CFD SIMULATION OF SHELL AND TUBE HEAT EXCHANGER IN ANSYS

Sidharth Nair[1], Vaibhav Nemade[2], Atharva Karulkar[3], Prof. R.L.Nibe[4], Prof R.V.Hinge[5]

[123] Students of Chemical Engineering Department, Pravara Rural Engineering College, Loni-413736, Maharashtra.

[45]Professor, Department of Chemical Engineering, Pravara Rural Engineering College, Loni-413736, Maharashtra.

ABSTRACT: *It has been observed that CFD has been employed for the wide areas of study in various types of heat exchangers, fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase. From these simulations the quality of the solutions obtained are largely within the acceptable range which proves that CFD is an effective tool for estimating the behaviour and performance of a wide variety of heat exchangers. Shell and tube heat Exchanger is the most common type of heat exchanger and mainly used in oil refinery and other large chemical processes because it is suitable for high pressure applications. The Processes in analysing the simulation consist of modelling using ANSYS Fluent R1 2021 and then, the boundary condition will be set before. The detailed knowledge on the heat transfer in this will provide the basis for further optimization of shell-and-tube heat exchangers.*

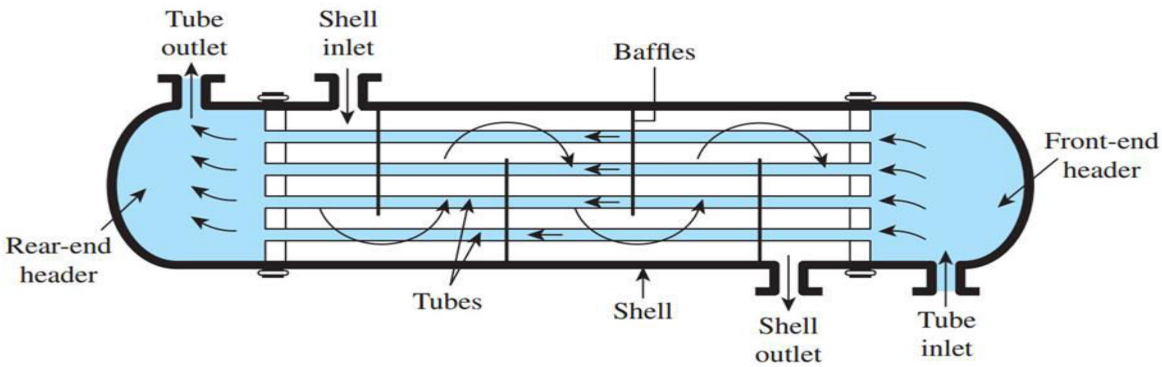
Keywords: STHE, CFD Simulation, Thermal performance.

1.INTRODUCTION

A heat exchanger is a system used to transfer heat among one or more fluids. Heat exchangers are utilized in both cooling and heating processes. They are broadly utilized in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The fluids can be separated through a solid wall to prevent mixing or they will be in direct contact. The classic example of a heat exchanger is found in an internal combustion engine wherein a circulating fluid called engine coolant flows through radiator coils and air flows beyond the coils, which cools the coolant and heats the incoming air. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated through an electronic or a mechanical tool to a fluid medium, often air or a liquid coolant. Heat exchangers are broadly used in industry for both cooling and heating large scale industrial processes. The type and size of heat exchanger used may be tailored to fit a process relying on the type of fluid, its phase, temperature, density, viscosity, pressures, chemical composition and various different thermodynamic properties. In many industrial processes there's waste of energy or a heat stream that is being exhausted, heat exchangers may be used to get better this heat and put it to use by heating a different stream in the process. This practice saves loads of money in industry. Heat exchangers are devices designed to transfer heat between one or more fluids i.e., liquids, vapours, or gases of different temperatures. Depending on the type of heat exchanger employed, the heat transferring process may be gas-to-gas, liquid-to-gas, or liquid-to-liquid and arise through a solid separator, which prevents mixing of the fluids, or direct fluid contact. Other design characteristics, which includes construction materials and components, heat transfer mechanisms, and flow configurations, additionally assist to categorize and classify the types of heat exchangers available. Finding application throughout a wide range of industries, a wide choice of these heat exchanging devices are designed and manufactured to be used in both heating and cooling processes.

