



SIMULATION BASED PERFORMANCE ANALYSIS OF HEAT EXCHANGERS: A REVIEW

Krishna dutta pandey^{1*}, Ankur Gupta¹, Anand kushwah¹, Mr.Gaurav Saxena²

¹PG scholars Department of Automobile Engineering, Rustamji Institute of Technology, BSF Academy, Tekanpur, Gwalior,(M.P.), India

²Asst.Professor Department of Automobile Engineering, Rustamji Institute of Technology, BSF Academy, Tekanpur, Gwalior, (M.P.), India

Abstract:-This review paper has an emphasis on the purpose of simulating software over the operation of heat exchangers and outcomes in different performing parameter. Due to quality of solution, it has been found that Simulation software's has a paramount position for the following areas of study in various types of heat exchangers such as unequal fluid flow, fouling, pressure drop ,turbulence of fluid ,temperature difference between two fluid ,thickness of material ,flow rate of fluid and thermal analysis in the design and optimization phase. Thus heat exchanger has becomes the core area to study about. The various design modifications which are implemented and studied through simulation software or compared with experimental and numerically modeled analysis are also discussed in the paper.

Keywords: Simulation, Heat Exchangers, experimental vs. simulation analysis, flow rate, pressure drop, fouling.

I. INTRODUCTION

Applications of energy in the form of heat are important for the oil and gas industries. At the same time transferring the heat from one place to another is important for efficient operation, in order to efficient working of such industries, heat exchangers has an imperative role in increasing process efficiency, reducing maintenance, conserving energy, and ensuring employee safety of many industries like oil refineries (double pipe, pipe, plate, shell and tube, aerial coolers type of heat exchangers) power plant, Gas industries or other applicable work place.

Heat exchanger is device which either heats the cold process steam using hot process fluid or they cool heat process steam by using cold process fluid. The term heat exchangers is to describe different type of equipment used in heat transfer .these equipment serves different functions like cooler, heater, condenser, vaporizer, and reboiler.

As coolers reduces the temperature of liquid or gas using water to remove heat, heaters increases the temperature of liquid or gas by adding heat using condensed steam and other heat sources. Condenser removes heat from gas by changing into liquid. Vaporizer add heat to liquid changing it a gas, reboilers provide heat to liquid in bottom of distillation tower.

1.1 THERMAL PERFORMANCE ANALYSIS OF HEAT EXCHANGER

For calculation of heat exchanger performance if only the inlet temperature are known, it is preferable to use the effectiveness no. of heat transfer (ϵ -NTU) method, which simplifies the algebra involved in predicating the performance of complex flow arrangement and compact heat exchangers the mechanism of heat transfer and pressure drop is fairly complex, and as a result, analytical derivation of ϵ -NTU relations is a difficult task [1]. The logarithmic mean temperature difference (also known as log mean temperature difference or simply by its initialism LMTD) is used to determine the temperature driving force for heat transfer in flow systems, most notably in heat exchangers[2]. These conventional methods considered for thermal analysis has a many drawbacks.

The LMTD is a steady-state concept, cannot be used in dynamic analysis In particular, if the LMTD were to be applied on a transient in whom, for a brief time, the temperature differential had different signs on the two sides of the exchanger, the argument to the logarithm function would be negative, which is not allowable [3]. To

indifferent these constraints, simulation is to be used for economical and assessable reasons or because simulation provides better solution accuracy and consistency. In the field of simulation analysis advancement are coming at a very fast rate, so I expect superior solver performance in the near future. Simulation brings unsurpassed multi-physics capability simply not found in these conventional methods.

Simulation enables every designer and engineer to carry out thermal analysis at any stage of design to ensure that every component and assembly performs properly within expected temperature ranges, and spot safety issues before they arise [4]

There is a wide variety of heat exchangers for diverse kinds of uses; hence the construction also would differ widely. However, in spite of the variety, most heat exchangers can be classified into some common types based on some fundamental design concepts. There are three basic types of heat exchangers: Direct transfer type (recuperative), Storage type (regenerative), direct contact type (shown in figure no.1)

