CFD ANALYSIS OF A 3D MODELLED SHELL & TUBE HEAT EXCHANGER

Vivek Pratap Singh[1], Dharamveer Singh[2], Ashok Kumar Yadav[3],

1. Abstract

Heat transfer is a very common phenomenon of applied physics. It takes place due to the difference in temperature difference between two objects. It has a very prominent role in industries based on thermal engineering. Although many equipment performs this action such as Heat Exchangers, Evaporators, and cooling towers. The effectiveness of a Heat exchanger is a very important factor where such types of operations are being used. Fouling factor is considered while analyzing a heat exchanger due to the deposition of dirt, soot, and dust particles in the tubes of a shell and tube heat exchanger. It lowers the effectiveness of a heat exchanger. The two methods preferred to analyze the performance of a heat exchanger are (i) LMTD and (ii) NTU.

The present paper throws light on the various parameters used in the study and analysis of a heat exchanger and how these parameters are interrelated with each other. It also contains some points obtained by comparing the overall heat transfer coefficient to reduce the fouling factor which will be proved beneficial for designing heat exchangers for industrial as well as research purposes in future. For study in the present paper, the modelling is done on SOLIDWORKS and the mesh generation and analysis of the model is completed in ANSYS workbench. This study can be proved useful in designing of heat exchangers for both analytical and industrial purposes by considering the various parameters of heat exchanger design. It can also be proved helpful enhance the effectiveness of heat exchanger by using the given techniques provided to minimize the fouling factor.

Keywords: Effectiveness, Heat transfer, Fouling, LMTD, NTU, Meshing

[1*,2,] R.D. Engineering College Ghaziabad, UP, India
[3]Raj Kumar Goel Institute of Technology,. Ghaziabad, UP, India.

***Corresponding Author:-** Vivek Pratap Singh
R.D. Engineering College Ghaziabad, UP, India hodme040@gmail.com

DOI: 10.48047/ecb/2023.12.si10.00494

2. INTRODUCTION

Heat Exchanger is a mechanical device which has an important role in industries where the thermal operations (various heating & cooling processes) on a liquid are performed.

A wavy fin and tube heat exchanger was mathematically and practically analysed by (Wolf et al., 2006). Utilising the method of computational fluid dynamics, the provided model was solved, and the solution's accuracy was assessed using experimental data and Wang's correlations. It was discovered that the empirically measured outcomes and the coefficients for the friction and Colburn factors are consistent. The model that was presented showed accurate heat transport predictions. By numerically simulating a tiny heat exchanger, (Ozden & Tari, 2010) explored the outer shell design of a heat exchanger made of shells and tubes, that is, the baffle spacing, baffle cut, and shell diameter relationships of heat transfer coefficients and the pressure drop. Results have been obtained from a series of CFD simulations for an individual shell and single tube passage heat exchanger with a varying number of baffles and turbulence in the flow. Most of the time, the variation in total thermal transfer rates between Bell-Delawere and CFD forecasts is less than 2%. This approach demonstrated the effectiveness of computational fluid dynamics (CFD) techniques as a tool for heat exchanger design.

In the fluidized bed model of a vertically heat exchanger composed of shells and tubes with counter flow, (Kang et al., 2011) looked into the impact of circulating solid fragments on the characteristics of the flow of fluid, the transfer of heat, and cleaning effect. A variety of solid substances, including glass, aluminium, steel, copper, and sand, were used. They also examined seven different solid particles that had the same volume as well as the effects on multiple parameters. The results that were obtained demonstrate that larger density solid particle flow velocity range for particle impact to the wall was greater. (Aslam Bhutta et al., 2012) investigated the use of CFD applications in heat exchanger development using LMTD and NTU techniques and came to the conclusion that commercial CFD software can satisfy the need for CFD examination of multiple kinds of heat exchangers and full heat exchange device design and optimisation involving an extensive variety of turbulence mathematical models and incorporating available strategies in CFD section.

In the case of a heat exchanger using perforated plates, (Li et al., 2013) analysed the forecasts on precipitation and particle fouling using various geometric factors, including plate high, plate separation, and plate angle. In a 3D numerical simulation, the realisable k-model with greatest equilibrium exterior wall function is used to calculate the Nusselt value and wall shear stress, and the Von-Kerman an analogue is used to calculate the coefficient for the transfer of mass. It was discovered that BRM01 had the greatest performance for heat transfer due to its longest plate angle of 65 degrees and the smallest correlations pitch of 7 mm. (Misra et al., 2013) examined the impacts of time duration, thermal conductivity of soil, pipe diameter, and velocity of flow on the thermal properties of EATHE using tests and CFD. And it was discovered that for soil with a coefficient of thermal conductivity of 0.52 W/mK, cooling of the air decreased from 18.7 ^{o}C to 16.6 ^{o}C under transient conditions whereas an impairment of 18.8 0C was recorded under steady state conditions.

