International Journal of Inventive Engineering and Sciences (IJIES)
ISSN: 2319–9598, Volume-4 Issue-6, June 2017

Analytical and Numerical Simulation of Shell and Tube Heat Exchanger

Shivaprakash K S, Devaraja A.S, Mohan A. E, Vijetha Vardhan R. N.

Abstract: Shell and tube heat exchanger is the most common type of heat exchanger and widely used in oil refinery and other large chemical processes because it suite for high pressure application. In this project design of shell-and-tube heat exchanger is developed using analytical method. Then the CFD simulation will be carried out. Computational Fluid Dynamic (CFD) is a useful tool in solving and analysing problems that involve fluid flows, in this process in solving the simulation consist of modeling and meshing the basic geometry of shell and tube heat exchanger using the CFD package ANSYS Meshing Then, the boundary condition will be set before been simulate in Fluent 12.1 based on the analytical parameters. Then the CFD simulation of heat transfer in shell and tube heat exchanger model is validated against analytical method. Finally the optimization of baffle will be carried out to improve the stagnant zone and heat transfer in the segmented Shell and Tube Heat Exchanger.

Keywords: computational fluid dynamics, Ansys 12.1, Shell and tube heat exchanger, Baffles cuts and Baffles orientation

I. INTRODUCTION

Heat exchangers are one of the mostly used equipments in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involves cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purposes. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfill the heat transfer requirements. The purpose of a heat exchanger is just the exchange of heat.

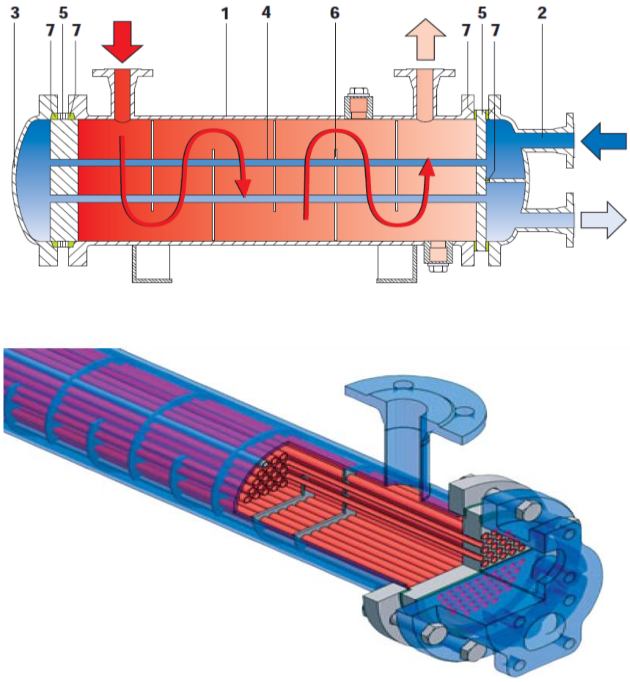


FIG-1 1 Heat exchanger shell 2 Connection chamber 3 Guide chamber 4 Internal tubes 5 Tube sheets 6 Baffles 7 Apparatus seal

There are three principle means of achieving heat transfer, conduction, convection, and radiation. Heat exchangers run on the principles of convective and conductive heat transfer. Radiation does occur in any process. However, in most heat exchangers the amount of contribution from radiation is miniscule in comparison to that of convection and conduction. Conduction occurs as the heat from the hot fluid passes through the inner pipe wall. To maximize the heat transfer, the inner-pipe wall should be thin and very conductive. However, the biggest contribution to heat transfer is made through convection.

There are two forms of convection; these are natural and forced convection. Natural convection is based on the driving force of density, which is a slight function of temperature. As the temperature of most fluids is increased, the density decreases slightly. Hot fluids therefore have a tendency to rise, displacing the colder fluid surrounding it. This creates the natural "convection currents" which drive everything from the weather to boiling water on the stove. Forced convection uses a driving force based on an outside source such as gravity, pumps, or fans. Forced convection is much more efficient, as forced convection flows are often turbulent.

