ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Design and Analysis of Heat Exchanger with Different Baffles

D. Gireesh
Department of Mechanical Engineering,
SISTAM College, JNTUK, India.

J. Bala Bhaskara Rao
Department of Mechanical Engineering,
SISTAM College, JNTUK, India.

ABSTRACT

In present days shell and tube heat exchanger is the most common type heat exchanger, it is widely used in oil refinery and other large chemical processes, because it is suitable for high pressure applications. The process in solving simulation consists of modelling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 15.0. The objective of the project is to design the shell and tube heat exchanger with different baffles and study the pressure drop inside the shell using FLUENT software.

The heat exchanger contains 7 tubes, 600mm shell length and 90mm diameter. The baffles of shell and tube are varied with normal baffle, 30^0 inclined baffle and helical baffle.In simulation it shows, how the pressure vary in shell due to different baffles. 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. Baffle spacing is the centerline-to centerline distance between adjacent baffles. It is the most vital parameter in STHE design. Closer spacing will result in poor bundle penetration by the shell side fluid and difficulty in mechanically cleaning the outsides of the tubes.In this project, by varying different types of baffles, Shell side pressure drop is calculated. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in an effective pressure drop in the heat exchanger. So helical baffles are preferred for effective pressure drop that is 2858.646 Pascal's in shell side of Shell and Tube Heat Exchanger.

Key words: ***heat exchangers, cfd, catia ,helical, inclined, baffle***

1. INTRODUCTION

Heat Exchanger

A heat exchanger is a device used to transfer heat between a solid object and a fluid, or between two or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment.

Heat exchangers are of two types: -

1) Where both media between which heat is exchanged are in direct contact with each other is direct contact heat exchanger,

2) Where both media are separated by a wall through which heat is transferred so that they never mix, indirect contact heat exchanger.

1.2 Double pipe heat exchanger:

A typical double-pipe heat exchanger consists of one pipe placed concentrically in side another of larger diameter with appropriate fittings to direct the flow from one section to the next, as shown in figure (1.2). Double-pipe heat exchangers can be arranged in various series and parallel arrangements to meet pressure drop and mean temperature difference requirements. The major use of double-pipes exchangers is for sensible heating or cooling of process fluids where small heat transfer areas are required. This configuration is also very suitable.

When one or both fluids is at high pressure. The major disadvantage is that double-pipe heat exchangers are bulky and expensive per unit transfer surface. Inner tube being may be single tube or multi-tubes Fig.(1.3). If heat transfer coefficient is poor in annulus, axially finned inner tube (or tubes) can be used. Double-pipe heat

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

exchangers are built in modular concept, i.e., in the form of hair fins.

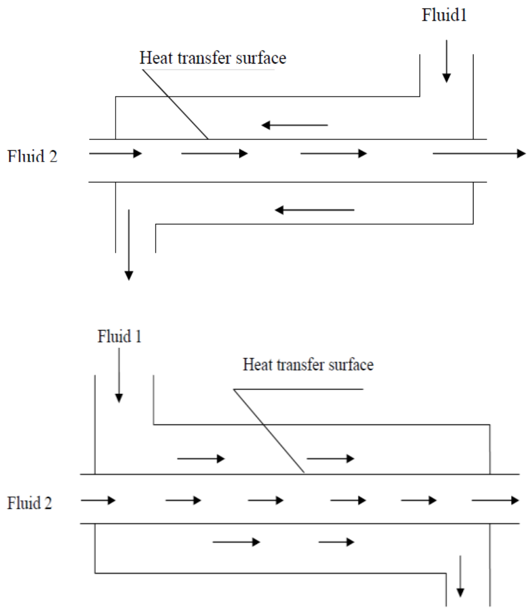

Fig 1.1: Double pipe heat exchanger

1.3 Heat Transfer Considerations

The energy flow between hot and cold streams, with hot stream in the bigger diameter tube, is as shown in Figure 2.10. Heat transfer mode is by convection on the inside as well as outside of the inner tube and by conduction across the tube. Since the heat transfer occurs across the smaller tube, it is this internal surface which controls the heat transfer process. By convention, it is the outer surface, termed Ao, of this central tube which is referred to in describing heat exchanger area. Applying the principles of thermal resistance,

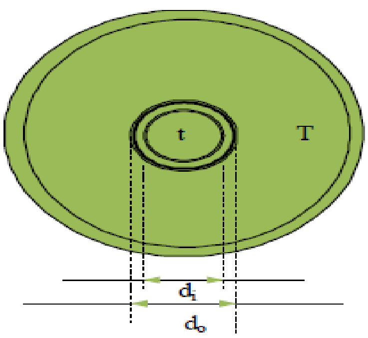

Fig 1.2: End view of a tubular heat exchanger

$$R = \frac{1}{h_o A_o} + \frac{\ln\left(r_o/r_i\right)}{2\pi \cdot kl} + \frac{1}{h_i A_i}$$

If we define overall the heat transfer coefficient, Uc, as:

$$U_c = \frac{1}{RA_o}$$

Substituting the value of the thermal resistance R yields:

$$\frac{1}{U_c} = \frac{1}{h_o} + \frac{r_o \ln\left(r_o/r_i\right)}{k} + \frac{A_o}{h_i A_i}$$

Standard convective correlations are available in text books and handbooks for the convective coefficients, *ho* and *hi*. The thermal conductivity, *k*, corresponds to that for the material of the internal tube. To evaluate the thermal resistances, geometrical quantities (areas and radii) are determined from the internal tube dimensions available.

1.4 Log Mean Temperature Differences

Heat flows between the hot and cold streams due to the temperature difference across the tube acting as a driving force. As seen in the Figure 7.3, the temperature difference will vary along the length of the HX, and this must be taken into account in the analysis.