The highest possible drop in temperature was achieved for a pipe with a length of 100 mm, diameter 0.2-meter, velocity 5 m/s, and is 18.40C, 18.70C, and 18.40C for different soil conductivities, according to (Bansal et al., 2013) evaluation of a new term "Derating Factor" and studies approximately for an EATHE via a test and the CFD. In a study by (Bai et al., 2014), 6 distinct exhaust exchangers for heat were created within the exact same shell, and the CFD simulation models were compared with a typical vehicle driving cycle. The study's findings revealed that the greatest heat transfer of 1737 W was achieved by using 7 baffles, which improved heat transmission. This rate was higher than the distinct plate with holes and the pair of parallel plate construction by 35% and 26%, respectively.

In their analysis of the many turbulence models offered by general-purpose clouds commercialized CFD tools including the k-model and the k-SST model, (Patel et al., 2015) shown the value of CFD in predicting both the behaviour and the efficiency of a wide range of HE. It is one of the best techniques for investigating these kinds of heat sources. Through the use of CFD, (Babu & Gugulothu, 2015) improved the efficiency of heat transfer for a HE employing passive ways that don't require power from the outside. They discovered that the improvement in heat transfer rose by 10%–15% at the expense of the permitted pressure decrease.

(Petrik et al., 2019) used CFD to do a parametric study on the heating capacity of a small automobile radiator. The physical fin model is suitable, they decided. The permeable model is a good substitute. Utilizing the ANSYS programming tool, (Kishan et al., 2020) researched and created three different types of heat exchangers. The optimum form of zig-zag pattern was discovered to identify the rate of heat transmission of HE advances among three varieties, including parallel S type and zig-zag pattern.

In order to decrease consumption of energy in structures, (Congedo et al., 2020) suggested using a heat pump with an air source system in conjunction with a horizontal earth to air heat exchanger. By comparing how it performed to the conventional ASHP and utilizing various coefficients using a 3 m pipe length, the performance was examined. (Khan et al., 2023) used round and hexagonal tubes in their CFD modelling and experimental investigation of a shell and tube heat exchanger. They looked at temperature decline, the transfer of heat coefficients, and the rate of heat transmission. In comparison to round tubes, hexagonal tubes had a better dispersion of flow.

3. METHODOLOGY

Heat exchanger is an equipment used to lower the temperature of a working fluid with the use of a coolant (another fluid having lowered temperature with respect to the working fluid). In the present study, the given heat exchanger model is created in SOLIDWORKS. The basic model created by this software is being shown in figure 3.1.

Fig. (3.1) : Model Created in SOLIDWorks with different views.

this software is frequently used to create the 3D models and to perform their static analysis in real time but this not suitable for thermal analyses of those models. Hence, there is no need to create pre analysis real prototyping. The various steps involved in this present study from creating the model to analyze it are shown in fig (3.2)

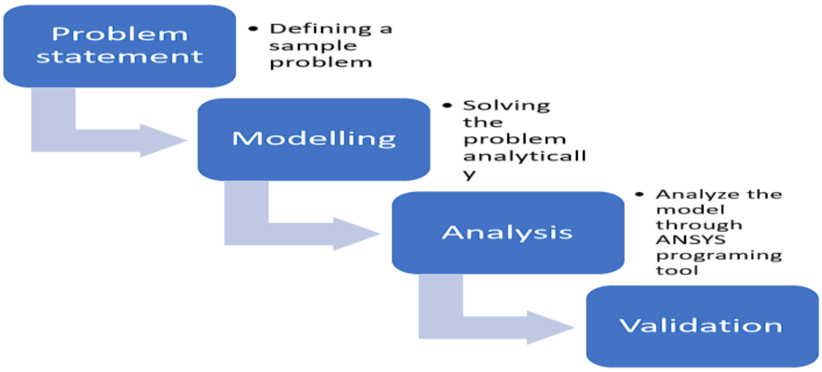

Fig. 3.2 : Various steps involved in this research

There are four cases considering the question problem. These four cases are solved numerically as well as by using ANSYS programming tool. The first case is associated with the length of tube of heat exchanger. In this the length of heat exchanger tube is varying from 2 m to 3 m, and the variation in heat transfer rates are calculated for this 1 m change in length of tube, the results are obtained. In second case the study is done by varying the internal and outer diameter of tube, the corresponding change in values of diameter and heat transfer rates are numerically calculated.

In the third case, there is a variation of number of tubes of heat exchanger and by analytical method, the heat transfer rates are calculated. The fourth case is associated with the different materials which are used in making the tubes of the given heat exchanger. There are some assumptions which are made during the study of presented heat exchanger model. The coefficient of transfer of heat is uniform throughout the material. While during study and analysis, the flow conditions are steady. The two fluids used in this heat exchanger are supposed to have uniform specific heats and rates of mass flow. There are some negligible changes assumed in the kinetic energy as well as the potential energy of the fluids. During heat transfer, there is no condition of phase change as phase change is the function of temperature and pressure (specific heats are uniform). The changes found in the temperature and pressure are not enough to change the state of working fluids. Heat exchanger is insulated properly to prevent the loss of heat to the surroundings.