1.1 Shell and Tube Heat Exchanger:

A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common kind of heat exchanger in oil refineries and different big chemical processes, and is acceptable for higher-pressure applications. As its name implies, this type of heat exchanger includes a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat among the two fluids. The set of tubes is known as a tube bundle, and can be composed of numerous types of tubes: plain, longitudinally finned, etc. Two fluids, of different starting temperatures, flow through the heat exchanger. One flows through the tubes (the tube side) and the other flows outside the tubes however with inside the shell (the shell side).Heat is transferred from one fluid to the other through the tube walls, either from tube side to shell side or vice versa. The fluids may be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a big heat transfer area must be used, leading to the usage of many tubes. In this manner, waste heat may be put to use. This is an effective manner to preserve energy. Heat exchangers with only one phase (liquid or gas) on each side may be referred to as one-phase or single-phase heat exchangers. Two-phase heat exchangers may be used to heat a liquid to boil it right into a gas (vapour), sometimes known as boilers, or to cool the vapours and condense it right into a liquid (known as condensers), with the phase change generally occurring at the shell side. Boilers in steam engine locomotives are generally large, normally cylindrically-shaped shell-and-tube heat exchangers. In big power plants with steam-driven turbines, shell-and-tube surface condensers are used to condense the exhaust steam exiting the turbine into condensate water which is recycled back to be changed into steam with inside the steam generator. Counter current heat exchangers are most efficient due to the fact they allow the highest log mean temperature difference between the hot and cold streams. Many organizations however do now no longer use pass heat exchangers with a u-tube due to the fact they could break easily in addition to being more costly to build. Often multiple heat exchangers may be used to simulate the counter current flow of a single large exchanger. The simple design of a shell and tube heat exchanger makes it a perfect cooling solution for a different kind of applications. One of the most common applications is the cooling of hydraulic fluid and oil in engines, transmissions and hydraulic power packs. With the proper choice of materials they also can be used to cool or heat different mediums, such as swimming pool water or charge air. There are many advantages to shell and tube technology over plate. One of the huge benefits of the usage of a shell and tube heat exchanger is that they may be often easy to service, mainly with models wherein a floating tube bundle is available (in which the tube plates are not welded to the outer shell). The cylindrical design of the housing is extremely resistant to pressure and allows all levels of pressure applications.



1.2 Fluid allocation for a Shell and Tube Heat Exchanger:

Fluid with higher flow rate is generally preferred to be kept on the shell side. Service where a large temperature change is desired is also kept on the shell side, where temperature change can be achiever by increasing the length of the exchanger. A fluid with a tendency to foul generally should go on the tube side. Cleaning straight tubes normally is easier than cleaning the shell — even if a relatively large tube pitch or a square tube pattern is used to make the shell side easier to clean. Every shell and tube heat exchanger will have two fluids: one on the tubeside and the other on the shellside. Chances are carefully considered our options for fluids and came to a conclusion based on the needs of the process and the properties of our choices. You should take the same thoughtful deliberation when deciding which fluid to put on the shellside and which to allocate to the tubeside. The location for each makes a big difference in the effectiveness of heat transfer, maintenance needs and the cost of the exchanger and any replacements.

Table 1 : Fluid allocation for a Shell and Tube Heat Exchanger

TUBE SIDE FLUID	SHELL SIDE FLUID
Corrosive fluid	Viscous fluid
Fouling fluid	Fluid with higher flow rate
Less viscous fluid	Fluid with large temperature difference
Hot fluid	
High pressure fluid	

2. ANSYS:

Ansys is the finite detail analysis code broadly use in computer aided engineering(CAE) field. ANSYS software help us to construct computer models of structure, machine, components or system, apply operating masses and different design criteria, study physical response consisting of stress level temperature distribution, pressure etc.

- In Ansys following Basic step is followed:

1. During pre processing the geometry of the problem is defined. Volume occupied by fluid is divided into discrete cells(the mesh). The mesh can be uniform or non uniform. The physical modelling is defined. Boundry condition is defined. This includes specifying the fluid behaviour of the problem. For transient problem boundry condition are also defined.
2. The simulation is started and the equation are solved iteratively as consistent state or transient.
3. Finally a post procedure is used for the evaluation and visualisation of the resulting problem.

