CFD analysis & Experimental Investigation of Shell & Tube Type Heat Exchanger with modified Slotted Baffles

Mr. Dinesh Bhalchandra Ahire [1], Prof. Manaklal hiraji patil [2]

[1] PG scholar, Mechanical Engineering, P.S.G.V.P. MANDAL'SD.N.PATEL COLLEGE OF ENGINEERINGSHAHADA, DIST-NANDURBAR (M.S.), Maharashtra, India

2 Professor, Mechanical Engineering, P.S.G.V.P. MANDAL'SD.N.PATEL COLLEGE OF ENGINEERINGSHAHADA, DIST-NANDURBAR (M.S.), Maharashtra, India

ABSTRACT

In process industry such as Oil and Gas, Power, Petrochemical and Refinery etc. Several types of heat exchangers According to Design of Plant's Requirement. Now here we are discussing on the topics of shell and tube type heat exchangers. Shell and tube type heat exchangers are Design and Assembled by Various and Heavy components. In which baffle is used to direct shell side fluid and to support the tubes. The Segmental baffle type having drawback that produces large pressure drop of flow induced vibration. To remove these drawback by one way is to direct the flow as longitudinal on the shell side by use of a new baffle arrangement which will produce the longitudinal flow on the shell side. The baffle is called as Slotted baffle.

Keyword : - *Heat Exchanger, modified Slotted Baffles, CFD & FLUENT*

---

1. INTRODUCTION

Here we introduced one Advanced type of heat Exchangers Family are "Slotted baffle exchangers "And the technology represents a new heat exchanger design that uses axial shell side flow to reduce the pressure drop. Other exchanger designs available that provide axial shell side flow are very expensive and do not provide adequate vibration mitigation at bundle entrance and exit areas. Axial flow designs are especially valuable when the shell side pressure drop must be minimized. For example, if the shell side stream passé through compressor, its power usages would be highly dependent on the shell side pressure drop and all other parameters equal to a standard exchanger with single segmental baffles would be five or more times greater shell side pressure drop than that offered by the slotted baffle exchanger technology[1]. Thus slotted baffle exchanger could have a substantial impact on energy usage. Slotted baffle exchanger design provides an economical bundle design along with providing vibration mitigation throughout the bundle. "Slotted Baffle" (SB) is provided at about 2 to 3 ft (600 to 900 mm) in slotted baffle exchanger, with alternate horizontal and vertical ribs. Preferably, these SBs with ribs may be produced by welding flat bars or rods to a window. Also preferably, SBs with horizontal ribs should be placed where slings are expected to be positioned during bundle handling. Slotted baffles repeats the sequence that throughout the bundle. The ribs in each SB are slightly undersized compared to tube spacing.

2. LITERATURE REVIEW

M. Prithiviraj, M. J. Andrews [1], based on three dimensional, collocated, fully implicit, control volume based calculation procedure. HEATX (5) has been used to simulate fluid flow and heat transfer in shell-and-tube heat exchangers. The three-dimensional numerical model uses the distributed resistance method along with volumetric porosities and surface permeability to model

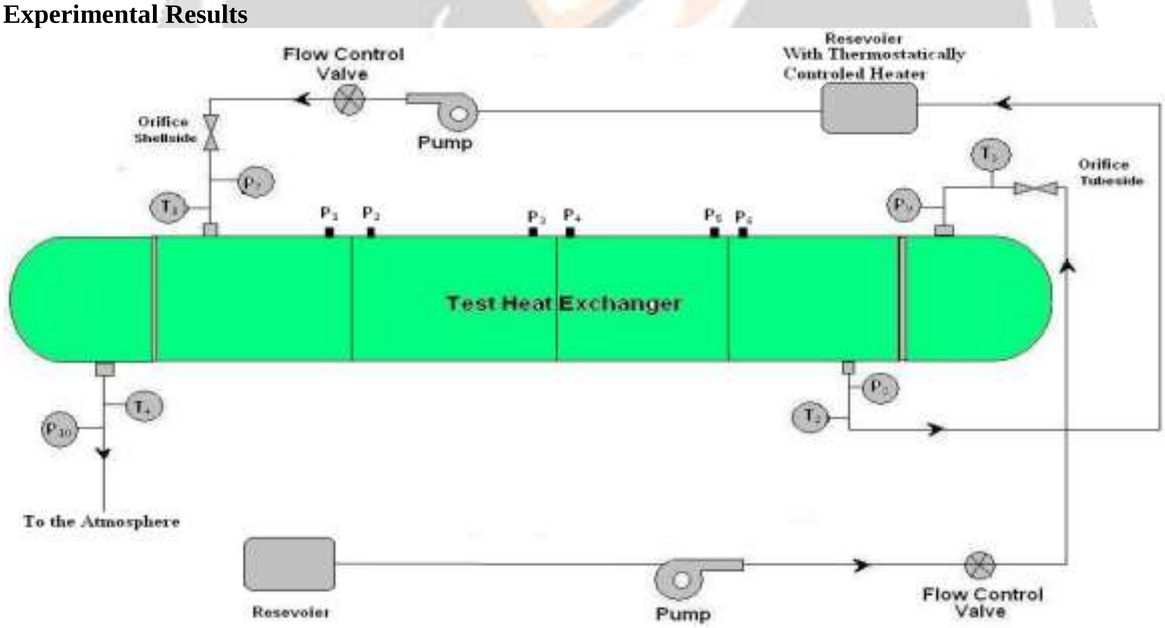
Tubes in the heat exchanger. Turbulence effects are modeled using a modified k-ε model with additional source terms for turbulence generation and dissipation by tubes. Shell and baffle walls are modeled using the wall function approach. Tubes and baffles are modeled using volumetric porosities and surface permeability. Baffle-shell and baffle-tube leakages are modeled using Bernoulli type formulation. Specialized geometry generators compute baffle, nozzle, and tube region porosities and permeability. Article presents the foundation and fluid mechanics of the problem. The three-dimensional numerical model is validated by comparison of computed pressure drops with the experiments. The effect of baffle cut and baffle spacing on the pressure drop is also studied.

R.H. Clarke and F. Nicolas [2] studied fouling in shell & tube HX through CFD, 14-baffle crude oil heater was modeled using scalar-CFD methods based on the single tube model, although in addition to the fouling model Heat transfer coefficient and friction factor correlations were used with the RNG k-ε model. Reduction of porosity inside the shell was used to represent the bundle, with impermeable interfaces used for the baffles. Fouling was predicted using the Ebert and Panchal model [4]. As the deposits grew viscosity was artificially increased, cell by cell, as for the single tube to account for the increased pressure drop. Variation of fouling resistance as variation in time below "threshold conditions".

