International Conference on Advances in Materials, Computing and Communication Technologies
In Association with International Journal of Scientific Research in Science and Technology
Volume 9 | Issue 1 | Print ISSN: 2395-6011 | Online ISSN: 2395-602X (www.ijsrst.com)

CFD Analysis of Shell and Tube Heat Exchanger with Different Baffle Arrangements

K.Adithya[1], S.Naveen Gandhi[2], M.P.Prasannamoorthi[2], M.Srithar Nishanth[2], K.Vijay[2]
[1]Assistant Professor, Department Of Mechanical Engineering, Nandha College Of Technology, Erode, Tamil Nadu, 638052, India.
[2]UG Scholar, Department Of Mechanical Engineering, Nandha College Of Technology, Erode, Tamil Nadu, 638052, India.

ABSTRACT

The shell and tube heat exchanger is a condenser type heat exchanging device it can be used in thermal power plant, oil and gas industries in this shell side baffle support is used to holding the tubes and create the turbulence flow in the shell side it will increase the heat transfer rate, the baffles are classified as a single segmented, double segmented and triple segmented which is used to reduce the pressure drop, in this research heat transfer rate and pressure drop in the single, double and triple segmented heat exchanger analysis through CFD Methodology, and it was concluded with double segmented get higher heat transfer coefficient and higher pressure outlet.

Keywords: shell and tube heat exchanger, heat transfer coefficient, pressure drop.

I. INTRODUCTION

Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense are known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and pressure drop. A more better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements.

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. In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modeling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained.

II. METHODS AND MATERIAL

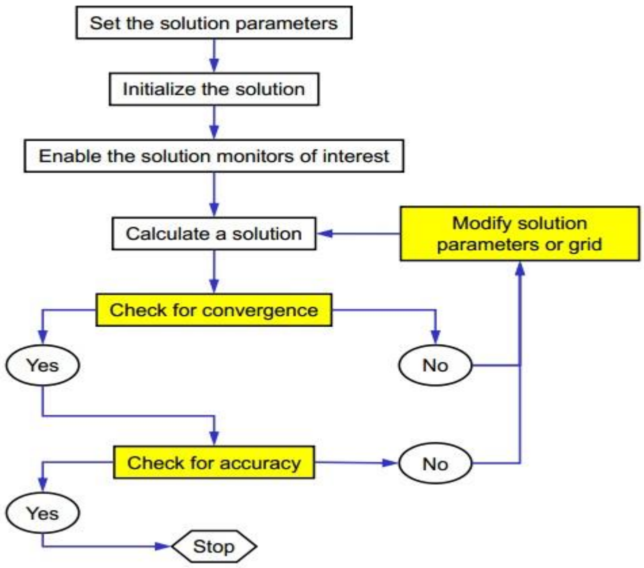
FLUENT requires high quality mesh to avoid numerical diffusion. Several Mesh Quality Metrics are involved in order to quantify the quality. However, the skewness is the primary metric however the skewness is the primary metric. The aspect ratio and cell size change mesh metrics are also very important.

- Length of Heat exchanger, (L) 570mm
- Inner diameter of Shell (Di) 135mm
- Outer diameter of Tube (do) 20mm
- Tube bundle geometry and pitch (Triangular) 30mm
- Number of tubes (Nt) 7
- Number of baffles (Nb) 5
- Central baffle spacing B 110mm
- Baffle inclination angle 0 to 50 deg

III. RESULTS AND DISCUSSION

Basically, a service may be single phase (such as the cooling or heating of a liquid or gas) or two-phase (such as condensing or vaporizing). Since there are two sides to an STHE, this can lead to several combinations of services. Broadly, services can be classified as follows:

- Single-phase (both shell side and tube side);
- Condensing (one side condensing and the other single-phase);
- Vaporizing (one side vaporizing and the other side single-phase); and
- Condensing/vaporizing (one side condensing and the other side vaporizing).