ANSYS comes with two different state-of-the-art computational fluid dynamics software for process and product design optimization

2.1 ANSYS CFX

ANSYS CFX is a general-purpose high-performance computational fluid dynamics program that is used to solve complex wide range fluid problems. ANSYS CFX is its innovative solver technology that is the main driving force to achieve highly reliable and accurate results in a much more efficient manner as compared to other CFD solvers. ANSYS CFX advanced solver lays the foundation of the users to abundant choices of physical models to virtually capture any type of phenomenon that is connected with fluid flow. Let it be turbulent to laminar flow or fully compressible to incompressible fluid or let is be combusting or non-reacting fluids, ANSYS CFX solver capabilities covers almost all the spectrum of fluid dynamics simulations.

3.METHODOLOGY

Fig 1.Overview

Preprocessing is done like importing mesh files created in the ICEM into the CFX preprocessor and then defining the problem and setting up the boundary conditions.In Simulation run, the launching of CFX solver and running the Simulation is carried out. At last post-processing like importing the CFD-post and post processing results takes place.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 05 Issue: 08 | Aug - 2021 ISSN: 2582-3930

3.1 Problem Description:-

The task is to model the heat exchange between two fluids at different temperatures, flowing in opposite directions. In the shell of the heat exchanger, steam enters at a mass flow rate of 1kg/s and a temperature of 20°C, passing through the baffles and leaving on the other side. Hot water at 80°C is entering the tubes at a mass flow rate of 1kg/s.

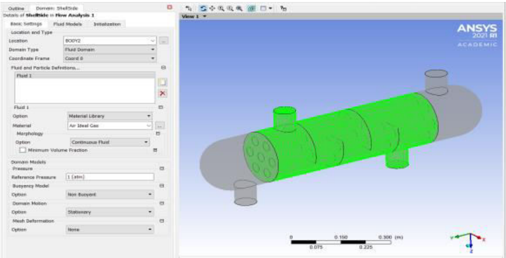


Fig 2.Assigning shell side fluid (Air)

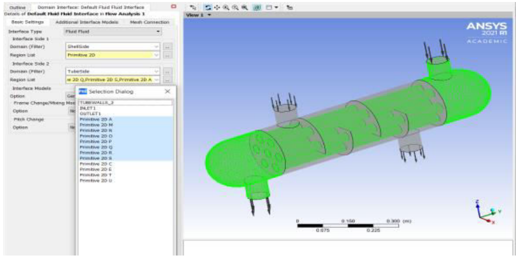


Fig 6.Details of interface (Tube side)

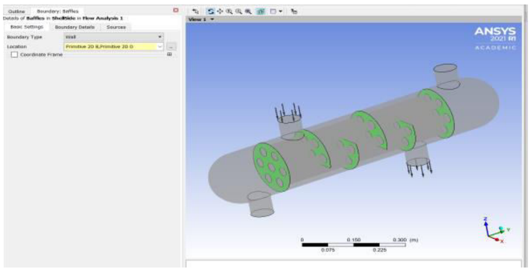


Fig 3.Details of Baffles

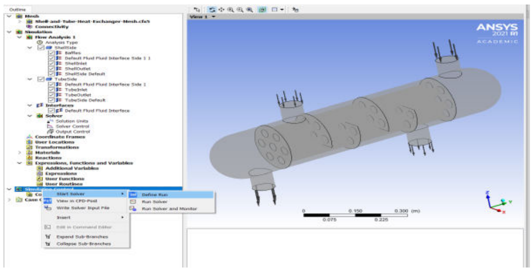


Fig 7.Start solver and define Run

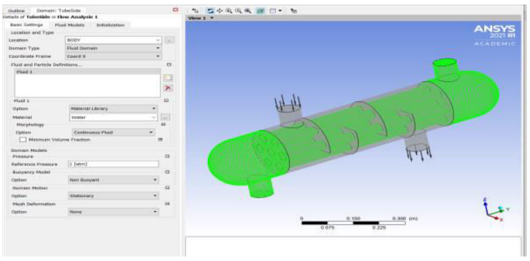


Fig 4.Assigning tube side fluid (Water)

