



INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

COMPARATIVE STUDY OF DIFFERENT CFD MODELS TO EVALUATE HEAT TRANSFER AND FLOW PARAMETERS IN STHE

Anshul Jain*, Prof. K. K. Jain, Prof. Sudarshan Patel

(*ME student, Prof. & Head, Prof. SRIT, Jabalpur)

* Mechanical Engineering Department, SRIT Jabalpur, India

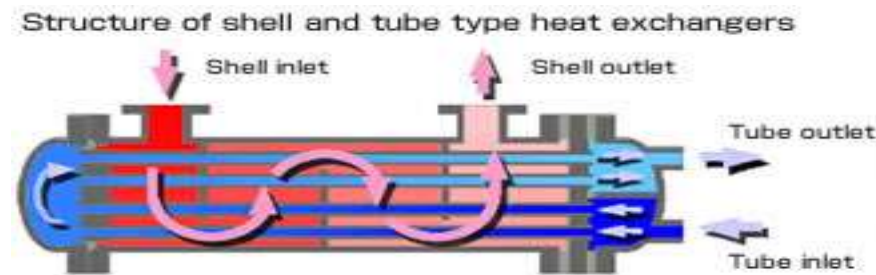
ABSTRACT

Computational fluid dynamics (CFD) simulations can be very useful to investigate heat transfer and visualize the temperature fields & fluid flow characteristics of shell and tube heat exchanger. A shell and tube heat exchanger is modeled to find the heat transfer parameters. The heat exchanger contains tubes inside with baffle arrangement. The flow and temperature fields are resolved using CFD package (ANSYS FLUENT). The experimental investigation has been also performed for comparison purpose. The CFD turbulence models considered for investigation are k-epsilon, SST, Eddy Viscosity and Laminar model. Laminar flow is considered for understanding the significance of turbulence in the flow field. The boundary conditions taken for the computational domain are derived out of the experimental investigation results. Transient analysis has been performed for the physical time scale of 1800 seconds. Unstructured meshing method is used to create mesh on the domain. It has been found out that k-epsilon model came out to be the best model to predict the flow parameters, heat transfer coefficient and behavior of present case of STHE. Reasonable agreement is found between the simulation and experimental data.

KEYWORDS: Shell & tube; Turbulence ; CFD; heat transfer parameters.

INTRODUCTION

Heat exchangers are one of the most important heat transfer apparatus that are used in industries like chemical engineering, oil refining, electric power generation etc. Shell-and-tube types of heat exchangers (STHXs) have been commonly and most effectively used in Industries over the years. The shell-and-tube heat exchangers are still the most common type in use. They have larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures, and their construction facilitates disassembly for periodic maintenance and cleaning. The shell-and-tube heat exchangers consist of a bundle of tubes enclosed within a cylindrical shell. One fluid flows through the tubes and a second fluid flows within the space between the tubes and the shell. Typical Shell-and-Tube heat exchanger is shown in Figure 1.1.



Its unavoidable need has necessitated work on efficient and reliable designs leading towards optimum share in the overall system performance. The Log Mean Temperature Difference (LMTD) method and the number of heat transfer units (NTU) method have been used for heat exchanger design. These methods have some shortcomings associated with them i.e. iterative in nature and need of a prototype to implement the design. Due to these reasons, these methods are time consuming as well as expensive especially for large scale models. However, economical access to powerful micro processors has paved the way for evolution of Computational Fluid Dynamics (CFD) during the design phase. (V. Kumar, S. Saini et al 2006)

CFD is a science that can be helpful for studying fluid flow, heat transfer, chemical reactions etc by solving mathematical equations with the help of numerical analysis. It is equally helpful in designing a heat exchanger system from troubleshooting and optimization by suggesting design modifications.

CFD employs a very simple principle of resolving the entire system in small cells or grids and applying governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients, flow parameters and the like in a shorter time at a lower cost because of reduced required experimental work (Y. Wang, Q. Dong, M. Liu et al 2007)

In the present work a shell and tube heat exchanger is modeled to investigate the heat transfer parameters. The heat exchanger contained 14 tubes inside a 1025 mm long and 156 mm diameter shell with baffle arrangement. The flow and temperature field inside the shell and tube are resolved using CFD package (ANSYS CFX 13.0). A set of CFD simulations is performed for a single shell and tube bundle and is compared with experimental results.

The four CFD models are considered and are compared to find the best suitable model for the present case. The modeling has been done in design modular module in the Ansys package. The meshing has been done using un-structural Tetrahedral mesh element. And the mesh of the element fixed after grid dependency test has been found to be 779491 elements. (ref. Appendix.A)

The results are found to be good for the turbulence model. Further the k-epsilon model predict the flow parameters and heat transfer-coefficient more accurately then other models.

COMPUTATIONAL MODELLING

The first step of computational modeling is Geometry Modeling. It requires the geometric parameters of the model. It is a mathematical model that requires extensive computational resources to study the behavior of complex system by computer simulation. Instead of deriving the analytical solution of the problem by solving complex mathematical equations, experimentation with the model is done by setting the parameters of the system in the computer. CFD resolves the entire system in small cells and apply the governing equations to find numerical solutions with regard to fluid flow and temperature distribution. It creates a virtual prototypes of the system and gives the numerical solution in a shorter time and lower cost due to reduced required experimental work. The basic approach of used CFD are

(A) Pre-processor (B) solver (C) Post-processor.