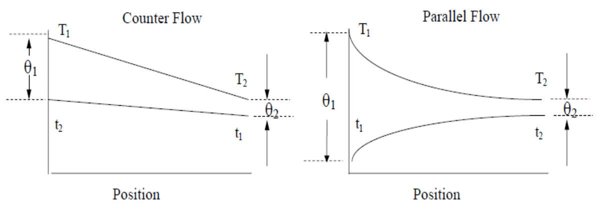

Fig.1.11:Log Mean Temperature Differences

1.5: Temperature Differences between Hot and Cold Process Streams

From the heat exchanger equations shown earlier, it can be shown that the integrated average temperature difference for either parallel or counter flow may be written as:

$$\Delta\theta = LMTD = \frac{\theta_1 - \theta_2}{\ln\left(\frac{\theta_1}{\theta_2}\right)}$$

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The effective temperature difference calculated from this equation is known as the log mean temperature difference, frequently abbreviated as LMTD, based on the type of mathematical average that it describes. While the equation applies to either parallel or counter flow, it can be shown that $\Delta\theta$ will always be greater in the counter flow arrangement. Another interesting observation from the above Figure is that counter flow is more appropriate for maximum energy recovery.

2. LITERATURE REVIEW

M. Thirumarimurugan, T.Kannadasan and E.Ramasamy [1] have investigated heat transfer study on a solvent and solution by using Shell and Tube Heat Exchanger. In which Steam is taken as the hot fluid and Water and acetic acid-Water miscible solution taken as cold fluid. A series of runs were made between steam and water, steam and Acetic acid solution .The flow rate of the cold fluid is maintained from 120 to 720 lph and the volume fraction of Acetic acid is varied from 10-50%. Experimental results such as exchanger effectiveness, overall heat transfer coefficients were calculated. . MATLAB program was used to simulate a mathematical model for the outlet temperatures of both the Shell and Tube side fluids. The effect of different cold side flow rates and different compositions of cold fluid on the shell outlet temperature, tube outlet temperature and overall heat transfer coefficients were studied.

Usman Ur Rahman [2] had investigated an un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient and pressure drop by numerically modelling. The heat exchanger contained 19 tubes inside a 5.85m long and 108mm diameter shell. For this reason, Realizable k – ε model is used with standard and then Non-equilibrium wall functions.Thus in order to avoid this and to include the low Reynolds

Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao [3] developed a method for design and rating of shell-and-tube heat exchanger with helical baffles based on the public literatures and the widely used Bell–Delaware method for shell-and-tube heat exchanger with segmental baffles (STHXSB). The accuracy of present method is validated with experimental data. Four design cases of replacing original STHXsSB by STHXsHB are taken. And comparison result shows that all shell and tube heat exchanger with helical baffles have better performance than the original heat exchanger with segmental baffles.

Muhammad Mahmoud Salam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, AhmerRaisKhan,KanwarNaveed Ahmad, SarfarazKhani[4] , It focuses on the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD employed for the fluid flow mal-distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models such as standard, realizable and RNG, k – ε, RSM, and SST k - ε with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO and etc. have been adopted to carry out the simulationsThe simulations results ranging from 2% to 10% with the experimental studies. In some exceptional cases, it varies to 36%.

ŽarkoStevanović, GradimirIlić, NenadRadojković, MićaVukić, VelimirStefanovićand GoranVučković [5] has developed an iterative procedure for sizing shell-and-tube heat Exchangers according to given pressure drop and the thermo-hydraulic calculation and the geometric optimization on the basis of CFD technique have been carried out. A numerical study of three-dimensional fluid flow and heat transfer is described. The baffle and tube bundle was modelled by the 'porous media' concept. Three turbulent models were used for the flow processes.

Ender Ozden, Ilker Tari. [6] Has investigated the design of shell and tube heat exchanger by numerically modeling in particular the baffle spacing, baffle cut and shell diameter dependencies of heat transfer coefficient and pressure drop. The flow and temperature fields are resolved by using a commercial CFD package and it is

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. It is observed that the CFD simulation results are very good with the Bell-Delaware methods and the differences between Bell-Delaware method and CFD simulations results of total heat transfer rate are below 2% for most of the cases.

3.Catia Modeling

3.1MODELING:

In the process of the Catia modelling of Shell and Tube Heat Exchanger we have to design four Parts. They are,

Tube Sheet
Tubes
Baffles
Shell

3.1.1 TUBE SHEET:

Dimensions:

Diameter	= 100mm
Pitch	= 30mm
Hole bundle geometry	= Triangular
No. of Holes	= 7
Hole diameter	= 20mm

Used Catia Tools:

Circle, Rectangular Pattern, Circular Pattern, Pad, Pocket and Plane.

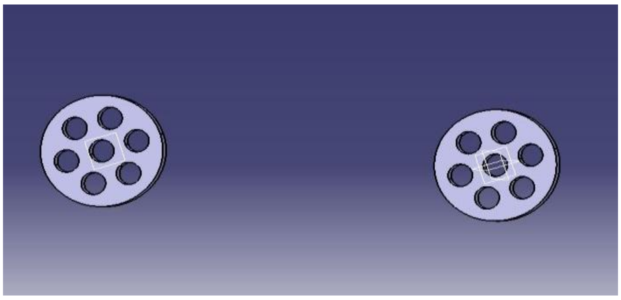

Fig.3.1 Designed Catia model of Tube Sheet

3.1.2 TUBES:

Dimensions:

Tube outer Diameter	= 20mm
Thickness	= 1mm
Tube Length	= 600mm

Used Catia Tools:

Project 3D Elements and Pad.