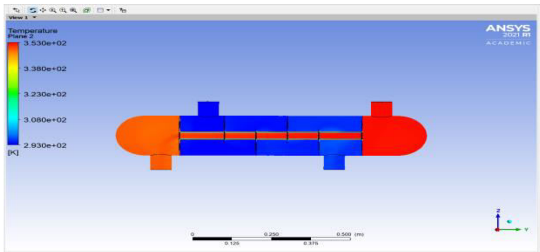


Fig 8.Temperature Distribution

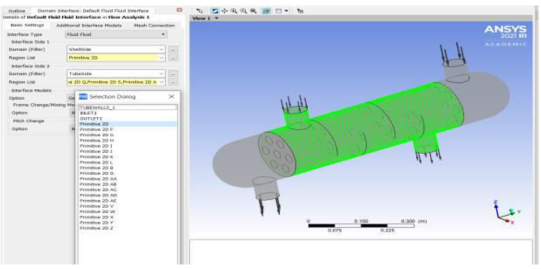


Fig 5.Details of interface (Shell side)

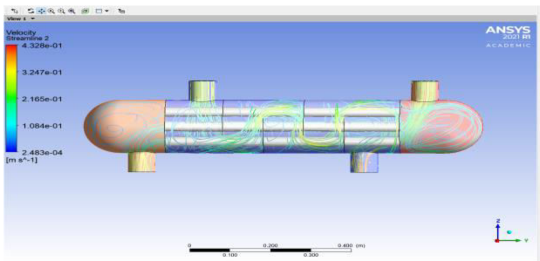


Fig 9.Velocity Distribution

The 3-D model is then discretized in ICEM CFD. In order to capture both the thermal and velocity boundary layers the entire model is discretized using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into three fluid domains Fluid_Inlet, Fluid_Shell, and Fluid_Outlet. The heat exchanger is discretized into solid and fluid domains in order to have better control over the number of nodes. The fluid mesh is made finer for simulating conjugate heat transfer phenomenon. The discretized model is checked for quality. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS CFX pre-processor.

3.2 Governing equations:

- The 3-D flow through the shell-and-tube heat exchanger has been simulated by solving the appropriate governing equations, eq. (1) to eq. (5). viz. conservation of mass, momentum and energy using ANSYS CFX code. Turbulence is taken care by shear stress transport (SST) k-w model of closure which has a blending function that supports Standard k-w near the wall and Standard k-e elsewhere.

$$\textit{Conservation of mass: } \nabla(\rho\vec{V}) = 0 \tag{1}$$

$$\textit{x-momentum: } \nabla(\rho u\vec{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \tag{2}$$

$$\textit{y-momentum: } \nabla(\rho u\vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g \tag{3}$$

$$\textit{z-momentum: } \nabla(\rho u\vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \tag{4}$$

$$\textit{Energy: } \nabla(\rho e\vec{V}) = -p\nabla\vec{V} + \nabla(k\nabla T) + q + \phi \tag{5}$$

3.3 Boundary condition set-up:-

- In ANSYS CFX pre-processor, the various fluid domains are defined. The flow in this study is turbulent, hence SST k-w turbulence model is chosen.
- The boundary conditions are specified in ANSYS CFX pre-processor and then the file is exported to the ANSYS CFX. The same procedure is adopted for the other two models.

Heat exchanger length	900 mm
Shell inner diameter	220 mm
Tube outer diameter	50 mm
Pitch type	Triangular Pitch
Number of tubes	7
Number of baffles	4

Table 2: Geometric dimensions of shell and tube heat exchanger.

4. RESULTS:

In the present study, for the present heat exchanger, the sensitivity of the results of the heat transfer to the varying mass flow rate is investigated for four different mass flow rate. Then, with the selected model, suitable solver set up and discretization scheme, the variations in shell side heat transfer with the different flow rate are investigated. Contours of Temperature for different mass flow rate are showed in below Figure-10, Figure-11, Figure-12 and Figure-13. Variation of Shell side outlet temperature(k), tube side outlet temperature(k) for different mass flow rate are shown in Table 3. The graph-1 shows that the variation of Shell side Outlet temperature (k) with respect to mass flow rate (kg/s) is increasing by a significant value.

