# **CFD** analysis Compact Heat Exchanger

<sup>1</sup>Sushil Kumar, <sup>2</sup>Pawan Yadav

<sup>1,2</sup>Department of Mechanical Engineering, R.D. Engineering College, Duhai, Gaziabad (U.P),

India

Corresponding author-<u>sushilmit07@gmail.com</u>

Abstract: One of the essential components of machinery, gadgets, and industrial processes is heat transfer, which helps to maintain their functionality and produce goods of higher quality. Therefore, to drain excess heat from the process or device and to maintain the optimum operating temperatures, heat exchangers of various types and sizes are utilised in these applications. However, the size of a heat exchanger is crucial to take into account for any kind of process or equipment because it establishes how much room the machine, device, or treatment facility will require. The goal of this work is to theoretically investigate the design process of a heat exchanger before employing computer-aided fluid dynamics to analyse and optimise its performance. A counter-current heat exchanger was considered for design purposes and its length was theoretically calculated using the LMTD method, while the pressure drop and energy consumption were also calculated with the Kern method. In the CFD analysis, the three-case model was used in this study to analyze the behavior of heat transfer, mass flow rates, pressure drops, flow velocities and vortices of the bundle flows in the heat exchanger. Theoretical and CFD results showed only a 1.15% difference in the cooling performance of hot fluids. The axial pressure drops showed positive correlations with the total heat transfer coefficient and the required pumping power. Overall, the results of this study confirm that CFD modeling can be promising for the design and optimization of heat exchangers and that it can test many design options without producing physical prototypes.

### Keywords: CFD, Heat Exchanger, LMTD, ANSYS.

#### I. INTRODUCTION

Heat exchangers are among the most commonly used devices in the process industry. Heat exchangers are used to transferring heat between two process flows. Their use shows that any process involving cooling, heating, condensation, boiling, or evaporation requires a heat exchanger for this purpose. Process liquids are generally heated or cooled before the processor is subject to a phase change. Different heat exchangers are named according to their application. For example, the heat exchangers used for condensation are called condensers, in the same way the heat exchangers for cooking are called boilers. The performance and efficiency of heat exchangers are measured using the amount of heat transfer using the smaller heat transfer area and the pressure drop [1]-[2]. Efficiency can best be represented by calculating the total heat transfer coefficient. The pressure drop and the area required for a particular heat transfer provide an overview of the investment costs and energy requirements (operating costs) of a heat exchanger.

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

### A. Heat exchangers are of two types

- If the two fluids between which heat is exchanged are in direct contact with each other, the heat exchanger is in direct contact.
- If the two fluids are separated by a wall through which heat is transferred so that they never mix, contact the heat exchangers with indirect contact. A typical heat exchanger, typically for applications with higher pressures up to 552 bar, is the tube bundle heat exchanger. Tube bundle heat exchanger, indirect contact heat exchanger. It consists of a series of tubes through which one of the liquids flows. The bowl is the container for the bowl liquid. In general, it has a cylindrical shape with a circular section, although in some applications shells of different shapes are used. For this particular study, a hull is considered, which a single-passage hull [3] is generally. A shell is often used because of its low cost and simplicity and has the highest correction factor for the log- mean temperature difference (LMTD). Although the tubes may have one or more passages, there is a passage on the side of the housing while the other liquid in the housing flows over the tubes to be heated or cooled. Liquids on the side of the tube and on the side of the bowl are separated by a tube plate.
- B. There are following objective of this research work:
- The main objective of the study calculated the total heat transfer coefficient.
- Improve the heat transfer rate by using ANSYS CFD.
- During the CFD calculations of the flow in internally ribbed tubes.
- Calculated the temperature distribution and pressure inside the tube by using ansys.