For numerical calculations, there are two methods generally used to calculate heat transfer rate and effectiveness. Theses methods are (i) Logarithmic Mean Temperature Difference (LMTD) (ii) Number of Transferred Units (NTU).

$$\theta_{lm} = \frac{\theta_{t1} - \theta_{t2}}{\ln(\frac{\theta_{t1}}{\theta_{t2}})}$$

Where,
θ_{lm} = Logarithmic Mean Temperature Difference.
θ_{t1} = Difference in inlet temperatures of both fluids.
θ_{t2} = Difference in outlet temperatures of both fluids.

Effectiveness of heat exchanger,

$$\in_{HE} = \frac{\left[1 - e^{-\{NTU(1-R)\}}\right]}{\left[1 - Re^{\{-NTU(1-R)\}}\right]}$$

Where,
R = Capacity Ratio
NTU = No. of Transfer Units.

This research article is completely based upon the modelling and analysis of a shell and tube heat exchanger for B. L. Agro. Industries pvt. Ltd, an edible oil production Company. The design parameters for this heat exchanger are taken from the maintenance department of this company. These parameters are listed in table (1)

Table (1) : DESIGN PARAMETERS FOR HEAT EXCHANGER

S. No.	DESIGN PARAMETERS	VALUES
1	Internal diameter of tube	20 mm
2	Outer diameter of tube	23 mm
3	Water flow rate through tube	0.3 Kg/s
4	Oil flow rate through tube	0.2 Kg/s
5	Water inlet temperature	20 ^{o}C
6	water outlet temperature	30 ^{o}C
7	Oil inlet Temperature	75 ^{o}C
8	oil outlet temperature	60 ^{o}C
9	Water side film coefficient	4500 W/m^{2} ^{o}C
10	Oil side film coefficient	1250 W/m^{2} ^{o}C
11	conductivity of Tube wall	350 W/m ^{o}C
12	Fouling factor for water	0.0004
13	Fouling factor for oil	0.001
14	length of tube	2.4 m

4. RESULTS AND DISCUSSION

Heat Transfer Rate,
$Q_t = U_o A_H \Theta_{lm}$
Where,
Q_t = Total heat transfer rate in
U_o = Overall heat transfer coefficient
Θ_{lm} = Logarithmic mean temperature difference
A_H = total heat transfer area = $\pi * d_o * L$
d_o = Outer diameter of heat exchanger tube
L = length of tube

Length of tube is varying from 2 m to 3 m and the results are obtained for heat transfer rates which are being shown in table (2)and figure 4.1.

Table (2) : Variation in Heat transfer w.r.t. length of tube

S. No.	LENGTH	IN TEMP	OUT TEMP	SPECIFIC HEAT	MASS FLOW	NO	HEAT TRANSFER AREA	Θ_1	Θ_2	LMTD	HEAT TRANSFER RATE
1	2	20	30	4.187	0.3	1	0.14451326	45	40	42.451	2429.841128
2	2.05	20	30	4.187	0.3	1	0.14812609	45	40	42.451	2490.587156
3	2.1	20	30	4.187	0.3	1	0.15173893	45	40	42.451	2551.333184
4	2.15	20	30	4.187	0.3	1	0.15535176	45	40	42.451	2612.079212
5	2.2	20	30	4.187	0.3	1	0.15896459	45	40	42.451	2672.825241
6	2.25	20	30	4.187	0.3	1	0.16257742	45	40	42.451	2733.571269
7	2.3	20	30	4.187	0.3	1	0.16619025	45	40	42.451	2794.317297
8	2.35	20	30	4.187	0.3	1	0.16980308	45	40	42.451	2855.063325
9	2.4	20	30	4.187	0.3	1	0.17341591	45	40	42.451	2915.809353
10	2.45	20	30	4.187	0.3	1	0.17702875	45	40	42.451	2976.555382
11	2.5	20	30	4.187	0.3	1	0.18064158	45	40	42.451	3037.30141
12	2.55	20	30	4.187	0.3	1	0.18425441	45	40	42.451	3098.047438
13	2.6	20	30	4.187	0.3	1	0.18786724	45	40	42.451	3158.793466
14	2.65	20	30	4.187	0.3	1	0.19148007	45	40	42.451	3219.539494
15	2.7	20	30	4.187	0.3	1	0.1950929	45	40	42.451	3280.285523
16	2.75	20	30	4.187	0.3	1	0.19870574	45	40	42.451	3341.031551
17	2.8	20	30	4.187	0.3	1	0.20231857	45	40	42.451	3401.777579
18	2.85	20	30	4.187	0.3	1	0.2059314	45	40	42.451	3462.523607
19	2.9	20	30	4.187	0.3	1	0.20954423	45	40	42.451	3523.269635
20	2.95	20	30	4.187	0.3	1	0.21315706	45	40	42.451	3584.015663
21	3	20	30	4.187	0.3	1	0.21676989	45	40	42.451	3644.761692