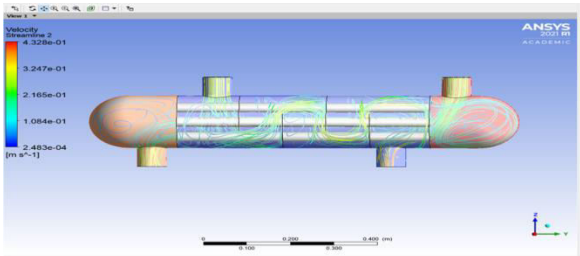


Fig 10: Velocity Distribution (Flowrate 1 kg/s)

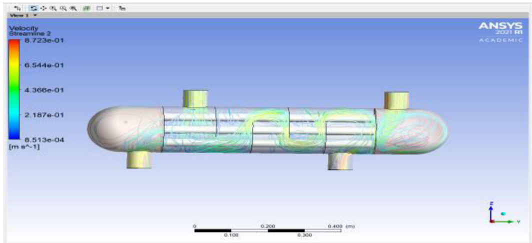


Fig 11: Velocity Distribution (Flowrate 1.5 kg/s)

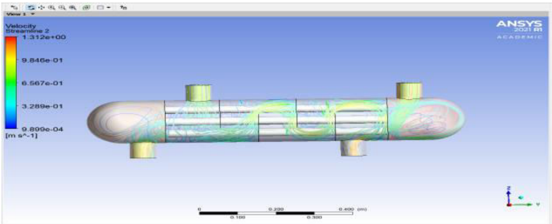


Fig 12: Velocity Distribution (Flowrate 2 kg/s)

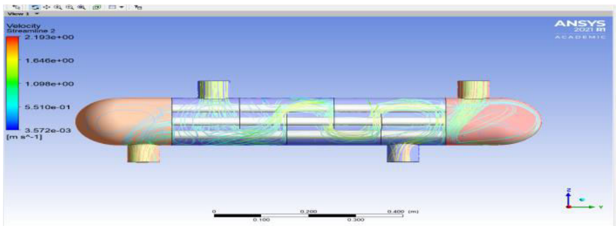
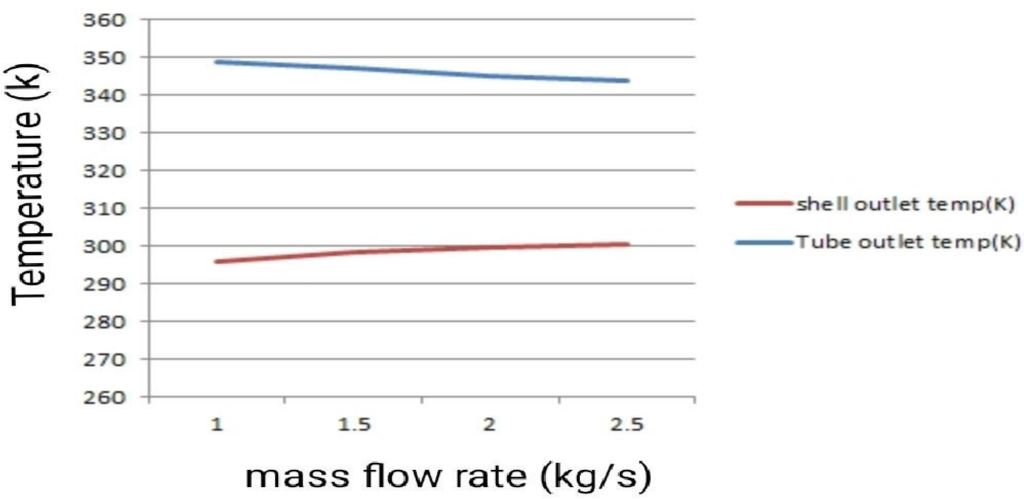


Fig 13: Velocity Distribution (Flowrate 2.5 kg/s)

Mass flow rate (kg/s)	Shell outlet temperature (K)	Tube inlet temperature (K)
1	296	349
1.5	298.4	347
2	299.6	345.3
2.5	300.5	344

Table 3: Results



5 CONCLUSION:

The Shell side CFD analysis of a small shell-and-tube heat exchanger is modelled with sufficient detail to resolve the flow and temperature parameters. The resulted values of shell side outlet temperature and tube side outlet temperature are obtained from the CFD simulation of fixed tube wall and shell inlet temperatures. K-ε standard turbulence model with first order discretization and fine mesh is selected for simulation approach. By varying the mass flow rate 1 ,1.5, 2 and 2.5 kg/sec shell side flow rates, the simulation results are found. From the above results we can conclude that the Shell side outlet temperature (K) has a significant change with respect to the mass flow rate (Kg/s). Heat transfer increases with respect to the mass flow rate Kg/s. For better performance of STHE one should design with considerable changes depending upon the need and heat transfer characteristics one can predict and decide from the above mass flow rate (Kg/s) as the best operating parameters for the STHE.

