Modelling and CFD Simulation of Prototype of AC Plant Chiller on-Board Marine Ship

Amolkumar Musale[1], Pravin Hadgekar[2]

[1]M.Tech Student, Dept. of Mechanical Engineering, Defence Institute of Advanced Technology, Pune, MH India
[2]Assistant professor, Dept. of Mechanical Engineering, Defence Institute of Advanced Technology, Pune, MH India

---

Abstract - *In present day shell and tube heat exchanger (STHX) is the most common type of heat exchanger broadly used in marine ships, due to its high pressure application. The AC plants fitted on-board Marine ships consist of a Chiller i.e. parallel flow heat exchanger with single segmental baffles. The make of the Chiller is Alfa Laval Ltd. and that of AC plant is Heinen and Hopman ltd. Kolkata. The heat exchanger contains 234 tubes and 2692 mm length. The water is cooled by using refrigerant R134a in this chiller. This project mainly deals with modelling the prototype of basic geometry of shell and tube heat exchanger using Solidworks and Space claim 2017, meshing using ICEM CFD and simulation run using CFD package ANSYS 17.0. The objective of the project is to model the shell and tube heat exchanger with single segmental baffles and to achieve the temperature outputs as that factory acceptance trials (FATs) and to study the flow and temperature distribution inside the shell and tube using ANSYS software tools with parallel flow. In CFD analysis we will show how the temperature varies in shell due to different mass flow rates. The stream pattern in the shell with single segmental baffles was required to be rotational, which outcomes in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger. The CFD outcomes will be compared with that of actual readings obtained from marine ship.*

Key Words: Shell and Tube Heat Exchanger, Chiller, Temperature Distribution, CFD Analysis, single segmental baffles, Pressure drop, etc.

1. INTRODUCTION

A heat exchanger is a device built for effective and proficient heat transfer amid two or more media. The media may be parted by a solid wall, in order not to mix with each other. They are largely used in various chemical, petroleum, refrigeration and air conditioning, petrochemical applications and cryogenics field applications. Heat exchangers are also used in various sewage treatment and natural gas processing plants.

1.1 Heat Exchanger Classification

Heat exchangers are mainly classified according to their flow arrangement. Firstly, in parallel-flow heat exchangers, the two fluids enter the exchanger at the same end, and travel parallel to one another to the other side. Secondly for counter-flow heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is most efficient as it can transfer the most heat from the heat (transfer) medium. And finally in a cross-flow heat exchanger, the fluids travel roughly perpendicular to one another through the exchanger. Fig 1 shows the parallel and counter flow heat exchangers' flow arrangements.

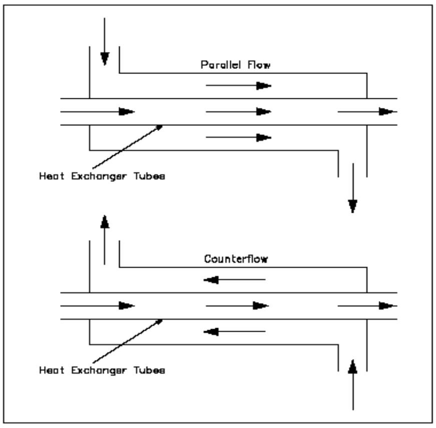

Fig-1: Flow arrangement in heat exchanger

1.2 Shell-and-Tube Exchangers

This heat exchanger is commonly built of a bundle of curved tubes fixed in a cylindrical shell with the tube axis parallel to that of the shell. One fluid flows inside the shell and the other flows inside the tubes. The major components of this exchanger are tubes (or tube bundle), shell, front-end head, rear-end head, baffles, and tube sheets, and are described briefly. A range of different internal constructions are used in shell-and-tube exchangers, depending on the preferred heat transfer and pressure drop performance. Fig 2 depicts the construction and component details of the STHX. STHX are used by methods employed to,

- lessen thermal stresses,
- prevent leakages,
- deliver for ease of cleaning,
- contain operating pressures and temperatures,
- control corrosion,
- house highly asymmetric flows, and so on.