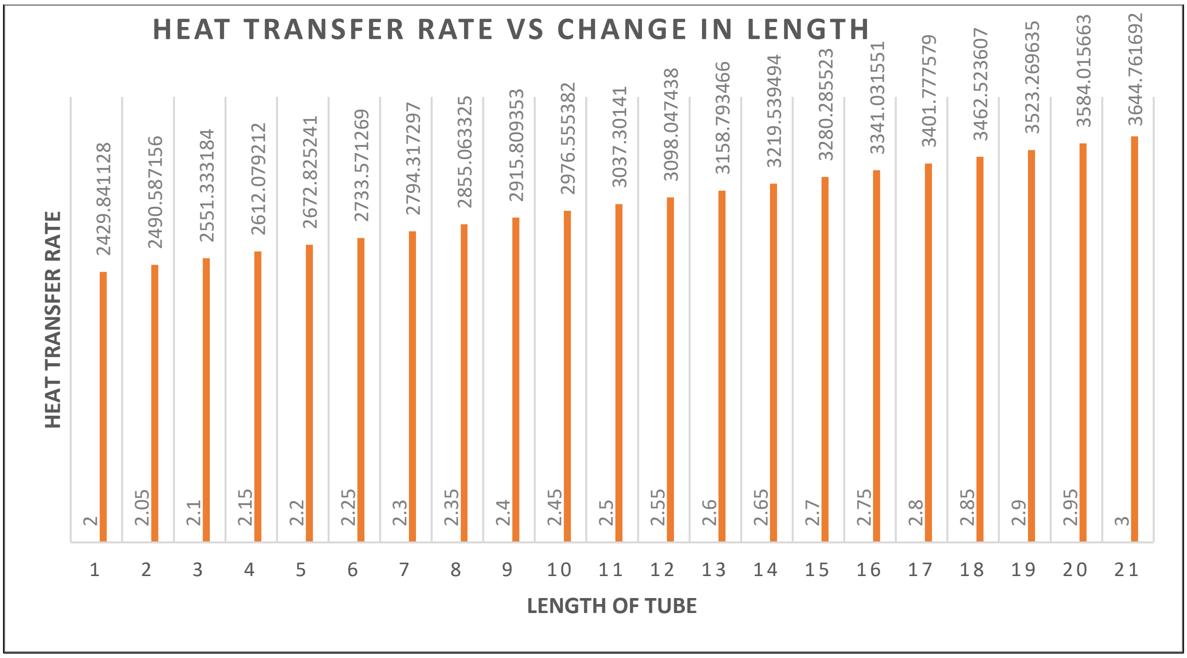

Fig 4.1 : Variation in Hear transfer rate with respect to change in length

Now,

In second case the inner diameter of tube varies from 20 mm to 50 mm and outer diameter varies from 23 mm to 53 mm. the results for heat transfer rates are obtained for the change in diameter and the same are being shown in table (3) and figure 4.2.

Table (2) Variation in heat transfer rate with respect to change in outer diameter

S. No.	INLET DIA	OUTLET DIA	L	NO.	HEAT TRANSFER AREA	OVERALL H TRANS COEFF	HEAT TRANSFER RATE
1	0.02	0.023	2	1	0.144513262	396.81232	2434.334
2	0.02	0.023	2	1	0.15393804	397.84349	2599.833
3	0.02	0.023	2	1	0.163362818	398.74454	2765.255
4	0.02	0.023	2	1	0.172787596	399.53865	2930.614
5	0.02	0.023	2	1	0.182212374	400.24377	3095.92
6	0.02	0.023	2	1	0.191637152	400.87408	3261.181
7	0.02	0.023	2	1	0.20106193	401.44089	3426.405
8	0.02	0.023	2	1	0.210486708	401.95332	3591.597
9	0.02	0.023	2	1	0.219911486	402.41885	3756.76
10	0.02	0.023	2	1	0.229336264	402.84363	3921.9
11	0.02	0.023	2	1	0.238761042	403.23279	4087.018
12	0.02	0.023	2	1	0.24818582	403.59063	4252.118
13	0.02	0.023	2	1	0.257610598	403.92079	4417.201
14	0.02	0.023	2	1	0.267035376	404.22635	4582.27

15	0.02	0.023	2	1	0.276460154	404.50997	4747.326
16	0.02	0.023	2	1	0.285884931	404.77392	4912.37
17	0.02	0.023	2	1	0.295309709	405.02019	5077.403
18	0.02	0.023	2	1	0.304734487	405.2505	5242.427
19	0.02	0.023	2	1	0.314159265	405.46634	5407.443
20	0.02	0.023	2	1	0.323584043	405.66904	5572.451
21	0.02	0.023	2	1	0.333008821	405.85977	5737.451