Mass velocity strongly influences the heat-transfer coefficient. For turbulent flow, the tube side heat-transfer coefficient varies to the 0.8 power of tube side mass velocity, whereas tube side pressure drop varies to the square of mass velocity. Thus, with increasing mass velocity, pressure drop increases more rapidly than does the heat-transfer coefficient.

Fluent is a computational fluid dynamics computer code developed and marketed by Fluent Inc. The code solves the equations for conservation of mass, momentum, energy and other relevant fluid variables using a cell-centred finite-volume method. First the fluid domain is divided into a large number of discrete control volumes (also known as cells) using a pre-processor code which creates a computational mesh on which the equations can be solved. Once the fluid domain has been meshed, the governing equations (in integral form) are applied to each discrete control volume and used to construct a set of non-linear algebraic equations for the discrete dependent variables.

The shell side calculations are far more complex than those for the tube side. This is mainly because on the shell side there is not just one flow stream but one principal cross-flow stream and four leakage or bypass streams. There are various shell side flow arrangements, as well as various tube layout patterns and baffling designs, which together determine the shell side stream analysis.

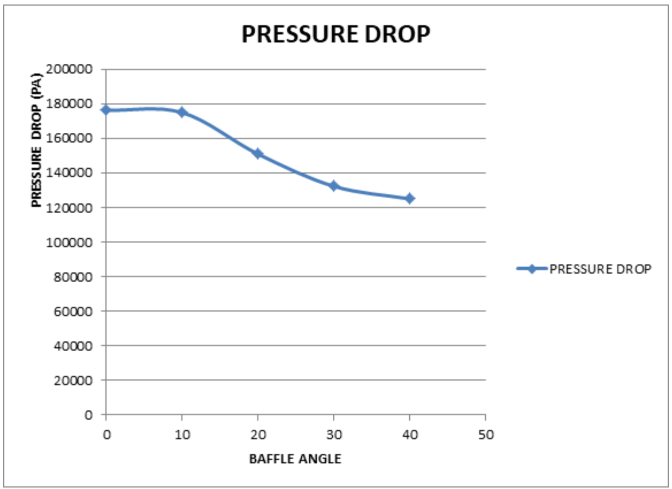


Figure 1: The graphical representation of variation of pressure drop and baffle degree angle of the shell and tube heat exchanger by using CFD analysis software. Here, The pressure drop is reducing according to the baffle degree angle.

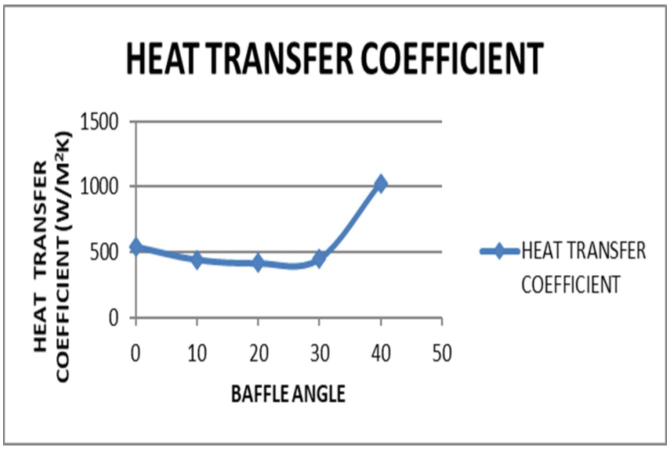


Figure 2: The explains about the graphical representation of variation of Heat transfer coefficient and baffle degree angle of the shell and tube heat exchanger by using CFD analysis software. Here, The Heat transfer coefficient is increasing according to the baffle degree angle.

Solve time-averaged Navier-Stokes equations All turbulent length scales are modeled in RANS Various different models are available This is the most widely used approach for calculating industrial flows There is not yet a single, practical turbulence model that can reliably predict all turbulent flows with sufficient accuracy.

