CFD Simulation Analysis for Different Tube Shapes Configuration in Shell & Tube Heat Exchanger by using SOLIDWORKS Software

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

Abstract - Traditionally Shell-and-tube heat exchangers are designed using solid modeling software & analyzed by using Computational Fluid Dynamics (CFD) software, by using CFD now it is possible to design small heat exchangers. In this paper, shell-and-tube heat exchangers are modeled in SOLIDWORKS software and analyzed using CFD flow simulation of SOLIDWORKS. The designed heat exchangers are relatively small, have single shell and tube passes. The leakage effects are not permissible into account in the design process. Hence, we are preventing leakage from baffle orifices and no gap between baffles and the shell. This study is focused on improving the efficiency of heat exchanger. First, only Design the shell and tube type heat exchanger with straight tubes & analyzed with SOLIDWORKS CFD simulations, similarly two other designs are created with step and taper tube assembly. The designed model is analyzed on SOLIDWORKS CFD simulation. The results are compared with design models having different tube shapes by using different materials. The results are compared in temperature drop in shell & tube heat exchanger with straight tube, taper tube & step tube model. By this analysis we can find out which model gives better heat transfer rate.

Keywords: Computational Fluid Dynamics (CFD), SOLIDWORKS CFD Simulation, Temperature Drop, Mass Flow, Heat Transfer, Shell & Tube Heat Exchanger, Efficiency of Heat Exchanger, Tube Shape, Different Tube Shapes Configuration, Different Tube Materials 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, buildup, 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. Despite the fact that 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. 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.

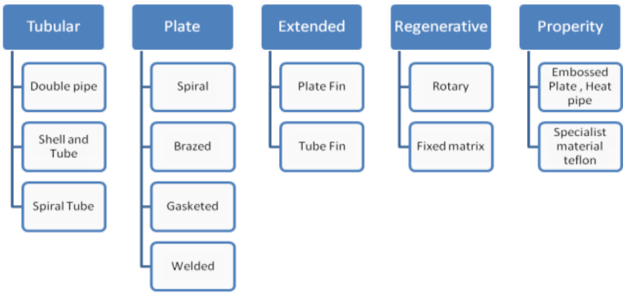

Figure 1: Types of Heat Exchanger

1.1.1 Parallel & Counter Flow Heat Exchanger

Two kinds of stream game plan are conceivable in a twofold line heat exchanger: equal stream and counter stream. In equal stream, both the hot and cold liquids enter the heat exchanger at a similar end and move a similar way, as appeared in Figure 2. In counter stream, the hot and cold liquids enter the heat exchanger at far edge and stream inverse way, as appeared in Figure 3.

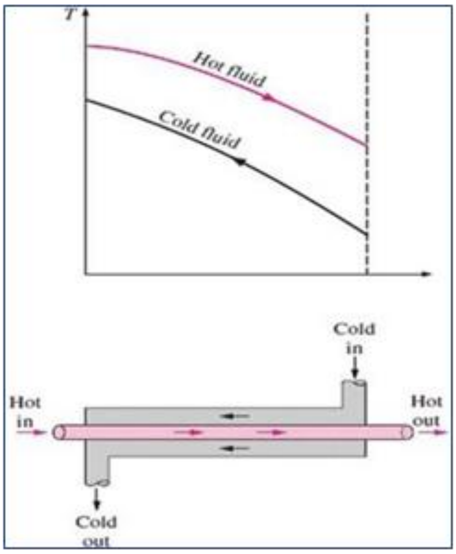

Figure 2: Patellar Flow Heat Exchanger

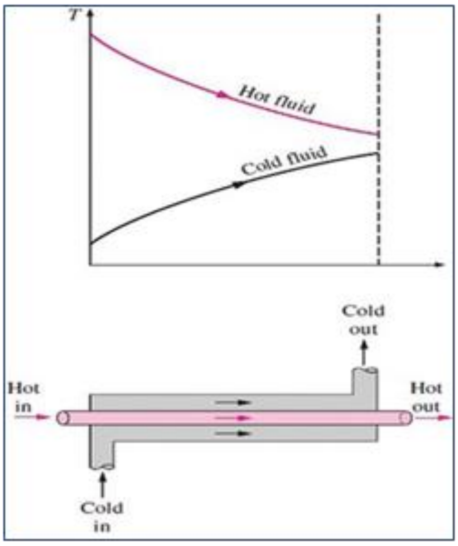

Figure 3: Counter Flow Heat Exchanger

1.1.2 Multi pass & Crossflow Heat Exchanger

In the compact heat exchangers, the 2 fluids usually move perpendicular to every other, and such flow configuration is named crossflow. Another way of classification crossflow is unmixed and mixed flow, depending on the flow.