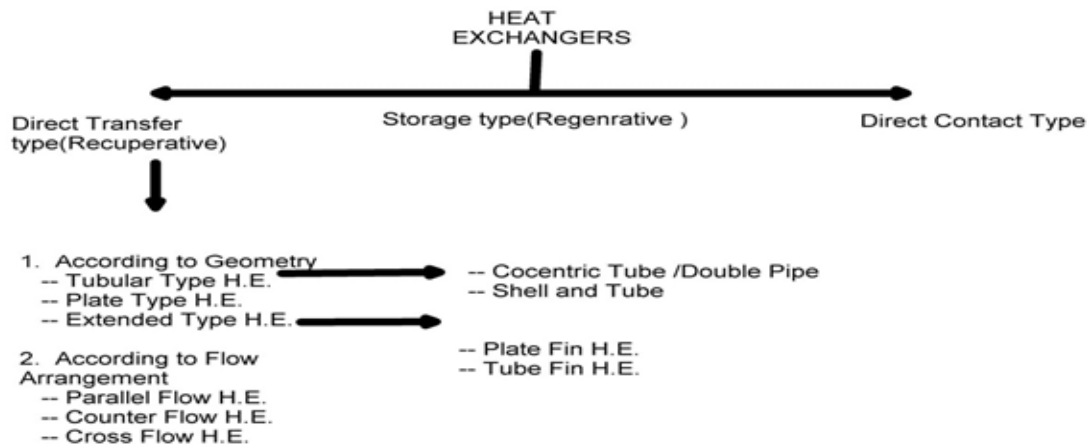


Figure 1.classification of heat exchangers

II. PREVIOUS STUDIES ON SIMULATION ANALYSIS OF HEAT EXCHANGERS

A simulation of heat exchangers is the operation of a model of the heat exchanger. The heat exchangers can be reconfigured and experimented with usually, this is impossible, too expensive or impractical to do in the system it represents. The operation of the heat exchangers can be studied, and hence, properties concerning the behavior of the actual heat exchanger or its subsystem can be inferred. In its broadest sense, simulation is a tool to evaluate the performance of a heat exchanger, existing or proposed, under different configurations of interest and over long periods of real time. Simulation is used before an existing system is altered or a new system built, to reduce the chances of failure to meet specifications, to eliminate unforeseen bottlenecks, to prevent under or over-utilization of resources, and to optimize system(heat exchangers) performance.[6]

Rehman Usman Ur [5] had deliberated design of unbaffled shell-and-tube heat exchanger with respect to heat transfer coefficient and pressure drop with the help of numerically modeling. The heat exchanger incorporates 19 tubes inside a 5.85m long and 108mm diameter shell. The flow and temperature fields are resolved using a commercial CFD package and it is performed for a single shell and tube bundle and is compared with the experimental results. **Khan Sarfaraz et al.** [6] selected CFD technique for the fluid flow mal-distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models such as standard, realizable and RNG, $k - \epsilon$, RSM, and SST $k - \epsilon$ with velocity-pressure coupling schemes such as SIMPLE, SIMPLER, PISO and etc. have been adopted to carry out the simulations .The simulations results ranging from 2% to 10% with the experimental studies. In some exceptional cases, it varies to 36%. **Ozden Ender et al.** [7] has examined the design of shell and tube heat exchanger by numerically modeling for the baffle spacing, baffle cut and shell diameter dependencies of heat transfer coefficient and pressure drop. The flow and temperature fields are examined by using a commercial CFD package for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow further best turbulent model among the one is selected to compare with the