GOVERNING EQUATIONS

The flow is governed by the continuity equation, the energy equation and Navier-Stokes momentum equations. Transport of mass, energy and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume where $i, j, k = 1, 2, 3$ correspond to the three dimensions.

Continuity Equation

The continuity equation describes the conservation of mass and is written as in equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_1}{\partial x_1} + \frac{\partial \rho U_2}{\partial x_2} + \frac{\partial \rho U_3}{\partial x_3} = 0$$

or

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0, i = 1, 2, 3$$

(b) Momentum Equations (Navier- Stokes Equations)

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + g_i$$

(c) Energy Equation

$$\frac{\partial (h_m)}{\partial t} = -U_j \frac{\partial (h_m)}{\partial x_j} + P \frac{\partial U_i}{\partial x_i} - \frac{\partial (P U_i)}{\partial x_i} - \frac{\partial}{\partial x_j} (\tau_{ij} U_i) - \tau_{ij} \frac{\partial U_i}{\partial x_j} + \rho g U_i$$

Turbulence Model :

Since the flow in this study is turbulent, so turbulence effect should be considered using turbulence modeling. The choice of turbulence is very critical in CFD simulations. However, there is no universal criterion for selecting a turbulence model. On the basis of literature available, and for comparing the performance of the turbulence model; three turbulence model are consider in this study. One laminar model is also consider to understand the effects of turbulence. In this study k – ε turbulence model, k – ω SST model, Eddy-viscosity model and laminar viscosity model are considered.

(i) k – ε Model : The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation, ε. There respective modeled transport equations are as under

For k,

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = \nu_T \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right] - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

And for ε

$$\frac{\partial \varepsilon}{\partial t} + \langle U_j \rangle \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \nu_T \frac{\varepsilon}{k} \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right] + C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

The closure coefficients for k – ε models are $C_\mu=0.09$, $C_{\varepsilon 1}= 1.44$, $C_{\varepsilon 2}= 1.92$, $\sigma_k=1.00$, $\sigma_\varepsilon= 1.34$

The physical interpretation of the ε equation is,

1. Accumulation of ε
2. Convection of ε by the mean velocity
3. Production of ε
4. Dissipation of ε
5. Diffusion of ε

(ii) k – ω SST Model- It has been a problem to accurately predict the flow separation. The modeled equation for k is as under

$$\frac{\partial k}{\partial t} + \langle U_j \rangle \frac{\partial k}{\partial x_j} = \nu_T \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right] - \beta k \omega + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

and the modeled equation for ω is

$$\frac{\partial \omega}{\partial t} + \langle U_j \rangle \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} \nu_T \frac{\varepsilon}{k} \left[\left(\frac{\partial \langle U_i \rangle}{\partial x_j} + \frac{\partial \langle U_j \rangle}{\partial x_i} \right) \frac{\partial \langle U_i \rangle}{\partial x_j} \right] - \beta^* \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]$$

Closure Coefficients for k – ω Model are $\alpha = 5/9$, $\beta = 3/40$, $\beta^* = \sigma_k = 1/2$, $\sigma_\omega = 1/2$

(iii) **Eddy-Viscosity Model** : The concept behind the eddy-viscosity model are that the unknown Reynolds stresses, a consequence from the averaging procedure, are modeled using flow parameters (strain rate tensor & rotation tensor) and an eddy viscosity

$$-\rho \overline{u'_i u'_j} = \underbrace{\mu_T}_{\text{Eddy viscosity}} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\rho k + \mu_t \frac{\partial \bar{u}_m}{\partial x_m} \right)$$

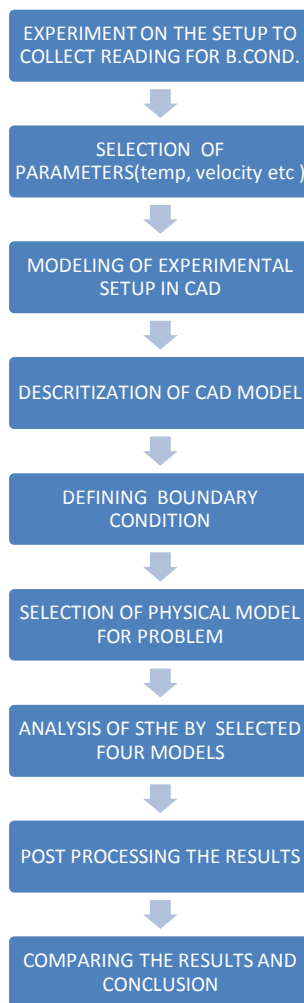
$$R_{ij} = -\rho \overline{u'_i u'_j} = \mu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_T \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \rho k \delta_{ij}$$

(iv) **Laminar viscosity model** : The laminar viscosity model is used for specifying the viscous conditions of flow. It defines the laminar flow, it is based on Navier Stokes equation