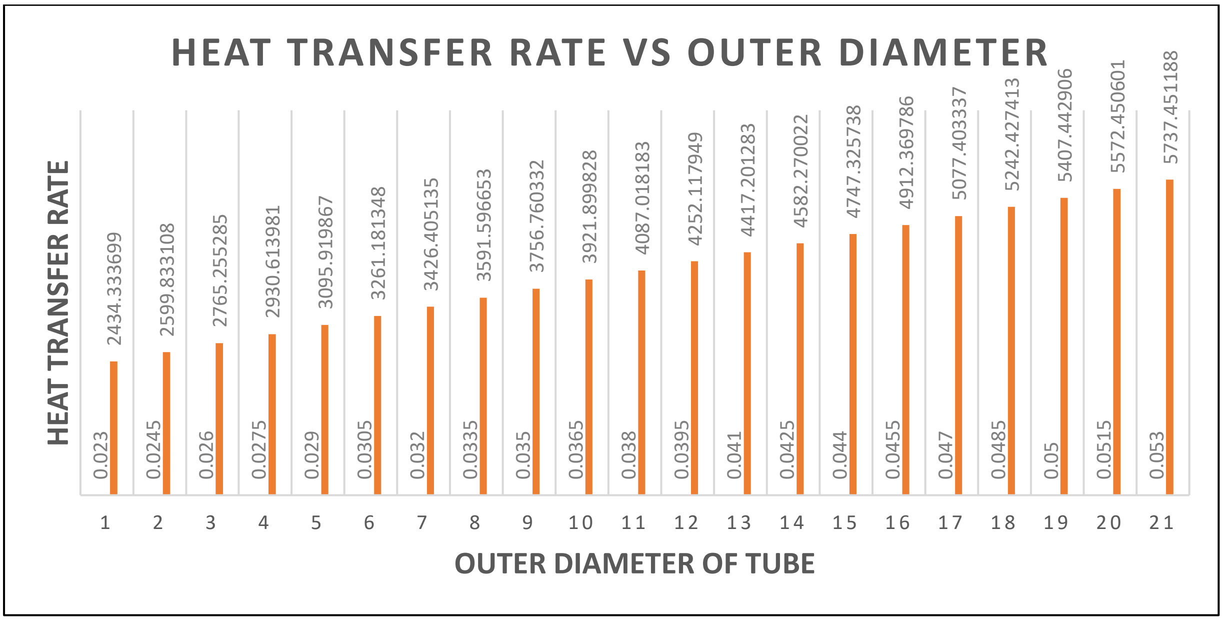

Fig. (4.2) : Variation in Heat transfer rates with respect to change in outer diameter

In third, each and every parameter of sample problem statement remains same and the number of tubes varies from 1 to 21. The results obtained for this variation are being shown in table (3) and figure 4.3

Table (3) Variation in Heat transfer rate with respect to number of tubes

S. No.	INLET DIA	OUTLET DIA	LENGTH	NO	HEAT TRANSFER AREA	HEAT TRANSFER RATE
1	0.02	0.023	2	1	0.144513262	2429.841128
2	0.02	0.023	2	2	0.289026524	4859.682256
3	0.02	0.023	2	3	0.433539786	7289.523383
4	0.02	0.023	2	4	0.578053048	9719.364511
5	0.02	0.023	2	5	0.72256631	12149.20564
6	0.02	0.023	2	6	0.867079572	14579.04677
7	0.02	0.023	2	7	1.011592834	17008.88789
8	0.02	0.023	2	8	1.156106097	19438.72902
9	0.02	0.023	2	9	1.300619359	21868.57015
10	0.02	0.023	2	10	1.445132621	24298.41128
11	0.02	0.023	2	11	1.589645883	26728.25241
12	0.02	0.023	2	12	1.734159145	29158.09353
13	0.02	0.023	2	13	1.878672407	31587.93466
14	0.02	0.023	2	14	2.023185669	34017.77579
15	0.02	0.023	2	15	2.167698931	36447.61692
16	0.02	0.023	2	16	2.312212193	38877.45804
17	0.02	0.023	2	17	2.456725455	41307.29917
18	0.02	0.023	2	18	2.601238717	43737.1403
19	0.02	0.023	2	19	2.745751979	46166.98143
20	0.02	0.023	2	20	2.890265241	48596.82256
21	0.02	0.023	2	21	3.034778503	51026.66368

