[Pandita*, 4.(6): June, 2015] **ISSN: 2277-9655**
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CFD ANALYSIS OF A SINGLE SHELL AND SINGLE TUBE HEAT EXCHANGER AND DETERMINING THE EFFECT OF BAFFLE ANGLE ON HEAT TRANSFER

Manoj Kumar Pandita*, Prof Ashok Kumar Gupta, Kailash Kumar Lahadotiya
* Dept. of Mechanical Engineering Lakshmi Narain College of Technology & Science, Indore (MP) India.
Dept. of Mechanical Engineering Lakshmi Narain College of Technology & Science, Indore (MP) India.
Dept. of Mechanical Engineering Lakshmi Narain College of Technology & Science, Indore (MP) India.

ABSTRACT

The energy present in the exit stream of many energy conversion devices such as I.C engine gas turbine etc. goes as waste, if not utilized properly. The present work has been carried out with a view to predicting the performance of a shell and tube heat exchanger. The performance of the heat exchanger has been evaluated by using the CFD package Fluent for different baffle angles. An attempt has been made to calculate the performance of the heat exchanger by varying the baffle angles and the results obtained have been compared. The baffles inclination dependencies of the heat transfer coefficient and the pressure drop are investigated by numerically modelling a small heat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD software tool ANSYS FLUENT

In this present work, attempts were made to investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for three different baffles inclination angles namely 0°, 45° and -45°. The simulation results for shell and tube heat exchangers, one with segmental baffles perpendicular to fluid flow and two with segmental baffles inclined to the direction of fluid flow are compared for their performance. The results are observed to be sensitive to the baffle angles. The steady state heat transfer is found to be more in the case of 45 degrees as compared to the -45 degrees case.

KEYWORDS: Shell-and-tube heat exchanger, CFD, Conjugate Temperature Distribution, Pressure drop, Baffle inclination angle, turbulence models.

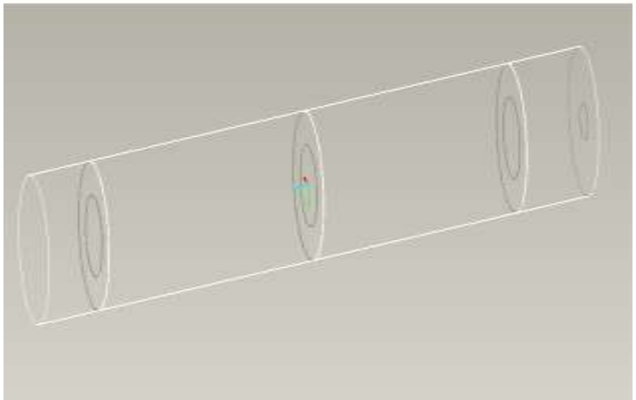
INTRODUCTION

Heat exchangers have always been important part to lifecycle and operation of many systems. A heat exchanger is a device built for efficient heat transfer from one medium to another in order to carry and process energy. Typically one medium is cooled while the other is heated. They are widely used in petroleum refineries, chemical plants, petrochemical plants, natural gas processing, air conditioning, refrigeration and automotive applications. One common example of a heat exchanger is the radiator in a car, in which it transfers heat from the water (hot engine-cooling fluid) in the radiator to the air passing through the radiator. There are two main types of heat exchangers. Direct contact heat exchanger, where both media between which heat is exchanged are in direct contact with each other. Indirect contact heat exchanger, where both media are separated by a wall through which heat is transferred so that they never mix. Shell and tube type heat exchanger is an indirect contact type heat exchanger as it consists of a series of tubes, through which one of the fluids runs. The tube side and shell side fluids are separated by a tube sheet, Gaddis [1], Schlunder [2], Mukherjee [3]. The heat exchanger model used in this study is a small sized one, as compared to the main stream, all of the leakage and bypass streams do not exist or are negligible, Ender Ozden and Ilker Tari [4] , Uday Kapale and Satish Chand[5], Thirumarimurugan et al. [6]. Baffles are used to support the tubes for structural rigidity, preventing tube vibration and sagging and to divert the flow across the bundle to obtain a higher heat transfer coefficient. Baffle spacing is the centre line distance between two adjacent baffles, Sparrow and Reifschneider [7], Li and Kottke [8], Su Thet Mon Than et al. [9]. Baffle is provided with a cut (Bc) which is expressed as the percentage of the segment height to shell inside diameter. Baffle cut can vary between 15% and 45% of the shell inside diameter, Kakac and Liu [10], Gay et al. [11], Emerson [12]. In general, conventional shell and tube heat exchangers result in high shell-side pressure drop and formation of recirculation zones near the baffles. Most of the researches now a day are carried on helical baffles, which give better performance then single segmental baffles but they involve high manufacturing cost, installation cost and maintenance cost. The effectiveness and cost are two important parameters in heat exchanger design. So, In order to improve the thermal performance at a reasonable cost of the Shell and tube heat exchanger, baffles in the present study are provided with some inclination in order to maintain a reasonable pressure drop across the exchanger Yong-Gang Lei et al. [13]. The complexity with experimental techniques involves quantitative description of flow phenomena using measurements dealing with one quantity at a time for a limited range of problem and operating conditions. Computational Fluid Dynamics is now an established industrial design tool, offering obvious advantages Versteeg and Malalasekera [14].