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + g_i$$

METHODOLOGY AND EXPERIMENTAL WORK

Methodology:



Experimental setup description :



HEAT EXCHANGER DIMENSIONS			
No.	Description	Unit	Value
1	Shell outside diameter	mm	156
2	Shell inside diameter	mm	150
3	Tube outer diameter	mm	12
4	Tube inner diameter	mm	10
5	Number of tubes		14
6	Shell/Tube length	mm	1025
7	Number of baffles	mm	4
8	Baffle spacing	mm	205
9	Baffle inclination		0°

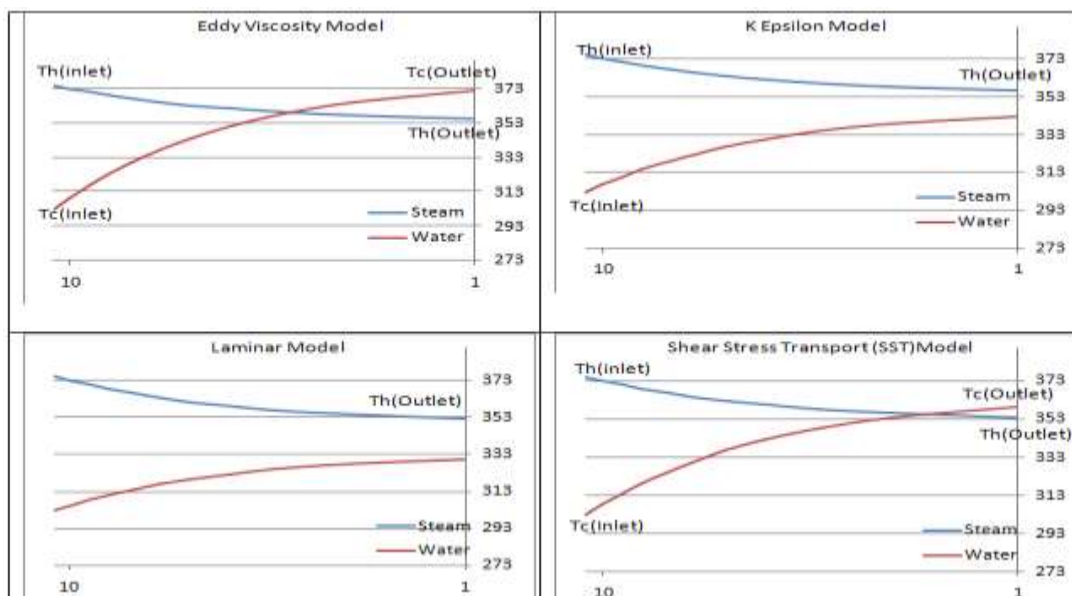
The Experimental observations are as follows

1. Steam pressure (gauge) (kg/cm²) = 0.25 (kg/cm²) = 0.245 bar
2. Water inlet temperature = 29.5°C.
3. Water outlet temperature = 72.4°C.
4. Water flow rate = 11.5 cc/sec.
5. Steam inlet temperature = 102.1°C.
6. Steam Outlet temperature = 81.2°C.
7. Condensate volume collected = 340 ml.
8. Time of collection = 60 sec.

RESULTS AND DISCUSSION

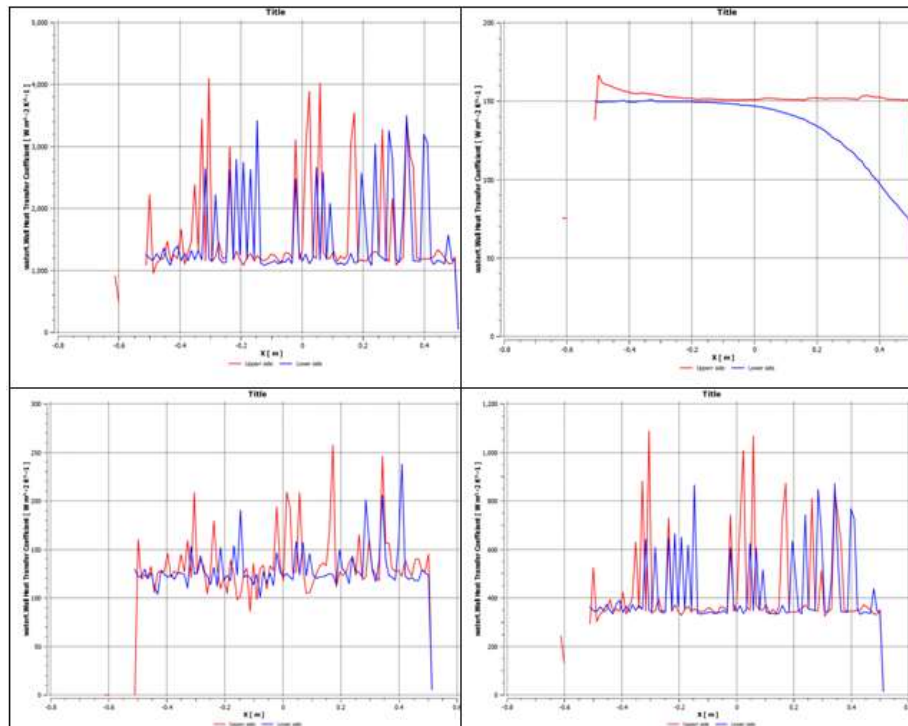
Figure 1,2,3,4 represents the results obtained by eddy viscosity model, k-epsilon model, laminar model and shear stress transport(SST) model respectively.