REFERENCES:

1)Indian Standard (IS: 4503-1967): Specification for Shell and Tube Type Heat Exchangers, BIS 2007, New Delhi.
2)R. K. Sinnott, Coulson & Richardson‟s Chemical Engineering: Chemical Engineering Design (volume 6), Butterworth-Heinemann, 3rd ed. 1999.
3)D.Q. Kern, Process Heat Transfer, McGraw-Hill Book Company, Int. ed. 1965.
4)Dutta B.K. „Heat Transfer-Principles and Applications‟, PHI Pvt. Ltd., New Delhi, 1st ed. 2006.
5)Emerson, W.H. Shell-side pressure drop and heat transfer with turbulent flow in Segmentally baffled shell-tube heat exchangers, Int. J. Heat Mass Transfer, 6 (1963), pp. 649–66.
6)James R. Couper; W. Roy Penney, James R. Fair, Stanley M. Walas, Chemical Process Equipment: selection and design, Elsevier Inc., 2nd ed. 2005.
7)Karno A, Ajib S. Effects of baffle cut and number of baffles on pressure drop and Heat transfer in shell-and-tube heat exchangers – numerical simulation. Heat Exchangers 2006;7:299–322.
8)Sparrow EM, Reifschneider LG. Effect of interbaffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger. Int J Heat Mass Tran1986; 29:1617-28.
9)Eryener D. Thermoeconomic optimization of baffle spacing for shell and tube heat exchangers. Energ Convers Manage 2006; 47:1478–89.
10)Kern DQ. Process heat transfer. New York, N.Y.: McGraw-Hill; 1950.
11)Mahmood Aslam Bhutta, Nasir Hayat, Muhamma Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan, CFD Applications in Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2012, pp. 1-12.
12)Mukherjee R. Effectively design shell-and-tube heat exchangers. Chem Eng Progress 1998;94(2):21–37.
13)R. Mukherjee. Practical Thermal design of shell and tube heat exchangers.
14)Standards of Tubular exchanger manufacturers association (TEMA) (Ninth Edition).
15)Li, H. and Kottek .V. Effect of baffle spacing on pressure drop and local heat transfer in Shell and tube heat exchangers for staggered tube arrangement, Int.J. Heat Mass Transfer, 41:10, (1998), pp. 1303–1311.
16)Stehlík, P., Nemcansky, J., Kral, D. and Swanson, L.W. Comparison of correction factors for Shell-and-tube heat exchangers with segmental or helical baffles, Heat Transfer Engineering, 15(1), (1994) pp. 55-65.
17)Usman, U. E. and Goteborg. Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD, Master's Thesis, Sweden, 09 (2011).
18)Lutcha, J. and Nemcansky, J. Performance improvement of tubular heat exchangers by helical Baffles, Chemical Engineering Research and Design, 68, (1990) pp.263-270.
19)Ambekar Aniket Shrikant, R. Sivakumar, N. Anantharaman, M. Vivekenandan. CFD simulation study of shell and tube heat exchangers with different baffle segment configurations.
20)Dogan Eryener. Thermoeconomic optimization of baffle spacing for shell and tube heat exchangers.
21)Kiran K, A M Mulla, Manoj and Umesh C J. Investigation of baffle spacing effect on shell side heat transfer characteristics in shell and tube heat exchanger using computational fluid dynamics.
22)Ender Ozden, Ilker Tari. Shell side CFD analysis of a small shell-and-tube heat exchanger.
23)Yue Sun , Xinting Wang, Rui Long, Fang Yuan and Kun Yang. Numerical Investigation and Optimization on Shell Side Performance of A Shell and Tube Heat Exchanger with Inclined Trefoil-Hole Baffles.
24)Uday C. Kapale, Satish Chand. Modeling for shell-side pressure drop for liquid flow in shell-and-tube heat exchanger.
25)Qiuwang Wang , Qiuyang Chen, Guidong Chen, Min Zeng. Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles.