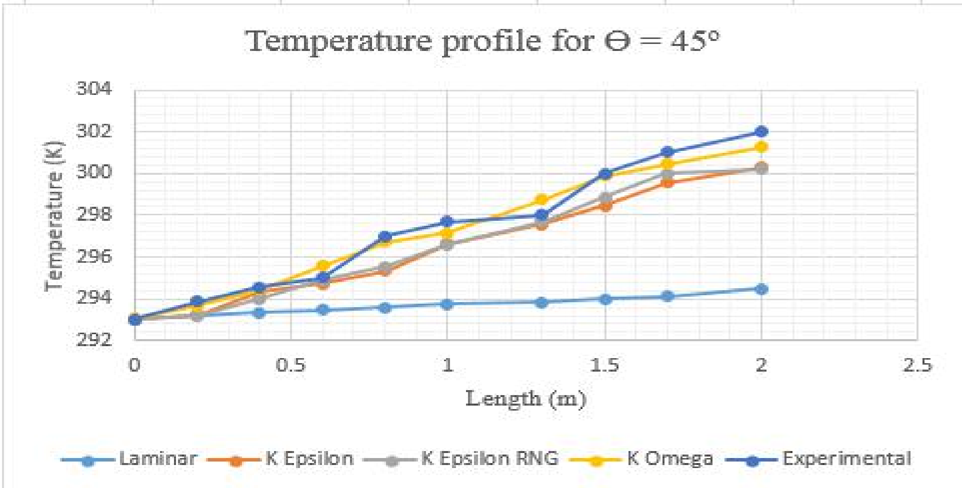

Fig 13. Temperature profile for 45°

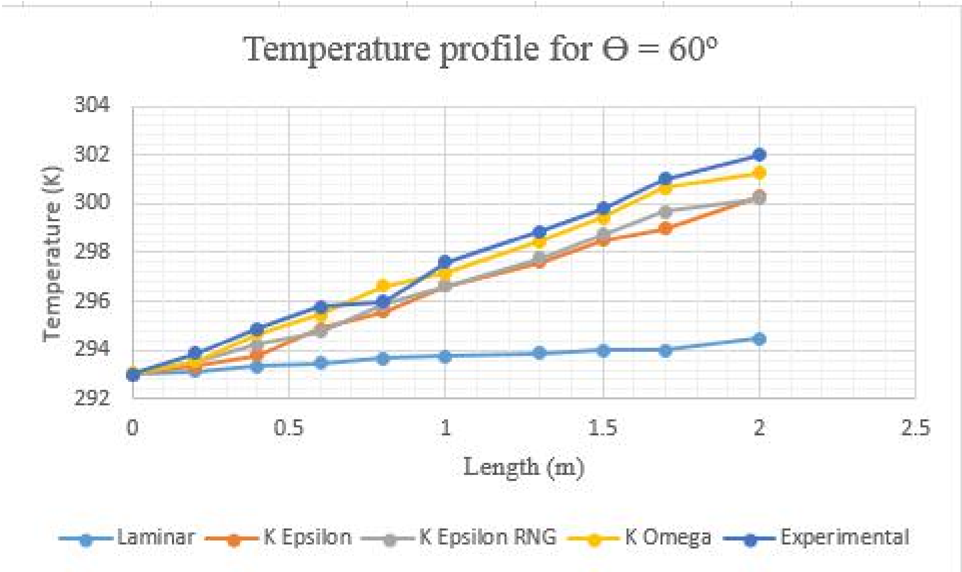

Fig 14. Temperature profile for 60°

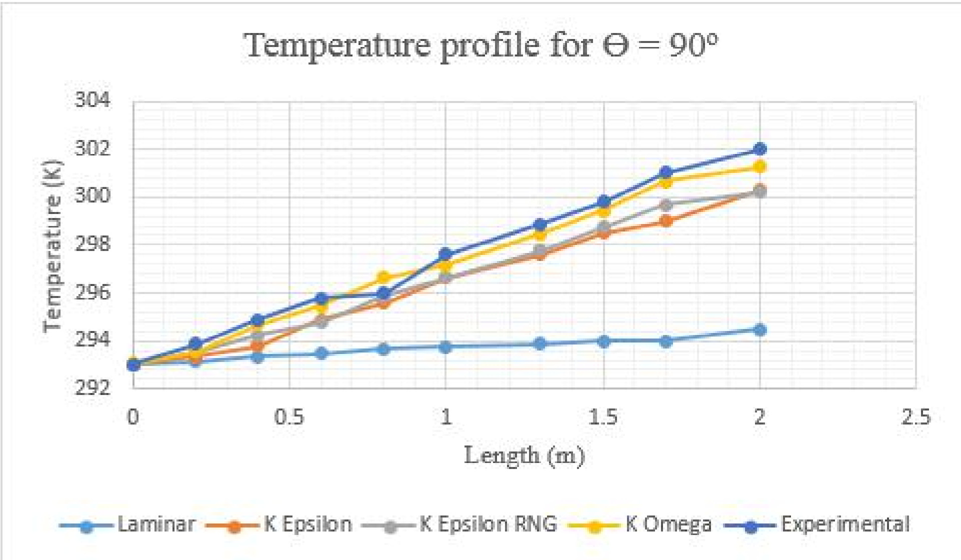

Fig 15. Temperature profile for 90°

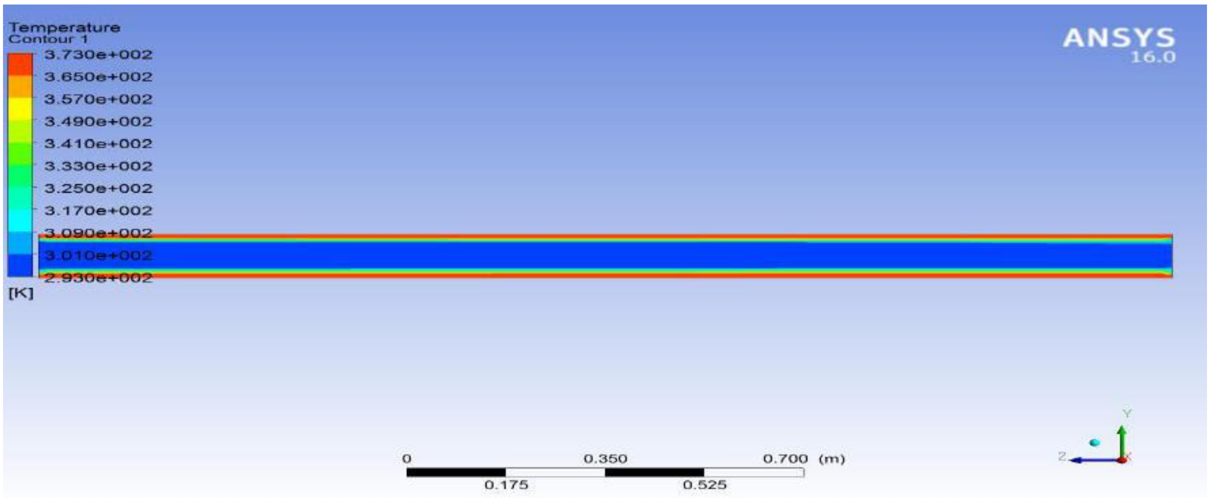

Fig 16. Temperature contour plot for 0°

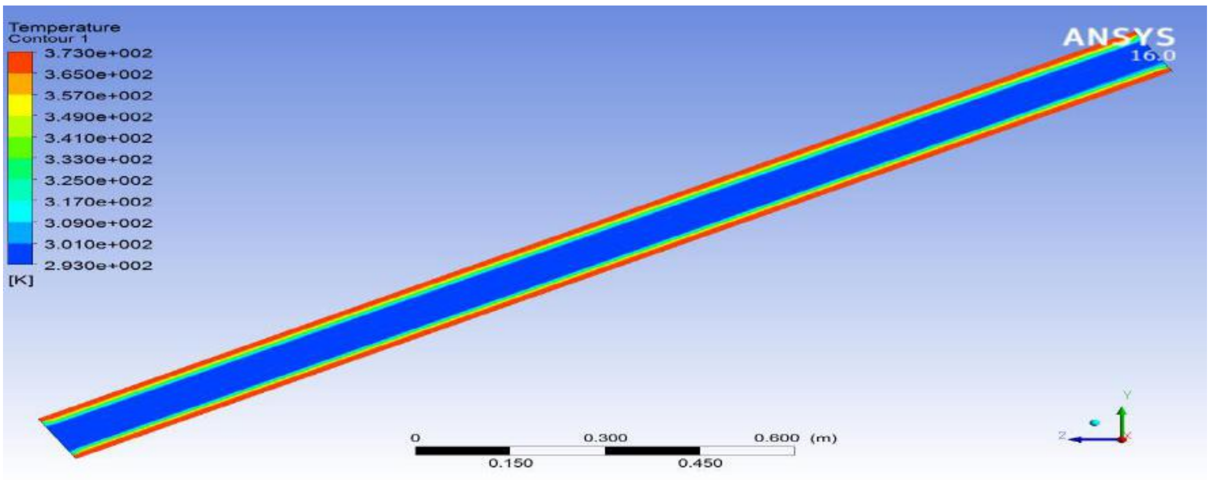

Fig 17. Temperature contour plot for 30°

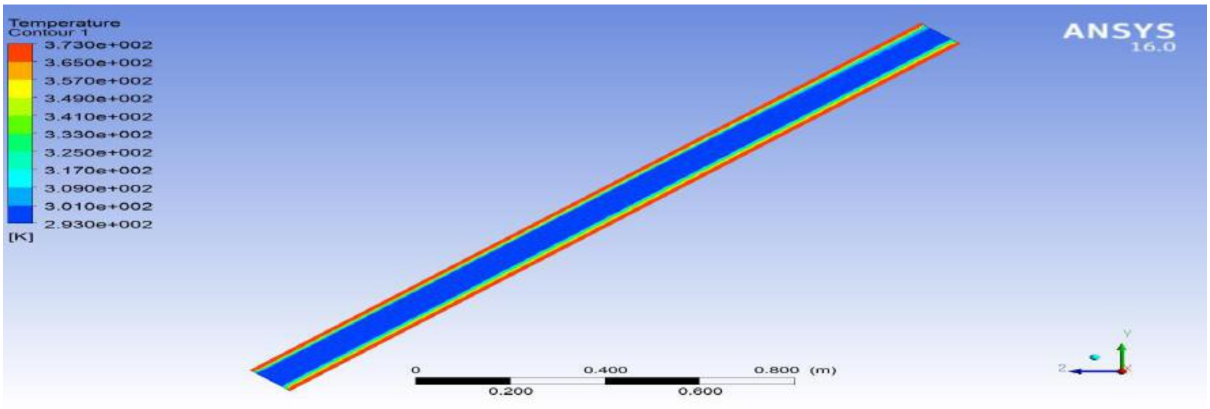

Fig 18. Temperature contour plot for 45^o

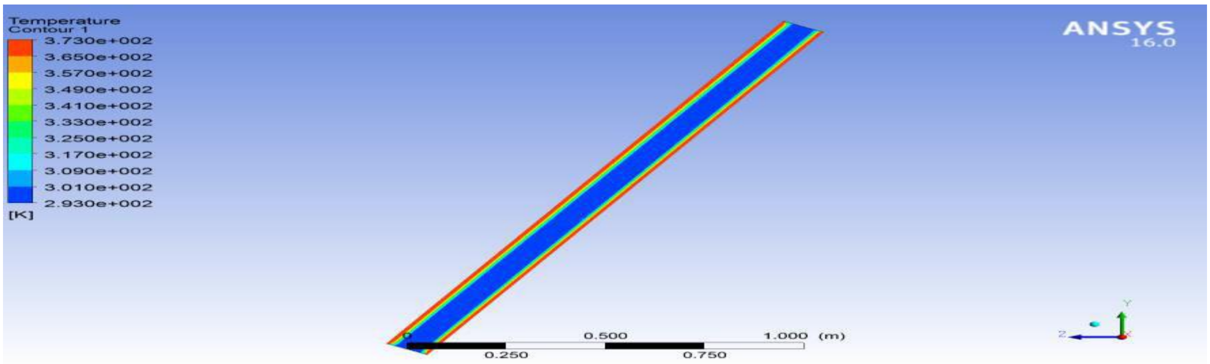

Fig 19. Temperature contour plot for 60^o

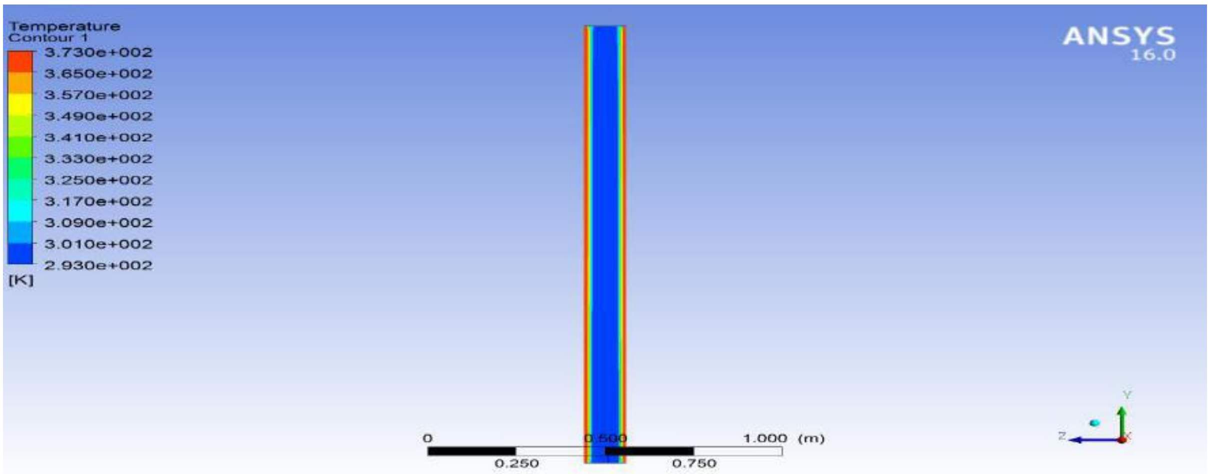

Fig 20. Temperature contour plot for 90^o

4. CONCLUSION:

The paper determines the competence of the laminar model and various turbulent model such as Standard K-epsilon, RNG K-epsilon and K-Omega in codes of Computational fluid dynamics through comparison of test results of the pipe with inclinations 0^o, 30^o, 45^o, 60^o and 90^o. The models of a pipe with inclinations 0^o, 30^o, 45^o, 60^o and 90^o are created by using ANSYS SPACECLAIM, meshing is done with help of