International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 10, Issue 8, August 2021)

CFD Simulation Analysis of Various Types of Heat Exchangers: A Literature Review

Maniratnam[1], Amber Gupta[2]

[1]*M.Tech. Research Scholar (Machine Design),* [2]*Assistant Professor, Department of Mechanical Engineering Saraswati Institute of Engineering and Technology, Jabalpur (M.P.) India*

Email: [1]manircit@gmail.com, [2]amber_gupta_2@yahoo.com

***Abstract* -- Heat exchangers are utilized to move heat from liquid at high temperature to liquid at lower temperature. Heat exchangers are utilized in modern purposes in synthetic businesses, atomic force plants, treatment facilities, food preparing, and so on Measuring of heat exchangers assumes exceptionally huge part for cost streamlining. Likewise, productivity and adequacy of heat exchangers is a significant boundary while choice of mechanical warmth exchangers. Strategies for development on warmth move have been worked upon for a long time to get high productivity with ideal expense. In this research work, we design a shell & tube heat exchanger with straight tube and same model with variant shape of tubes and analyze the flow and temperature drop using Solidworks Simulation CFD 2017. When comparing the CFD analysis with different model with copper and steel materials we will found that the heat transfer rate with step tube shows better results as compare to others.**

***Keywords*-- Mass Flow, Heat Transfer, Shell & Tube Heat Exchanger, Different Tube Shape Assembly, Computational Fluid Dynamics (CFD), SOLIDWORKS etc.**

I. INTRODUCTION

In the ventures heat exchangers are generally utilized gear in the enterprises. Heat exchangers are utilized to move heat between two cycle streams. One can understand their utilization that any cycles which include cooling, heating, build-up, bubbling or vanishing will require a heat exchanger for these reasons. Cycle liquids, as a rule are warmed or cooled before the cycle or go through a stage change. Diverse heat exchangers are named by their application. For instance, heat exchangers being utilized to gather are known as condensers, correspondingly heat exchanger for bubbling objects are called boilers. Execution and effectiveness of heat exchangers are estimated through the measure of heat move utilizing least zone of heat move and weight drop. A superior introduction of its productivity is finished by figuring over all heat move coefficient. Weight drops and region needed for a specific measure of heat move, gives an understanding about the capital expense and force prerequisites (Running expense) of a heat exchanger.

For the most part, there is bunches of writing and hypotheses to plan a heat exchanger as indicated by the prerequisites.

Heat exchangers are of two sorts: -Where both media between which heat is exchanged are in direct contact with each other is Direct contact heat exchanger, where both media are separated by a wall through which heat is transferred so that they never mix, indirect contact heat exchanger the greater part of the heat exchangers is as a rule for higher weight work up to 552 bars is the shell and cylinder heat exchanger. Shell and cylinder type heat exchanger, circuitous contact type heat exchanger. It comprises of a progression of cylinders, through which one of the liquids runs. The shell is the holder for the shell liquid. For the most part, it is round and hollow fit as a fiddle with a roundabout cross segment. For this specific examination shell is thought of, which is commonly a one pass shell. A shell is the most ordinarily utilized because of its ease and straight forwardness and has the most noteworthy log-mean temperature-distinction (LMTD) remedy factor. Even though the cylinders may have single or different passes, there is one pass on the shell side, while the other liquid streams inside the shell over the cylinders to be warmed or cooled. The cylinder side and shell side liquids are isolated by a cylinder sheet. The unpredictability with exploratory strategies includes quantitative portrayal of stream marvels utilizing estimations managing each amount in turn for a restricted scope of issue and working conditions. Computational Fluid Dynamics is currently a set up mechanical plan apparatus, offering evident focal points. In this investigation, a full 360° CFD model of shell and cylinder heat exchanger is thought of. By displaying the math as precisely as could reasonably be expected, the stream structure and the temperature circulation inside the shell are gotten.

1.1 Types Of Flow In Heat Exchanger

Based on the constructional design & mode of heat transfer a wide variety of heat exchangers is in used various process industries.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 10, Issue 8, August 2021)

Plate type heat exchanger (PHE), shell-and-tube heat exchanger, vertical mantle heat exchanger and micro heat exchanger are among the most popular once.

Figure 1 below enlists various other types of heat exchanger & their specific classification.