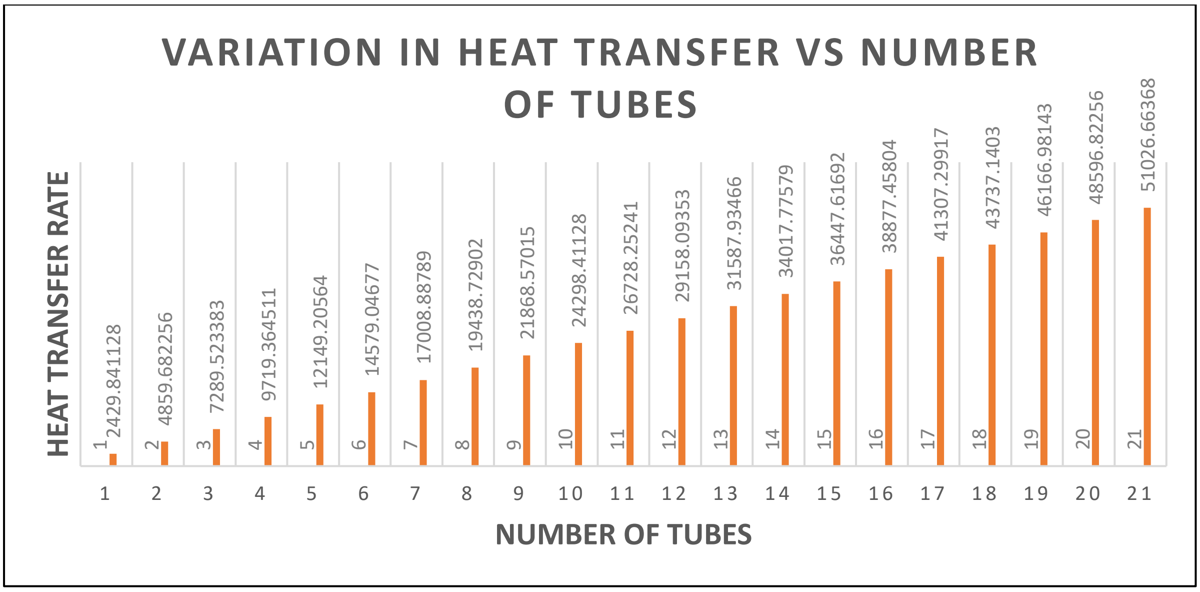

Fig. (4.3) : Variation in Heat transfer rates with respect to change in number of tubes

As the fourth case deals with the material of tube, heat transfer rates are directly proportional to the thermal conductivity of the tube material. Higher the thermal conductivity of the material, higher the heat transfer rates obtained. The component of electronics "k_e" and the interconnected component "k_g" are thought to be added to the thermal conductivity component, which is typically denoted by the symbol "k".

$k = k_e + k_g$ for a particular material, it can be different at different temperatures. The conductivity of some selective materials is as follows as being shown in table (4)

Table (4) : Thermal conductivity for different materials

S. No.	MATERIAL	TEMPERATURE	THERMAL CONDUCTIVITY
1	Aluminium	-73	237
		0	236
		127	240
		327	232
		527	220
2	Alumnium – duralium	20	164
3	aluminium – Silicon	20	164
4	Aluminium - bronze	0 - 25	70
5	Aluminium Alloy 3003	0 - 25	190
6	Aluminium Alloy 2014	0 - 25	190
7	Aluminium Alloy 360	0 - 25	150
8	Chromium	-73	112.11
		0	94.8
		127	87.31
		327	80.35
		527	71.23
		727	65.53
		927	62.64
9	Copper	-73	412.9
		0	400.99
		127	392.12
		327	383.17
		527	371.16
		727	357.15
		927	342.19
10	Iron	-73	94.3
		0	82.8
		127	69.45
		327	54.73
		527	43.39
		727	32.62
		927	28.25

The above data validates the newton's law of cooling in which the thermal conductivity is inversely proportional to the temperature, that is higher the temperature of material, the thermal conductivity decreases.

5. CONCLUSION

The above results which are obtained from the analytical data and from ANSYS programing tool, it is found that the heat transfer varies linearly with span of tube of heat exchanger. Hence, it is very common to say that the heat transfer rate is directly proportional to the length of tube. The heat transfer gradient is also linear with change n outlet diameter of tube.

Thus, it is obvious that the heat transfer is a direct function of length and diameter which is also a linear function of the heat transfer area.
The numerical data is very similar to the data obtained by the ANSYS programming tool. It is concluded that the CFD programming software like ANSYS, Simscale or STAR CCM+ etc gives the best results or very close to real values. Hence, these software are very effective in real time analysis of a mechanical equipment. This present study is very beneficial for the research scholars working in the optimization and effectiveness of a shell & Tube heat exchanger and in thermal industries in which the heat transfer is a very common phenomenon, it will serve the academia and scholars as well as heat exchanger designers (Design Engineers) that they will come to know how these explained parameters effect the design of a Heat Exchanger.