Revised Version Manuscript Received on May 23, 2017.

Shivaprakash K. S, Assistant Professor, Department of Mechanical Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore, India, E-mail: shivaprakash72@gmail.com

Dr. Devaraja A. S, Professor & Head, Department of Mechanical Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore, India, E-mail: mech.hod80@gmail.com

Mohan A. E, Assistant Professor, Department of Mechanical Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore, India, E-mail: aemohan999@gmail.com

Vijetha Vardhan R. N, Assistant Professor & Head, Department of Mechanical Engineering, East West College of Engineering, Bangalore, India, E-mail: racharlavv@gmail.com

Turbulent flows undergo a great deal of mixing which allow the heat to be transferred more quickly.

BAFFLING: Type of baffles: It is apparent that higher heat transfer coefficient result when a liquid is maintained in a state of turbulence. To induce turbulence outside the tubes it is customary to employ baffles which cause the liquid to flow through the shell at right angle to axes of the tubes. This cause considerable turbulence even when a small quantity of liquid flows through the shell. Baffles are used to support tubes, enable a desirable velocity to be maintained for the shell side fluid, and prevent failure of tubes due to flow-induced vibration. There are two types of baffles: plate and rod. Plate baffles may be single-segmental, double segmental, or triple-segmental, as shown in Figure 2.

Baffle spacing: Baffle spacing is the centreline-to-centreline distance between adjacent baffles. It is the most vital parameter in STHE design. The TEMA standards specify the minimum baffle spacing as one-fifth of the shell inside diameter or 2 in., whichever is greater. Closer spacing will result in poor bundle penetration by the shell side fluid and difficulty in mechanically cleaning the outsides of the tubes. Furthermore, low baffle spacing results in a poor stream distribution.

The maximum baffle spacing is the shell inside diameter. Higher baffle spacing will lead to predominantly longitudinal flow, which is less efficient than cross-flow, and large unsupported tube spans, which will make the exchanger prone to tube failure due to flow-induced vibration.

Optimum baffle spacing. For turbulent flow on the shell side (*Re* > 1,000), the heat-transfer coefficient varies to the 0.6–0.7 power of velocity; however, pressure drop varies to the 1.7–2.0 power. For laminar flow (*Re* < 100), the exponents are 0.33 for the heat-transfer coefficient and 1.0 for pressure drop. Thus, as baffle spacing is reduced, pressure drop increases at a much faster rate than does the heat-transfer coefficient. This means that there will be an optimum ratio of baffle spacing to shell inside diameter that will result in the highest efficiency of conversion pressure drop to heat transfer. This optimum ratio is normally between 0.3 and 0.6.

Baffle cut: As shown in Figure 2, baffle cut is the height of the segment that is cut in each baffle to permit the Shell side fluid to flow across the baffle. This is expressed as a percentage of the shell inside diameter. Although this, too, is an important parameter for STHE design, its effect is less profound than that of baffle spacing.

Baffle cut can vary between 15% and 45% of the shell inside diameter. Both very small and very large baffle cuts are detrimental to efficient heat transfer on the shell side due to large deviation from an ideal situation, as illustrated in Figure 2.7. It is strongly recommended that only baffle cuts between 20% and 35% be employed. Reducing baffle cut below 20% to increase the shell side heat-transfer coefficient or increasing the baffle cut beyond 35% to decrease the shell side pressure drop usually lead to poor designs.