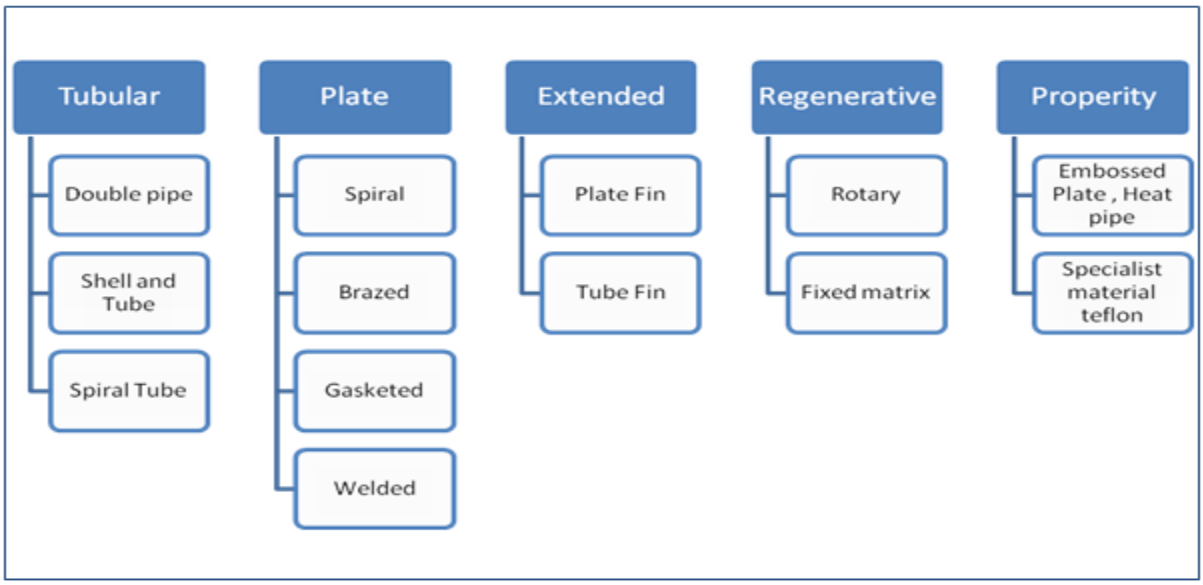

Figure 1: Types of Heat Exchanger

1.2 Classification Of Heat Exchangers

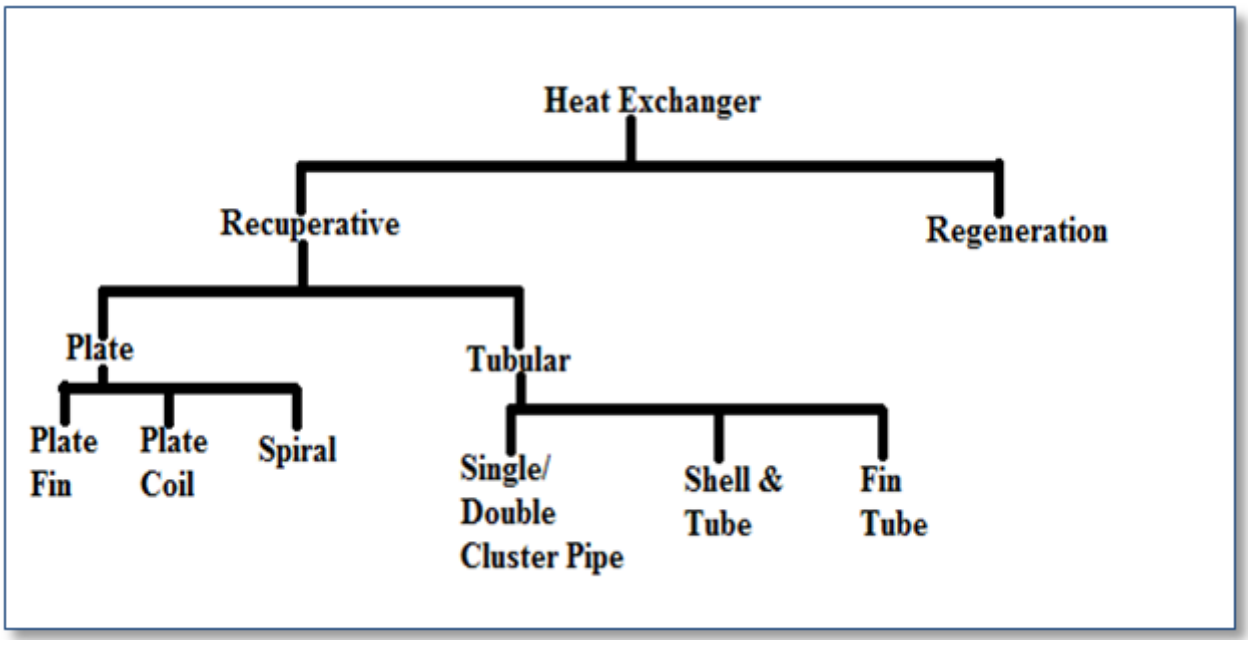

Figure 2: Classifications of Heat Exchangers on The Basis of Construction

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 10, Issue 8, August 2021)