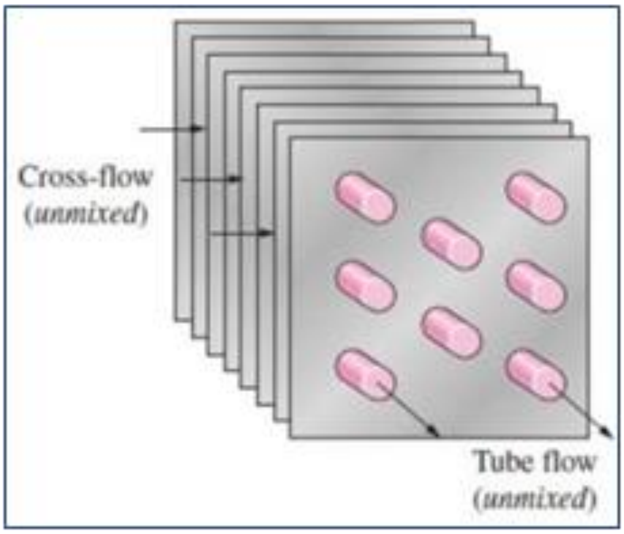

Figure 4: Heat Exchanger Crossflows

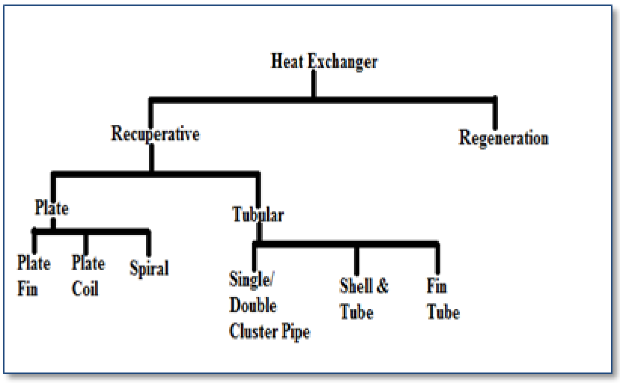

Figure 5: Classifications of Heat Exchangers

1.2 Classification of Heat Exchangers

1.2.1 Recuperative Heat Exchangers

A recuperate may be a special purpose counter-flow energy recovery heat exchange positioned within the availability and exhaust air streams of an air handling system, o r within the exhaust gases of a process, so as to recover the waste heat. The image on the proper shows the three major configurations. The term cooperator refers also to liquid-liquid counter flow heat exchanges used for heat recovery within the chemical and refinery industries and in closed processes like ammonia-water or Libra-water absorption refrigeration cycle.

1.2.2 Plate Heat Exchangers

A plate heat exchanger is a kind of heat exchanger that utilizations metal plates to move heat between two liquids. This has a significant bit of leeway over a regular heat exchanger in that the liquids are presented to a lot bigger surface territory on the grounds that the liquids spread out over the plates. This encourages the exchange of heat, and significantly speeds up the temperature change. Plate heat exchangers are currently normal and little brazed adaptations are utilized in the high temp water areas of millions of mix boilers. The high heat move effectiveness for quite a little actual size has expanded the homegrown heated water (DHW) stream pace of blend boilers. The little plate heat exchanger has had an incredible effect in homegrown heating and boiling water. Bigger business forms use gaskets between the plates, while more modest renditions will in general be brazed.