3.EXPERIMENTAL SETUP AND INSTRUMENTATION

During the dissertation the test setup for shell and tube type heat experimented was fabricated. During the experiment the readings were taken for different values of mass flow rate and the inlet temperature on both shell side and tube side. The experimental results are compared with the CFD results of the of the same. During the CFD analysis SIMPLE algorithm was used for analysis. In the CFD analysis is done for three dimensional, steady state conditions were considered. Standard k-ϵ model was employed and the energy equation was considered during CFD analysis. Second order upwind scheme was considered for discretization of energy equation and momentum equation. CATIA V5R16 software is sued for solid modeling of the heat exchanger and the CFD analysis was carried out in FLUENT 6.0. The results of the experimentation and CDF analysis are compared.

Experimental Results

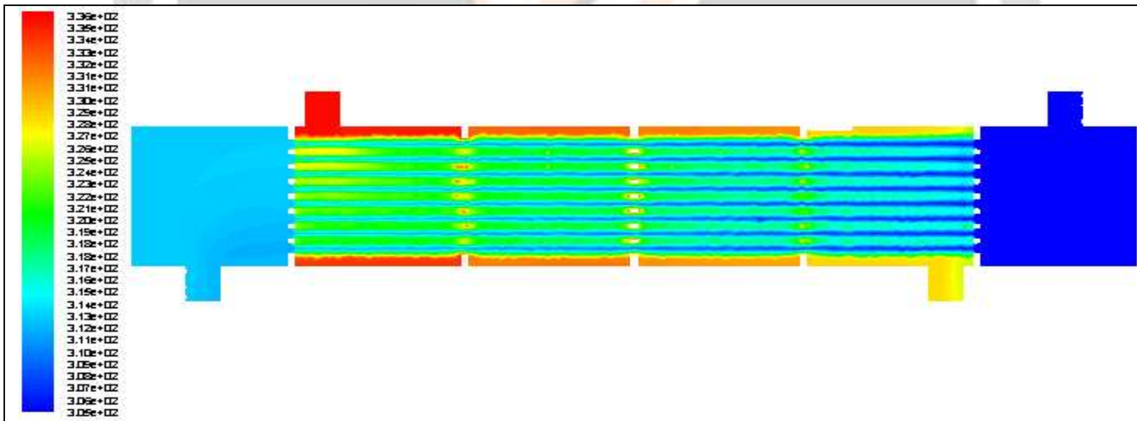


The above figure shows the schematic diagram of the experimental setup. In the setup the provision to record the temperature of fluid on shell side as well as tube side were made. Along with this the u tube manometers are fitted to measure the pressure difference on the both sides of the baffles. During the experiment different values of mass flow rate on shell side and tube side were adjusted. For the adjusted flow rates the temperature and pressure readings were recorded as below

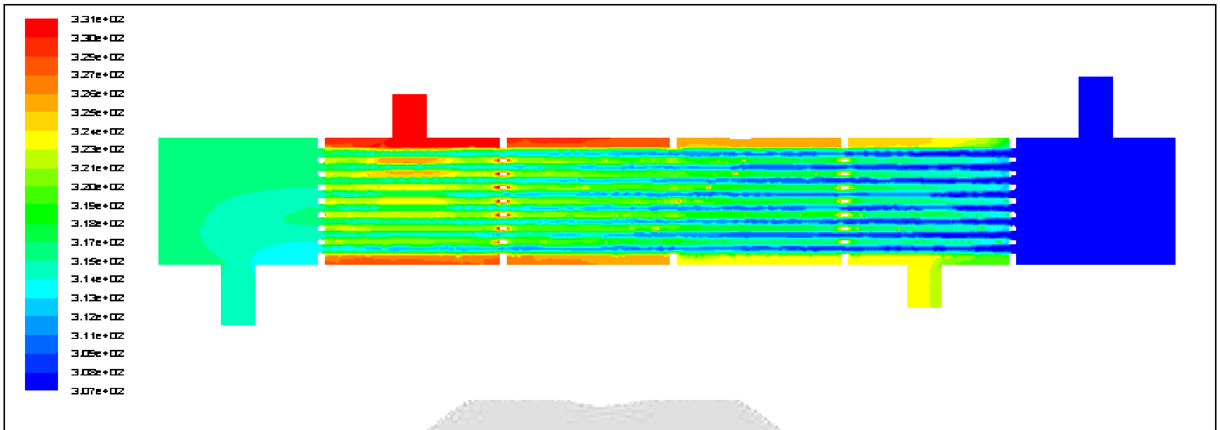
3.1 Temperature

Case No.	Mass flow rate in Shell (Kg/s)	Mass flow rate in Tube side (Kg/s)	Hot waterinlet temp.(Thi) (K)	Hot water outlet temp. (Tho) (K)	Coldwaterinlet temp.(Tci) (K)	Coldwateroutlettemp.(Tci) (K)	Ambienttemp. (Tatm.) (K)
1.	0.2852	0.3414	336	329.5	305.5	310.5	305
2.	0.25153	0.25742	331	326	307	311	305
3.	0.3223	0.47416	336	328	303	308	306
4.	0.3977	0.2773	331.5	328	304	310	306
5.	0.3977	0.2719	337.5	332	305.5	313	306
6.	0.3006	0.2092	328	325.5	307	310.5	307

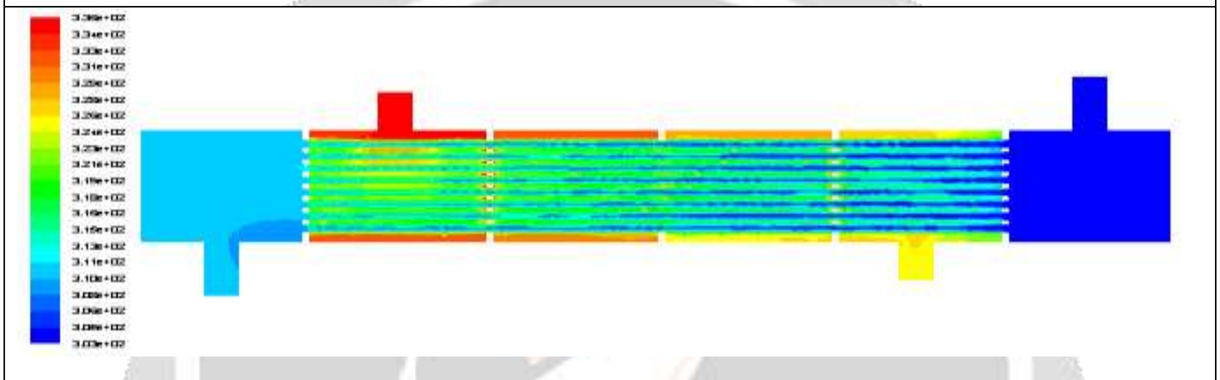
The above set of results were used as the boundary conditions in the CFD analysis to validate the values of the temperature on shell side and tube side and pressure difference across the baffles. Temperature contour for the each case of mass flow rate settings are as below.



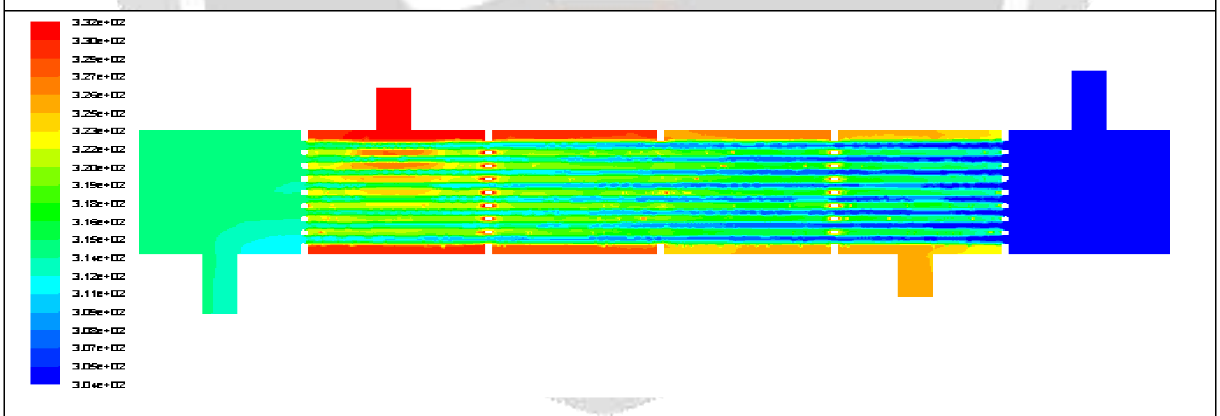
Case 1 Mass flow rate on shell side and tube side 0.2852 & 0.3414 respectively



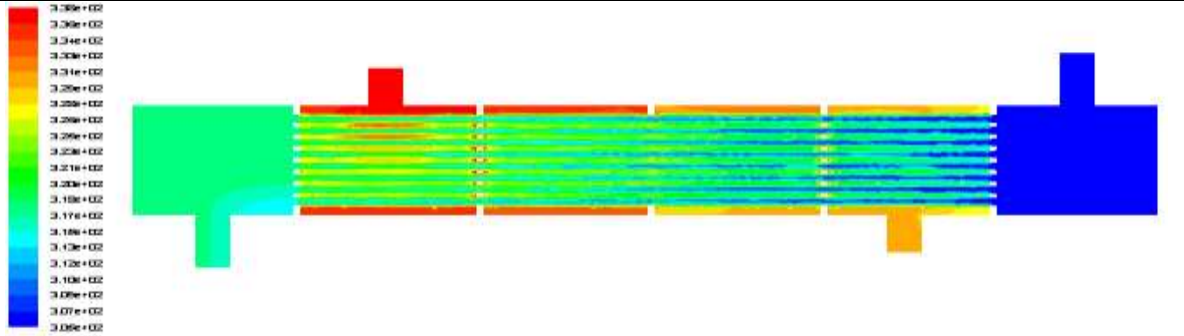
Case 2 Mass flow rate on shell side and tube side 0.2515 & 0.25724 respectively



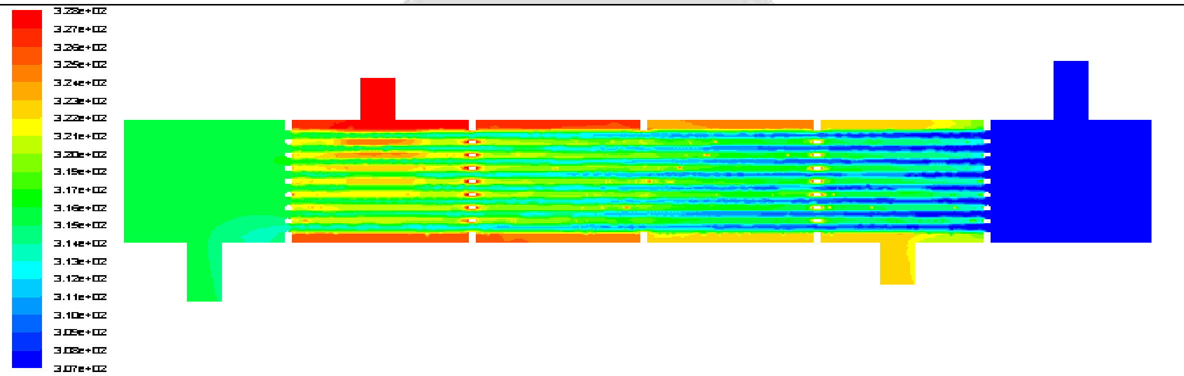
Case 3 Mass flow rate on shell side and tube side 0.3223 & 0.4721 respectively



Case 4 Mass flow rate on shell side and tube side 0.3977 & 0.2771 respectively



Case 5 Mass flow rate on shell side and tube side 0.3977 & 0.2719 respectively



Case 6 Mass flow rate on shell side and tube side 0.3006 & 0.2092 respectively

The pattern of the temperature counters obtained after CFD analysis is in good agreement with the published literature. From the contours it is clearly seen that the mass flow rate variation on inlet and outlet side have reverse impact on the temperature variation on inlet and outlet side. The results for temperature for shell side and tube side obtained from CFD analysis are as below.

Reading No.	Mass flow rate in Shell (Kg/s)	Mass flow rate in Tube side (Kg/s)	Hot wateri nlet temp.(Thi) (K)	Hot water outlet temp.(Tho)(K)	Cold water inlet temp. (Tci)(K)	Cold water outlet temp.(Tci) (K)
1.	0.2852	0.3414	336	328	305.5	312.5
2.	0.25153	0.25742	331	324	307	315
3.	0.3223	0.47416	336	326.5	303	310.5
4.	0.3977	0.2773	331.5	325.3	304	313.5
5.	0.3977	0.2719	337.5	329.5	305.5	316.6
6.	0.3006	0.2092	328	322	307	315.7