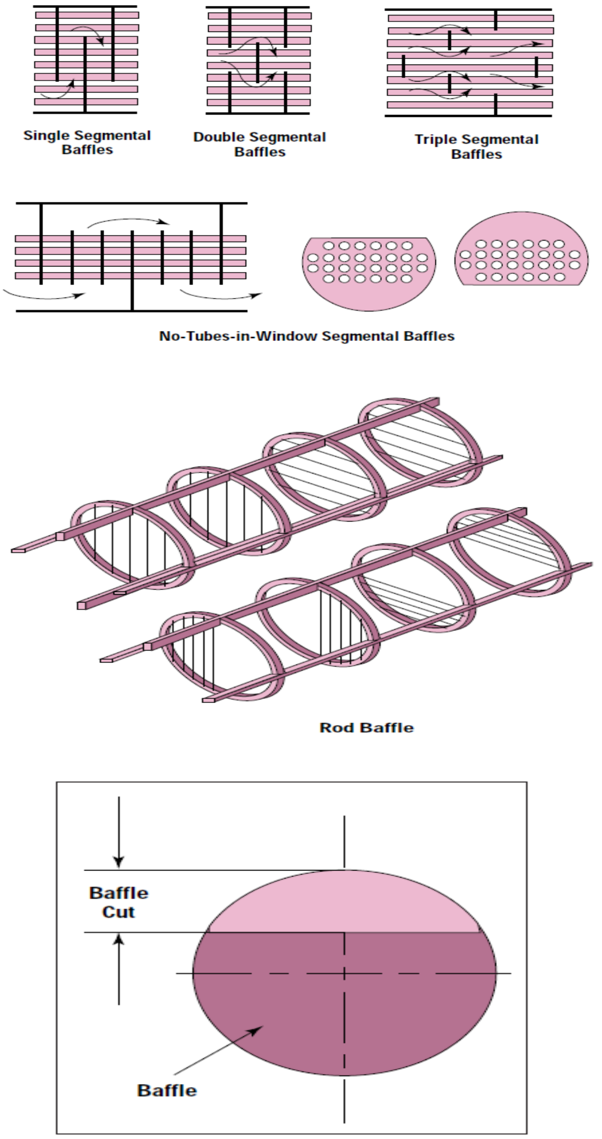


FIG - 2. Types of Baffles & Baffle Cuts

II. OBJECTIVE

1. Analytical study of shell & tube Exchanger
2. To perform CFD simulation of single phase and single segmental baffled shell-and-tube heat exchanger with variable number of baffle and baffle cut by using commercial CFD package FLUENT 12.0.
3. Analyse the heat transfer coefficient and pressure drop on shell side by comparing CFD results with the analytical result.
4. Validate simulation results to analytical results.

III. METHODOLOGY & ANALYTICAL CALCULATIONS

1. Geometric modelling of shell and tube heat exchanger as per reference using CAD software CATIA [1].
2. CFD meshing using ANSYS Meshing.
3. Transient CFD analysis of shell and tube heat exchanger with using ANSYS Fluent using water as fluid.
4. Validate the transient temperature results with available experimental data.

Design methods of shell and tube heat exchangers

First step in designing of heat exchanger, there is two way to design heat exchanger.

1. LMTD
2. NTU Method.

General equation of heat exchanger is

Q = UAF ΔTM (4.1)

Where ΔT is the Temperature difference between hot and cold fluid

In terms of energy flow for heat exchanger, we can use this equation for hot fluid,

Q= -MC$_P\Delta$TH (4.2)

Where ΔT is the Temperature difference between hot fluids

In terms of energy flow for heat exchanger, we can use this equation for cold fluid,

Q= MCpΔTC (4.3)

Where ΔT is the Temperature difference between cold fluids

A. Log Mean Temperature Difference Method

Heat flows between the hot and cold streams due to the temperature difference across the tube acting as a driving force. The difference will vary with axial location. Average temperature or effective temperature difference for either parallel or counter flow may be written as:

$$\text{LMTD } (\Delta TM) = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1/\Delta T_2)} \quad (4.4)$$

Normal practice is to calculate the LMTD for counter flow and to apply a correction factor F_T, such that CORRECTION FACTOR:

$$R = T_1\text{-}T_2/t_2\text{-}t_1 \quad s = t_2 - t_{1/}\,T_1 - T_2 \quad (4.5)$$

$$((\Delta T)_{\text{CORRECTION FACTOR}} = F_T.\text{LMTD}$$

The correction factors, F_T, can be found theoretically and presented in analytical form. The equation given below has been shown to be accurate for any arrangement having 2, 4, 6… 2n tube passes per shell pass.