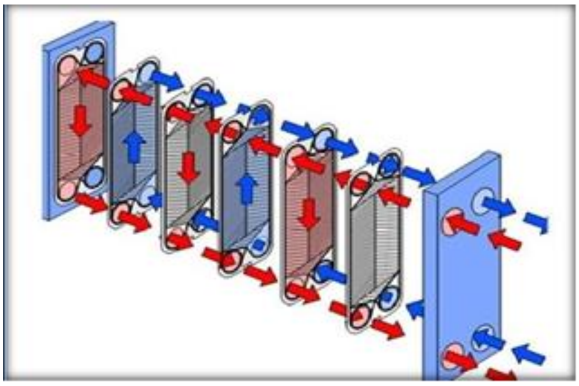

Figure 6: Plate Heat Exchanger

1.2.3 Tubular Heat Exchanger

Shell and tube heat exchanges comprise of arrangement of cylinders. One bunch of those cylinders contains the liquid that has got to be either warmed or cooled. the next liquid runs over the cylinders that are being warmed or cooled so it can either give the heat or retain the heat required. A bunch of cylinders is understood because the cylinder package and may be comprised of a couple of kinds of cylinders: plain, longitudinally finned, then on Shell and cylinder heat exchanges are ordinarily

utilized for high-pressure applications (with pressures more prominent than 30 bar and temperatures more noteworthy than 260 °C). this is often on the grounds that the shell and cylinder heat exchangers are powerful due to their shape.

Figure 7: Tubular Heat Exchanger

1.2.4 Regenerative Heat Exchanger

A regenerative device, or more commonly a regenerator, may be a sort of device where heat from the recent fluid is intermittently stored during a thermal data-storage medium before it's transferred to the cold fluid. To accomplish this hot fluid is brought into contact with the heat data-storage medium, then the fluid is displaced with the cold fluid, which absorbs the heat. In regenerative heat exchangers, the liquid on one or the opposite side of the heat exchanger are often an identical liquid. The liquid may experience an outdoor handling step, and afterward it's streamed back through the heat exchanger the opposite way for extra preparing. Normally the appliance will utilize this cycle consistently or repetitively. Regenerative heating was one among the most advancements created during the economic Revolution when it had been utilized within the hot shoot measure on impact heaters, it had been later utilized in glass and steel making, to create the proficiency of open-hearth heaters, and in high weight boilers and substance and different applications, where it keeps on being significant today.

II. LITERATURE REVIEW

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., (2021) 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]**

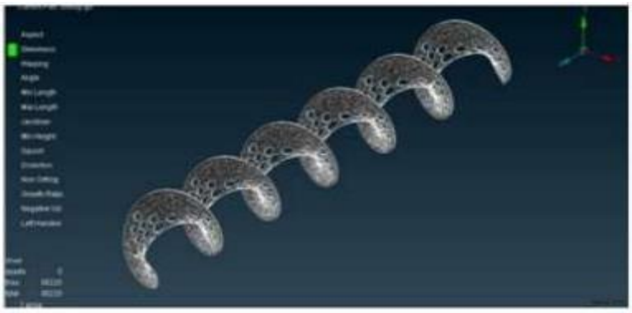

Figure 8: Surface meshes with Helical Baffle

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. Sathishkumar, (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 reenactment 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 aluminum 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 drop 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]**

III. CAD MODELLING

3.1 Modelling

Cad modeling or CAD (Computer Aided Design) provides a platform to engineers and designers to generate realistic models of parts and assemble them. These models can be then printed by using 3D printers. The entire solid modeling of heat exchanger has been completed on SOLIDWORKS.

3.1.1 Introduction of SOLIDWORKS

SOLIDWORKS is computer graphics software for modeling various mechanical designs for performing related design and manufacturing operations. This software is developed by Dassault Systems. This software is used for 3D solid modeling system as the core and applies the feature base parametric modeling method. In short it is a feature based parametric solid modeling system with many extended design and manufacturing applications.

3.1.2 Feature of SOLIDWORKS

3.1.2.1 Ease of Use

SOLIDWORKS was designed to start where the planning engineers begins with features and style criteria. SOLIDWORKS menus flow within the manner that's easily understand. This makes it simple to find out and utilize even for the foremost casual user. Because SOLIDWORKS provides the power to sketch directly on the solid model, feature placement is straightforward and accurate.

3.1.2.2 Full Associatively

SOLIDWORKS supported one arrangement with the power to form change built into the system. Therefore, when a change is formed anywhere within the development process it's propagated throughout the whole design through manufacturing process, ensuring consistency altogether engineering deliverables.

3.1.2.3 Parametric & Feature Based Modeling

SOLIDWORKS features are process plans with embedded intelligence and are easy to use while at the same time powerful enough to most complex geometry.

3.2 Modeling of Heat Exchanger in SOLIDWORKS

For creating the geometry of Heat Exchanger, we are adopted these three workbenches of SOLIDWORKS software.