In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modelling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained. In this study, a small shell-and-tube heat exchanger is modelled for CFD simulations. A commercial CFD package, ANSYS Fluent is used as a mesh generation software and analysis.

MODELLING DETAILS

In this study, a small heat exchanger is selected in order to increase the model detail and to make solid observations about the flow inside the shell. The material of the shell and tube is Mild Steel with a thermal conductivity of 50 w/mK .Some of the design parameters and the predetermined geometric parameters are presented below.. The geometric model with three baffles is shown in Figure.



The working fluid of the shell side is air. In this study three baffles are placed along the shell in alternating orientations with cut facing up and cut facing down. The geometric model is optimized by varying the baffle inclination angle *i.e.*, 0°, 45° and -45°. The computational modelling involves pre-processing, solving and post- processing.

Geometric Parameters

Heat exchanger length:	3000 mm
Shell inner diameter:	300 mm
Shell Thickness:	3 mm
Tube outer diameter:	100 mm
Tube Thickness:	3 mm
Number of tubes:	1
Number of baffles:	3
Central baffle spacing:	1100 mm
Baffle inclination angle:	0° ,45° and -45°
Baffles Outer Diameter:	300 mm
Baffles Inner Diameter:	120 mm
Baffles: Thickness:	3mm

Governing equations

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady, time dependent parameters are dropped from the equations. The resulting equations are:

Conservation of mass: $\nabla(\rho Vr) = 0$ (1)

x-momentum: $\nabla.(\rho u Vr) = -\,\partial p/\partial x + \partial\tau xx/\partial x + \partial\tau yx/\partial y + \partial\tau zx/\partial z$ (2)
y-momentum: $\nabla.(\rho v Vr) = -\,\partial p/\partial y + \partial\tau xy/\partial x + \partial\tau zy/\partial y + \partial\tau zx/\partial z + \rho g$ (3)
z-momentum: $\nabla.(\rho w Vr) = -\,\partial p/\partial z + \partial\tau xz/\partial x + \partial\tau yz/\partial y + \partial\tau zz/\partial z + \rho g$ (4)

Energy: $\nabla.(\rho e Vr) = -p\nabla.Vr + \nabla.(k.\nabla T) + q\cdot\ \varphi$ (5)

In Eq. (5), φ is the dissipation function that can be calculated from

$\varphi = \mu\ [\ 2[(\partial u/\partial x)^2 + (\partial v/\partial y)^2 + (\partial w/\partial z)^2] + (\partial u/\partial y + \partial v/\partial x)^2 + (\partial u/\partial z + \partial w/\partial z)^2 + (\partial v/\partial z + \partial w/\partial y)^2] + \lambda(\nabla.Vr)^2$ (6)

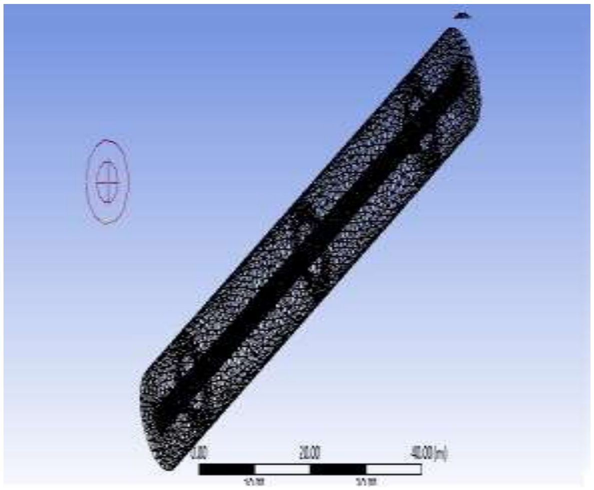
Boundary conditions

Boundary conditions:-

1. The working fluid of the shell side is air
2. The air entering tube or tube inlet temperature is set to 300 K,
3. The air entering the Shell from the other side is at 500 K
4. Zero gauge pressure is assigned to the outlet of the tube.
5) The mass flow rate of cold air inlet in the shell is 0.0942 Kg/s
6) The mass flow rate of hot air inlet in the shell is 0.7536 Kg/s
5. The inlet velocity profile is assumed to be uniform,
6. No slip condition is assigned to all surfaces,
7. The zero heat flux boundary condition is assigned to the shell outer wall