$$F_{1\text{-}2} = [\sqrt{R^2+1}/R\text{-}1]\text{Ln}(1\text{-}p/1\text{-}pr)/\text{Ln}[2/p\text{-}1\text{-}R+\sqrt{R^2+1}/2/P\text{-}1\text{-}R\text{-}\sqrt{R^2+1}] \quad (4.6)$$

4.3 Effectiveness-NTU Method

In the thermal analysis of shell-and-tube heat exchangers by the LMTD method, an equation (4.1) has been used. This equation is simple and can be used when all the terminal temperatures are known. The difficulty arises if the temperatures of the fluids leaving the exchanger are not known. In such cases, it is preferably to utilize an altogether different method known as the effectiveness-NTU method. Effectiveness of shell-and-tube heat exchanger is defined as:

The group **UA/ C$_{min}$** is called number of transfer units, NTU.

PRESSURE DROP: The shell side pressure drop [20] is calculated as a summation of the pressure drops for the inlet and exit sections (ΔP_e), the internal cross flow sections (ΔP_c), and the window sections (ΔP_w). For a shell-and-tube exchanger, the combined pressure drop is given as:

SHELLSIDE

1) $$\Delta P_S = \frac{f*G_g^{2*}D_g*(N+1)}{5.22*10^{10}*D_S*\Phi_{S*}S}$$

TUBE SIDE

1) $$\Delta P_t = \frac{f*G_t^{2}*L*N}{5.22*10^{10}*D_{T*}S*\Phi_t}$$

S=1for water

n= no. of passes L=length

D_S=ID/12

Return crosses $\Delta P_r=4*n/s*v^2/2g$

No. of crosses (N+1) =12L/B

Total pressure drop of entire tube $\Delta P_T=\Delta P_t+\Delta P_r$

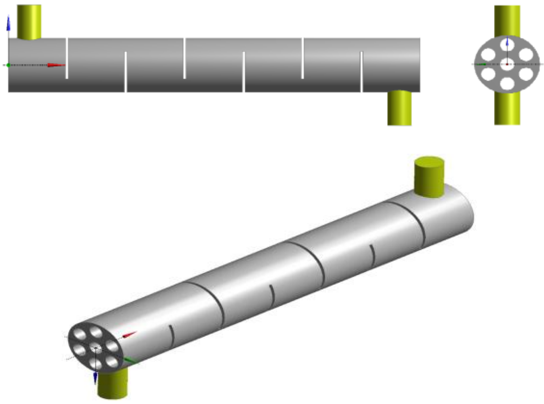


Fig- 3 Geometric Model of SATHE

IV. RESULTS AND DISCUSSION

Type the results and discussion here. Follow the guidelines given for forming the Figures and Tables.

In this study, small heat exchanger is selected in order to increase the model detail and to make solid observations about the flow inside the shell. Some of the design parameter and predetermined geometric parameter are presented in the table1. The Geometric model with 6 baffles is shown in Fig 3. Two different baffle cut value is selected: 25% baffle cut is common in shell-and-tube heat exchanger design; whereas 36% baffle cut value is selected to place the cut just below or above central row of tubes. The working fluid for the shell is water.

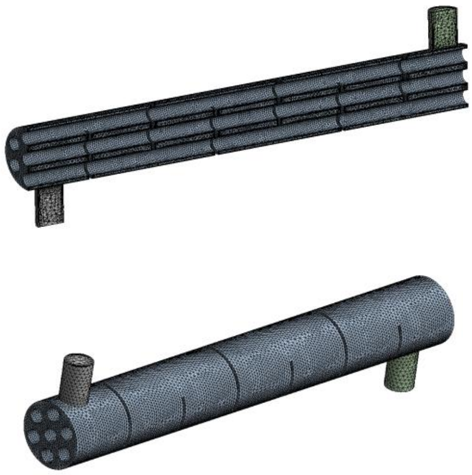
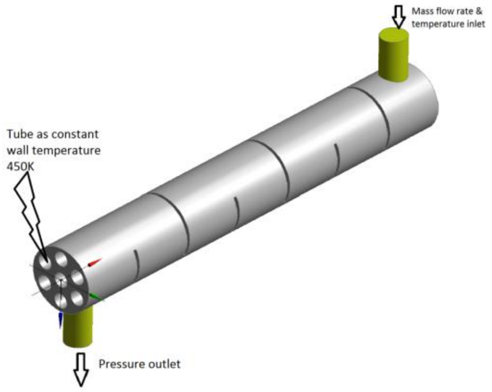