II. Application Of CFD In Various Aspect Of Heat Exchanger (Literature Review)

The reason for this part is to give a literature audit of past research work in various journals or articles on the topic of shell and tube heat exchanger and computational fluid dynamics (CFD) analysis on it. Moreover, review of other relevant research studies is made to provide more information in order to understand more on this research.

The performance evaluation of heat exchanger on the basis of pressure drops, fouling, fluid mal distribution & thermal performance, Geometry is carried out using different CFD codes. FLUENT, CFX, STAR CD, FIDAP, ADINA, CFD2000, PHOENICS are few among these.

2.1 JAYACHANDRAIAH, B. AND PATEL, C.D.K., (2020) In this article is manages a shell- and-tube heat exchanger is planned in CATIA V5 and examined utilizing Autodesk Simulation CFD 2015. The external temperature of the shell-side liquid is 56.57 °C which is around equivalent to the re-enactment results. The external temperature of the cylinder side liquid is 49.62 °C which is additionally almost equivalent to the recreation result. In this research work on the **[1]**

2.2 ANAND, R.S., DAVID, S., GAJENDIRAN, M. AND STANLEY, K., (2020) This article is managing the displaying and investigation of small-scale shell and tube heat exchanger (MSTHE) for low temperature applications which is under 250°C. The plan of the heat exchanger is made with nine cylinders which are of 6 mm distance across and shell of 41 mm width. As traditional plan doesn't bring about the inward heat move, computational liquid elements conspire is embraced to plan the altered heat exchanger by receiving the conditions, for example, speed of cylinder liquid and weight drop. The demonstrating of MSTHE is finished by Pro/E though CFD investigation is finished with ANSYS. The form got from the examination demonstrates that the MSTHE is material for the temperature under 250oC and can possibly move heat viably.**[2]**

2.3 YADAV, P.K. AND KUMAR, G.S, (2019) In this investigation the heat exchanger contains seven containers of width 20 mm and the shell length comprises of 600 mm long and the breadth 90 mm. the helixes point goes from 00 to 200. Here the recreation shows how the weight changes inside the shell because of various helix point and stream rate. The stream design is compelled to be rotational with proceeds with helical perplex present inside the shell. Subsequently we see the outcomes in an ascent of heat move coefficient per unit pressure drop in this heat exchanger. The confuse cut here is 36 percent.

The heat exchanger is viewed by differing its mass stream rate and puzzle tendency point. Deciding of the shell side source temperature, pressure drop, for the given heat exchanger is known utilizing computational liquid elements. According to the mathematical exploratory information the outcome here acquired is increment in the exhibition of heat exchanger in helical perplex rather than segmental puzzle**.[3]**

Figure 3: Surface meshes with Helical Baffle

2.4 KALIAPPAN, A.S. AND MOTHILAL T, (2018) A helical loop heat exchanges with a helix point of 30o utilizing CREO programming was planned and manufactured. These days copper is supplanted via Carbon steel in ventures. As of now the goal is to utilize ANSYS CFX 15.0 programming to construe the reparability of copper with two diverse inward cylinder materials, for example, POCO HTC graphite and ASTM SA 179 carbon steel which likewise have calculable heat move qualities and great erosion obstruction than copper. A laminar hot liquid stream is the heat source medium. Likewise, the expansion in heat move rate with increment in mass stream rate is additionally noticed**.[4]**

2.5 STEPHENRAJ V., M.K. SATHISH KUMAR, (2018) this venture manages heat move productivity that relies upon both plan of heat exchanger and property of working liquid. Some significant plan boundaries, for example, the pitch proportion, tube length, and cylinder layer just as astound dividing. In this task, the heat move productivity is improved by executing the full confound plan and travel tube plan and dissecting it through CFD stream reproduction to locate the inexact heat move rates. From the re-enactment results the ideal bewilder plan and travel tube plan for greatest heat move rate is distinguished. Likewise, this venture manages locate the reasonable liquid for most extreme heat move rate.[5]