- Part Design
- Assembly Design:
- Design Documentation (Drawings)

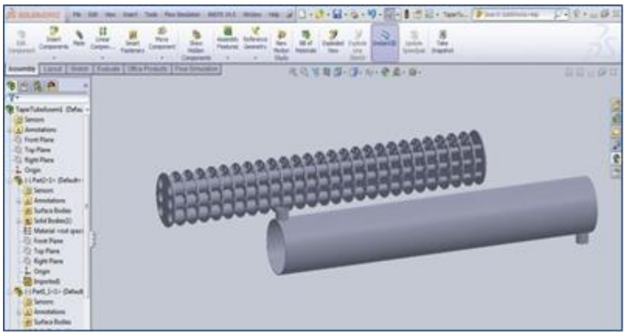

Figure 9: Disassemble view of Shell &Tube Type Heat Exchanger

3.2.1 Part Design

In part designing section we are using some basic commands for making parts of heat exchanger in the SOLIDWORKS software. Those basic commands are used for many parts.

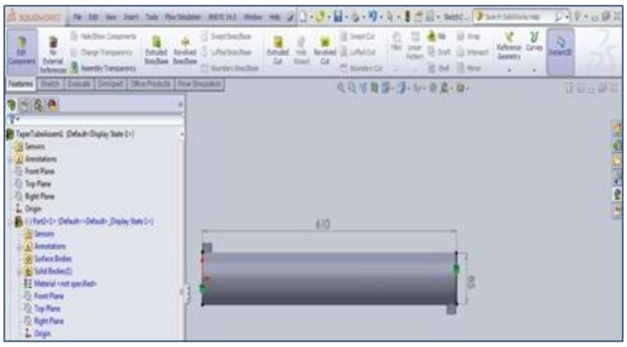

Figure 10: Shell of Heat Exchanger

IV. CFD ANALYSIS

1.4 Introduction Of CFD

The equations of hydraulics which are known for over a century are solvable only for a limited no. of flows. The known solutions are extremely useful in understanding fluid flow but rarely used directly in Engineering Analysis or design. CFD makes it possible to gauge velocity, pressure, temperature, and species concentration of fluid flow throughout a solution domain, allowing the design to be optimized before the prototype phase.

Availability of fast and computer makes techniques popular among engineering community. Solutions of the equations of hydraulics on computer has become so important that it now occupies the eye of a perhaps a 3rd of all researchers in hydraulics and therefore the proportion remains is increasing. This field is understood as computational fluid dynamics. At the core of the CFD modeling may be a three-dimensional flow solver that's powerful, efficient, and simply extended to custom engineering applications. In designing a replacement mixing device, injection grid or simply an easy gas diverter or a distribution device, design engineers got to ensure adequate geometry, pressure loss, and duration would be available. More importantly, to run the plant efficiently and economically, operators and plant engineers got to know and be ready to set the optimum parameters.

1.5 Techniques for Numerical Discretization

In order to unravel the governing equations of the fluid motion, first their numerical analogue must be generated. this is often done by a process mentioned as discretization. within the discretization process, each term within the partial equation describing the flow is written in such a fashion that the pc is often programmed to calculate. There are various techniques for numerical discretization. Here we will introduce three of the most commonly used techniques, namely:

(1) Finite Difference Method

(2) Finite Element Method

(3) Finite Volume Method

1.6 CFD Methodology

CFD could also be wont to determine the performance of a component at the planning stage, or it are often used to analyses difficulties with an existing component and cause its improved design. For instance, the pressure drop through a component could also be considered excessive: the primary step is to spot the region of interest: The geometry of the region of interest is then defined. If the geometry already exists in CAD, it are often imported directly. The mesh is then created. After importing the mesh into the pre-processor, other elements of the simulation including the boundary conditions (inlets, outlets, etc.) and fluid properties are defined. The flow solver is run to provide a file of results which contain the variation of velocity, pressure and therefore the other variables throughout the region of interest. The results are often visualized and should provide the engineer an understanding of the behavior of the fluid throughout the region of interest. this may cause design modifications which can be tested by changing the geometry of the CFD model and seeing the effect.