### II. **Related Work**

Neeraj Kumar Nagayach et al. [4] this work provides a summary of the research work of the last decade on the expansion of heat transfer in circular and non-circular tubes. Active and passive methods are wont to increase the heat transfer coefficient within the heat exchanger; Passive methods don't require external power, as within the case of active methods. The effectiveness of the active and passive methods strongly depends on the sort of heat transfer, which may vary from single-phase free convection to the boiling of a dispersed flow film. During this work, the stress is on works involving the staggered use of circular tubes (rotating threaded insert, threaded insert, spiral threaded insert, wire mesh insert), circular (triangular, rectangular) and non-analysis tubes on CFD in laminar and flow.

Nasir Hayat et al. [5] this review of the literature focuses on the applications of CFD within the field of heat exchangers. It's been discovered that CFD is employed within the following investigation areas in several sorts of heat exchangers: misalignment of the liquid flow, pollution, pressure drop and thermal analysis within the design and optimization phase. They need been adopted for the implementation of the simulations. The standard of the solution obtained from these simulations falls within the suitable range, which shows that CFD is an efficient tool for predicting the behavior and performances of a spread of heat exchangers.

Hosseini et al. [6] studied the influence of particle size on deposition in compact heat exchangers. In his work, ANSYS Fluent software was used to solve Navies Stokes equations mediated by Reynolds. Appropriate turbulence models were assessed and a standard epsilon turbulence model with a standard wall function was found appropriate. The semi-implicit equations relating to pressure were chosen as the coupling scheme for the printing speed. As a

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

result, a numerical investigation has shown that particle deposition increases with increasing particle size.

Ismail et al. [7] in this paper presented are the worked with CFD on compact fins for heat exchangers. Analyzes were performed using Ansys Fluent commercial software to predict the factors f and j for the corrugated and offset fins. Periodic constraints have been applied to eliminate the input effects of the inputs. It is emphasized that there are many correlations between the f and j factors of the band moved in the literature, and these correlations show deviations from each other. Furthermore, new correlations of fe<sub>j</sub> factors have been developed as a function of Reynolds number and the geometric parameters of the ribs for the wavy fins. Finally, these correlations have been compared with other correlations from the literature.

Rao et al. [8] studied the factors few of the compact fin heat exchanger for single fins using CFDs. In their study, only simple fins were considered. The rectangular flow geometry was modeled taking one quarter of the section due to symmetry. The analyzes were performed using a standard k standard swirl model with better wall function close to the wall treatment. The model was validated with experimental data from the literature, therefore new correlations of the factors f and j were presented for the smooth rectangular fins.

Girgin [9] studied the effects of the configurations of the vortex generators of compact heat exchangers on the heat transfer via CFD. In his office; the common upward flow, the common downward flow and the orientations of the mixed vortex generators with different angles of attack were analyzed. Additionally, the number of vortex generator pairs has been increased from two to three. Ansys Fluent was used as a CFD tool. As a result, three pairs of common drainage configurations with 30  $^{\circ}$  and 45  $^{\circ}$  attack angles gave the best heat transfer performance.

Yaïci et al. [10] investigated the effects of incorrect flow distribution on heat exchanger performance. They examined fin and tube heat exchangers using CFDs. The distributions of the intake air flow and the geometric parameters were mathematically analyzed for various longitudinal, transverse and lamellar slopes. The Coburn factor j, the fan-shaped friction factor and the j / f factor were determined by the analyzes. Flow misalignment and geometric parameters have been reported to have a strong impact on thermal and hydraulic performance.

Ozden et al. [11] studied the effects of design parameters on the hull side on pressure drop and heat transfer by CFD. In his study, numerous CFD simulations were performed to predict the most appropriate turbulence model. The results were compared with the analysis results obtained with the Bell Delaware method. Two impact cutoff values and the relationship between impact distance and hull diameter with different flow rates were examined. The k-f turbulence model feasible with a fine mesh network and spatial discretization of the first order was found to be the best method compared to the results of the Bell-Delaware analysis.

Rehman et al. [12] examined an undisturbed tube bundle heat exchanger with CFD. The heat exchanger contained 19 smooth pipes and a 5.85 m long hull. A network independence study and a series of CFD analyzes were performed and the results were compared with the experimental results. Different turbulence models with different wall treatments were also examined and it was found that the turbulence model of the k-shear stress transport gave better results than the other models. Another conclusion from the CFD analysis was that 2/3 of the envelope's lateral flow

### International Journal of Research in Engineering and Applied Sciences (IJREAS) Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

bypassed the tubes and caused an ineffective heat transfer. As a result of this study, changes were made to the design of the heat exchanger to improve heat transfer.