1. Logarithmic temperature distribution of heat exchanger pipe along the length.



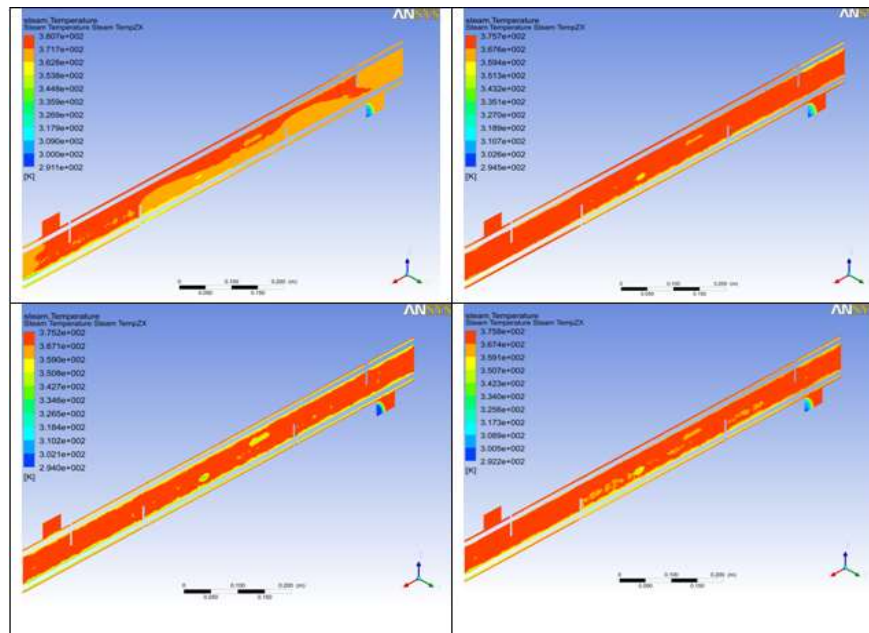
Comments :

Eddy Viscosity model and Shear stress transportation model generated the unsatisfactory results due to violation of exit temperature criterion. Laminar viscosity model generated the realistic results but due to turbulences encountered in steam and water particle practically this results are also not giving the satisfactory correlation with experimental results. So the best suited criterion for this heat exchanger problem solving is K-Epsilon model. So the best suited criterion for heat exchanger problem solving is K-Epsilon model. Results are matched and validated by the experimental results, because this model having a fear balance between the turbulence and shear at wall.

2 . Heat transfer coefficient of heat exchanger pipe along the length.**Comments :**

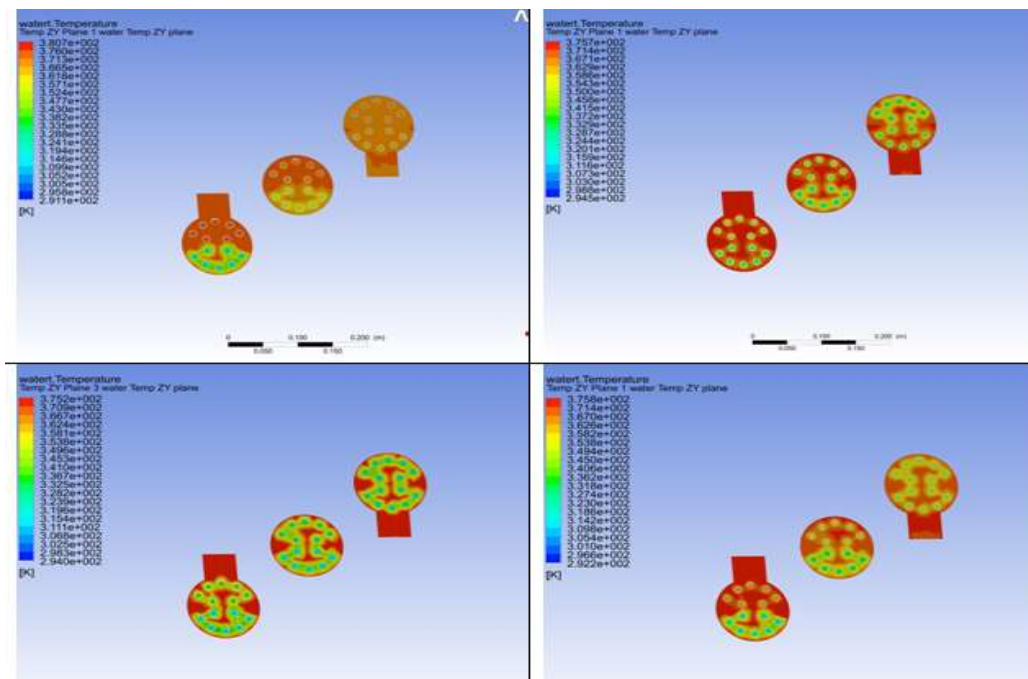
Eddy Viscosity model and Shear stress transportation model generated the unsatisfactory results due to dominance of the turbulences criterion consideration by these two models. So the heat transfer coefficient is vulnerable throughout the flow. In Laminar viscosity model the flow is stable and free from eddy but not a practical case. Results are also not giving the satisfactory correlation with experimental results. So for the mix flow (steam and water) problems of heat transfer the K-Epsilon model gives the best results that is because, this model having a fear balance between the turbulence and shear at wall for moderate fluid flow conditions.