Fig.4. CFD Meshing details

The shell inlet temperature is set to 300K. Zero gauge pressure is assigned to the outlet nozzle, in order to obtain the relative pressure drop between inlet and outlet. The inlet velocity profile is assumed to be uniform. No slip condition is assigned to all surfaces. The zero heat flux boundary condition is assigned to the shell outer wall, assuming the shell is perfectly insulated outside.

Since tube side flow is easy to resolve, the present study concentrated on the shell side flow. After modeling the tubes as solid cylinders, the constant wall temperature of 450K is assigned to the tube wall.



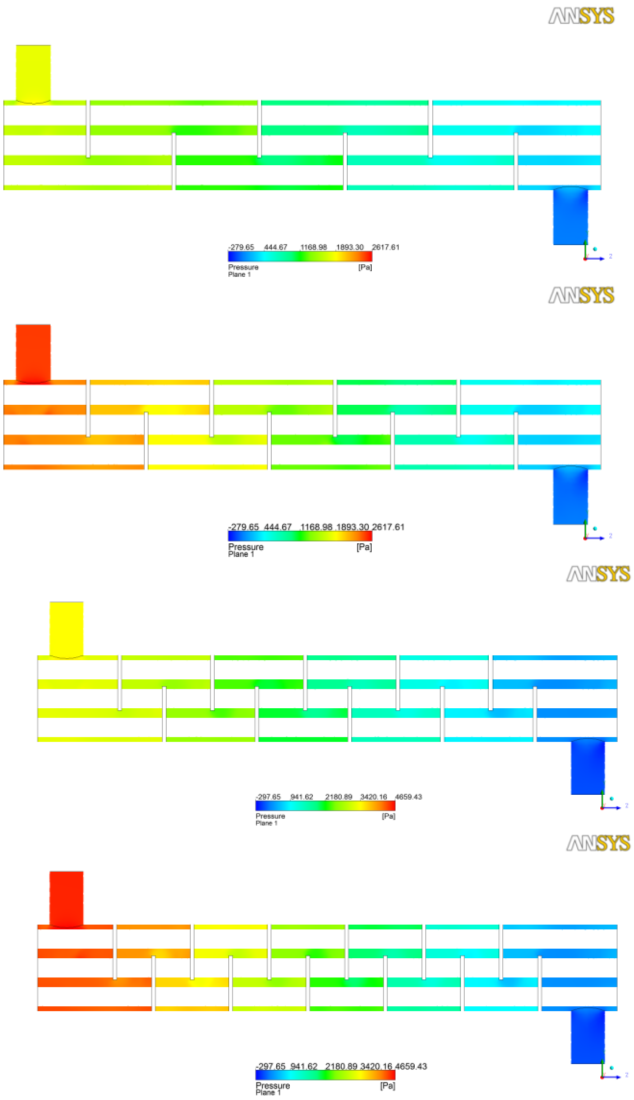
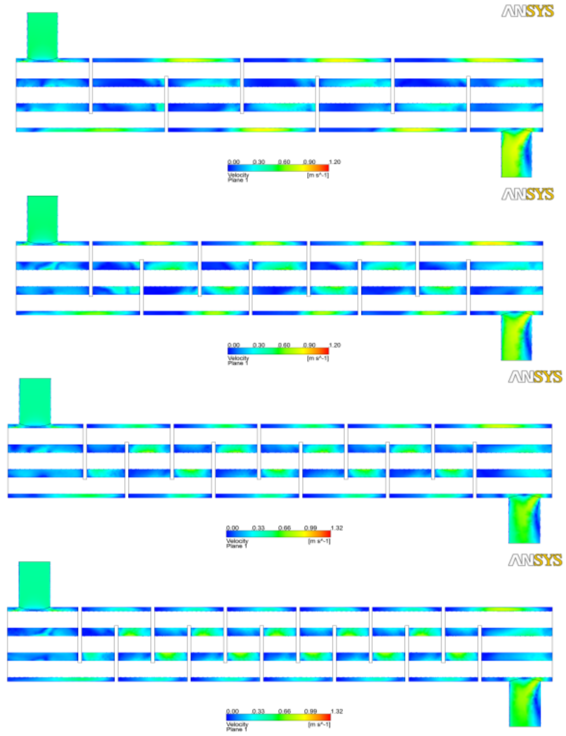
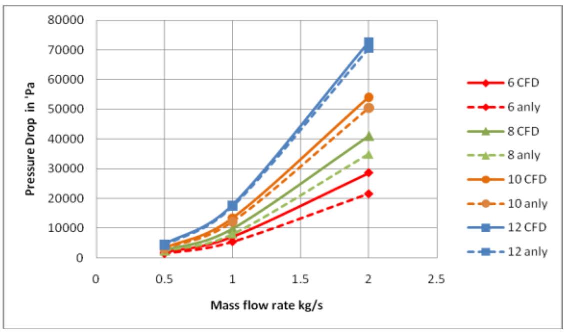


Fig.6. Contours of static Pressure at Vertical-Mid plane on shell side when Bc=36%, N_b 6,8,10,12

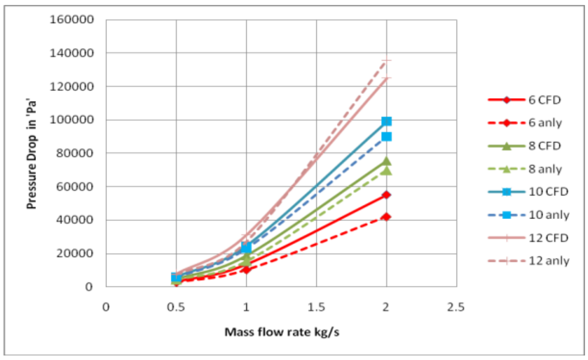
In all of the preliminary simulation, flow inside the shell is observed to be turbulent, viscous model is selected to be K-ε turbulent model. Sensitivity of the result is investigated using the heat exchanger model with 6, 8, 10 & 12 baffles for two different baffle cut which is shown in above figure 6 and 7.



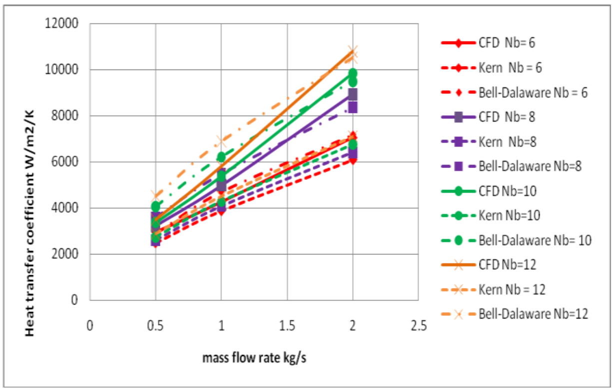
. Fig.7 Contours of velocity at Vertical-Mid plane on shell side when Bc=36%, N_b 6,8,10,12