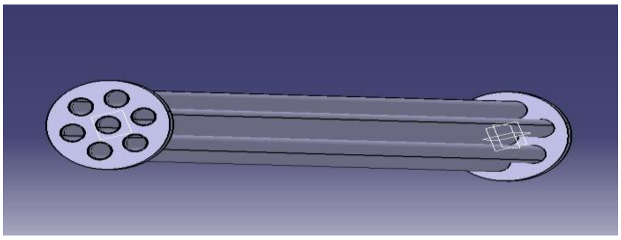

Fig 3.2 Designed Catia model of Tubes

3.1.3 BAFFLES:

1) Normal Baffle:

Dimensions:

Baffle Diameter	= 90mm
Baffle thickness	= 2mm
Baffle cut	= 36%
Baffle spacing	= 86mm
No. of Baffles	= 6

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

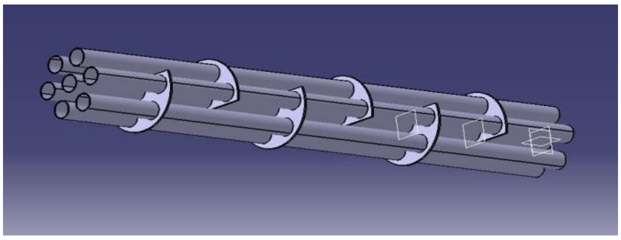

Fig 3.3 Designed Catia model of Normal Baffle

2) Inclined Baffle:

Dimension:

Baffle Inclination	= 30^o
Baffle Diameter	= 90mm
Baffle thickness	= 2mm
Baffle cut	= 36%
Baffle spacing	= 86mm
No. of Baffles	= 6

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

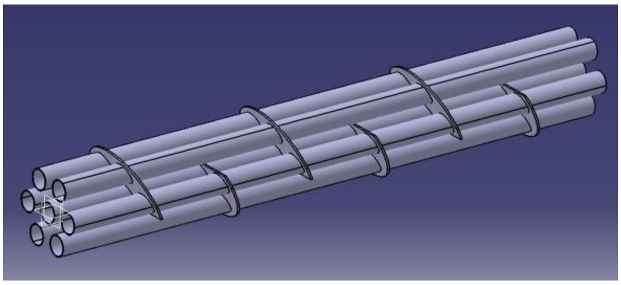

Fig 3.4 Designed Catia model of Inclined Baffle

3) Helical Baffle:

Dimensions:

Helix Diameter = 90mm
Helix Length = 600mm
Helical Pitch = 200mm
Baffle Thickness = 2mm

Used Catia Tools:

Helix, Point, Line, Rectangle and Pad.

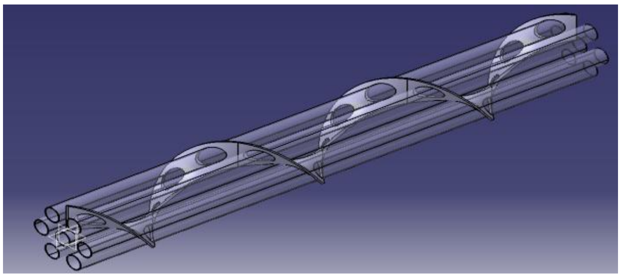

Fig 3.5 Designed Catia model of Helical Baffle

3.1.4 SHELL:

Dimensions:

Shell inner Dia = 90mm
Shell Thickness = 5mm
Shell Length = 600mm

Used Catia Tools:

Project 3D Elements, Pad, Plane and Pocket.

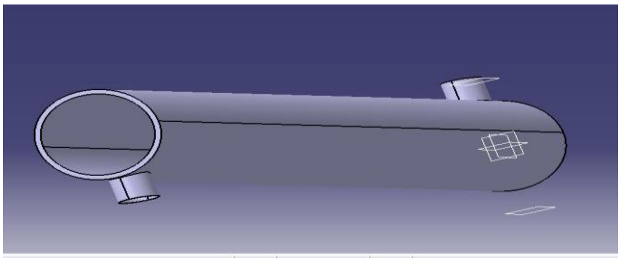

Fig 3.6 Designed Catia model of Shell

3.2 DESIGNED CATIA MODEL:

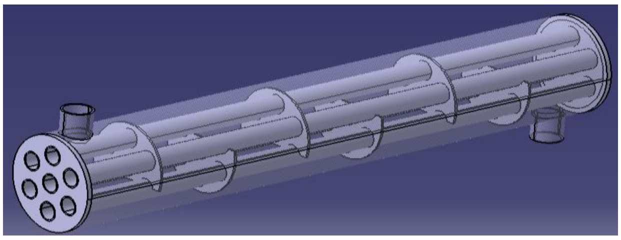

Fig 3.7 Designed Catia model of STHE with Staggered Baffle

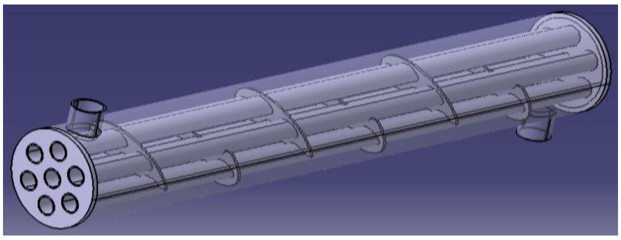

Fig.3.8 Designed Catia model of STHE with Inclined Baffle

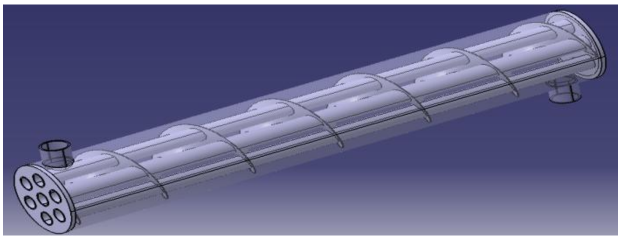

Fig 3.9 Designed Catia model of STHE with Helical Baffle

4. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction

4.1 GEOMETRY:

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeler opens as a new window as the geometry is double clicked.