3. Steam Temperature distribution of heat exchanger pipe.

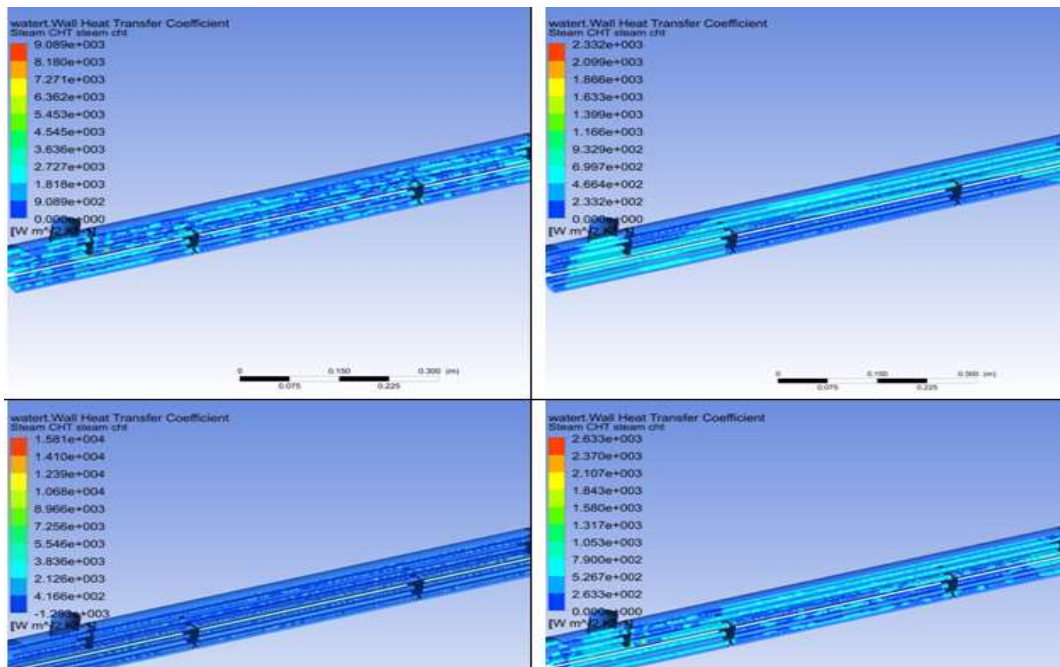


Comments: Steam Temperature distributions inside the casing of heat exchanger are shown in the figures. The high heat transfer zones or where the turbulence involves the temperature of the steam reduces abruptly. This abrupt behavior is undesirable, so as the K-epsilon model shows the minimum inconsistency across the flow line.