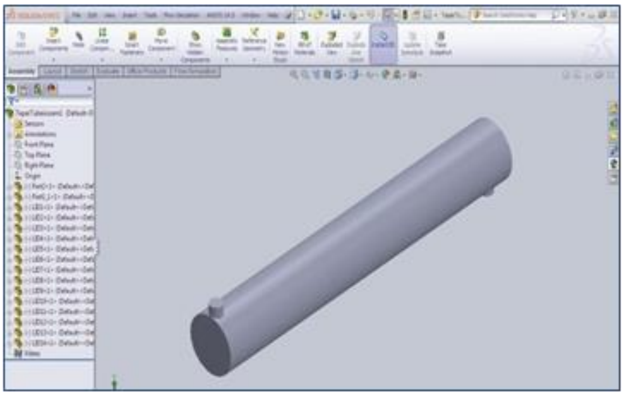

Figure 11: Geometry of Heat Exchanger in SOLIDWORKS

1.6.1 Geometry creation

The first task to accomplish in a numerical flow simulation is the definition of the geometry followed by the grid generation. This step is that the most important step for the study of isolated impeller considering an axis symmetric flow simplifies the domain to one blade passage.

1.6.2 Solis works Flow Simulation

SOLIDWORKS Flow Simulation is an intuitive Computational Fluid Dynamics (CFD) solution embedded within SOLIDWORKS 3D CAD that permits you to quickly and simply simulate liquid and gas flows through and around your designs to calculate product performance and capabilities.

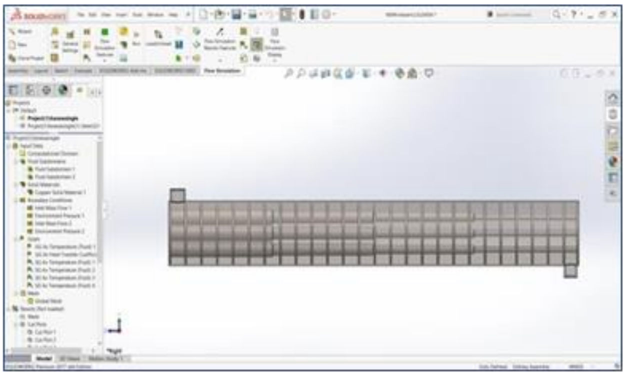

Figure 12: Photo of Heat Exchanger on Flow Simulation of SOLIDWORKS

V. RESULTS & DISCUSSION

In this analysis we are creating the model of heat exchanger with three different types of tubes and work with flow simulation analysis in SOLIDWORK for analysis use. During this analysis we are finding out the temperature difference between hot fluids. For this analysis flow rate is constant. This analysis is consisting of two parts.

- Heat Exchanger with Steel Material
- Heat Exchanger with Copper Material

1.7 Heat Exchanger with Steel Material

In this part we are doing the analysis on heat exchanger. For this analysis we are using steel material and for getting the efficient heat transfer we are creating three variants of tubes.

- Straight tubes
- Step tubes
- Taper tubes

1.8 Heat Exchanger with Copper Material

In this part we are doing the analysis on heat exchanger. For this analysis we are using copper material and for getting the efficient heat transfer we are creating three variants of tubes.

- Straight tubes
- Step tubes
- Taper tubes

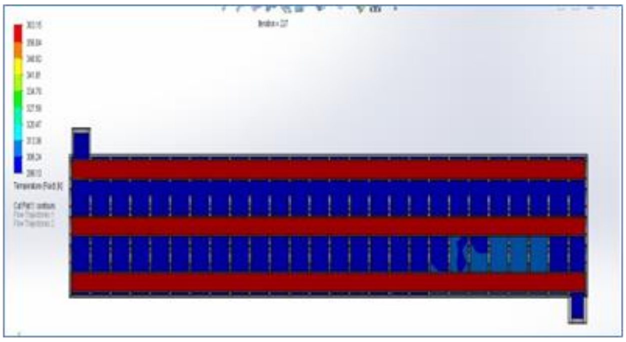

Figure 13: Temperature distribution – straight baffle with uniform tube with steel Material

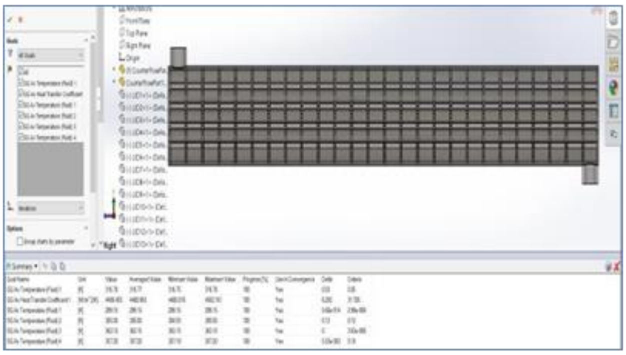

Figure 14: Temperature distribution output – straight baffle with uniform tube with steel material

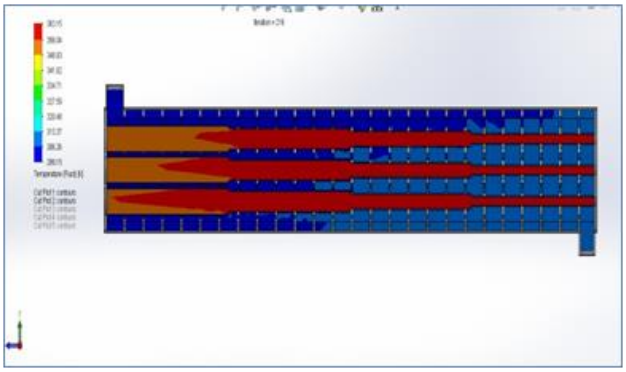

Figure 15: Temperature Distribution – Straight Baffle with Step Tube with Steel Material

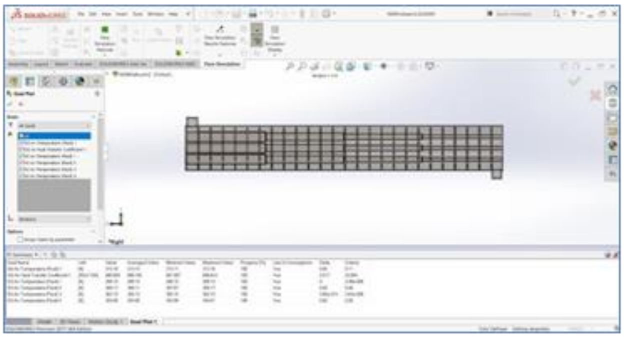

Figure 16: Temperature Distribution Output – Straight Baffle with Uniform Tube with Steel Material

Table 1: Various Tube Analysis Result Table For Steel Material

S.No.	Mass Flow Rate	Type of tubes	Steel		
			Inlet Tempe-rature	Outlet Tempe-rature	Tempe-rature Drop
1	2.1	STRIGHT TUBES	363	257	6
2		STEP TUBES	363	253	10
3		TAPER TUBES	363	356	7

Table 2: Various Tube Analysis Result Table For Copper Material

S.No.	Mass Flow Rate	Type of Tubes	Copper		
			Inlet Tempe-rature	Outlet Temperature	Tempe-rature Drop
1	2.1	STRIGH TUBES	363	353.25	9.75
2		STEP TUBES	363	348.62	14.38
3		TAPER TUBES	363	352.32	10.68

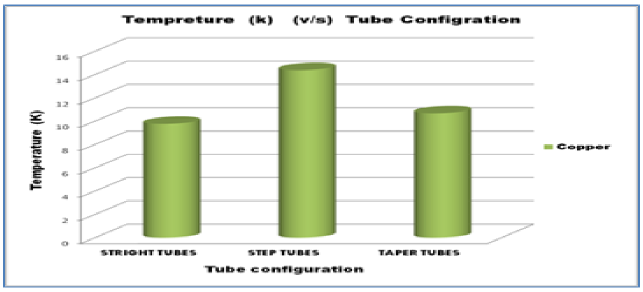

Figure 17: Result Graph for Copper Materials

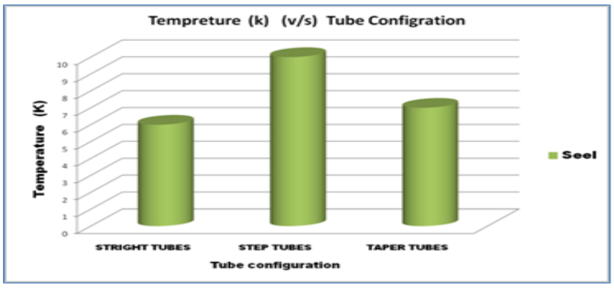

Figure 18: Result Graph For Steel Materials

VI. CONCLUSION

The small shell-and-tube heat exchanger is designed in SOLIDWORKS with sufficient detail to resolve the flow and temperature fields. From the CFD simulation the point wise results are listed below

- Initially three model of shell and tube heat exchanger with straight tube, taper tube & step tube is designed in SOLIDWORKS with using steel and copper material.
- These designs of heat exchanger are analyzed in CFD flow simulation part of SOLIDWORKS.
- This analysis having constant mass flow rate in various cross section tubes shows the results in temperature difference with straight, taper & step tube.