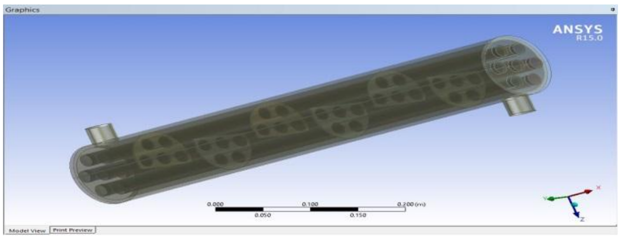

Fig. 4.1 Imported model in geometry

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

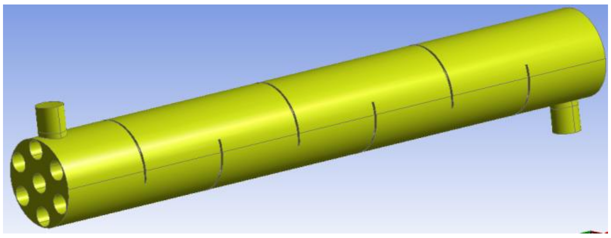

Fig. 4.2 shell side fluid domain in normal baffle

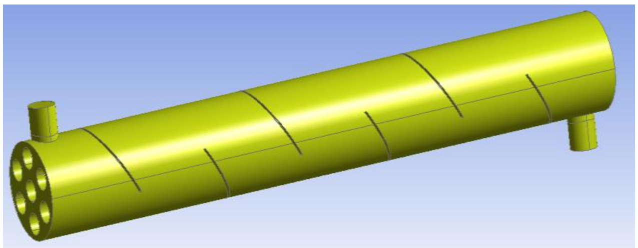

Fig. 4.3 shell side fluid domain in inclined baffle

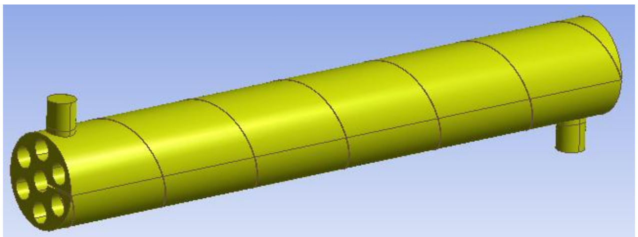

Fig. 4.4 shell side fluid domain in helical baffle

PART NUMBER	PART OF THE MODEL	STATE TYPE
1.	INNER FLUID	FLUID
2.	OUTER FLUID	FLUID
3.	BAFFLES(6)	SOLID
4.	SHELL(1)	SOLID
5.	TUBES(7)	SOLID
6.	TUBE SHEET(2)	SOLID

Table.4.1 geometry type and model

4.1.1 The Main Solver

The solver is the heart of CFD software. It sets up the equation set according to the options chosen by the user and meshes points generated by the pre-processor, and solves them to compute the flow field. The process involves the following tasks:

- selecting appropriate physical model,
- defining material properties,
- prescribing boundary conditions,
- providing initial solutions,
- setting up solver controls,
- set up convergence criteria,
- solving equation set, and
- saving results

Once the model is completely set up, the solution starts and intermediate results can be monitored in real time from iteration to iteration. The progress of the solution process is displayed on the screen in terms of the residuals, a measure of the extent to which the governing equations are not satisfied.

4.2 MESHING

Initially a relativelycoarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

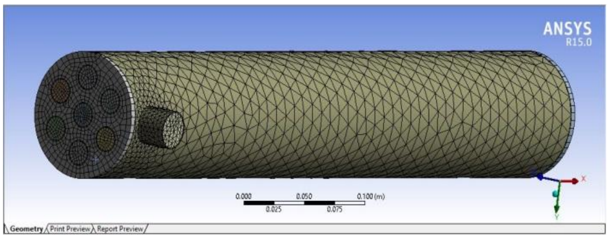

Fig 4.5 Shell and tube model after Meshing

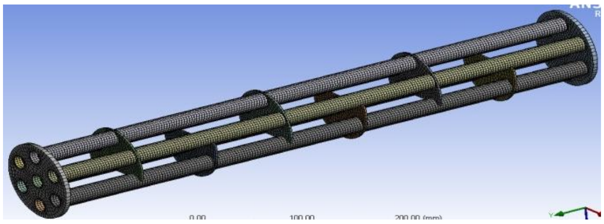

Fig 4.6 Shell and tube model of normal baffle after Meshing

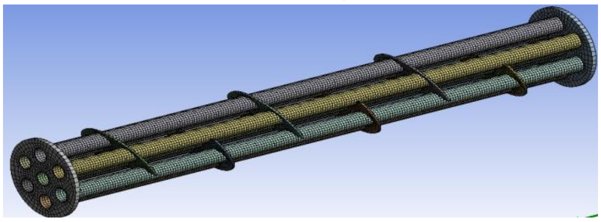

Fig 4.7 Shell and tube model of inclined baffle after Meshing

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

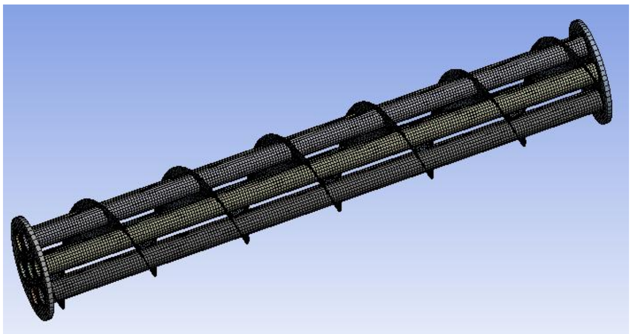

Fig 4.8 Shell and tube model of helical after Meshing

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids.

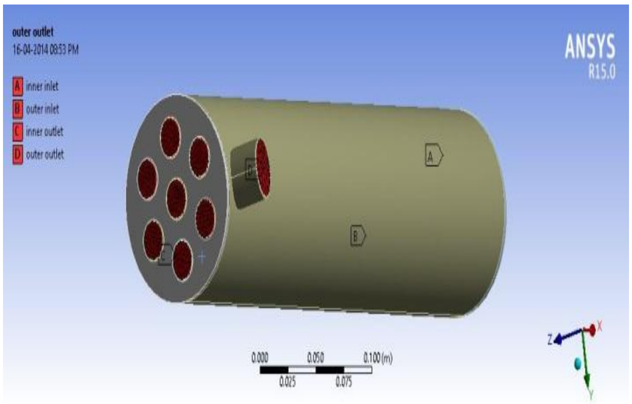

Fig 4.9 Named selections

Save project again at this point and close the window. Refresh and update project on the workbench. Now open the setup. The ANSYS Fluent Launcher will open in a window. Set dimension as 3D, option as Double Precision, processing as Serial type and hit OK. The Fluent window will open.

4.3 SETUP:

The mesh is checked and quality is obtained.

4.3.1 MATERIALS:

The create/edit option is clicked to add water-liquid, steel and copper to the list of fluid and solid respectively from the fluent database

4.3.2 CELL ZONE CONDITIONS: In cell zone conditions, we have to assign the conditions of the liquid and solid.

Table 4.2 cell zone conditions

Sno	PART/BODY	MATERIAL
1.	INNER FLUID	WATER-LIQUID
2.	OUTER FLUID	WATER-LIQUID
3.	TUBE SHEET	STEEL
4.	TUBES	COPPER
5.	BAFFLES	COPPER
6.	SHELL	STEEL

4.3.3 BOUNDARY CONDITIONS:

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table as given below.

Table 4.3 boundary conditions

	BOUNDARY CONDITION TYPE	MASS FLOW RATE(kg/s)	TEMPERATURE (k)
INNER INLET	Mass flow inlet	1	450
INNER OUTLET	Pressure outlet	-	-
OUTER INLET	Mass flow inlet	1	300
OUTER OUTLET	Pressure outlet	-	-

4.4 SOLUTION:

RUN CALCULATION:

After giving the boundary conditions to the inner and outer fluid, finally we have to run thecalculations. The number of iteration is set to 500 and the solution is calculated and various contours, vectors and plots are obtained.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

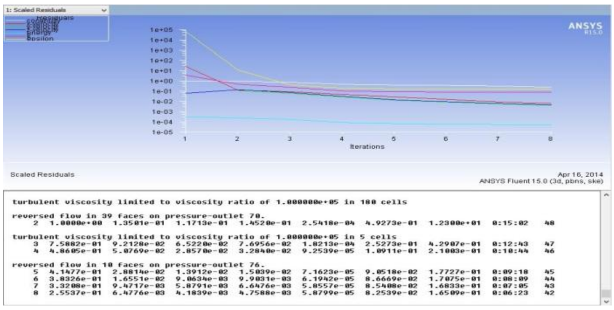

Fig 4.10 Calculations was running

4.5The Post-processor

The post-processor is the last part of CFD software. It helps the user to examine the results and extract useful data. The results may be displayed as vector plots of velocities, contour plots of scalar variables such as pressure and temperature, streamlines and animation in case of unsteady simulation. Global parameters like drag coefficient, lift coefficient, Nusselt number and friction factor etc. may be computed through appropriate formulas. These data from a CFD post-processor can also be exported to visualization software for better display.

Several general-purpose CFD packages have been published in the past decade. Prominent among them are:PHOENICS,FLUENT, STAR-CD,CFX, CFD-ACE,ANSWER, CFD++, FLOW-3D and COMPACT. Most of them are based on the finite volume method. CFD packages have also been developed for special applications; FLOTHERM and ICEPAK for electronics cooling, CFX-TASCFLOW and FINE/TURBO for turbo machinery and ORCA for mixing process analysis are some examples. Most CFD software packages contain their own grid generators and post processors. Software such as ICEM CFD, Some popular visualization software used with CFD packages are TECPLOT and FIELDVIEW.

4.6 OVERVIEW OF FLUENT PACKAGE

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D FLUENT also allows user to refine or coarsen grid based on the flow solution.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The user interface is written in a language called Scheme, a dialect of LISP. The advanced user can customize and enhance the interface by writing menu macros and functions.

5. RESULTS AND DISCUSSIONS

The objective of this project is to evaluate the maximum pressure drop in shell side of Shell and Tube Heat Exchanger. When pressure difference is more, it will caused for increasing of entering velocity and discharge. Baffles are the main parameters which influence the pressure drop. These Baffles are optimized by using fluid flow analysis.

The Effect of Staggered Baffle Arrangement over Pressure Drop in Shell and Tube Heat Exchanger

The baffle member plays an important role in STHE and it supports tube bundle and equally distribute the fluid in shell side. When segmental baffles are used in STHE which have many disadvantages .The low heat transfer is achieved due to the flow stagnation i.e., dead zones which are created at the corners between baffle and shell wall . It requires higher pumping power and it creates high pressure drop under the same heat load. The heat transfer rate, velocity and pressure drop in shell side are calculated below

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

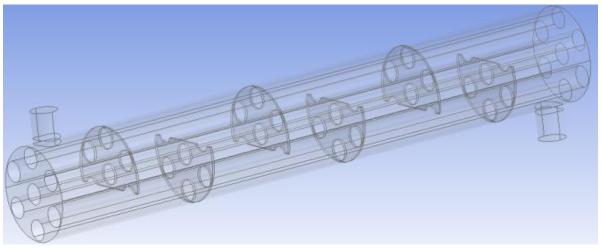

Fig 5.1 Shell and tube model with normal baffle

5.1 REPORTS:

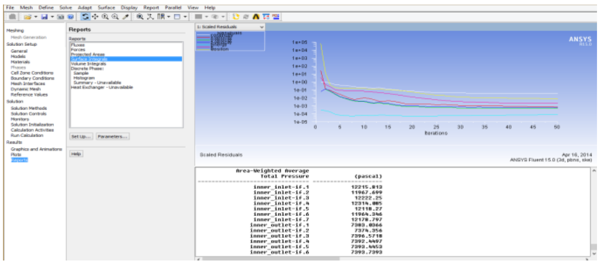

Fig 5.2 verifying the reports.

5.2 Normal Baffle

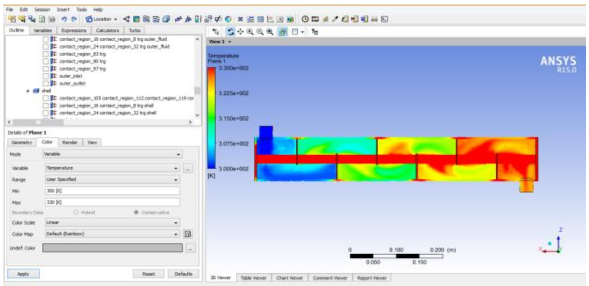

Fig 5.3 Temperature variation in normal baffle

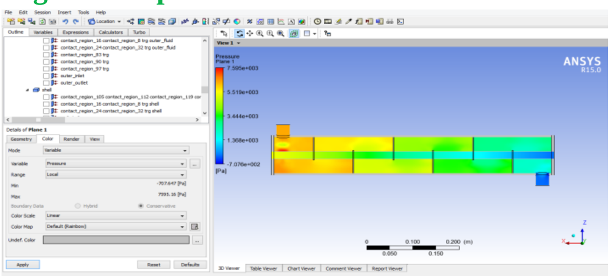

Fig 5.4 Pressure variation in normal baffle.

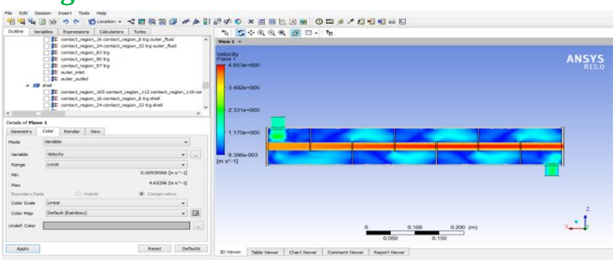

Fig 5.5 Velocity variation in normal baffle

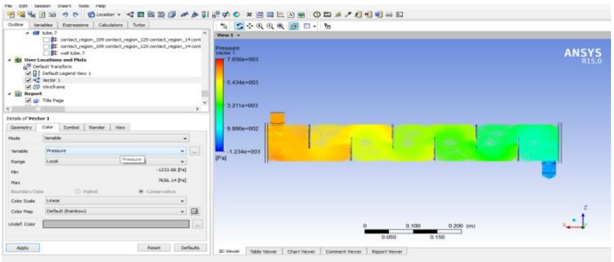

Fig 5.6 Pressure variation in normal baffle

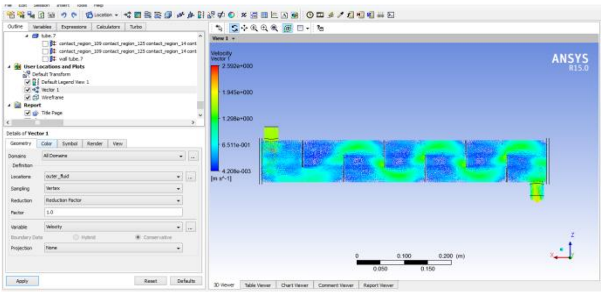

Fig 5.7Velocity variation in normal baffle

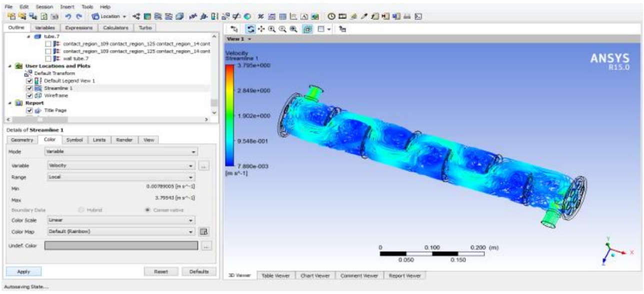

Fig 5.8 Velocity variation in normal baffle.

5.3 INCLINED BAFFLE:

Staggered baffles are replaced by inclined baffles and thermal properties are calculated. Due to inclination of the baffle it reduces the stagnation points and dead zones. The flow path increased in shell side of shell and tube heat exchanger with these inclined baffles. The flow pattern and pressure drop are calculated and tabulated. As compare to normal baffles these baffles gives less pressure drop in shell side of shell and tube heat exchanger.

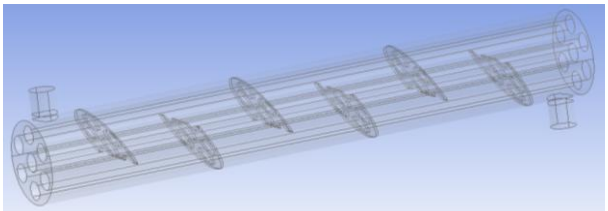

Fig 5.9 Shell and tube model with inclined baffle

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

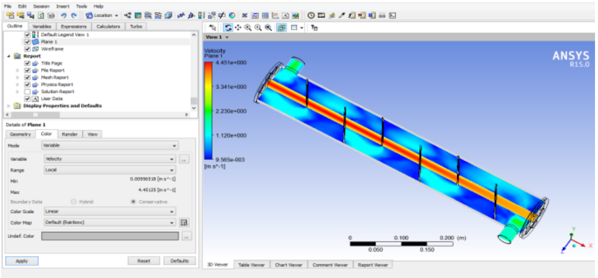

Fig 5.10 Velocity variation in inclined baffle.

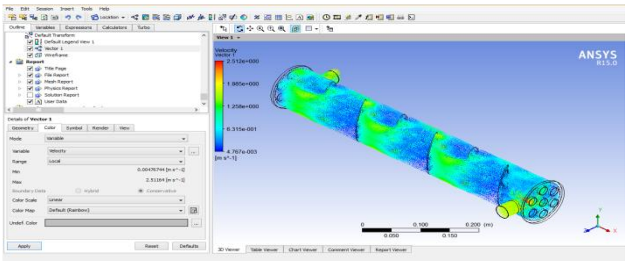

Fig 5.11 Velocity variation in inclined baffle

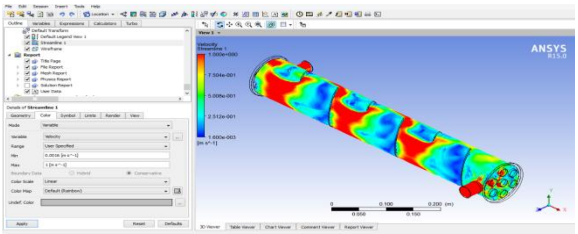

Fig 5.12 Velocity variation in inclined baffle.

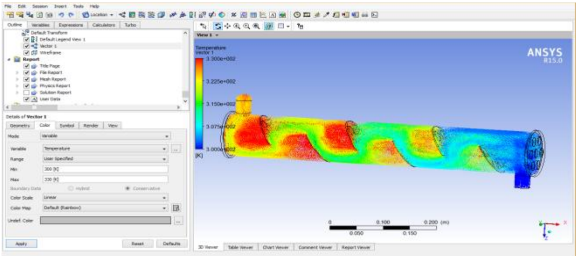

Fig 5.13 Temperature variation in inclined baffle.

5.4 Helical Baffle:

In this case normal baffles and inclined baffles are replaced by helical baffles. These helical baffles shows very good results as compare to other two baffles. The thermal analysis results and flow patterns are shown below. Due to helical shape of the baffle it reduces the dead zones in between shell and baffle space of shell and tube heat exchanger and also it increases the flow path in shell side and it increases the heat transfer rate and reduces the pressure drop.