grid sensitivity analysis and the analysis is carried out in ANSYS FLUENT. The outlet temperature of the fluid in a pipe with inclination 0^o, 30^o, 45^o, 60^o and 90^o have been gathered and selected laminar and various pertinent turbulent model such as Standard K-epsilon, RNG K-epsilon and K-Omega which generated results close to data of test and the pipe is subjected to constant surface heat flux. Temperature distribution is presented for the specified inlet and outlet boundary conditions. The appraisement of outlet temperatures are well matched with the simulation results obtained by using K omega model. The influence of angle of inclination of the pipe on temperature is not significant. There is no widespread turbulent model. The designer has to select an appropriate model through assessment with data of test. This research work confirms that the numerical analysis validation and created models useful in the design and performance of the pipe.

REFERENCES:

1. Henderson D. (2007), “Experimental and CFD investigation of an ICSSWH at various inclinations”, Renewable and Sustainable Energy Review 11 1087–1116.
2. Rajesh Khatri, (May 2012) “Laminar flow analysis over a flat plate by Computational fluid dynamics”, International Journal of Advances in Engineering & Technology, , ISSN: 2231-1963.
3. Sahu M.,(Feb.2009) “Developed Laminar Flow In Pipe Using Computational Fluid Dynamics”,7th International R&D Conference on Development and Management of water and Energy Resources, 4-6 Bhubaneshwar, India.
4. White, Frank M.,(1991)”Viscous Fluid Flow”, International Edition, McGraw-Hill,
5. Bansal R.K. (2010). Fluid mechanics and hydraulic machines. Laxmi publication (p) ltd,
6. Abdulwahhaba Mohammed, Injetib N K, Dakhilc Sadoun Fahad (2012) : “CFD Simulation and flow analysis through a T junction pipe” IJEST, 4(7), 3393-3407,
7. Crawford N. M, Cunningham .G and Spedding P. L, “Prediction of Pressure Drop for Turbulent Fluid Flow in 90° Bends,” Proceedings of the Institution of Mechanical Engineers, Vol. 217, No. 3, 2003, pp. 153-155.
8. Spedding P. L, Benard .E and McNally G. M,(2004) “Fluid Flow through 90° Bends,” Developments in Chemical Engineering and Mineral Processing, Vol. 12, No. 1-2, pp. 107-128.
9. Keon B.J.Mc, M.V.Zagarola, A.J.Smits, (2005), A new friction factor relationship for fully developed pipe flow, J. Fluid Mech. vol. 538, pp. 429–443
10. Romero-Gomez, P., C. K. Ho, and C. Y. Choi. (2008). Mixing at Cross Junctions in Water Distribution Systems – Part I. A Numerical Study, ASCE Journal of Water Resources Planning and Management, 134:3, pp. 284-294.
11. Lahiouel Y., Haddad A., Chaoui K., ―Evaluation of head losses in fluid Transportation networks Sciences & Technologies B – N°23, jun (2005), pp. 89-94
12. Ansys, Inc. http://http://www.idac.co.uk/products/downloads/Meshing.pdf
13. Anand B. Desamala, (2013)“CFD Simulation and Validation of Flow Pattern Transition Boundaries during Moderately Viscous Oil-Water Two-Phase Flow through Horizontal Pipeline”, World Academy of Science, Engineering and Technology, 73.
14. Aslam A. Hirani, C. Udaya Kiran, (Nov. - Dec. 2013), “CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe”, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 10, Issue 1 PP 31-34.