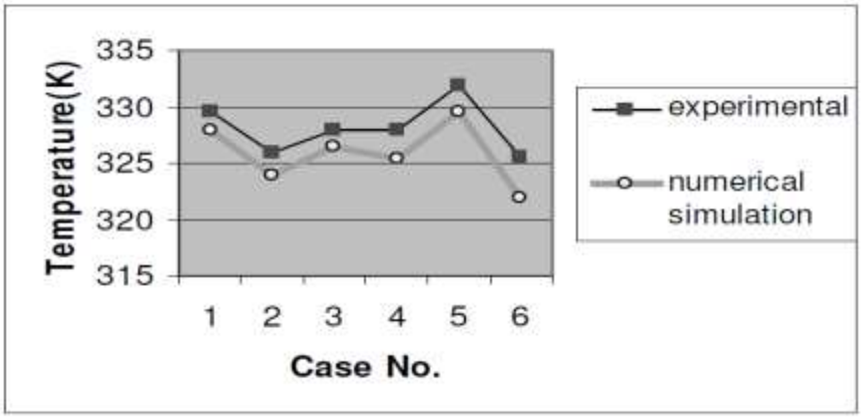


Chart 1 Experimental and CFD Variation of Shell Side temperature

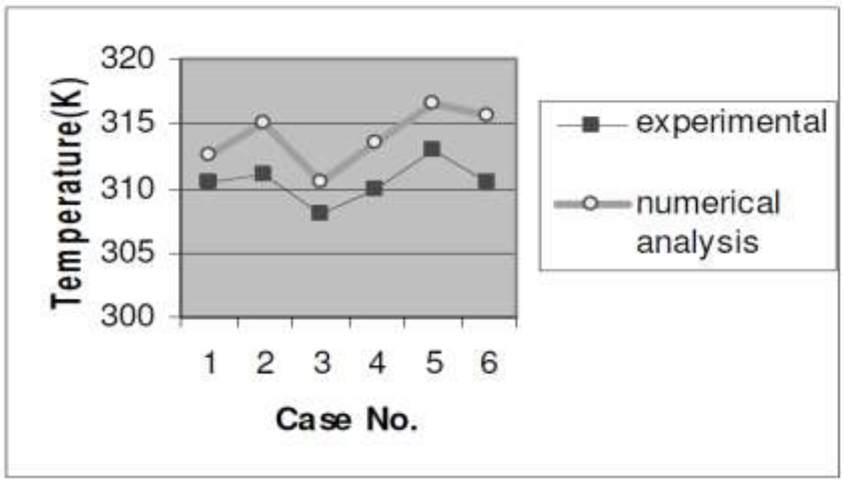


Chart 2 Experimental and CFD Variation of Tubel Side temperature

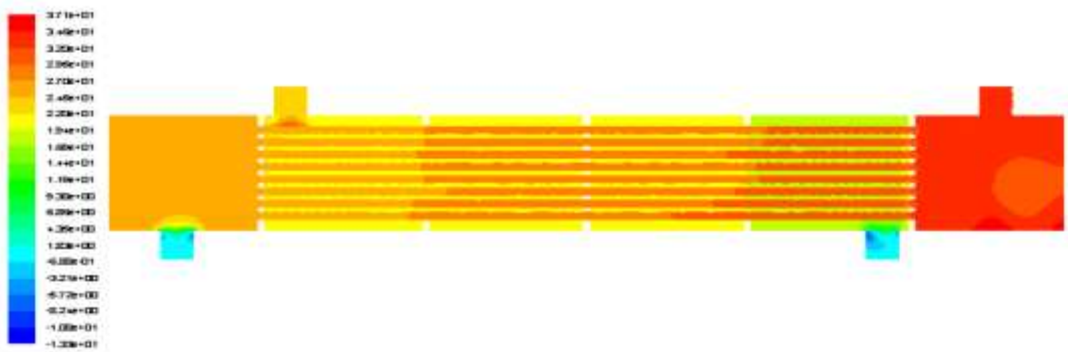
The variation between experimental and simulation results can be seen in above Figs .This is due to fact that heat loss to atmosphere and fouling factor of heat exchanger are not considered in CFD simulation. But it can be seen that the profile followed in both cases matches with the experimental results and hence CFD simulation can be used to predict the behavior of heat exchanger for different baffle geometry and boundary conditions.

3.2 Pressure Variation across Baffles

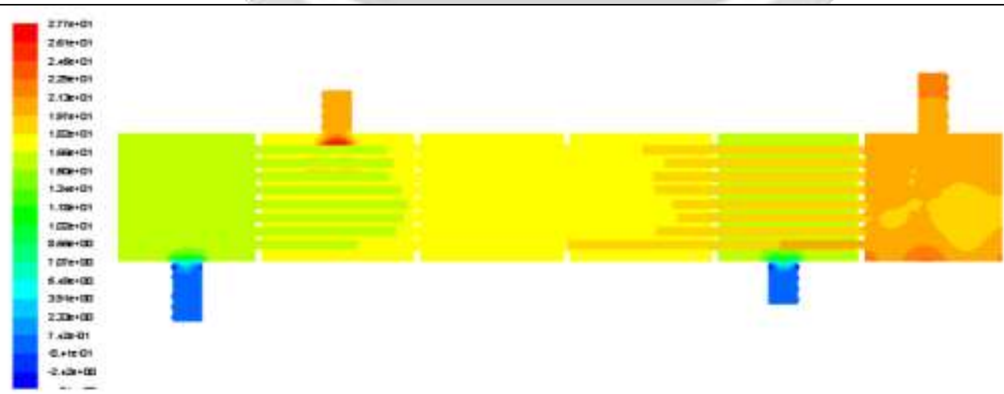
For recording the pressure difference across the baffles u tube manometers were used. The various pressure readings for corresponding values for different values of mass flow rate are tabulated below

Reading No.	Mass flow rate in Shell(Kg/s)	Mass flow rate in Tube side(Kg/s)	P. D. Ist Baffle (mm H2O)	P. D. IInd Baffle (mm H2O)	P. D. IIIrd Baffle (mm H2O)	P. D. Shell side (mm H2O)	P. D. Tube side (mm H2O)
1.	0.2852	0.3414	29	21	105	230	35
2.	0.25153	0.25742	11	15	34	220	23
3.	0.3223	0.47416	18	10	105	250	55
4.	0.3977	0.2773	10	7	125	290	25
5.	0.3977	0.2719	10	7	35	290	25
6.	0.3006	0.2092	13	18	83	202	21

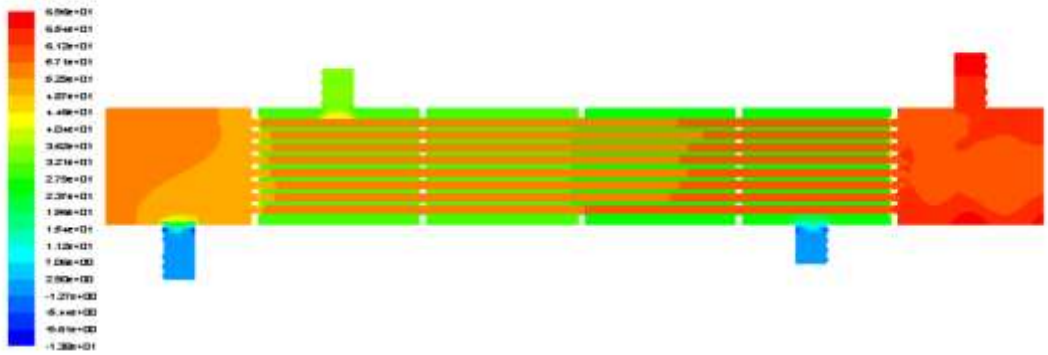
From the pressure contours as shown below it is clear that maximum pressure drop in the shell takes place when fluid goes through inlet/outlet nozzle. Across each baffle some pressure drop takes place. In between the two baffles the pressure drop is almost negligible as more flow area is available. The pressure drop on tube side takes place uniformly along the length of the tube.



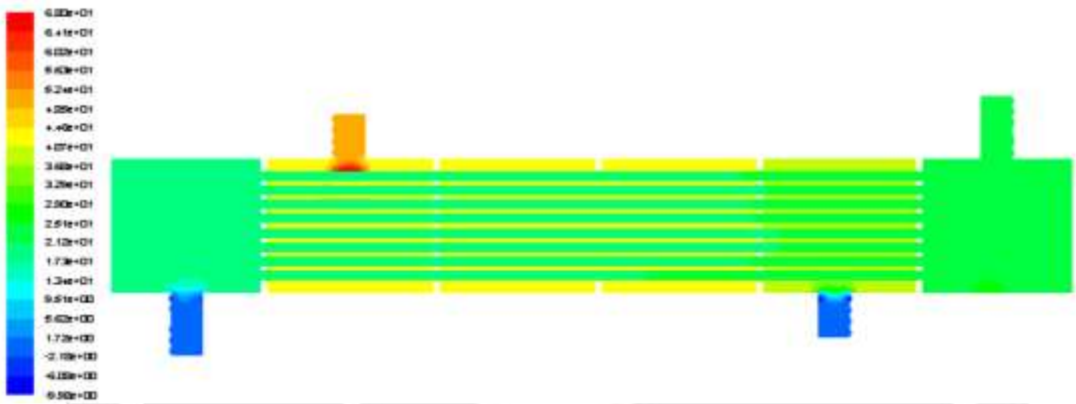
Case 1 Mass flow rate on shell side and tube side 0.2852 & 0.3414 respectively



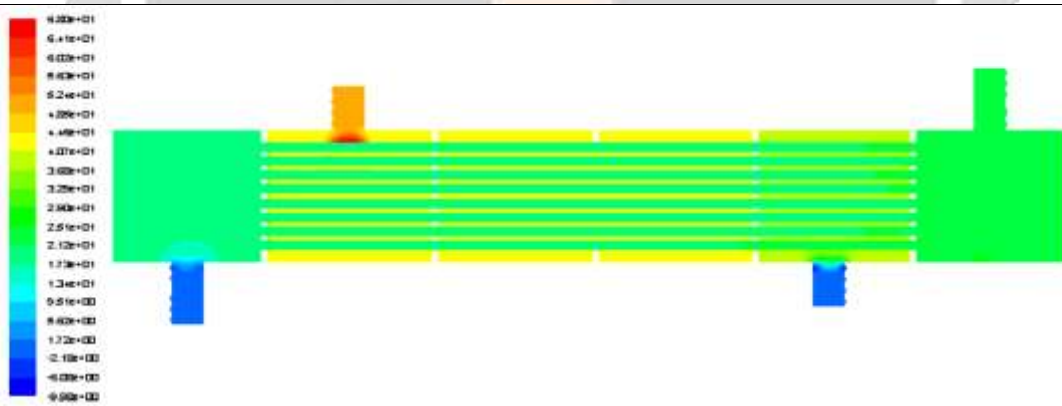
Case 2 Mass flow rate on shell side and tube side 0.2515 & 0.25724 respectively



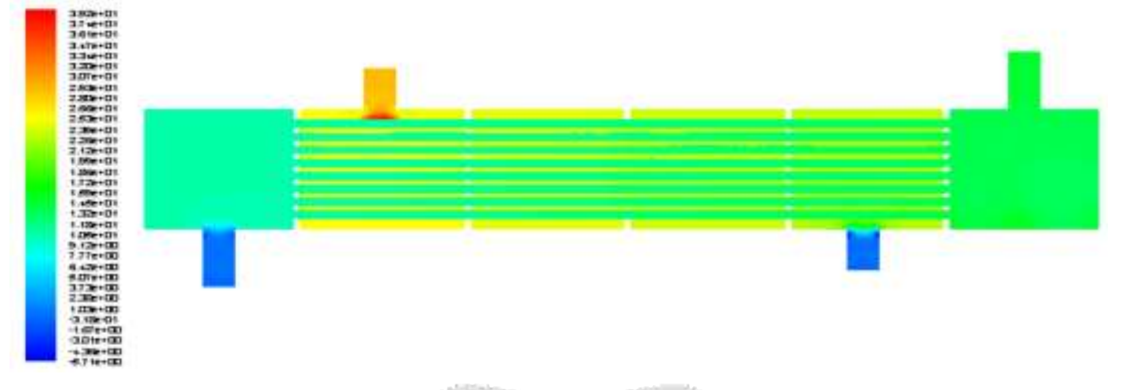
Case 3 Mass flow rate on shell side and tube side 0.3223 & 0.4721 respectively



Case 4 Mass flow rate on shell side and tube side 0.3977 & 0.2771 respectively



Case 5 Mass flow rate on shell side and tube side 0.3977 & 0.2719 respectively



Case 6 Mass flow rate on shell side and tube side 0.3006 & 0.2092 respectively

Though here turbulent boundary conditions are used, from the pressure contour it is clear that as mass flow rate through shell or tube increases corresponding pressure drop is also increases in that region. So it is clear that viscous forces plays an important role in pressure drop in heat exchanger.

4. EFFECT OF BAFFLE SPACING ON THE VARIATION OF TEMPERATURE AND PRESSURE

As the CFD results and experimental results are in good agreement the case formulated for CDF analysis is validated. Same case is used to investigate the effect of baffle spacing on temperature and pressure on shell side and tube side. In this case the effect of baffle spacing on the performance of the slotted baffle heat exchanger is studied. Six case studies are done with baffle spacing of 50mm, 100mm, 150mm, 200mm, 500 mm in five cases and the sixth one is shell without baffle. For all the cases the inlet boundary condition is kept same as given below

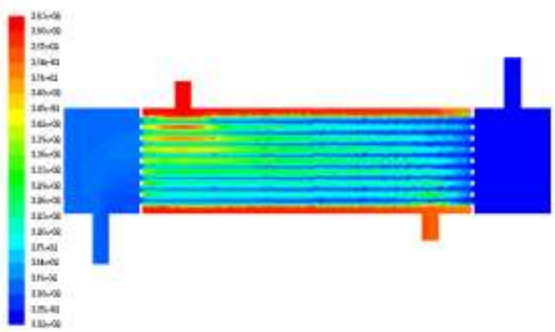
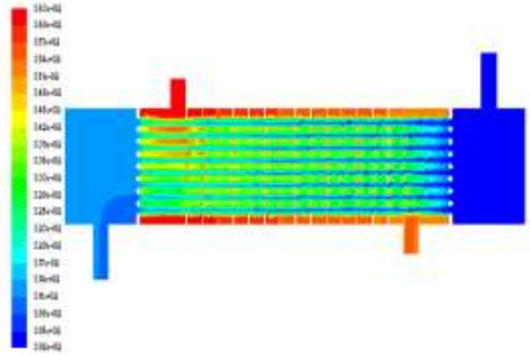
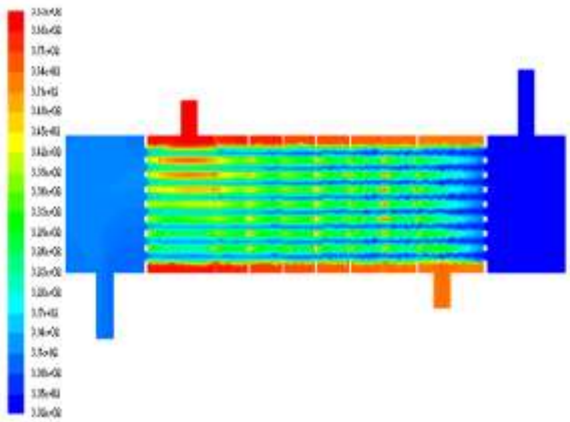
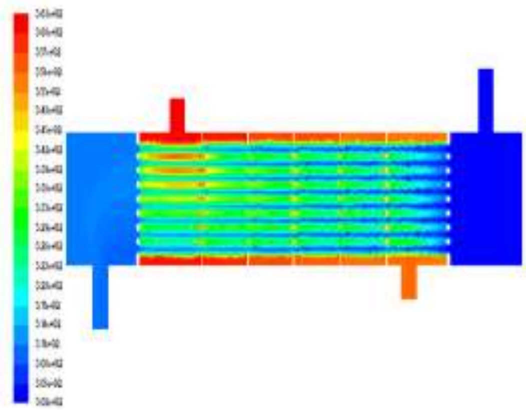
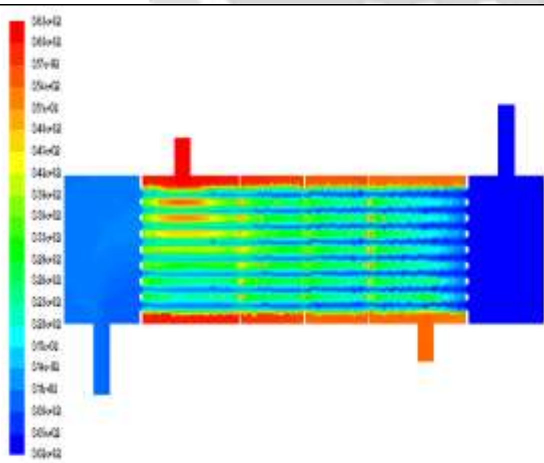
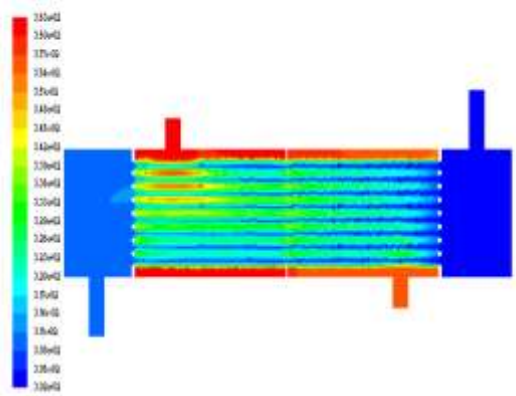
Boundary Conditions

Mass flow rate through shell(Kg/s)	1.5
Mass flow rate through tube(Kg/s)	1.5
Shell inlet temperature(K)	363
Tube inlet temperature(K)	302

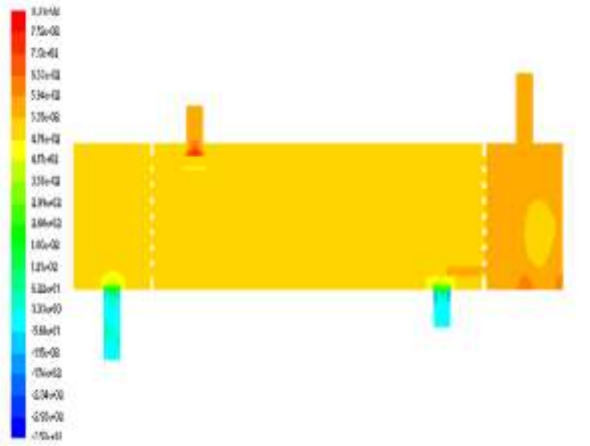
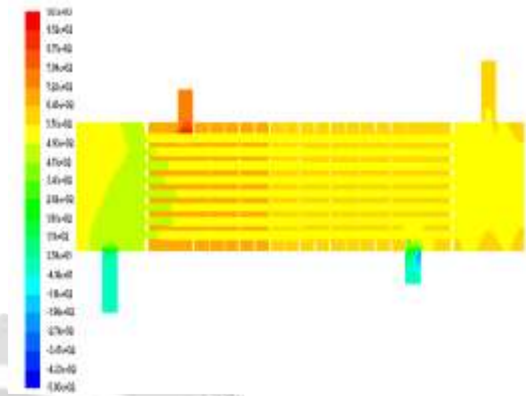
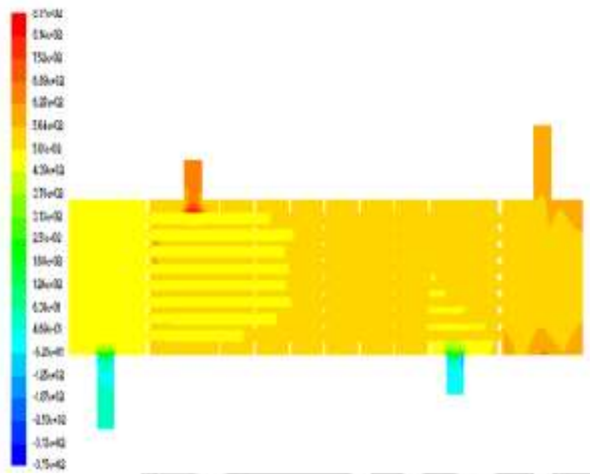
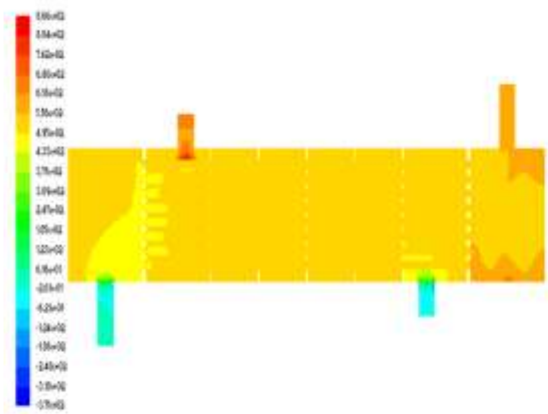
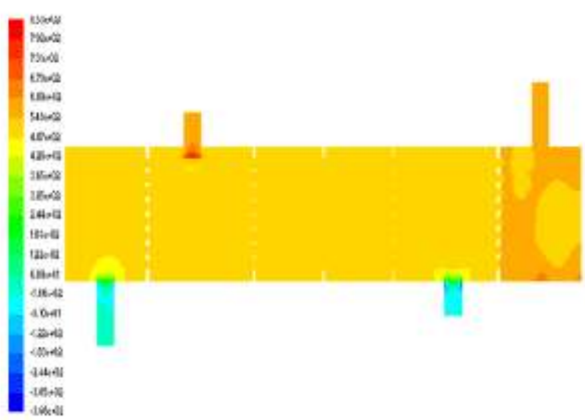
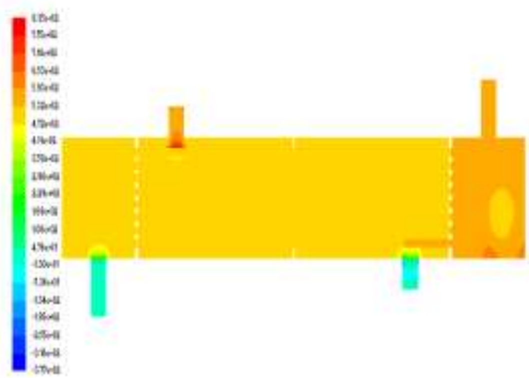
Geometrical condition and physical properties of tube