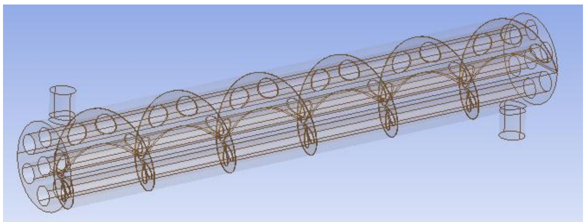

Fig 5.14 Shell and tube model with helical baffle

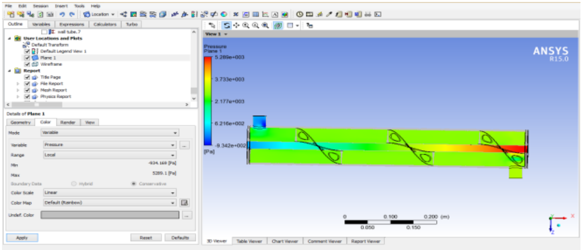

Fig 5.15 Pressure variation in helical baffle

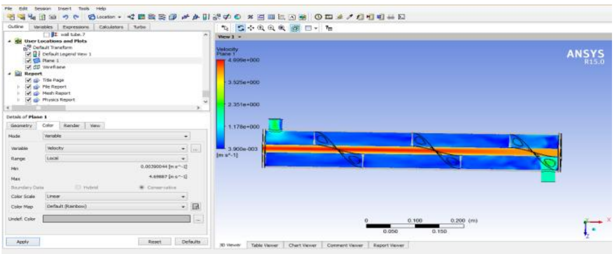

Fig 5.16 Velocity variation in helical baffle

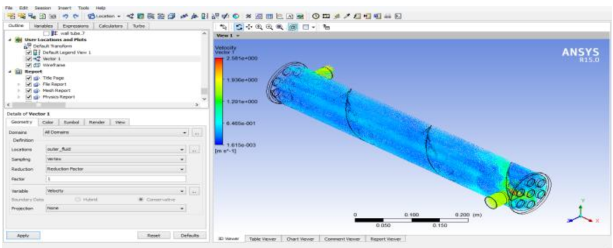

Fig 5.17Velocity variation in helical baffle

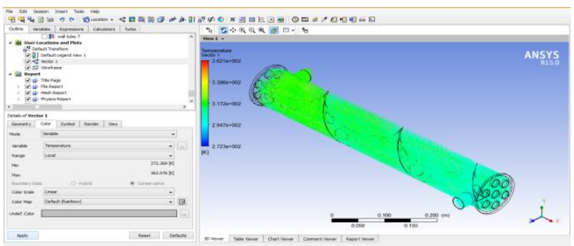

Fig 5.18 Temperature variation in helical baffle

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

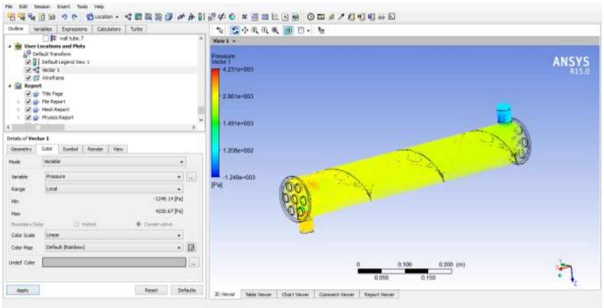

Fig 5.19 Pressure variation in helical baffle

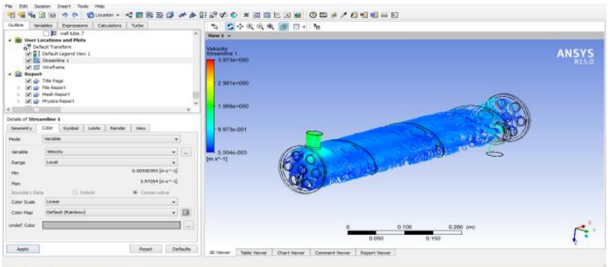

Fig 5.20 Velocity variation in helical baffle.

In this project we are calculate the shell side pressure drop of Shell and Tube Heat Exchanger by varying different types of baffles(normal baffle, inclined baffle and helical baffle). Among those, helical baffle gives effective pressure drop. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in an effective pressure drop in the heat exchanger. So helical baffle is preferred in pressure drop conditions.

Table 5.1 Calculated pressure drop by using different baffles.

Baffle Type	Shell/Tube	Pressure Drop(Pascal)
Normal Baffle	Tube	5183.781
	Shell	6291.379
Helical Baffle	Tube	5172.153
	Shell	2858.646
Inclined Baffle	Tube	5164.011
	Shell	5768.581

So helical baffles are preferred for effective pressure drop that is 2858.646 Pascal's in shell side of Shell and Tube Heat Exchanger.

6.CONCLUSION

The Shell Side Pressure Drop is discussed in detailand proposedmodel is compared withdifferent baffles. The CFDresultswhencompared withtheresultsfromdifferentstudieswere wellwithintheerrorlimits.Theassumptionworkedwell inthisgeometryandmeshing expecttheoutletandinletregionwhere rapidmixing andchange inflow directiontakesplace.Thusimprovementisexpectediftheheli calbaffleused inthe model should have complete contact with the surface of the shell, it will help in more turbulence acrossshell sideand the pressure drop willincreaseMoreoverthemodelhasprovidedthereliableres ultsby considering thestandardk-epsilonmodel. Furthermore the enhancewall function arenotuseinthisproject,butthey can be veryuseful Thepressure drop ispoorbecause mostof the fluidpasseswithoutthe interactionwithbaffles Thus the designcanbemodifiedforbetterpressure dropintwowayseitherthedecreasing theshell diameter,sothatitwillbeapropercontact withthehelicalbaffleorbyincreasing thebaffleso thatbaffleswillbepropercontact withtheshell.

In this project we are calculate the shell side pressure drop of Shell and Tube Heat Exchanger by varying different types of baffles(normal baffle, inclined baffle and helical baffle). Among those, helical baffle gives effective pressure drop. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in an effective pressure drop in the heat exchanger. So helical baffle is preferred in pressure drop conditions. So helical baffles are preferred for effective pressure drop that is 2858.646 Pascal's in shell side of Shell and Tube Heat Exchanger.

7.REFERENCES

[1] M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis Of Shell And Tube

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Heat Exchanger Using Miscible System, American Journal Of Applied Sciences

[2] Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011.

[3] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, A Design And Rating Method For Shell-And-Tube Heat Exchangers With Helical Baffles, Journal Of Heat Transfer, May 2010.

[4] Muhammad Mahmoud AslamBhutta, Nasir Hayat, Muhammad Hassan Bashir, AhmerRais Khan, KanwarNaveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review,

[5] ŽarkoStevanović, GradimirIlić, Design of Shell-And-Tube Heat Exchangers by Using CFD Technique, University Of Niš, Fr, 2002.

[6] Ender Ozden, Ilker Tari, Shell Side CFD Analysis of a Small Shell and Tube Heat Exchanger, Middle East Technical University, 2010.

[7] Kern DQ. Process heat transfer. New York, N.Y.: McGraw-Hill; 1950.

[8] 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),

[9] G. V. Gaddis ES, "Pressure drop on the shell side of shell-and-tube heat exchangers withsegmental baffles,"

[10] S. Kakac and H. Liu, "Heat exchangers: Selection, rating, and thermal performance," 1998.

[11] Ozden E. Detailed design of shell-and-tube heat exchangers using CFD, M.S.Thesis, Middle East Technical University, Ankara, Turkey, 2007.

[12] Kano, A., and A jib, S., 2006, "Effect of tube pitch on heat transfer in shell-and-tube heat exchangers—new simulation software Heat Mass Transfer 42, pp. 263–270.

[13] KhairunHasmadi Othman, CFD Simulation of Heat Transfer in Shell and Tube Heat Exchanger, University Malaysia Pahang, April 2009.

[14] Tinker, T., "Shell side Characteristics of Shell-and-tube Heat Exchangers:

[15] Willis, M.J.N. and D. Johnston, 1984. A New and Accurate Hand Calculation Method for Shell Side Pressure
Drop and Flow Distribution.

[16] KakaQ. S., & Liu. H. Classification of heat exchangers in, Heat exchangersSelection, rating, and thermal design. CRC Press LLC, Florida, 2002.