- Comparing the results with steel material for all models the heat transfer rate for step tube gives the better result as compare to other one. This result is also compared with copper material.
- Compare the results with steel and copper material with three different designs of heat exchanger, result shows the heat transfer rate by using step tube gives the better result compared to other design, so if we use this design for shell & tube heat exchanger then we will get better output.

REFERENCES

[1] Jayachandraiah, B., and C. Dinesh Kumar Patel. "Design of Shell-and-Tube Heat Exchanger with CFD Analysis." *Trends in Mechanical and Biomedical Design* (2021): 393-400.

[2] Anand, R. S., et al. "Design and analysis of shell and tube heat exchanger for low temperature applications using CFD." *IOP Conference Series: Materials Science and Engineering*. Vol. 912. No. 4. IOP Publishing, 2020.

[3] Yadav, Pintu Kumar, and G. Satish Kumar. "Design and Computational Analysis of Shell and Tube Heat Exchanger Considering various Parameters." (2019).

[4] KALIAPPAN, Ahamadu6 S., and T. Mothilal. "Thermal analysis of shell and tube heat exchanger." *International Journal of Pure and Applied Mathematics* 119.12 (2018): 14299-14306.

[5] Raj, V. S., and M. K. Sathishkumar. "Designand Analysis of Heat Exchangerfor Maximum Heat Transfer Rate (Multi Model Optimisation Technique)." *International Research Journal of Engineering and Technology (IRJET)* 5.01 (2018).

[6] De, Dipankar, Tarun K. Pal, and Santanu Bandyopadhyay. "Helical baffle design in shell and tube type heat exchanger with CFD analysis." *International Journal of Heat and Technology* 35.2 (2017): 378-383.

[7] Katarki, Santosh K., and Anandkumar S. Malipatil. "CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities." *International Journal of Engineering and Techniques* 3.6 (2017).

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

[9] Jadhav, Avinash D., and Tushar 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 (2014): 1-7.

[10] Samal, Anil Kumar. *Shell and tube heat exchanger design using CFD tools*. Diss. 2013.

[11] Raj, V. Antony Aroul, and R. Velraj. "Heat transfer and pressure drop studies on a PCM-heat exchanger module for free cooling applications." *International Journal of Thermal Sciences* 50.8 (2011): 1573-1582.

[12] Wang, Qiuwang, et al. "Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles." *International Journal of Heat and Mass Transfer* 52.5-6 (2009): 1214-1222.

[13] De Bonis, Maria Valeria, and Gianpaolo Ruocco. "Conjugate fluid flow and kinetics modeling for heat exchanger fouling simulation." *International Journal of Thermal Sciences* 48.10 (2009): 2006-2012.

[14] Jayakumar, J. S., et al. "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers." *Chemical engineering research and design* 86.3 (2008): 221-232.

[15] Mohammadi, K., W. Heidemann, and H. Müller-Steinhagen. "Numerical Investigation of the effect of baffle orientation and baffle cut on heat transfer and pressure drop of a shell and tube heat exchanger." *Proceedings of the International Heat Transfer Conference IHTC-13, Sydney*. 2006.

[16] Perrotin, Thomas, and Denis Clodic. "Thermal-hydraulic CFD study in louvered fin-and-flat-tube heat exchangers." *International Journal of Refrigeration* 27.4 (2004): 422-432.

[17] Grijspeerdt, Koen, Birinchi Hazarika, and Dean Vucinic. "Application of computational fluid dynamics to model the hydrodynamics of plate heat exchangers for milk processing." *Journal of food engineering* 57.3 (2003): 237-242.

[18] Kumar, Vikas, et al. "CFD analysis of cross flow air to air tube type heat exchanger." *PHOENICS 10th International User Conference*. 2003.