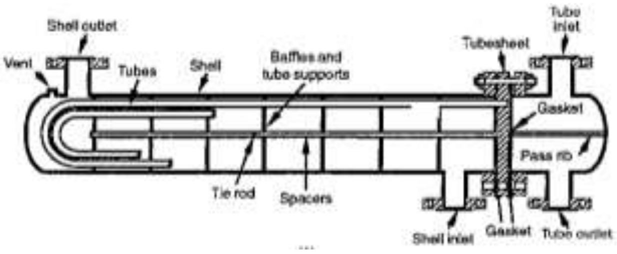

Fig-2: Shell and tube type heat exchanger

Generally, STHX is cylindrical in shape with a circular cross section, although shells of diverse contour are used in specific applications. For this particular study one pass shell is considered. It is the most frequently used due to its low cost and simplicity, and has the utmost log-mean temperature difference (LMTD) correction factor. Though the tubes may have single or multiple passes, there is one pass on the shell side, while the other fluid flows within the shell over the tubes to be heated or cooled. Baffles are used to support the tubes for structural rigidity, preventing vibration and sagging of tubes and to divert the flow across the tube bundle to obtain a higher heat transfer coefficient. In this study, a full CFD model of shell and tube heat exchanger is designed. The geometry is modelled so accurately that the temperature distribution and flow pattern inside shell and tube are evidently obtained.

1.3 Principle

A shell and tube heat exchanger is a type of heat exchanger designs that are predominantly used in marine ships, chemical processing plants, and oil refineries and is apposite for higher-pressure applications. As its name infers, this type of heat exchanger consists of a shell (a large cylindrical pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to achieve the heat transfer between the two fluids. The set of tubes named as a tube bundle is composed of numerous types of tubes viz. plain, longitudinally finned, etc.

2. LITERATURE SURVEY

Varieties of works were previously carried out by researchers on the Shell and tube heat exchangers in order to study the heat transfer mechanism and temperature contours across shell and tube. Some of them are enlisted here:

Nakka Sita and Rama Raju et al. [1] undertook the thermal analysis of STHX to portray the heat transfer proficiencies by using various materials. The study mainly focused on a simplified model of counter flow STHX designed to cool water from 55 to 45^{0}C by using water at room temperature. The design was made by using Kern's method in order to obtain various dimensions of shell, tubes, baffles etc. A computer model of STHX was prepared by using ANSYS 14.5 and thermal analysis was carried out for thermal materials viz. copper, copper-nickel alloys, steel, etc. **Moses Omolayo petinrin et al [2]** revealed in his paper performance analysis of STHX by varying tube layouts. The paper includes the study of STHX with different geometrical tube layout patterns viz. triangular, rotated triangular and the combined. The results were obtained by solving the governing continuity, momentum and energy equations. Study mainly represents that bulk of the heat transfer and pressure drop occur during the cross-flow of shell-fluid through the tube bundles. Evaluation also showed that the triangular pattern was more desirable followed by the combined one as they exhibit higher heat transfer coefficient than the rotated triangular for the same pressure drop in the shell-side. **Thundil Karuppa et al. [3]** in his research of numerical analysis of STHX with simple baffels investigated using computational fluid dynamics software. Flow and temperature fields were analyzed inside the shell and tubes using CFD software. Investigations were carried out on the k-ω SST model which provides better results. Three meshes, coarse, medium and fine were investigated resulting that aspect ratio had no significant effect. Thus, lastly a mesh comprising 4.2 million elements was used. Profiles of temperature and velocity obtained of the model were compared with experimental data and it had an acceptable adaptation. It is perceived that flow due to the existence of baffle does not remain in parallel with tube. As a result, the heat transfer level advances and thus the heat transfer surges. **Kevin Shah et al [4]** studied the shell and tube heat exchanger used as chiller plant for transfer of waste heat from the injection moulding machine to the cooling water in order to advance the efficiency of the injection moulding machine. Same transformations were dependent on the heat exchange capacity of heat exchangers. To increase the heat exchange capacity of heat exchanger optimization was carried out which pursues to identify the best parameter mishmash of heat exchangers. The tube diameter was used as an input variable and the output parameter was the maximum temperature difference of shell and tube heat exchanger. Nine models were studied on the basis of Taguchi method in NX 10.00 and CFX analysis is carried out in ANSYS 14.5. Result found from same delivers the best dimension of heat exchanger for minimum outlet temperature of water. **Ender Ozden et al [5]** depicts that the heat transfer coefficient and the pressure drop are dependent of the baffle spacing, baffle cut and shell diameter. This is explored by numerically modelling a small heat exchanger. The flow and temperature fields inside the shell are determined using a commercial CFD package. CFD models were executed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The outcomes are perceived to be sensitive to the turbulence model selection. The best turbulence model among the ones considered is obtained by relating the CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell–Delaware method outcomes.