Mesh selection

Mesh generation is performed using ANSYS..The shell volume is meshed using tetragonal elements. The meshed model is shown in the figure below:



Turbulence model

Since the flow in this study is turbulent, turbulence effects should be taken into account using turbulence modelling. The choice of turbulence model is very critical in CFD simulations. However, there is no universal criterion for selecting a turbulence model. In this study, k-ε turbulence model are tried. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy *k* and its dissipation rate ε. For steady state, *k* and *ε* are obtained from the following transport equations:

$\partial/\partial xi\ (\rho k ui) = \partial/\partial xj\ [\ (\mu + \mu i/\sigma k)\ \partial k/\partial xj] + Gk + Gb - \rho\varepsilon + Sk$ (7)

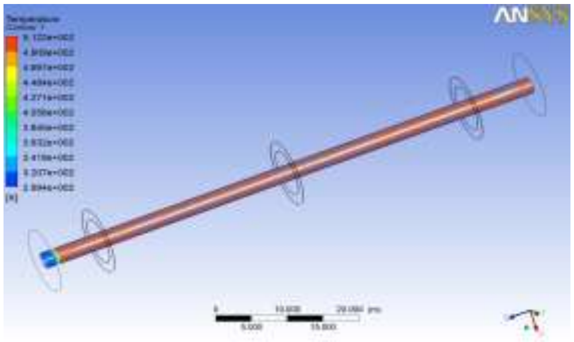
$\partial/\partial xi\ (\rho\varepsilon ui) = \partial/\partial xj\ [\ (\mu + \mu i/\sigma\varepsilon)\ \partial\varepsilon/\partial xj] + C1\varepsilon\ \varepsilon/k(Gk + C3\varepsilon Gb) - C2\varepsilon\rho\varepsilon^2/k + Sk$ (8)

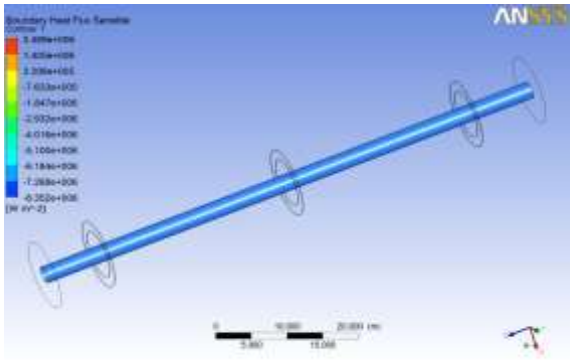
And the turbulent viscosity is defined by the following equation:

$\mu t = \rho C\mu k^2/\varepsilon$ (9)

RESULTS AND DISCUSSION

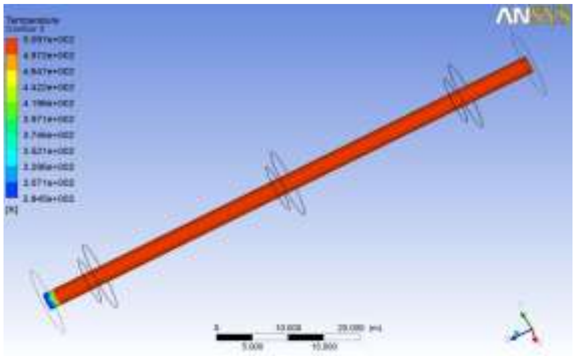
The results (Temp Profile and Heat Flux) for heat exchanger with 0 degree baffles in a counter flow heat exchanger are as follows:

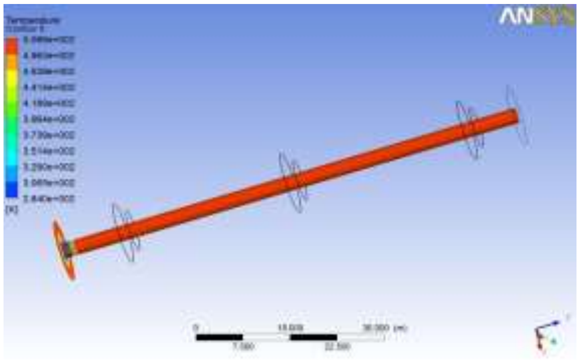




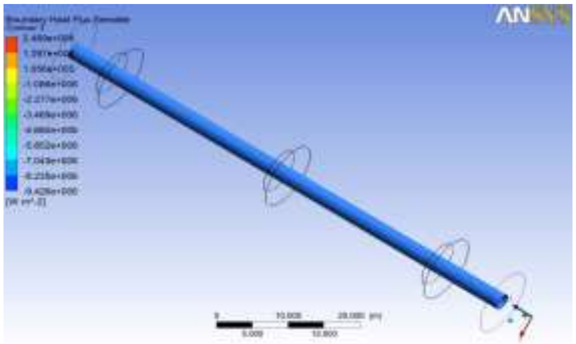
For the counter flow case with air as fluid the temperature distribution, heat flux of the heat exchangers with 45 degrees baffles and -45 degrees baffles are compared below:

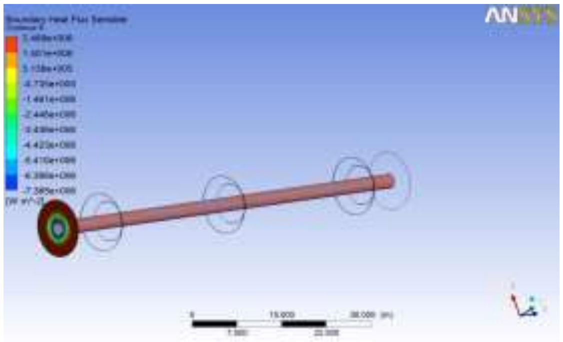
a) Temperature Distribution





b) Heat Flux





It turns out that the temperature distribution of the steady state counter flow heat exchanger has minor differences but the steady state heat flux turns out to be more in the case of 45 degrees baffles than the -45 degrees baffles. The reason for the same is that the shell side fluid is supported by 45 degree baffle direction and is opposed by -45 degree baffles direction. The heat flux for the zero degree case falls between the 45 degrees and -45 degrees case.

Table: Heat Flux in Different Baffle Profiles

S.No	Geometry	Heat Flux in 10^6 W/m2
1	-45 Degrees	-7.385
2	0 Degrees	-8.352
3	+45 Degrees	-9.426

CONCLUSIONS

A small shell-and-tube heat exchanger is modelled with sufficient detail to resolve the flow and temperature fields. Following conclusions are drawn from the present study:

1. There were minor differences in the steady state temperature distribution of the counter flow heat exchanger modelled with +45 degree baffles and -45 degrees baffles.
2. The steady state heat flux comes out to be more in the case of +45 degree baffles case than -45 degrees baffles case.
3. The heat flux comes of 0 degree baffles come out intermediate between 45 and -45 degrees cases.

Thus any inclination of the baffles in the positive direction will increase the heat transfer and any angle or inclination in the negative direction will reduce the heat flux.

REFERENCES

1. Gaddis, D., editor. Standards of the Tubular Exchanger Manufacturers Association, ninth ed, Tarrytown (NY): TEMA Inc, 2007.
2. Schlunder, E.V, Heat Exchanger Design Handbook, Hemisphere Publishing Corp., New York, Bureau of Energy Efficiency, 1983.
3. Mukherjee, R., Practical Thermal Design of Shell-and-Tube Heat Exchangers, Begell House.Inc, New York, 2004.
4. Ender Ozden, Ilker Tari, Shell side CFD analysis of a small shell-and-tube heat exchanger, Energy Conversion and Management 51 (2010), pp. 1004 – 1014.
5. Uday Kapale, C., Satish Chand, Modeling for shell-side pressure drop for liquid flow in shell-and-tube heat exchanger, International Journal of Heat and Mass Transfer 49 (2006), pp. 601–610
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. Sparrows, E. M., Reifschneider, L. G., Effect of inter baffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger, International Journal of Heat and Mass Transfer 29 (1986), pp. 1617-1628.
8. Li, H., Kottke, 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 (1998), 10, pp. 1303–1311
9. Su Thet Mon Than, Khin Aung Lin, Mi Sandar Mon, Heat Exchanger Design, WorldAcademy of Science, Engineering and Technology 46, 2008.
10. Kakac, S., Liu, H., Heat Exchangers Selection, Rating and Thermal Design, CRC press,second ed, Washington D.C., 2002, pp. 318–335
11. Gay, B., Mackley, N.V., Jenkins, J. D., Shell-side heat transfer in baffled cylindrical shell andtube exchangers, Int. J. Heat Mass Transfer 19 (1976), pp. 995-1002.
12. Emerson, W.H., Shell-side pressure drop and heat transfer with turbulent flow in segmentally baffled shelltube heat exchangers, Int. J. Heat Mass Transfer 6 (1963), pp. 649–668.
13. Yong-Gang Lei, Ya-Ling He, Rui Li, Ya-Fu Gao, Effects of baffle inclination angle on flow and heat transfer of a heat exchanger with helical baffles, Chemical Engineering and Processing 47 (2008), pp. 2336– 2345.
14. Versteeg, H.K., Malalasekera, W., An introduction to computational fluid dynamics: the finite volume method, first ed, Essex (England): Pearson, 1995.
15. Incropera FP, Dewitt DP. Fundamentals of heat and mass transfer. 4th ed. New York: J. Wiley; 1996.