ISSN: 2456–5474 RNI No.UPBIL/2016/68367 Vol-2* Issue-9* october- 2017
Innovation The Research Concept

A Study on the Design of Shell and Tube Heat Exchanger with Helical Baffle

Abstract

Now a days, Shell and tube heat exchanger is that commonest kind device wide employed in refinery and alternative giant natural process, as a result of it suits high pressure application.The method in resolution simulation consists of modeling and meshing the fundamental mathematics of shell and tube device by CFD package ANSYS 13.0.The target of the project is to design of shell and tube device with spiraling or helical baffle and study the flow and temperature field within the shell by ANSYS code tools. This device contains seven tubes and 600 millimeter length shell diameter ninety millimeter.The angle of spiraling or helical baffle are varied from 0 degree to 20 degree. In simulation can show however the pressures vary in shell because of totally different angle and rate. The flow pattern within the shell side of the heat exchanger with continuous spiraling or helical baffles was forced to be motion and spiraling or helical because of the pure mathematics of the continual spiraling baffles, which ends during a vital increase in heat transfer cofficient per unit pressure drop in the device.

Kanchan Kumari
Student,
Deptt.of Mechanical Engg.,
Patel Institute of Engineering and Science,
Bhopal, M.P.

Bittu Pathak
Assistant Professor,
Deptt.of Mechanical Engg.,
Patel Institute of Engineering and Science,
Bhopal, M.P.

Amit Kumar
Assistant Professor,
Deptt.of Mechanical Engg.,
Patel Institute of Engineering and Science,
Bhopal, M.P.

Keywords: Heat Exchanger, Helical Baffle, CFD, ANSYS Tools, Simulation.

Introduction

Whenever the heat exchanger is using in industry, the main purpose is to increase the maximum efficiency of heat exchanger in future amount.To achieve maximum efficiency many problems come .Most important problems is fouling this causes corrosion and erosion fouling. It is eliminated by reducing velocity of stream. Another problem will appear exactly after reducing the velocity of the stream. The problem is to determine the second stream condition because in heat exchanger there are minimum two streams. These two streams will transfer energy each other without having a contact each other. If there are any changes made to one stream, it must be changes for another stream.

Heat exchangers square measure is one among the principally used instrumentality within the method industries. Heat exchangers square measure used to transfer heat between 2 method streams. One will notice their usage that any method which at involve cooling, heating, condensation, boiling or evaporation would require a device for these purpose. Method fluids, sometimes square measure heated or cooled before the method or endure a physical change. Completely different heat exchangers square measure named in line with their application. As an example, heat exchangers being employed to condense is understood as condensers, equally device for boiling functions square measure referred to as boilers. Performance and potency of warmth exchangers square measure measured through the number of warmth transfer mistreatment least space of warmth transfer and pressure drop. a more robust presentation of its potency is finished by conniving over all heat transfer constant. Pressure drop and space needed for a definite quantity of warmth transfer, provides AN insight concerning the cost of capital and power necessities (Running cost) of a device. Usually, there's voluminous literature and theories to style a device in line with the wants.

Heat exchangers square measure are of 2 types:

1. Where each media between that heat is changed square measure in direct contact with one another is Direct contact device,
2. Where each media square measure separated by a wall through that heat is transferred so they ne'er combine, indirect contact device.

To remove problems we take this topics because in this research heat transfer and pressure drop reduces with an average error of 20%.

Objective of the Study

1. To study the process in solving simulation consists of modeling and meshing the basic geometry of shell and tube heat exchanger
2. To design a shell and tube heat exchanger with helical baffle and study the flow and temperature field inside the shell using ANSYS software tools.

Review of Literature

In previous researches, a model has been developed to evaluate analysis of a helical and segmental baffle heat exchanger as well as the comparative analysis between the thermal parameters between the segmental and helical angle has been showed. From several numerical experimentation results from previous researches it is found that the performance of a tubular heat exchanger can be improved by helical baffles instead of segmental baffles. It is observed from the earlier studies that use of helical baffles in heat exchanger reduces shell side pressure drop, pumping cost, weight, fouling etc as compare to segmental baffle for a new installation.

Moreover, the ratio of heat to increased cross flow area resulting in lesser mass flux throughout the shell transfer coefficient to pressure drop as higher than that of segmental baffle. On the other hand, the pressure drop in helical baffle heat exchanger is appreciably lesser as compared to segmental baffle heat exchanger. Thus, it could be concluded from previous researches that helical baffle is the much higher than the segmental baffle because of reduced by pass effect &reduced shell side fouling. The helical baffle is three times higher than the segmental baffle.

