Modification of Shell Type Heat Exchanger to Helical Baffle Heat Exchanger

Pokkalath Anjali Shivdas
Heat and Power Engineering
University of Mumbai
Rajiv Gandhi Institute of Technology
a.pokkalath@gmail.com

Siddharth Saindane
Heat and Power Engineering
University of Mumbai
Rajiv Gandhi Institute of Technology

Abstract - These instructions give you the basic guidelines for preparing papers Shell and tube heat exchangers with helical baffles are used for improved performance by reducing pressure drop, vibration, and fouling while maintaining a higher heat transfer capability. In the present study, a 3D numerical simulation of a Shell and tube heat exchanger with a continuous helical baffle is carried out by using commercial codes of FLUENT 6.3. An experimental analysis and numerical comparison is provided that examines developments and improvements on a conventional Shell and Tube heat exchanger (STHX) and a Shell and tube heat exchanger with a continuous helical baffle (STHXHB).The analysis has been made for both cold and hot fluid. It was found that the increase in total heat transfer rate is 09% to 23% for the STHXHB compared with STHX for different hot fluid velocities. It is also concluded that STHXHB have a higher total heat transfer rate and a lower pressure drop when compared to the STHX for the same mass flow rate and inlet condition. There is good agreement between numerical and experimental results.

*****

I. Introduction

Heat exchangers play an important role in many engineering processes such as oil refining, the chemical industry, environmental protection, electric power generation, refrigeration, and so on. Among different types of heat exchangers, shell-and-tube heat exchangers have been commonly used in industries. It has reported that more than 35 – 40% of heat exchangers are of the shell-and-tube type, because of their robust construction geometry as well as easy maintenance and possibility of upgrades. In order to meet the special requirements of modern industries, various ways are adopted toenhance the heat transfer performance while maintaining a reasonable pressure drop for the STHXs. One useful method is by using baffles to change the direction of the flow in the shell side to enhance turbulence and mixing. For many years, various types of baffles have been designed, examples being, conventional segmental baffles with different arrangements, deflecting baffles, overlapping helical baffles, the rod baffles, and others. The most commonly used segmented baffles make the fluid flow in a tortuous, zigzag manner across the tube bundle in the shell side where they improve the heat transfer by enhancing turbulence and local mixing. However, the traditional STHXs with segmental baffles have many disadvantages, these being high pressure drop on the shell side due to the sudden contraction and expansion of the flow; and fluid impinging on the shell wall caused by these baffles; low heat transfer efficiency due to the flow stagnation in the so-called ―stagnation regions, which are located at the corners between baffles and shell wall; low shell-side mass velocity across the tube bundle due to the leakage between baffles and shell wall caused by inaccuracy in manufacturing tolerance and installation; and short operation time due to the vibration caused by shell-side flow normal to tube bundle. When the traditional segmental baffles are used in STHXs, higher pumping power is often needed to offset the higher pressure drop for the same heat load. During the past decades, deflecting baffles, rod baffles, and disk-and-doughnut baffles have been developed to solve these shortcomings. However, none of these baffle arrangements can solve all the principal problems mentioned earlier. New designs are still needed to direct the flow in plug flow manner, to provide adequate support to the tubes, and to provide a better thermodynamic performance. The shell-and-tube heat exchanger with helical baffles (STHXHB) is usually called a helix changer. It was invented in the Czech Republic and is commercially produced by ABB Lummus Heat Transfer. Helical baffles offer a possible alternative to segmental baffles by circumventing the aforementioned problems of conventional segmental baffles. They are accepted for their outstanding advantages,
including:

(1) Improved shell side heat transfer rates and pressure drop ratio
(2) Reduced bypass effects
(3) Reduced shell-side fouling
(4) Prevention of flow induced vibration
(5) Reduced maintenance.

In the past decades, the STHXHB types have been continuously developed and improved and have been widely accepted by engineers. Therefore, the objectives of this study are to develop STHX with continuous helical baffles and to investigate their performance.

In this work,
(1) STHX with continuous helical baffles were designed and tested,
(2) A simple and feasible method was developed to fabricate the continuous helical baffles used for STHX,
(3) The heat transfer rate and pressure drop of the STHX with continuous helical baffles were compared with those of the STHX with segmental baffles numerically and with STHXHB experimentally.

Purpose of Use of Helical Baffle:
A new type of baffle, called the helical baffle, provides further improvement. This type of baffle was first developed by Lutcha and Nemcansky. They investigated the flow field patterns produced by such helical baffle geometry with different helix angles. They found that these flow patterns were very close to the plug flow condition, which was expected to reduce shell-side pressure drop and to improve heat transfer performance. Stehlik et al. compared heat transfer and pressure drop correction factors for a heat exchanger with an optimized segmental baffle based on the Bell–Delaware method, with those for a heat exchanger with helical baffles. Kral et al. discussed the performance of heat exchangers with helical baffles based on test results of various baffles geometries. One of the most important Geometric factors of the STHXHB is the helix angle. Recently a comprehensive comparison between the test data of shell-side heat transfer coefficient versus shell-side pressure drop was provided for five helical baffles and one segmental baffle measured for oil-water heat exchanger. It is found that based on the heat transfer per unit shell-side fluid pumping power or unit shell-side fluid pressured drop, the case of 400 helix angle behaves the best. 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 a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger. Properly designed continuous helical baffles can reduce fouling in the shell side and prevent the flow-induced vibration as well. The performance of the proposed STHXs was studied experimentally in this work. The use of continuous helical baffles results in nearly 10% increase in heat transfer coefficient compared with that of conventional segmental baffles for the same shell-side pressure drop. Based on the experimental data, the non dimensional correlations for heat transfer coefficient and pressure drop were developed for the proposed continuous helical baffle heat exchangers with different shell configurations, which might be useful for industrial applications and further study of continuous helical baffle heat exchangers.

Computational Fluid Dynamics (CFD):
CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modelling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design. Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyse problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are on going research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger. Furthermore, motor vehicle manufactures now routinely predict drag forces, under bonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

APPLICATION OF CFD:
CFD not just spans on chemical industry, but a wide range of industrial and non-industrial application areas which is in below:

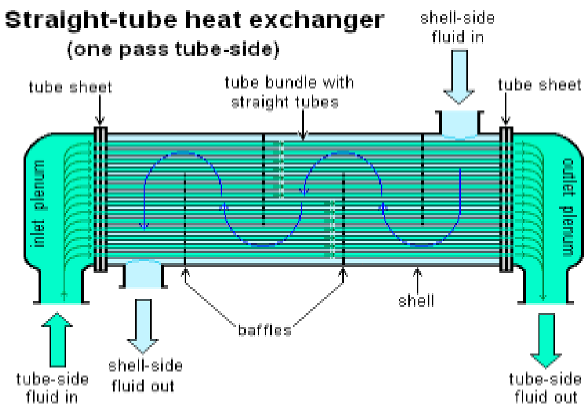
- Aerodynamics of aircraft and vehicle.
- Combustion in IC engines and gas turbine in power plant.
- Loads on offshore structure in marine engineering.
- Blood flows through arteries and vein in biomedical engineering.
- Weather prediction in meteorology.
- Flow inside rotating passages and diffusers in turbo-machinery.
- External and internal environment of buildings like wind loading and heating or
- Ventilation system.
- Mixing and separation or polymer moldings in chemical process engineering.
- Distribution of pollutants and effluent in environmental engineering.

ANSYS:

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

In Ansys following Basic step is followed:

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



In simple type heat exchangers, it has low heat transfer rate.

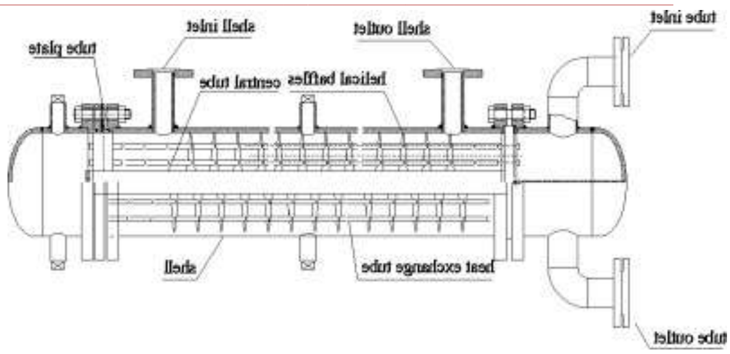
It also has high fouling factor.

Low efficiency.

Maintenance cost is high because of large size.

By pass factor is high.

Vibration is high.



As we can see there are many problems in shell type heat exchanger, so to rectify those problems, we should modify from straight baffle to helical baffle heat exchanger.

Advantages of Helical type heat exchanger:

Considering a Straight baffle type heat exchanger, assuming its efficiency is 'X'. For same input for helical type heat exchanger, we can get its efficiency as 'X+10'. So from above we can say that, so as to get same output from both heat exchangers, we can reduce the size of helical heat exchanger.From above points we can say that, we can reduce the 'SIZE' of helical type heat exchanger. As well as reducing its 'PRODUCTION COST'.Since drag in helical shape is less, therefore Pressure drop is negligible in case of helical type heat exchanger, so 'FOULING FACTOR' is reduced drastically.Vibration also decreases.Since fluid flow over helical baffle is dragless, therefore it has more heat transfer rate as compared to Simple heat exchanger.

ACKNOWLEDGMENT

The work done by many Authors around the world has given us the inspiration and motivation to go with this paper. We Specially thank Mr.Mandar Chalke to help us with the software part of the paper. We also take the opportunity to thank Prof: Deshmukh to help us learn the modifications that can be made on the existing design and thus increase the scope of this paper.

REFERENCES

[1] 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.

[2] Haseler, L.E., Wadeker, V.V., Clarke, R.H., (1992), "Flow Distribution Effect in a Plate and Frame Heat Exchanger", IChemE Symposium Series , No. 129, pp. 361-367.

[3] Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.

[4] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering,china.

[5] Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int.J. Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.

[6] Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (2008), pp. 548-552.

[7] Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD"

[8] Professor Sunilkumar Shinde, Mustansir Hatim Pancha / International Journal of Engineering Research and Applications (IJERA),"Comparative Thermal performance of shell and tube heat Exchanger with continuous helical baffle using ", Vol. 2, Issue4, July-August 2012