3. COMPUTATIONAL MODELLING AND ANALYSIS

By using the Solidworks the actual model of chiller has been prepared. But while meshing it has been observed that the geometry is too heavy to mesh and resulted in mesh count of **24456780** elements. The same mesh count was not that compatible for the PC to process and arrive at appropriate results. Hence it was indispensable to design a prototype from the actual heat transfer rate. For design of chiller Kern method and following formulae were used.

3.1 Formulae used

- Hot fluid flow : $Qh = Mh \times Cph \times \Delta Th$
- Cold fluid flow : $Qc = Mc \times Cpc \times \Delta Tc$
- One shell pass and multiple tubes
  : $Qh = Qc = Q$
- LMTD calculations :
  $\Delta Tlm = (\Delta Th - \Delta Tc)/Ln(\Delta Th/\Delta Tc)$
- Heat flow rate : $Q = U \times A \times \Delta Tlm$
- Specific heat of water
  Cph: 4185 J/kg k
- Specific heat of R134a
  Cpc: 1374 J/kg k
- Heat transfer area : $A = n\pi DL$

Where
$\Delta Th = (Thi - Tho)$
$\Delta Tc = (Tci - Tco)$
U = Overall heat transfer Coefficient
D = Diameter of tube
L = Length of tube

3.1 Modelling

In this study the shell and tube type heat exchanger is modelled with derived dimensions along with three baffles placed along shell. The baffels cut is 30% in order to avail the adequate heat transfer rate. This model depicts the fresh water inlet, outlet and refrigerant R134a inlet and outlet. Same model was used to conduct CFD analysis for six cases of loading, mass flow rate and type of flow. This prototype was prepared in ANSYS Spaceclaim software as it was easy to create 3D geometry. Fig-3 shows the full prototype of shell and tube chiller.

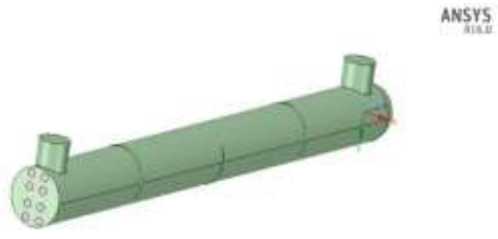

Fig-3: Prototype

3.2 Meshing

The '.stp' file of heat exchanger prototype is generated and imported to ICEM CFD. It is a very user friendly software to mesh a 3D geometry. Firstly, trilateral and quadrilateral meshing is used for surface mesh. Then the tetrahedral meshing is carried out in volume meshing of shell and tubes. Further fine mesh is obtained by prism and layering in the regions of high pressure and temperature gradients. Fig 4 depicts the meshing of the prototype of STHX and table 1 below shows the mesh metrics data:

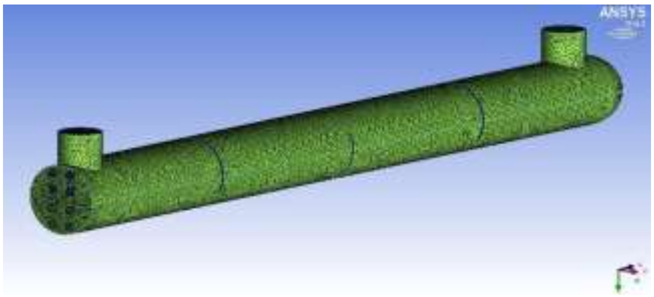

Fig-4: Meshing of the prototype

Table-1: Mesh metrics data

Domain	Nodes	Elements
Fluid shell	415078	1324737
Fluid tube	237102	601275
All Domains	652180	1926012

3.3 Boundary Conditions

The shell and tube type heat exchanger has two in no. inlets and outlets. One pair is for fresh water inlet and outlet and other is for that of refrigerant R134a. By adopting 100 and 75% loading with parallel flow total two in no. cases are studied for the development of temperature profile along shell and tubes. The temperature and streamline plots are acquired for all these two cases and compared. The boundary conditions of mass flow inlet and pressure outlet at all inlets and outlets respectively for six cases are tabulated in table-2 and 3 as below:

Table-2: Mass flow inlet

S. No.	Loading %	Shell /Tube	Mass flow rate kg/s	Temperature °C	Turbulent Intensity %	Hyd. Dia. mm
1	100	Shell	0.01	13	5	12
		Tube	0.123	4		
2	75	Shell	0.0075	13	5	12
		Tube	0.09	4		

Table-3 Pressure outlet

Parameter	Value
Gauge Pressure	0 pa
Temperature	NA
Turbulent Intensity	5%
Hydraulic Diameter	12 mm

3.4 Run Calculation:

Hybrid initialization is used to process the heat exchanger prototype simulation. Number of iterations were kept 2000 and fresh inlet temperature was 130^{0}C. Temperature distribution of the heat exchanger is found from CFD-Post. The temperature profile of shell and tube and streamline plot was obtained from simulation. Also the animation video of planes along and perpendicular to shell were acquired.

4. RESULTS AND DISCUSSIONS

Total 2 in no. cases of simulation were studied in the ANSYS fluent. The temperature contours, streamline figure and shell temperature contours of both cases are discussed below.

4.1 Case 1-100% loading with parallel flow:

By applying the boundary conditions as mentioned in table no-2 and 3 the simulation in ANSYS fluent was carried out. All figures are as depicted below:

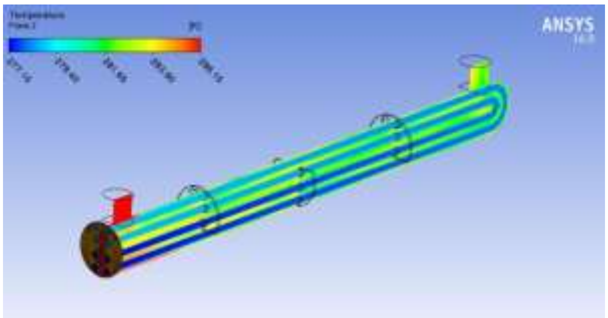

Fig-5: Temperature distribution in two planes

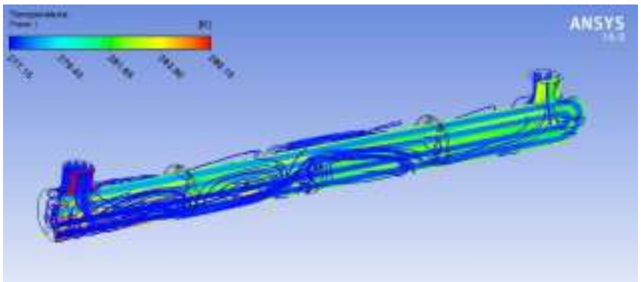

Fig-6: Streamline plot

4.1 Case 1-75% loading with parallel flow:

By applying the boundary conditions as mentioned in table no-2 and 3 the simulation in ANSYS fluent was carried out. All figures are as depicted below:

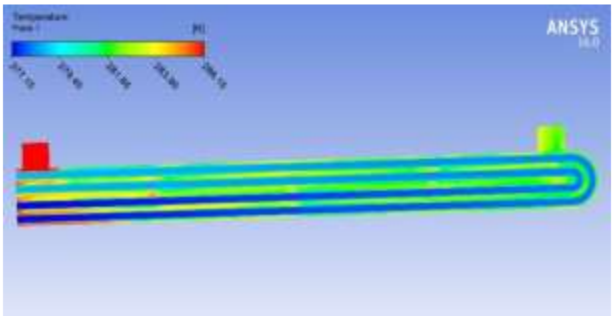

Fig-7: Temperature distribution across shell and tubes

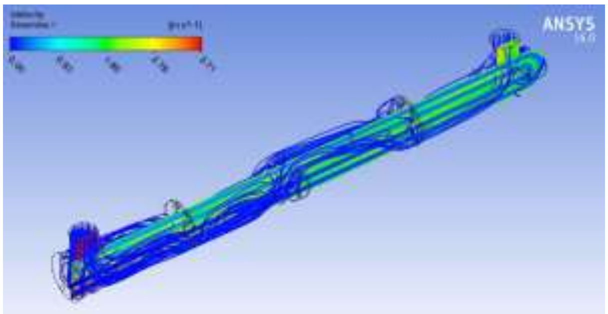

Fig-8: Streamline plot

4.3 Factory Acceptance Trials Data (FATs Data)

The data that has been collected from factory acceptance trials report and tabulated as shown in table no.-4 below:

Table-4: FATs Data

LOAD	%	100	100	100
Chilled water inlet temperature	^{0}C	13	12.5	12
Chilled water outlet temperature		7	6.5	6
Temperature		6	6	6

difference across chiller				
Water flow across chiller	CMH	36	36	36

Based on the data above, the heat flow rate for 100 and 75% loading is calculated as below:

Table no.-5: Actual Heat flow rate from FATs data

S. No.	Fluid	Loading %	Mass flow rate kg/s	ΔT °C	Heat flow rate W
1	Fresh Water	100	0.01	6	251.1
		75	0.0075	6	188.33
2	R134a	100	0.123	1.5	253.5
		75	0.09	1.5	185.49

Using the temperature outputs of both cases the temperature difference and heat flow rate across shell and tubes of prototype is calculated and compared with that of actual data. The comparison is as shown below:

Table no.-6 Comparison of actual and calculated temperature difference and heat flow rate

Case	Fluid	Loading %	Shell/Tube	Temperature °C		ΔT in °C	Heat Flow rate in W
				Inlet	Outlet		
1	Fresh Water	100	Shell	13	7.32	5.68	237.71
			Tube	4	5.67	1.67	282.23
2	R134a	75	Shell	13	7.38	5.62	176.4
			Tube	4	5.76	1.76	217.64

Table no.-7 Comparison of the data for fresh water

Case	Load %	Type of flow	ΔT in °C	ΔT from FATs in °C	Heat Flow rate from CFD in W	Calculated Heat Flow rate in W
1	100	Parallel	5.53	6	237.71	251.1
2	75	Parallel	5.47		176.40	188.33

5. CONCLUSIONS

The heat transfer and temperature flow distribution is conferred in detail in the research. The assumption has operated well in this geometry modelling and meshing. The mesh quality was so fine that the results obtained were indeed clear and adequately visible. The prototype is designed using Kern method and verified for both variations regarding flow direction and mass flow rates and following concluded:

- The design of STHX by using Kern method was found to be appropriate and satisfactory as overall heat transfer coefficient remains same for prototype and actual chiller.
- The temperature outputs at fresh water outlet and refrigerant outlet are found precise with an average error of circa 7%.
- The temperature difference across shell from actual and CFD derived when compared found precisely equivalent with an average error of circa 7%.
- The overall comparison of heat flow rate across shell and tubes of prototype with that of actual chiller is found precisely identical with an average error of circa 5%.

REFERENCES

1) Nakka Sita Rama Raju et al. “Thermal Analysis of STHE to Demonstrate the Heat Transfer Capabilities of Various Thermal Materials using ANSYS”
2) Moses Omolayo petinrin et al, “Performance of Shell and Tube Heat Exchangers with Varying Tube Layouts
3) Thundil Karuppa et al. “Numerical analysis of shell and tube heat exchanger with simple baffle by CFD”
4) Kevin Shah “Design and analysis of shell and tube type heat exchanger, by using CFD analysis”
5) Ender Ozden et al, “Shell side CFD analysis of a small shell -and-tube heat exchanger”
6) Nitesh B. Dahare, “Review on Experimental Analysis and Performance Characteristic of Heat Transfer in Shell and Twisted Tube Heat Exchanger”
7) http://www.engr.uconn.edu - Introductory to FLUENT Training.
8) https://en.wikipedia.org - Types of Meshing, Mesh Quality, Tetrahedron Meshing.
9) http://www.mathematik.uni-dortmund.de/~kuzmin/cfdintro/lecture1.pdf - Mathematical Modelling.

BIOGRAPHIES

Amolkumar Musale has done B.E. from Dr. J. J. Magdum College of Engineering Jaysingpur and currently pursuing M. Tech in Marine Engineering from Defence Institute of Advanced Technology Pune. Prior post-graduation the scholar has experience of 06 years on-board marine ship.

Pravin Hadgekar, working as Assistant Professor, Department of Mechanical Engineering, Defence Institute of Advanced Technology obtained B.E. (Mechanical) and M. Tech (Heat & Power from Vishwakarma Institute of Technology, Pune, guided 12 M. Tech and 10 B.E. thesis, 04 publications in various national and International journals, has 4.5 years of academic and 2.5 years of CFD industry experience.