Concept and Hypothesis

In this research we uses the basics concepts of heat exchanger and type of heat exchanger. We also uses the concept of helical and segmental baffle, ANSYS tools and CFD.

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:

1. During preprocessing 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 modeling is defined. Boundary condition is defined. This involves specifying the fluid behaviour of the problem. For transient problem boundary condition are also defined.
2. The simulation is started and the equations are solved iteratively as steady state or transient.
3. Finally a post procedure is used for the analysis and visualization of the resulting problem.

CFD

CFD could be a subtle computationally-based style and analysis technique. CFD code offers you the facility to simulate flows of gases and liquids, heat and mass transfer, moving bodies, polyphase physics, reaction, fluid-structure interaction and acoustics through pc modeling. This code may also build a virtual model of the system or device before are often apply to real-world physics and chemistry to the model, and therefore the code can give with pictures and knowledge, that predict the performance of that style.

Research Design

In current works we study the exchange of heat in shell and tube device with the help of helical baffle instead of segmental. Because helical baffle in device reduces shell side pressure, pumping cost, weight, fouling etc. This model improved by using ANSYS 13.0 tools and CFD. The angle of helix varies from 0 to 20 degree. With the help of this helix angle we measure the effect of pressure, temperature and velocity on heat exchanger. The universe of my study is to increase the efficiency of heat exchanger for present and future use.

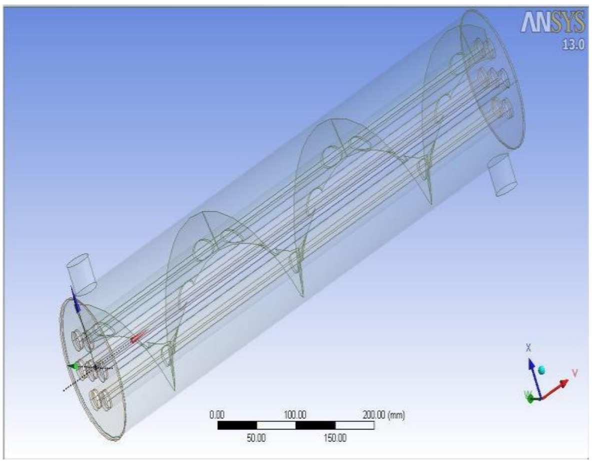
Computational Model

The computational model of an experimental tested Shell and Tube Heat Exchanger (STHX) with 10 helix angle is shown in fig., and the geometry parameters are listed in Table. As can be seen from Fig , the simulated STHX has six cycles of baffles in the shell side direction with total number of tube 7.The whole computation domain is bounded by the inner side of the shell and everything in the shell contained in the domain. The inlet and outlet of the domain are connected with the corresponding tubes.

To simplify numerical simulation, some basic characteristics of the process following assumption are made:

1. The shell side fluid is constant thermal properties
2. The fluid flow and heat transfer processes are turbulent and in steady state
3. The leak flows between tube and baffle and that between baffles and shell are neglected
4. The natural convection induced by the fluid density variation is neglected
5. The tube wall temperature kept constant in the whole shell side
6. The heat exchanger is well insulated hence the heat loss to the environment is totally neglected.

Isometric Views of Arrangement of Baffles and Tubes of Shell and Tube Heat Exchanger with Baffle Inclination



Geometric Dimensions of Shell and Tube Heat Exchanger

Heat exchanger length, *L*	600mm
Shell inner diameter, *Di*	90mm
Tube outer diameter, do	20mm
Tube bundle geometry and pitch Triangular	30mm
Number of tubes, *Nt*	7
Number of baffles. *Nb*	6
Central baffle spacing, *B*	86mm
Baffle inclination angle, θ	0 to 40^0

Geometry and Mesh

The model is designed according to TEMA (Tubular Exchanger Manufacturers Association) Standards Gaddis (2007).

Grid Generation

The three-dimensional 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 and six solid domains namely Solid_Baffle1 to Solid_Baffle6 for six baffles respectively.

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 then solid mesh for simulating conjugate heat transfer phenomenon. The three fluid domains are as shown in Fig. The first cell height in the fluid domain from the tube surface is maintained at 100 microns to capture the velocity and thermal boundary layers. The discretized model is checked for quality and is found to have a minimum angle of 18° and min determinant of 4.12. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS CFX pre-processor.