### III. METHODOLOGY

In this study, the design process of a heat exchanger will be theoretically examined, therefore its performance will be analyzed and optimized using computer-assisted fluid dynamics. A countercurrent heat exchanger was considered for design purposes and its length was theoretically calculated using the LMTD method. In this study, the HE30 heat exchange model was initially designed according to a permeable study and the HE30 model was validated. Then change the cutting speed of the deflectors up to 40%. The temperature contours show the outlet and intake temperatures of the heat exchanger with different deflector cutting conditions. Theoretical and CFD results showed only a 1.15% difference in the cooling performance of hot fluids. The axial pressure drops showed positive correlations with the total heat transfer coefficient and the required pumping power. Overall, the results of this study confirm that CFD modeling can be promising for the design and optimization of heat exchangers and that it can test many design options without producing physical prototypes.

A. Experimentation Algorithm





Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

### B. Steps of Ansys Analysis

The different analysis steps involved in ANSYS are mentioned below.

1. Preprocessor

The model setup is basically done in preprocessor. The different steps in pre-processing are

- Build the model
- Define materials
- Generation of element mesh
- 2. Building The Model
- Creating a solid model within Catia



Fig. 2 CAD model prepared in Catia (base paper model)



Fig. 3 CAD model prepared in CATIA

+

### **International Journal of Research in Engineering and Applied Sciences (IJREAS)** Available online at http://euroasiapub.org

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

C. Mechanical Properties of fluid

Properties	Value
Ph Density (kg/m^3)	1672
PC Density (kg/m^3)	0.275
Thavg	353.15 K
Tcavg	302.8K
CPh	4197 J.kg-1 K-1
CPC	4197 J.kg-1 K-1
uh	0:355X103 Pa.s
uc	0.718X103 Pa.s

Table 1. Mechanical Properties of Fluid

- D. Boundary Condition
- 1. Define boundary conditions.

In the previous paper, HE30 case was given the best result.so first validated the HE30 case result by using HE30 boundary conditions. Given in the table below. In current study working on different baffle cut ratio. The cut ratio increase the heat transfer rate and also improve the performance of the heat exchanger



Fig. 4. Cad model import into ANSYS

Apply inlet boundary conditions for CFD analysis

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |









Apply inlet boundary conditions for CFD analysis







Fig. 8 Tube outlet

Available online at http://euroasiapub.org

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

HE 30 CAE boundary conditions for base pape.		
Boundary conditions		value
	V inlet shell	1.25m/s
	V inlet tube	0.25m/s
	Shell inlet temperature	353k
	Tube inlet temperature	302k

Table 2 boundary conditions

### E. Validations

Point for paper validations: -

- Build HE30 cad model according base paper.
- In HE 30 case:-
- 1. Number of baffles: -25
- 2. Baffle cut of rations:-30%
- Validated outlet temperature of the shell and tube. Shown in the table below.

	Shelll temperature (K)	Tube temperature (K)	Pressure (Kpa)
base paper	313	332.42	5
Validations Result	309	332	4.5
Temperature Contour 4 3.530e+002 3.468e+002 3.345e+002 3.283e+002 3.283e+002 3.283e+002 3.036e+002 3.036e+002 2.913e+002 2.913e+002			

Fig. 9 Temperature contours

Average outlet temperature of tubes:-332.42K

**International Journal of Research in Engineering and Applied Sciences (IJREAS)** Available online at <u>http://euroasiapub.org</u>

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |



Fig. 9 Tube outlet temperature contours

Average outlet temperature of Shell:-309K



Fig. 10 Shell outlet temperature contours



Fig. 11 Pressure contours

**International Journal of Research in Engineering and Applied Sciences (IJREAS)** Available online at <u>http://euroasiapub.org</u>