IV. CONCLUSION

In this project modeling of the seven tube shell and tube heat exchanger by using the data from the literature survey and get the details of the boundary condition is apply on the geometry and complete the CFD analysis using the software of ANSYS fluent and post processing done in ANSYS CFD post and get the results of pressure, velocity, temperature and turbulence values from the results outlet get the more turbulences and velocity of the fluid is not vary in outlet and inlet temperature is increase in the outlet because of in the place of hot tube heat transfer is occurred and velocity stream line is show how the water flow in the shell and tube heat exchanger.

V. REFERENCES

[1]. Arjun K.S. and Gopu K.B.(2007) Design Of Shell And Tube Heat Exchanger Using Computational Fluid. Dynamics Tools. Department of Mechanical Engineering, Manav Bharti University, Solan, INDIA.

[2]. Ahmed F. Khudheyer And Mahmoud Sh. Mahmoud,(2011), "Numerical Analysis Of Fin-Tube Plate Heat Exchanger By Using CFD Technique", Department Of Mechanical Engineering, Nahrain University, Baghdad, Iraq, Journal of Engineering and Applied Sciences.

[3]. EnderOzdenIlkerTari,(2010), "Shell side CFD analysis of a small shell-and-tube heat exchanger", Energy Conversion and Management,Volume 51, Issue 5, May 2010, Pages 1004-1014.

[4]. FarajollahiS.Gh, EtemadM.Hojjat(2010), "Heat transfer of nanofluids in a shell and tube heat exchanger", International Journal of Heat and Mass Transfer, Volume 53, Issues 1–3.

[5]. Gurbir Singh , Hemant Kumar(2014) Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger Journal of Civil Engineering and Environmental Technology

Print ISSN: 2349-8404; Online ISSN: 2349-879X; Volume 1, Number 3. pp. 66-70

[6]. Hetal Kotwal, Dilip S Patel, Ravindrasinh R Parmar, Vipul M Prajapati,(2015), “CFD Analysis of Shell and Tube Heat Exchangers”, International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056 Volume: 02 Issue: 09, p-ISSN: 2395-0072.

[7]. Huadong Li And Volker Kottke,(1998) “ Effect Of The Leakage On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement”, Inl. J. Heat Mass Transfer, Elsevier Science Ltd. Volume 41, Issue 2, January 1998, Pages 425-433.

[8]. Patel.V.K, Rao.R.V,(2010), “Design optimization of shell-and-tube heat exchanger using particle swarm optimization technique”, Applied Thermal Engineering, Volume 30, Issues 11–12, August 2010, Pages 1417-1425.

[9]. Petrik, Mate, Szepesi, Gabor L,(2018), “Shell Side CFD Analysis of a Model Shell-and-Tube Heat Exchanger”, A journal of the Italian association of chemical engineering ,vol 18.

[10]. Prabhanjan.D.G, Raghavan.G.S.V, Rennie.T.J,(2002) “Comparison of heat transfer rates between a straight tube heat exchanger and a helically coiled heat exchanger”, International Communications in Heat and Mass Transfer, Volume 29, Issue 2, February 2002, Pages 185-191.

[11]. Salimpour.M.R,(2008) “Heat Transfer Coefficients Of Shell And Coiled Tube Heat Exchangers”, Isfahan University Of Technology, Iran. Experimental Thermal and Fluid Science, Volume 33, Issue 2, January 2009, Pages 203-207.

[12]. SepehrSanaye, Hassan Hajabdoll,(2010), “Multi-objective optimization of shell and tube heat exchangers”, Applied Thermal Engineering, Volume 30, Issues 14–15, October 2010, Pages 1937-1945.

[13]. Swapnaneel Sarma (2012), “CFD Analysis Of Shell And Tube Heat Exchanger Using Triangular Fins For Waste Heat Recovery Processes”, Mechanical Engineering Department NIT Silchar India. IRACST–Engineering Science and Technology.

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

[15]. Young Hwan Yoon, Jin Gi Paeng And Ki Chul Kim,(2008), “ Theoretical Analysis And CFD Simulation On The Ceramic Monolith Heat Exchanger”, Chemical Engineering Research and Design,Volume 86, Issue 3, March 2008, Pages 221-232.