4. Sectional view of water temperature distribution of heat exchanger pipe.

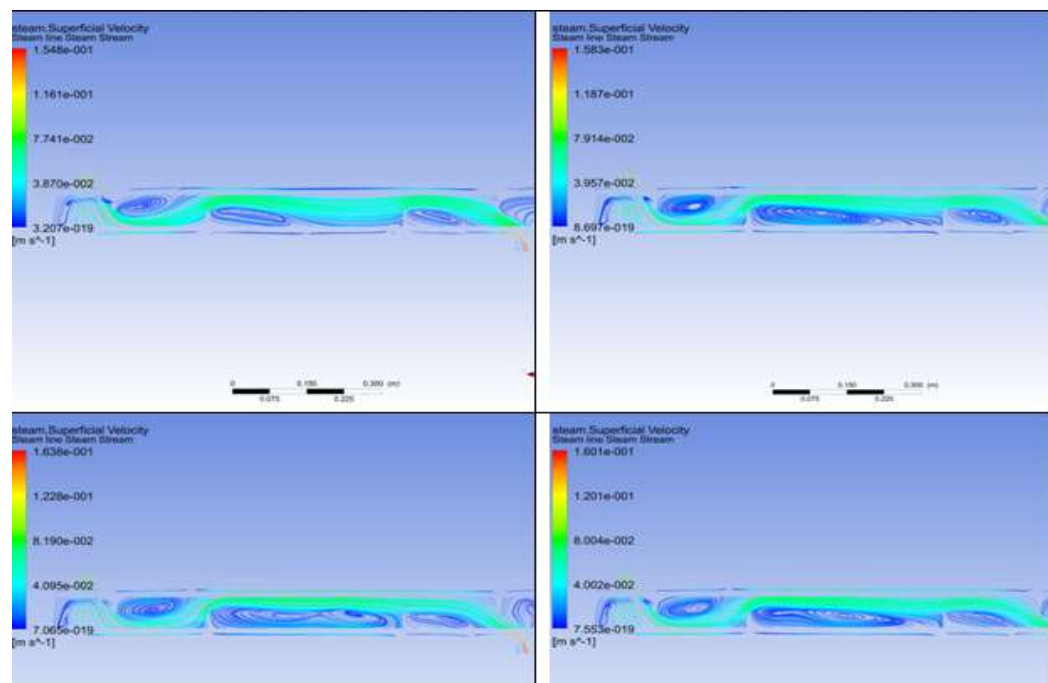


5. Heat transfer coefficient distribution of heat exchanger pipe.



Comment : There is a pictorial representation of Heat Transfer coefficient among the all models. K-epsilon model represent the consistent pattern of heat transfer coefficient whereas laminar. Eddy and SST models are non-uniform and random heat transfer coefficient generators. So for the low and moderate velocity force convective systems like our case, the K-ε Viscosity model is best suited.

6. Steam line for stream distribution of heat exchanger pipe.



Comments: Vorticity is generated due to shear or velocity gradient among the particle and also due to wall and fluid interface but the major role of vortex comes into the picture when the turbulent fluid interact with baffles inside the tubes. Eddy viscosity, laminar and SST model are not able to generate the ideal condition for problem solving, due to over-dominancy of turbulence in Eddy and SST model. Results are best fit for K-epsilon model. So for the general purpose discreet fluid interaction problems are solving by this particular method.

COMPARISION OF RESULTS AND DISCUSSION :

(a)Comparison of Results of Different CFD models with Experimental Results

Sr. No	Parameter	Experimental Result	Eddy Viscosity Model	Laminar viscosity Model	k-ε Model	Shear Stress Transport Model
1	Steam Inlet Temperature (°C)	102.1	102.1	102.1	102.1	102.1
2	Steam Outlet Temperature (°C)	81.2	82.8	79.1	83.2	89.9
3	Water Inlet Temperature (°C)	29.5	29.5	29.5	29.5	29.5
4	Water outlet Temperature (°C)	72.4	99.1	57.1	69.9	86.3
5	Water Wall side CHT (W/m ² K)	117.072	1407	133.5	143.5	391.5
6	Steam Wall side CHT (W/m ² K)	235.180	951.1	79.93	297.24	322.8

Table 5.1: Comparative study of results.

(b)Comparison of Water Outlet temperature.

The Water outlet temperature is more accurately predicted by k-ε model than the other CFD model considered. The outlet temperature of water predicted by k-ε model has a difference of 2.5 °C or 3.5 % with the experimental results, while the other models show greater variation. So a good agreement is shown between experimental and simulation result by k-ε model.

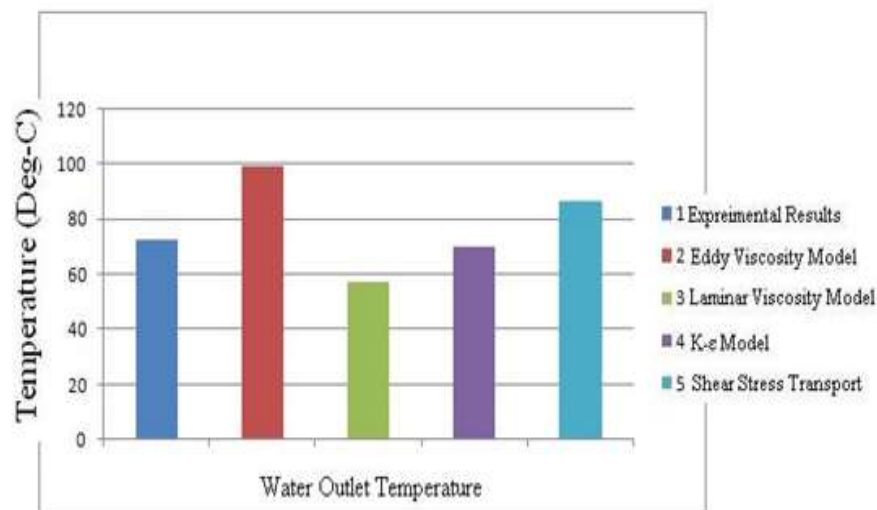


Figure: Water outlet temperature graph.

(c) Comparison of Steam outlet temperature.

The outlet temperature of steam obtained by the considered CFD models shows a close agreement with the experimental results except for the case of shear stress transport model which over predict the result by 10% while the other models predict the result by $\pm 2.5\%$. Moreover the Eddy viscosity model and k- ϵ model shows a close agreement with the experimental results. The k- ϵ model predict the steam outlet temperature with the difference of 2°C or less than 2.5%. So a good agreement is shown between experimental and simulation result by k- ϵ model.