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

Vel	ocity itour 1	ANSYS 160
	1.547e+000	
H	1.392e+000	
H	1.238e+000	
	1.083e+000	
H	9.283e-001	
H	7.736e-001	
	6.189e-001	[10] S. LARDER, M. M. MARD, AND M. AND MARD, AND M. AND MARD, A
	4.642e-001	
	3.094e-001	
	1.547e-001	
-	0.000e+000	
[m :	s^-1]	

Fig. 12 Velocity contours

### F. Case-1

In case-1 number of baffles 25 and cut ratio was 20%.and average temperature of the shell outlet temperature was 315k.and tube temperature 327k.



Fig. 13 Temperature contours







Fig. 15 Velocity contours

### International Journal of Research in Engineering and Applied Sciences (IJREAS) Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

### G. Case-2

In case-2 In case-2 number of baffles 25 and cut ratio was 25%.and average temperature of the shell outlet temperature was 305k.and tube temperature 335k











Fig. 18 Pressure contours

### H. Case-3

In case-3 number of baffles 25 and cut ratio was 35%.and average temperature of the shell outlet temperature was 298k.and tube temperature 340k.

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

/elocity Contour 1	ANSYS
1.947e+000	
1.392e+000	
1.238e+000	
- 1.083e+000	
9.283e-001	
7.736e-001	
6.189e-001	
4.642e-001	
3.094e-001	
1.547e-001	
0.000e+000 m s^-1]	

### Fig. 19 Velocity contours



### Fig. 20 Pressure contours



Fig. 21 Temperature contours

### I. Case-4

In case-4 number of baffles 25 and cut ratio was 40%.and average temperature of the shell outlet temperature was 302k.and tube temperature 337k.

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |











Fig. 24 Outlet temperature of Shell

Available online at <u>http://euroasiapub.org</u> Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |







Fig. 26 Velocity Graph

IV. **Results** 

In case 1 to 3 shell temperature decreases and tube temperature increase due to change in heat transfer rate. The graph1 show the shell outlet temperature. And graph 2 show the tube outlet temperature.

	Shelll temperature (K)	Tube temperature (K)	Pressure (Kpa)
CASE-1	315	327	2
CASE-2	305	335	2
CASE-3	289	340	3
CASE-4	302	337	3
base paper	313	332.42	5
Validations Result	309	332	4.5

Available online at <u>http://euroasiapub.org</u>

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |





In case 1 to 3 shell pressure increase due to change in heat transfer rate. The graph 3 show the shell axial pressure.



International Journal of Research in Engineering & Applied Sciences Email:- editorijrim@gmail.com, <u>http://www.euroasiapub.org</u> An open access scholarly, online, peer-reviewed, interdisciplinary, monthly, and fully refereed journals

Available online at <u>http://euroasiapub.org</u>

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |



In case 1 to 3 shell velocity decreases due to change in number of baffles. The graph 5 show the shell outlet velocity. And graph 6 show tube outlet velocity.

	Shell outlet velocity(m/ s)	Tube outlet velocity(m/ s)
CASE-1	1.4	0.45
CASE-2	1.5	0.5
CASE-3	1.3	0.5
Case-4	1.25	0.35
base paper	1.3	0.46
Validations results	1.45	0.5



International Journal of Research in Engineering & Applied Sciences Email:- editorijrim@gmail.com, <u>http://www.euroasiapub.org</u> An open access scholarly, online, peer-reviewed, interdisciplinary, monthly, and fully refereed journals