During the course of heating and cooling operations, fouling is the buildup of undesirable material deposits on heat transfer surfaces. It affects all sectors and the majority of heat exchangers, deteriorating heat transmission while also increasing the resistance to flow and pressure decreases. A decrease in pressure and inadequate heat transfer cause manufacturing inefficiency. Heavy fouled deposits demand additional equipment and maintenance time. Due to industry closures, it also causes a loss in production. Fouling can be caused by a number of factors, such as the pH of the water, the speed of the fluid, the roughness of the component surfaces, etc. Heat transmission and pressure decrease within the heat exchange mechanism can be used to identify fouling. The fouling elements cause a reduction in thermal transfer rates. If there is a rise in the pressure drop between the inlet and output, fouling is the likely source of the thermal exchanger's frictional barriers.

Physical factors including the speed of the fluid, liquid temperature, fluid chemistry, and manufacturing material can all help to lessen the impact of fouling. Due to an increase in shear stress in fluids caused by increased flow rates, fouling is reduced. However, if the contributions are particularly powerful, speeding up the flow can have little impact. With a rise in temperature, deposits of salt upon the heat exchangers sides grow. By limiting the region where the product may flow, fouling raises pressure reduction. If a fluid undergoes chemical changes, the difference may cause a coating of tube fouling. Choose units made with corrosion-resistant metallic materials and alloys if you want to avoid corrosion contamination from layers of heat resistance material.

The various methods explained in this study can help the designers to select materials matched best for the heat exchangers. There are also some methods discussed here to reduce the negative factor named as fouling.

REFERENCES

1. Aslam Bhutta, M. M., Hayat, N., Bashir, M. H., Khan, A. R., Ahmad, K. N., & Khan, S. (2012). CFD applications in various heat exchangers design: A review. In *Applied Thermal Engineering* (Vol. 32, Issue 1, pp. 1–12). Elsevier Ltd. https://doi.org/ 10.1016/ j.applthermaleng.2011.09.001
2. Vivek Pratap Singh, Dharamveer Singh & Ashok Kumar Yadav (2022). Analytical Investigation of Ribbed Channels for Gas Turbine in *International Journal of Research Trends & Innovation* (Vol. 7, Issue 8).
3. Babu, C. R., & Gugulothu, S. K. (2015). CFD Analysis of Heat Transfer Enhancement by Using Passive Technique in Heat Exchanger. *International Journal of Recent Advances in Mechanical Engineering*, *4*(3), 99–111. https://doi.org/10.14810/ijmech.2015.4308
4. Bai, S., Lu, H., Wu, T., Yin, X., Shi, X., & Chen, L. (2014). Numerical and experimental analysis for exhaust heat exchangers in automobile thermoelectric generators. *Case Studies in Thermal Engineering*, *4*, 99–112. https://doi.org/10.1016/j.csite.2014.07.003
5. Bansal, V., Misra, R., Agarwal, G. Das, & Mathur, J. (2013). "Derating Factor" new concept for evaluating thermal performance of earth air tunnel heat exchanger: A transient CFD analysis. *Applied Energy*, *102*, 418–426. https://doi.org/10.1016/j.apenergy.2012.07.027
6. Congedo, P. M., Baglivo, C., Bonuso, S., & D'Agostino, D. (2020). Numerical and experimental analysis of the energy

performance of an air-source heat pump (ASHP) coupled with a horizontal earth-to-air heat exchanger (EAHX) in different climates. *Geothermics*, *87*. https://doi.org/10.1016/ j.geothermics. 2020.101845