2.6 DE, D., PAL, T.K. AND BANDYOPADHYAY, S., (2017) The point of this work is to plan of shell and cylinder type heat exchanger with helical puzzle and contrasting and straight perplex with CFD investigation utilizing ANSYS FLUINT programming instruments. The model contains 7 Copper tubes each having 20 mm outside width and 17 mm inside breadth, length 600 mm and internal distance across of steel shell is 90 mm and external measurement 110 mm. 7 cylinders are hold by 6 straight or helical aluminium perplex, and the helix point of confuse is shifting from 0º to 30º. All the models are plan by utilizing CATIA programming apparatuses. In this paper how the weight drops and generally speaking heat move coefficient fluctuates because of various helix point has been contemplated when the stream rate stay same. The stream design in the shell side of the heat exchanger with consistent helical confuses are compelled to be rotational and helical because of the calculation of the constant helical astounds, which brings about a huge expansion in heat move coefficient per unit pressure drop in the heat exchanger.[6]

2.7 KATARKI, M.S.K. AND MALIPATIL, M.A.S., (2017) In this paper endeavours have been made to plan a heat exchanger by demonstrating in ANSYS programming which is having an inner distance across of 330mm and external measurement of 350mm for shell. Additionally, for tube inward width is 21.18mm and external measurement 25.4mm, length of cylinder is 1500mm, contains 36 cylinders. Here gathering of shell and cylinder are finished with water and steam as a medium. By utilizing results, the plan can be changed for better proficiency. Feasible k–ε (RKE) model handles top notch results.**[7]**

2.8 THAKUR, R.S.D.D.H., (2016) In this work we steadied how the heat transmission rate change by changing the progression of water it will be counter stream or equal stream with fluctuating its embellishment plan. The 3D displaying of the shell and cylinder heat exchanger is planned in INVENTER PROFESSIONAL and its examination is done in ANSYS-FLUENT

14.5 and 15. The Results Shows that the Maximum heath transmission occur in the round balance with astound having the counter progression of the water, which give more opportunity to the progression of water for heat transmission in a wavy structure and enormous surface region for the heat scattering for the cooling of water.**[8]**

2.9 JADHAV, A.D. AND KOLI, T.A., (2014) This project deals with the shell side design of a shell and tube heat exchanger; in this design work with baffle cut and shell diameter dependencies of the heat transfer coefficient and the pressure drop are investigated by numerically modeling a small heat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD package. A set of CFD simulations is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The results are observed to be sensitive to the turbulence model selection. The best turbulence model among the ones considered is determined by comparing the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell– Delaware method results. For two baffle cut values, the effect of the baffle spacing to shell diameter ratio on the heat exchanger performance is investigated by varying flow rate.**[9]**

2.10 SAMAL, A.K., (2013) the target of the undertaking is plan of shell and tube heat exchanger with helical confound and study the stream and temperature field inside the shell utilizing ANSYS programming devices. The heat exchanger contains 7 cylinders and 600 mm length shell measurement 90 mm. The helix points of helical confound will be shifted from 00 to 200. In re-enactment will show how the weight fluctuate in shell because of various helix point and stream rate. The stream design in the shell side of the heat exchanger with nonstop helical confuses had to be rotational and helical because of the math of the ceaseless helical confounds, which brings about a huge expansion in heat move coefficient per unit pressure drop in the heat exchanger.**[10]**

2.11 WANG Q., ET AL, (2009) performed the comparison of combined multiple shell-pass shell and tube type device with segmented baffles using CFD simulation. The comparative analysis of the results obtained from simulation of the 2 distinct models revealed that under similar mass and heat transfer rate with same pressure drop by the shell side the general heat transfer rate of CMSP-STHX is 5.6% above the SG- STHX**.[11]**

2.12 M.V.D. BONIS, G. RUOCCO, (2009) studied the fouling rates with reference to flow velocities. because the Re of flowing media is increased, a drastic increase within the fouling rate is encountered with a deposition of a touch quite 1 g for Re 1700 and almost 22 g for Re 3700 after a given period of time**.[12]**

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 10, Issue 8, August 2021)

Table 1:
CFD simulation of heat exchanger in fouling

Author	Type	CFD methodology		Results comparison
Soojin et al.(2005)	plate heat exchanger	CFD code	fluent	within 2 %
		meshing software	gambit	
		mesh	tetrahedral (831169 cells)	
Maria valeria et .al. (2009)	plate heat exchanger	pressure velocity coupling scheme	simplex	found to be in good agreement
		Environment mode	Frontier	

2.13 J.S. JAYAKUMAR, S.M. MAHAJANI, J.C. MANDAL, P.K. VIJAYAN, R. BHOI, (2008) studied thermal behavior of fluid media during a helical pipe followed by a shell and tube device with helically coiled tube bundles. The thermal properties were presented within the sort of Nu and De, considering them to be hooked into temperature alone. Firstly, three analyses were administered on the helical pipe; one for constant properties at ambient temperature, second for constant properties at mean temperature of the fluid and lastly for temperature dependent properties.**[13]**

III. CONCLUSIONS

Conventional methods used for the design and development of Heat Exchangers are largely tedious and expensive in today's competitive market. CFD has emerged as a cost-effective alternative, and it provides speedy solution to heat exchanger design and optimization. Easily accessible general purpose CFD software's can fulfil the requirements of CFD analysis of various types of heat exchangers including but not limited to Plate, Shell and Tube, Vertical Mantle, Compact and Printed Circuit Board Exchangers.

These are flexible enough to accommodate any kind of analysis requirement ranging from prediction of fluid flow behavior to complete heat exchanger design and optimization involving a wide range of turbulence models and integrating schemes available in CFD software's.

Heat exchangers are utilized to move heat from liquid at high temperature to liquid at lower temperature. Heat exchangers are utilized in modern purposes in synthetic businesses, atomic force plants, treatment facilities, food preparing, and so on Measuring of heat exchangers assumes exceptionally huge part for cost streamlining. Likewise, productivity and adequacy of heat exchangers is a significant boundary while choice of mechanical heat exchangers. Strategies for development on heat move have been worked upon for a long time to get high productivity with ideal expense. As review the work, we can design a shell & tube heat exchanger with model with variant shape of tubes and analyze the flow and temperature drop using Solid works Simulation CFD & comparing the CFD analysis with different model of shell and tube type heat exchanger.

REFERENCES

[1] B. Jayachandraiah and C. Dinesh Kumar Patel, "Design of SHELL-AND-TUBE heat exchanger with CFD Analysis," Lecture Notes in Mechanical Engineering, pp. 393–400, 2020.

[2] R. S. Anand, S. David, M. Gajendiran, and K. Stanley, "Design and analysis of shell and tube heat exchanger for low temperature applications using CFD," IOP Conference Series: Materials Science and Engineering, vol. 912, p. 042056, 2020.

[3] , P.K., Yadav, and G.S., Kumar, "Design and Computational Analysis of Shell and Tube Heat Exchanger Considering Various Parameters", 2019.

[4] A.S., KALIAPPAN, and T., Mothilal, "Thermal Analysis of Shell and Tube Heat Exchanger. International Journal of Pure and Applied Mathematics", 119(12), pp.14299-14306, 2018.

[5] V, Stephenraj. M.K. Sathishkumar, "Design and Analysis of Heat Exchanger for Maximum Heat Transfer Rate (Multi Model Optimization Technique)", 2018.

[6] D., De, T.K. Pal and S., Bandyopadhyay, "International Journal of Heat and Technology", 35(2), pp.378-383, 2017.

[7] M.S.K., Katarki, and M.A.S., Malipatil, "CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities", 2017.

[8] R.S.D.D.H., Thakur, "CFD Analysis of Shell and Tube Heat Exchanger", IJSRD-International Journal for Scientific Research & Development, 3(12), pp.2321-0613, 2016.

[9] A.D., Jadhav, and T.A., Koli, "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop", International Journal of Research in Aeronautical and Mechanical Engineering, 2(7), pp.1-7, 2014.

[10] A.K., Samal, "Shell and Tube Heat Exchanger Design Using CFD Tools (Doctoral dissertation)", 2013.

[11] Q. Wang, Q. Chen, G. Chen, M. Zeng, "Numerical investigation on combined multiple shell-pass shell- and-tube heat exchanger with continuous helical baffles", Int. J. Heat Mass Transf 52 (2009) 1214e1222, 2013.

[12] M. V., De Bonis, & G., Ruocco, "Conjugate fluid flow and kinetics modeling for heat exchanger fouling simulation", International Journal of Thermal Sciences, 48(10), 2006-2012, 2009.

[13] J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, P. K. Vijayan, and R. Bhoi, "Experimental and CFD estimation of heat transfer in helically COILED heat exchangers," Chemical Engineering Research and Design, vol. 86, no. 3, pp. 221–232, 2008.