Available online at <u>http://euroasiapub.org</u>

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |



#### V. CONCLUSION

Basic knowledge in thermodynamics, fluid dynamics, and CFD are obviously crucial for the design and optimization of a compact heat exchanger. In this work, three CFD models were developed and their accuracy was validated by a detailed theoretical calculation. Based on the results of the CFD, it has been recognized that careful selection of parameters such as the deflector cut ratio, the number of deflectors and pipes, the flow rate and the layout of the pipes is crucial to optimize performance. A tube bundle heat exchanger for a certain period of time. Of the three CFD models tested, the CASE-3 CFD model (with a diaphragm cutting speed of 35%, 16 tubes and 25 deflectors) gave the best results in completing the desired task, which is in good agreement with the theoretical results. A reduction in the cutoff frequency by impact increases the heat transfer coefficient on the side of the casing, but this also leads to an increase in the pressure drop. Obviously, the number of tubes used in a heat exchanger has an impact on the flow of the tube to maintain the required mass flow. The lower the number of pipes, the greater the pressure drop for a specific activity. It is even more important that the results of this work agree with the research results previously reported and this demonstrates the accuracy of the

International Journal of Research in Engineering & Applied Sciences Email:- editorijrim@gmail.com, <u>http://www.euroasiapub.org</u> An open access scholarly, online, peer-reviewed, interdisciplinary, monthly, and fully refereed journals

#### **International Journal of Research in Engineering and Applied Sciences (IJREAS)** Available online at http://euroasiapub.org

Vol. 9 Issue 4, April-2019, ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 |

results obtained. Overall, the results of this work confirm that CFD modeling is promising for the design and optimization of a heat exchanger.

### VI. **FUTURE SCOPE**

The total heat transfer rate of the heat exchanger is depend on the baffle design. For future study the design of the baffle also optimized by using ANSYS fluent. The changing in angle of baffle will also improve the total heat transfer rate and optimized the overall heat transfer rate.

### REFERENCES

- [1] Mohammed Irshad, Mohammed Kaushar "Design and CFD Analysis of Shell and Tube Heat Exchanger" IJESC, Vol. 7 Issue No.4, 2017.
- [2] Gurbir Singh, Hemant Kumar "Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger" Journal of Civil Engineering and Environmental Technology, Volume 1, Number 3; August, 2014 pp. 66-70.
- [3] DevvratVerma, Aanand Shukla "Design of Shell and Tube Type Heat Exchanger using CFD Tools" International Journal for Innovative Research in Science & Technology| Volume 4 | Issue 3 | August 2017.
- [4] NeerajkumarNagayach, Dr. AlkaBani Agrawal "Review Of Heat Transfer Augmentation In Circular And NonCircular Tube" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, Issue 5, September- October 2012, pp.796-802
- [5] Nasir Hayat, Muhammad MahmoodAslamBhutta "CFD applications in various heat exchangers design: A review" Applied Thermal Engineering, vol. 32, issue 1, pp. 1–12. January 2012
- [6] S. B. Hosseini, R. H. Khoshkhoo and M. Javadi, "Experimental and Numerical Investigation on Particle Deposition in a Compact Heat Exchanger," Applied Thermal Engineering, vol. 115, pp. 406-417, 2017.
- [7] S. Ismail and R. Velraj, "Studies on Fanning Friction (F) And Colburn (J) Factors of Offset And Wavy Fins Compact Plate Fin Heat Exchanger–A CFD Approach," Numerical Heat Transfer, vol. 56, pp. 987-1005, 2009.
- [8] R. B. S. Rao, G. Ranganath and C. Ranganayakulu, "Development of colburn 'j' factor and fanning friction factor 'f' correlations for compact heat exchanger plain fins by using CFD," Heat Mass Transfer, vol. 49, pp. 991-1000, 2013.
- [9] S. Girgin, "An Investigation of Heat Transfer Performance of Rectangular Channel by Using Vortex Generators," İstanbul Technical University, İstanbul, 2017.
- [10] W. Yaïci, M. Ghorab and E. Entchev, "3D CFD study of the effect of inlet air flow maldistribution on plate-fin-tube heat exchanger design and thermal– hydraulic performance," International Journal of Heat and Mass Transfer, no. 101, p. 527–541, 2016.
- [11] E. Özden and İ. Tarı, "Shell side CFD analysis of a small shell-and-tube heat exchanger," Energy Conversion and Management, vol. 51, pp. 1004-1014, 2010.
- [12] U. U. Rehman, "Heat Transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies," Chalmers University of Technology, Göteborg, 2011.