7. Kang, H. K., Lee, B. C., Ahn, S. W., Choi, J. W., & An, S. H. (2011). Numerical and experimental analyses of anti-fouling and heat transfer in the heat exchanger with circulating fluidized bed. *Canadian Journal of Chemical Engineering*, *89*(2), 240–253. https://doi.org/ 10.1002/cjce.20397
8. Dharamveer, Samsher, D. B. Singh, A. K. Singh, N. Kumar, Solar Distiller Unit Loaded with Nanofluid- A Short Review. Lecture Notes in Mechanical Engineering, Springer, Singapore, (2019) 241-247, https://doi.org/10.1007/978-981-13-6577-5_24.
9. Khan, A., Shah, I., Gul, W., Khan, T. A., Ali, Y., & Masood, S. A. (2023). Numerical and Experimental Analysis of Shell and Tube Heat Exchanger with Round and Hexagonal Tubes. *Energies*, *16*(2). https://doi.org/10.3390/ en 16020880
10. Dharamveer and Samsher, Comparative analyses energy matrices and enviro-economics for active and passive solar still, materials today: proceedings, https://doi.org/10.1016/j.matpr.2020.10.001 (2020)
11. Kishan, R., Singh Assistant Professor, D., Kumar Sharma, A., & Professor, A. (2020). Article ID: IJMET_11_02_001 Cite this Article: Ram Kishan, Devendra Singh and Ajay Kumar Sharma, CFD Analysis of Heat Exchanger Models Design Using Ansys Fluent. *International Journal of Mechanical Engineering and Technology (IJMET)*, *11*(2), 1–9. http://www.iaeme.com/ ijmet/issues.asp?JType=IJMET&VType=11&IType=2http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=11&IType=2
12. Dharamveer, Samsher, Anil Kumar, Analytical study of Nth identical photovoltaic thermal (PVT) compound parabolic concentrator (CPC) active double slope solar distiller with helical coiled heat exchanger using C$_u$O Nanoparticles, Desalination and water treatment, 233 (2021) 30-51, https://doi.org/10.5004/dwt.2021.27526
13. Li, W., Li, H. X., Li, G. Q., & Yao, S. C. (2013). Numerical and experimental analysis of composite fouling in corrugated plate heat exchangers. *International Journal of Heat and Mass Transfer*, *63*, 351–360. https://doi.org/10.1016/j.ijheatmasstransfer.2013.03.073
14. Misra, R., Bansal, V., Agrawal, G. Das, Mathur, J., & Aseri, T. K. (2013). CFD analysis based parametric study of derating factor for Earth Air Tunnel Heat Exchanger. *Applied Energy*, *103*, 266–277. https://doi.org/ 10.1016/j.apenergy.2012.09.041
15. Ozden, E., & Tari, I. (2010). Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy Conversion and Management*, *51*(5), 1004–1014. https://doi.org/10.1016/j.enconman.2009.12.003
16. Dharamveer, Samsher, Anil Kumar, Performance analysis of N-identical PVT-CPC collectors an active single slope solar distiller with a helically coiled heat exchanger using CuO nanoparticles, Water supply, October 2021, SCI-E Index, IWA Publication. I.F 1.275,https://doi.org/10.2166/ws.2021.348
17. Patel, D. S., Parmar, R. R., & Prajapati, V. M. (2015). CFD Analysis of Shell and Tube Heat Exchangers-A review. *International Research Journal of Engineering and Technology*. www.irjet.net
18. Petrik, M., Szepesi, G., & Jármai, K. (2019). CFD analysis and heat transfer characteristics of finned tube heat exchangers. *Pollack Periodica*, *14*(3), 165–176. https://doi.org/ 10.1556/606.2019.14.3.16
19. Wolf, I., Franković, B., Viličić, I., Jurkowski, R., & Bailly, A. (2006). A NUMERICAL AND EXPERIMENTAL ANALYSIS OF HEAT TRANSFER IN A WAVY FIN-AND-TUBE HEAT EXCHANGER. In *Energy and the Environment*.

AUTHORS' BIO

Vivek Pratap Singh is presently an M. Tech. scholar in the field of Thermal Engineering from R. D. Engineering College, Ghaziabad. He is pursuing research in Computational Fluid Dynamics (CFD). He has done B. Tech. in Mechanical Engineering from Dr. A. P. J. Abdul Kalam Technical university, Lucknow, U.P., India. He is also working as an asst. professor, Department of Mechanical Engineering at A. N. A. College of Engineering & Management Studies, Bareilly. He has participated in 6 International Conferences. He has published one paper in a reputed International Journal. Currently he is writing a book on fundamentals of CFD, which is going to be published soon. During his teaching career, he has guided more than 40 B. Tech. projects.

Dr. Dharamveer Singh is working as Professor, Deptt of Mech Engg in R D Engineering College Ghaziabad, U.P., India. He did B.Tech in mechanical engineering from Delhi College of Engineering, MTech in Thermal Engineering from RTU Kota, Rajasthan, MBA in HRM from IGNOU, New Delhi, and Ph.D. in Thermal Engineering from Delhi Technological University Delhi. He has total 20 years of experience. He has published more than 15 research papers in SCI/Scopus/UGC care list journal, 07 books/book chapters & 02 patents.. During his teaching career, he has guided more than 20 M.Tech Thesis. Dr. Singh works as a reviewer for many reputed SCI journals.

Dr. Ashok Kumar Yadav is presently working as an **Associate Professor & Head** in the Department of Mechanical Engineering at **RKGIT Ghaziabad**. He obtained both his Ph.D. and M. Tech in Mechanical Engineering from Jamia Millia Islamia (A Central University) New Delhi and B.E in Mechanical Engineering from Visveswaraiha Technological University, Belgaum. Dr. Yadav possess more than 16 years of teaching, research and industrial experience with leading institutions/organizations. His research interest includes alternative fuels for IC engines, energy technologies, sustainable energy & the environmental impact of energy. He has to his credit over 75 research articles in peer reviewed International Journals and conferences including **50 in reputed SCI/Scopus indexed journals**, 11 books/book chapters & four patents. He has supervised 10 M. Tech. thesis. Currently 3 Ph.D., scholars are working under his supervision. Dr. Yadav is on the editorial board of many journals and also works as a reviewer for many SCI journals.