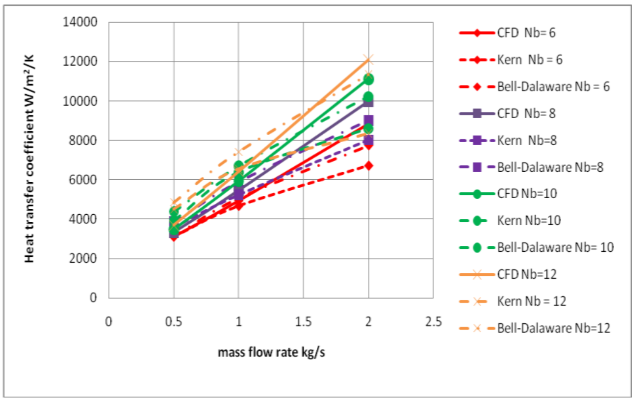
Pressure drop of CFD result with respect to analytical results for different baffle spacing B_c=36%



Pressure drop of CFD result with respect to analytical results for different baffle spacing B_c=36%



Heat transfer coefficient value of CFD result with respect to analytical results for different baffle spacing when 25% baffle cut



Deviation Heat transfer coefficient value of CFD result with respect to analytical results for different baffle spacing when 25% baffle cut

V. CONCLUSIONS

The effect of baffle spacing on the heat transfer and pressure drop are investigated for four different number of baffles (N_b) with 36% baffle cut. The corresponding central baffle spacing and B/D_s ratio values are presented in table. After adjusting the baffle spacing or N_b,

The percentage difference between the analytical calculation and CFD analysis result by taking the analytical one as are presented in the table. The corresponding data set for 6, 8, 10, and 12 baffles are given in table. By decreasing the baffle spacing (increasing N_b), the arrangement of the result is improved, as expected. In the overall heat transfer calculation, percentage difference with the kern method shows no improvement. For 8, 10, 12 baffles, the percentage difference result with respect to the Bell-delaware method are improved. For 0.5 kg/s and 1kg/s mass flow rates, the percentage decreases below 10%, by adjusting B/D_s ratio, but for 2 kg/sec the percentage difference is still high. The pressure drop result of the CFD analyses are also improved. In the 12 baffle case, the difference is reduced below 10%. There is also an improvement in the total heat transfer rate prediction. The percentage difference is reduced below 1% for 10 and 12 baffle cases

Here, simulations are repeated for baffle cut values of 25% and effect of the baffle cut values of 25% and effect of the baffle cut on the heat transfer and pressure drop are investigated. The calculation procedure is same as the previous sections. Similar to previous section, the shell side outlet temperature, shell side pressure drop and total heat transfer values are obtained directly from the CFD runs. Heat transfer coefficient value of CFD result with respect to analytical results for different baffle spacing when 25% baffle cut.

ACKNOWLEDGMENT

We express our humble pranamas to his holiness JAGADGURU Sri Sri Sri PADMABHUSHANA Dr. BALAGANGADHARANATHA MAHA SWAMIJI who has showered his blessings on us. Our heartiest gratitude to our honourable principal Dr. C.K Subbraya for creating the right kind of milieu. We express sincere thanks to our beloved H.O.D, Prof. T.N Krishnaiah, during the work and his moral support. We also take this opportunity to thank Mr. T.G Girish, Associate Professor and P G co-ordinator, Department of Mechanical Engineering, for his valuable suggestions.

REFERENCES

1. Shell and tube heat exchanger, Perry, Robert and Green, Don W. (1984). Perry's Chemical Engineers' Handbook, 6th Edition, McGraw-Hill.
2. More accurate exchanger shell-side pressure drop calculations by D.Gulley & associates, Tulska Oklahoma.
3. A Computer program for designing of shell and tube heat exchanger by Yusuf Ali Kara, Ozbilen guraras.
4. Design and rating shell and tube heat exchangers by John E. Edwards.
5. “ Effect of baffle cut and baffle spacing on the shell side main stream” by Ender ozden.Energy conversion and management 51 (2010) 1004-1014.
6. Data hand book for Design of heat transfer equipments for thermal power plant by Mr.T.G Girish and Mr. V.G Gunjalli.
7. “Transient heat exchanger response” by Juan yin, Michel K Jensen
8. “More accurate exchanger shell-side pressure drop calculations” by D. gulley & associates, Tulska Oklahoma.