Tube I.D.(mm)	17.50
Tube O.D.(mm)	19.50
Tube thickness(mm)	1
Thermal conductivity of tube material(K)(W/ m^2K)	51.7
Number of tubes	48
Inlet/Outlet dia. of shell side and tube side	50.8mm

4.1 Temperature Contours

	
Case 1: No Baffle	**Case 2**: Baffle Spacing 50 mm
	
Case 3: Baffle Spacing 100 mm	**Case 4**: Baffle Spacing 150 mm
	
Case 5: Baffle Spacing 200 mm	**Case 6**: Baffle Spacing 500 mm

4.2 Pressure Contours

	
Case 1: No Baffle	**Case 2**: Baffle Spacing 50 mm
	
Case 3: Baffle Spacing 100 mm	**Case 4**: Baffle Spacing 150 mm
	
Case 5: Baffle Spacing 200 mm	Case 6: Baffle Spacing 500 mm

Temperature contour, pressure contour and pressure drop graphs for different baffle spacing is shown in Fig. It is observed that there is a pressure drop taking place after each baffle in the shell. Relatively larger pressure drop takes place where water enters in to the shell. For baffle spacing of 50 mm, pressure drop is more than all other baffle spacing. As the water passes through the baffle, due to reduction in flow area, velocity gets increased. This causes turbulence in that region so more heat transfer takes place there. But when the baffle spacing is 50 mm, more heat transfer takes place due to continuation of turbulence intensity from one baffle compartment to next baffle compartment. In 150 mm as well as 200 mm baffle spacing pressure drop is almost same similarly for 500 mm baffle spacing & shell without baffle pressure drop is same.

5. EFFECT OF MODIFIED SLOTTED SEGMENTED BAFFLE ON PERFORMANCE OF HEAT EXCHANGER

In this case the effect of baffle geometry on the performance of the shell and tube type heat exchanger is studied. For that four slotted baffle and four 28.833 baffle cut single segmental baffle is simulated and the results of the simulation are compared. For both the cases the inlet boundary condition is kept same as given below.

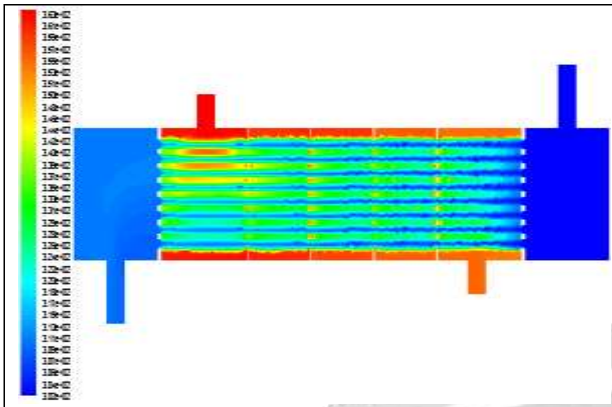
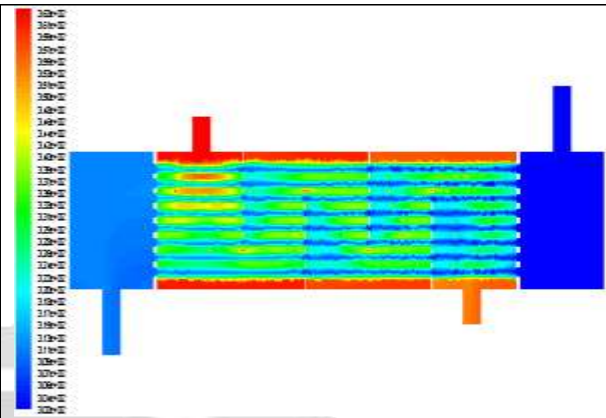
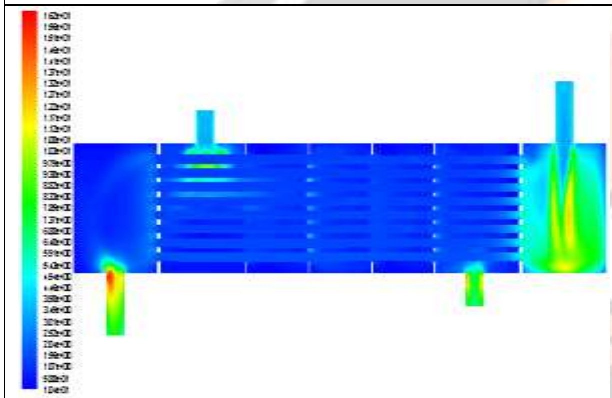
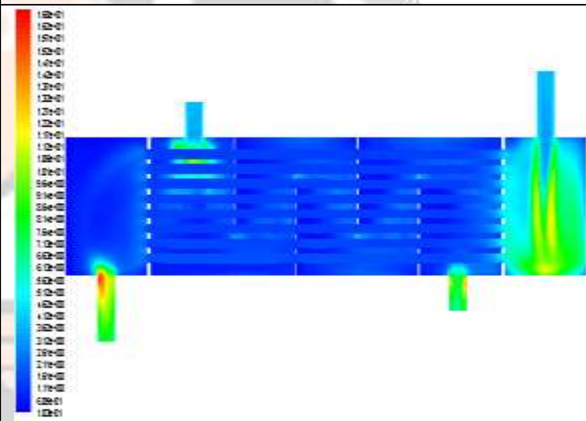
Boundary Conditions

Mass flow rate through shell(Kg/s)	1.5
Mass flow rate through tube(Kg/s)	1.5
Shell inlet temperature(K)	363
Tube inlet temperature(K)	302

Geometrical condition and physical properties of tube

Tube I.D.(mm)	17.50
Tube O.D.(mm)	19.50
Tube thickness(mm)	1
Thermal conductivity of tube material(K)(W/ m^2K)	51.7
Number of tubes	48
Inlet/Outlet dia. of shell side and tube side	50.8mm

Number of tubes	48
Baffle cut for single segmental baffle	28.833
Number of baffles	4
Baffle spacing	175mm.