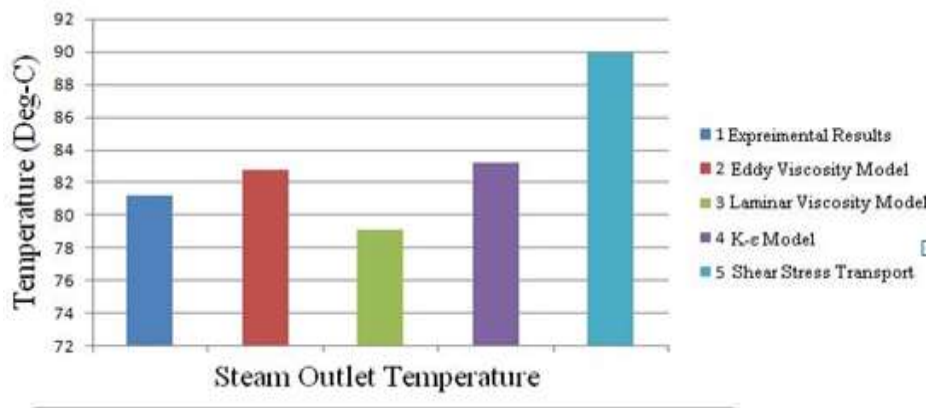


Figure: Steam outlet temperature graph.

(d) Comparison of Water wall side heat transfer coefficient.

The Water wall side heat transfer coefficient obtained by the experiment and the considered CFD models are shown in following graph. The water wall side CHT predict by Eddy Viscosity model is much higher than the experimental value, which is not feasible at all. Also the SST model over predict the CHT with a large difference. The laminar viscosity and k- ϵ model shows a fair good agreement with the experimental value of water wall side heat transfer coefficient. But the Pictorial representation of heat transfer coefficient by the laminar viscosity model is non uniform and it generates random heat transfer coefficient, while the k- ϵ model represent consistent pattern of heat transfer coefficient, so for the low and moderate velocity force convective system in the present case, k- ϵ viscosity model is best suited.

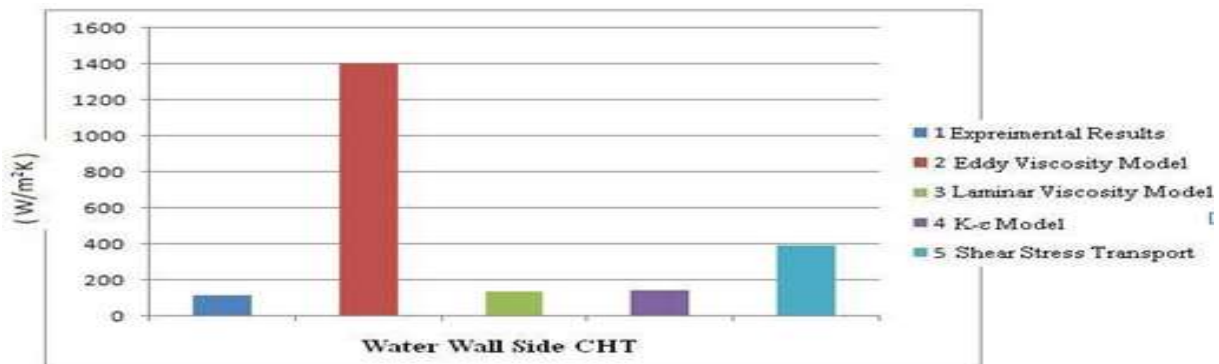


Figure: Water wall side CHT graph

(e) Steam Wall side heat transfer coefficient- comment.

The steam wall side heat transfer coefficient obtained by the experiment and the considered CFD models are shown in following graph. The eddy viscosity model over predict the steam side CHT by a much higher value than experimental result, so it is not feasible at all. More over the laminar viscosity model under predict the steam side CHT also the laminar viscosity model is not applicable at all in such case. The k-ε model shows a fair good agreement with the experimental value of steam side CHT. So for such type of models k-ε model is best suited.

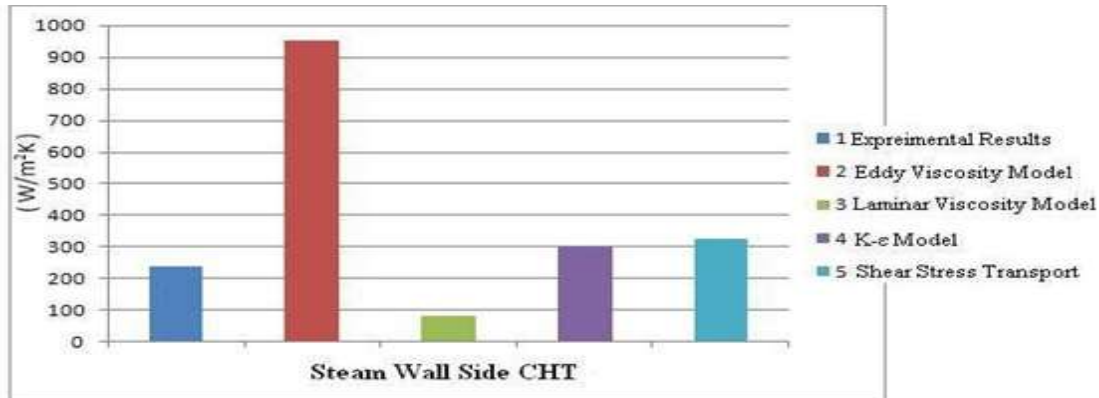


Figure : Steam wall side CHT graph

CONCLUSION

During the CFD analysis it has been found out that laminar model totally failed to predict the flow parameters as well as CHT parameters. And the other entire turbulence model shows significant improvement over laminar model. Out of three turbulence model, the SST model is over predicting the turbulence & separation due to which the thermal aspects of the flow shows the higher value. And the same happens with the eddy viscosity model, which predicts the high eddy formation in the baffle section which leads to the higher steam side CHT as well as water side which is not feasible at all. Further the K-epsilon is predicting the behavior moderately. It predicts the Water side heat transfer coefficient as well as Steam side heat transfer coefficient more closely to the experimental values. It also predict the flow parameters, water outlet temperature, steam outlet temperature more closely to the experimental results. It shows good agreement with the literature available on the use of k-epsilon model.