Meshing

Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. 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, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

Problem Setup

Simulation was carried out in ANSYS® FLUENT® v13. In the Fluent solver Pressure Based type was selected, absolute velocity formation and steady time was selected for the simulation. In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function.

In cell zone fluid water-liquid was selected. Water-liquid and copper, aluminum was selected as materials for simulation. Boundary condition was selected for inlet, outlet. In inlet and outlet 1kg/s velocity and temperature was set at 353k.Across each tube 0.05kg/s velocity and 300k temperature was set. Mass flow was selected in each inlet. In reference Value Area set as 1m2, Density 998 kg/m3, enthalpy 229485 j/kg, length 1m, temperature 353k, Velocity 1.44085 m/s, Ration of specific heat 1.4 was considered.

Solution Initialization

Pressure Velocity coupling selected as SIMPLEC. Skewness correction was set at zero. In Spatial Discretization zone Gradient was set as least square cell based, Pressure was standard, Momentum was First order Upwind, Turbulent Kinetic energy was set as First order Upwind, and Energy was also set as First order upwind. In Solution control, Pressure was 0.7, Density 1, Body force 1, Momentum 0.2, turbulent kinetic and turbulent dissipation rate was set at 1, energy and turbulent Viscosity was 1. Solution initialization was standard method and solution was initializing from inlet with 300k temperature.

Result and Finding

After research and study of this topic, model predicts that the heat transfer and pressure drop with average error of 20% by comparing with increasing baffle angle .The taken assumption play well in this geometry and meshing expects the working of outlet and inlet region of heat exchanger .This model only expected if helical baffle used should have complete contact with surface of shell,it may be help more turbulence near shell side and heat transfer rate will increase. According to the baffle inclination :

1. For Zero degree baffle inclination answer was converged at a hundred and seventieth iteration.
2. Simulation of 10^0 Baffle inclinations is converged at 133th iteration.
3. Simulation of 20^0 baffle inclination is converged at 138^{th} iteration.

Conclusion

The heat transfer and flow processes are explain in details and model is compared with increasing baffle inclination angle. The model finds that the heat transfer and pressure drop with an average error of 20%.Thuse the model can be improved. The assumption worked well in this geometry and meshing expects the outlet and inlet region where rapid mixing and change in flow direction takes place. So improvement is predicted if the helical baffle utilized in the model ought to have complete contact with the surface of the shell, it facilitate in additional turbulence across shell aspect and therefore the heat transfer rate will increase. If completely different rate is taken, it would be facilitate to get better heat transfer and to get better temperature difference between inlet and outlet. Moreover the model has provided the reliable results by considering normal| k-e and standard wall perform model, however this model over predicts the turbulence in regions with large normal strain. The heat transfer rate is poor as a result of most of the fluid passes while not the interaction with baffles. So the planning will be changed for higher heat transfer in 2 ways that either the decreasing the shell diameter, so it'll be direct contact with the coiled baffle or by increasing the baffle so baffles are going to be proper contact with the shell. It's as a result of the heat transfer space isn't utilised with efficiency. So the design will further be improved by making cross-flow regions in such how that flow doesn't stay parallel to the tubes. It'll permit the outer shell fluid to own contact with the inner shell fluid, so heat transfer rate can increase.

Suggestion

This model can also be improved by using Nusselt no. and Reynolds stress model, but with higher computational theory. Moreover the enhance wall perform aren't use during this project, however they will be terribly helpful. We also know heat exchanger is very useful for industries because of heat removal process.Without this the other process of industries can not work well. So in my opinion this model is very important and usefull.

References

1. *B. Sund´en. Introduction to heat transfer.*
2. *Khairun Hasmadi Othman, " CFD simulation of heat transfer in shell and tube heat exchanger",.*
3. *url:https://www.modelica.org/education/educationalmaterial/lecture-material/english/ModelicaOverview.pdf.*
4. *url: http : / / www . modelon . com / products / modelica-libraries/.*
5. *Uday V. Shenoy. Heat exchanger network synthesis. Gulf publishing company, 1995.*
6. *TEMA. Standard Tubular Exchanger Manufacturers Association, eight edition. Tubular Exchanger Manufacturers Association, Inc, 1999.*
7. *R. K. Shah and D. P. Sekulic. Fundamentals of heat exchanger design. John Wiley and Sons, Inc, 2003.*
8. *L. Friedel. Improved Friction Pressure Drop Correlations for Horizontal and Vertical Two-Phase Pipe Flow. 1979.*