	
Temperature Variation for Slotted Baffle	Temperature Variation for segmented Baffle
	
Pressure Variation for Slotted Baffle	Pressure Variation for segmented Baffle

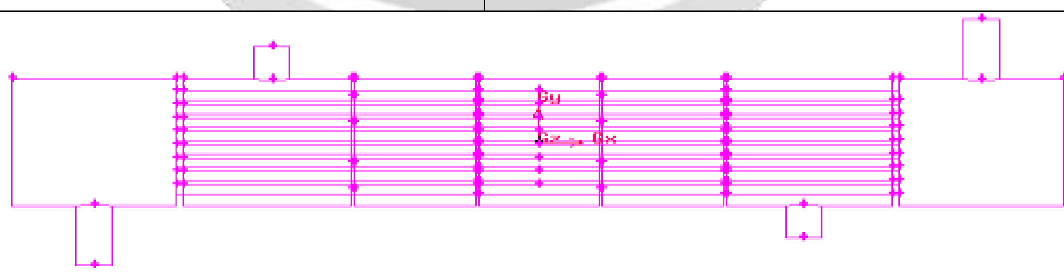


Fig.1 Heat Exchanger with Four Slotted Baffles

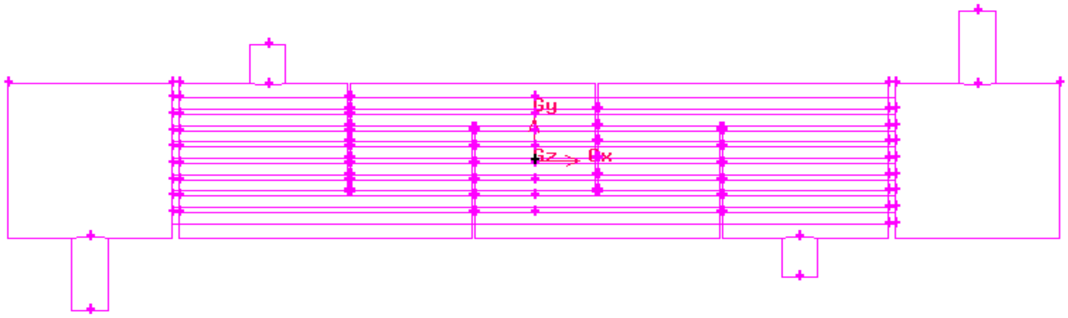


Fig.2 Heat Exchanger with Four modified Baffles

From fig it is clear that in segmental baffle more total pressure drop, turbulence and heat transfer takes place. From the turbulence intensity contour, as shown in fig. in case of more turbulence is created at the tip of the segmental baffle and dead zones is created on the back face of baffle whereas in slotted baffle the turbulence intensity is uniform throughout the shell and no dead zone is observed. Comparative study between slotted baffle and segmental baffle is given in table below.

Baffle Geometry	Shell inlet temp. (K)	Shello utlette mp. (K)	Tube inlet temp. (K)	Tube out let temp.(K)	Over all heat transfer coefficient(Uo) (W/ m^2K))
Slotted baffle	363	355.5	302	309.5	304.708
Segmental baffle (28.73baffle cut)	363	354	302	311	370.8955

Though the heat transfer rate in segmental baffle is more than slotted baffle it is clear from pressure graph that the total pressures drop in the segmental baffle is much higher than the slotted baffle. In slotted baffle pressure drop is just 33% of the segmental baffle while total heat transfer rate is about 82% of the segmental baffle.

6. CONCLUSIONS

The present work is focused on shell and tube heat exchanger with slotted baffle. During the course of dissertation, an attempt is made for geometry study of shell and tube heat exchanger with Slotted baffle. The pressure and temperature variation through the length of heat exchanger are obtained by flow simulation tool – Fluent 6.0. The validation of CFD results is achieved by comparison with Fluent 6.0 results and that through actual experiments.

The silent features of the present work may be summarized as follows.

1. The failure of tubes due to flow induced tube vibration problem is reduced due to; two point contact of tubes with Slotted baffle.
2. The outlet temperature profile followed in both experimental and numerical analysis is same but there is some variation between both results because heat loss to atmosphere and fouling factor is not consider in numerical simulation.
3. During the flow simulation, the flow is continuing in shell domain, so the problem of formation of dead zone is eliminated. This will reduce the fouling on shell side due to the elimination of low flow velocities.

4. As the number of baffle increases we can achieve more turbulence, pressure drop in shell side and better heat transfer coefficient.
5. Based on the results from present analysis and parametric studies it may be concluded that Slotted baffle can be used for the liquid/liquid and gas /gas case in process industry and the heat exchanger can be optimized with CFD analysis as suggested herein.
6. As the pressure drop across the slotted baffle is very less we can arrange more baffles in the shell to maintain turbulence in the shell and thereby improve heat transfer.

7. REFERENCES

[1] M. Prithiviraj, M. J. Andrews, "Three dimensional numerical simulations of shell & tube heat exchangers. Part I: Foundation & fluid mechanics.", Numerical Heat Transfer, Part - A, Vol. 33, pp.799-816, 1998.
[2] R.H. Clarke, F. Nicolas, "Computational fluid dynamics Investigation of Maldistribution Effects on Crude-oil fouling in Shell and Tube Exchangers.", Cal Gavin Ltd., Alcester, UK, 2001.
[3] Q.W. Dong, Y.Q. Wang, M.S. Liu, "Numerical & experimental investigation of shell side characteristics for ROD baffle heat exchanger.", Applied in Thermal Engineering, 2007.
[4] Himanshu D. Solanki, Dr. S.A.Channiwala, Mr. M.Venkatesh, "Heat transfer & flow analysis of shell & tube heat exchanger without baffle.", National conference AFFTS-2008, pp.112-118, 2008.
[5] S.V. Mokamati, R.C. Prasad, "Numerical Simulation of Fluid Flow & Heat Transfer in a Concentric Tube Heat Exchanger". Numerical Heat Transfer, Vol. 31, pp.612-618, 1998.