NOMENCLATURE

A = Heat transfer area (m^2)

h_m = Kinetic Energy

h_T = Thermal Energy

h_C = Chemical Energy

h = Total energy

h_h = Hot side heat transfer coefficient ($W/m^2.K$)

h_c = Cold side heat transfer coefficient ($W/m^2.K$)

K = Exchanger wall material thermal conductivity ($W/m.K$)

Q = Heat transfer rate (W)

R_f = Fouling coefficient ($W/m^2.K$)

T = Temperature (K)

t = Time (s)

U = Overall heat transfer coefficient ($W/m^2.K$)

u = Velocity (m/s)

v = Fluid flow velocity (m/s)

V = Volume (m³)
 ΔT_{LM} = Logarithmic mean temperature difference (K)
 Δx = Exchanger tube wall thickness (m)
 Φ = Potential Energy
 k = turbulence kinetic energy
 ε = turbulence dissipation rate.
 ω = specific dissipation rate
 τ = time constant for turbulence.
 C_{μ} , $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k , σ_{ε} = Closure Coefficients for $k - \varepsilon$ Model.
 α , β , β^* , σ_k , σ_{ω} = Closure Coefficients for $k - \omega$ Model
 μ_t = effective turbulent viscosity
 ρ = Density (kg/m³)
 x , y , z = Spatial coordinates (m)

REFERENCES

1. Ahmed F. Khudheyer and Mahmoud Sh. Mahmoud 2011. Numerical analysis of fin tube heat exchanger by using cfd technique. *ARPN journal of Engineering and applied sciences*, vol 6 p-7 2011.
2. Savitri Patel, D.s Patel "Computational modeling of STHE" *International Journal of Engineering science & Innovative Technology*, vol.3 Issue 2 March 2014.
3. Ozden, e. & Tari, I. 2010. Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy conversion and management*, 51(5):1004-1014.
4. Yongqing, W., Xin, G., KE, W. & Qiwu, D. 2011. Numerical investigation of shell-side characteristics of H-shape baffle heat exchanger. *Procedia engineering*, 18:53-58.
5. Dong, Q.W., Wang, Y.Q. & Liu, M.S. 2008. Numerical and experimental investigation of shellside characteristics for RODbaffle heat exchangers. *Applied thermal engineering*, 28:651-660.
6. Brennan, M.S. 2003. CFD simulations of gravity sluices. (In Proceedings of the Third International Conference on CFD in the Minerals and Process Industries held in Melbourne, Australia from 10 to 12 December 2003. Commonwealth Scientific and Industrial Research Organisation. Melbourne, p. 275-280.)
7. Tsai, Y.C., Liu, F.B. & Shen, P.T. 2009. Investigations of the pressure drop and flow distribution in a chevron-type plate heat exchanger. *International communications in heat and mass transfer*, 36(6):574-578.
8. Apu Roy, D.H.Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, *National Institute Of Technology*, 2011.
9. Yonghua You, Aiwu Fan, Suyi Huang, Wei Liu 2011. Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles
10. A. M. Prithviraj M, "Shell and tube heat exchangers. part 1: foundation and fluid mechanics," *Heat Transfer*, p. 33:799-816., 1998.
11. D. Kern, Process Heat Transfer. *McGraw-Hill*, 1950.
12. N. B. M. Bhutta, M.A.A Hayat, "Cfd applications in various heat exchangers design: A review," *Applied Thermal Engineering*, vol. 32, pp. 1-12, 2011.
13. Hetal kotwal, D.S patel, " Cfd analysis of shell and tube heat exchanger – A review" *IJESIT*, vol., marc2013.
14. E. Cao, Heat Transfer in Process Engineering. *McGraw Hill*, 2009
15. B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, and B. V. Wachem, *Computational Fluid Dynamics for Chemical Engineers. sixth ed., 2010.*
16. H. K. Versteeg and M. W., An Introduction to Computational Fluid Dynamics, The Finite Volume Method. *Pearson Education Limited*, 2007.
17. Domkundwar & Domkundwar ., *Heat and Mass Transfer data book, third ed revised, 2012.*
18. R. Yadav., Heat and Mass Transfer. *Central Publishing House. Sixth ed., 2010*
19. Muhammad Mahmood Aslam Bhutta, Nasir Hayat1, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan. CFD applications in various heat exchangers design: A review. *Applied Thermal Engineering* 32 (2012).
20. "Ansys fluent theory guide." <http://www.ansys.com>,.