CFD results of heat transfer coefficient, outlet temperature and pressure drop with the Bell-Delaware method result. By varying flow rate the effect of the baffle spacing to shell diameter ratio on the heat exchanger performance for two baffle cut value is investigated. Three turbulence models are taken for the first and second order discretization's to mesh density. It is observed that the CFD simulation results are very good with the Bell-Delaware methods and the differences between Bell-Delaware method and CFD simulations results of total heat transfer rate are below 2% for most of the cases. **Thirumarimurugan M. et al [8]** has modeled heat transfer study on a solvent and solution by using Shell and Tube Heat Exchanger. Steam is taken as the hot fluid and Water and acetic acid-Water miscible solution taken as cold fluid in it. A series of runs were made between steam and water, steam and Acetic acid solution. The flow rate of the cold fluid is maintained from 120 to 720 lph and the volume fraction of Acetic acid is varied from 10-50%. Experimental results such as exchanger effectiveness, overall heat transfer coefficients were calculated. MATLAB program was used to simulate a mathematical model for the outlet temperatures of both the Shell and Tube side fluids. From the comparisons it can be said that the mathematical model developed and simulated using MATLAB and compared with the experimental values for the system is very close. **Das D.H et. al [9]** has carried the performance of a shell and finned tube heat exchanger in the light of waste heat recovery application. Energy available in the exit stream of many energy conversion devices such as I.C engine gas turbine etc. goes as waste, if not utilized properly. The performance of the heat exchanger has been evaluated by using the CFD package fluent 6.3.16 and the available values are compared with experimental values. By considering different heat transfer fluids the performance of the above heat exchanger can also be predict. The performance parameters of heat exchanger such as effectiveness, overall heat transfer coefficient, energy extraction rate etc., have been taken in this work. **Tapre R. W. et al. [10]** has Reviewed on Heat Transfer in Spiral Heat Exchanger Energy saving is major matter in our global world, and heat exchanger is very useful for energy saving. Study on the various effects of feed flow rate and the coil diameter was done and concluded that on increasing the feed flow rate the pressure drop increases and vice versa. A relation between the pressure drop and the feed flow rate for the steady state Newtonian fluid into the Archimedean spiral tubes was developed. **Kumar R et al. [11]** has modeled 2D model using artificial neural network (ANN) concept and developed two models, deterministic and intelligent and then validated experimentally. Deterministic model accuracy was 75.3% whereas intelligent model accuracy was 72.6%. **Breesch H et al. [12]** formed a 3D simulation model TRNSYS/COMIS at Belgium, compared with experimental results and concluded that natural light ventilation was more effective than EAHE system for improving summer thermal comfort. Hybrid systems enhance the overall efficiency. **Khalajzadeh V et al. [13]** analyzed thermal performance of a hybrid system of ground heat exchanger (GHE) with indirect evaporative cooling during summer at Tehran, Iran. Hybrid system simulated on 3D CFD model and concluded that hybrid system was more efficient. **Bansal V et al. [14]** justified the combination of EAHE system with evaporative cooler by using 3D multiphase CFD modeling with FLUENT 6.3, at Ajmer in India. Combined cooling effect was 7609MJ in comparison to 4500MJ in EAHE system. **Bansal V et al. [15]** investigated a transient and implicit CFD based 3D model for economic study of combination of EAHE system with evaporative cooling and concluded that PBP with energy efficient blower was minimum 2years which was less than any other blowers. **Bansal Vet al. [16]** considered a new concept, de-rating factor for evaluating thermal performance of EAHE system through 3D transient CFD and experimental analysis. A new term de-rating factor was related to evaluate deterioration in thermal performance of EAHE under various operating conditions. It was observed that de-rating factor played key role in evaluating the performance of EAHE systems. It was concluded that de-rating factor mainly depended on thermal conductivity of soil, duration of continuous operation and length of pipe. **Bansal V et al. [17]** experimented on setup at Ajmer in India and investigated thermal performance regarding pipe length, soil thermal conductivity and time period of continuous operation and then validated with CFD and found experimental and simulated values had a minor difference of 3.4–8%. It was concluded that initial length of buried pipe and thermal conductivity of soil, play important role in the thermal performance of EAHE systems. Important observations of EAHE systems are presented in Table 5. It is observed from Table 5 that there is close agreement between experimental and simulated results. EAHE system with other renewable technology options i.e. evaporative cooler give higher efficiency and lower PBP. Most of the researchers opted 2D and 3D models for their studies but **Kujawa T et al. [18]** modeled a 1D computational model for estimating the temperature of geothermal water extracted to the earth's surface in addition to the temperature of water injected into a deposit level **Magraner T et al. [19]** analyzed a 2D model then compared the designed and actual energy performance of a HVAC ground coupled heat pump system. Predictions were performed with TRNSYS software tool and compared with experimental results. Predicted values were 15–20% higher. It was concluded that discrepancies between experimental results and simulation outputs were mainly due to heat pump efficiency degradation for being used at partial load. **Cocchi S et al. [20]** conferred a model to simulate an air conditioning system integrated with GSHP system. TRNSYS 17 software was used in order to refine the sizing. It was concluded that after simulation of system for the period of more than 5 years, the optimum

configuration would be 14 vertical heat exchangers in series with distance and length of 10 m and 100 m respectively. It could reduce 35% of total and operating cost. **Zheng Z et al. [21]** simulated finite element numerical simulation 3D model by virtue of MATLAB for analyzing the problem of underground vertical U-tube heat exchangers. It was obtained from simulation solution that larger center distance of the tube legs i.e. 100–200 mm was more apposite to ensure greater heat flow and reduce the probability of heat short-circuit occurrence. **Congedo P M et al. [22]** performed simulation for different configurations for horizontal type GSHP systems. Calculations were done with CFD code fluent for whole year for climatic conditions of the South of Italy. It was observed that important key factors were thermal conductivity of the ground around the heat exchanger, velocity of the heat transfer fluid inside the tubes. Depth of fitting of the horizontal GSHP systems did not play significant role and the helical heat exchanger arrangement performed best. **Zeng H et al. [23]** conferred quasi-3D model for vertical GSHP systems related to the fluid axial convective heat transfer and thermal “short-circuiting” among U-legs. It was found that double U-tubes in parallel offer enhanced thermal performance than those in series. Double U-tube boreholes reduced 30–90% borehole resistance than those of single U-tube. **Grijspeerdt K et al. [24]** employed CFD Code FINE-Turbo with Lomax turbulent stress model for the simulation of plate heat exchangers to determine the effect of fluid flow maldistribution. It was observed that recirculation bubbles are formed, which hinder steady flow **Li YZ et al. [25]** simulation CFD Code FLUENT has been used by researchers with SIMPLE-pressure velocity coupling scheme; standard $k-\epsilon$ turbulence model and finest mesh of 150,000 cells. They simulated the inlet configuration to optimize the header design in a plate-fin heat exchanger for fluid flow maldistribution. **Li YZ et al. [26]** further studied plate-fin heat exchangers using CFD code FLUENT with Semi Implicit SIMPLER pressure-velocity coupling scheme ,second-order up wind turbulence model with the finest mesh of 245,817 cells. They incorporated baffles with holes to uniformly distribute the fluid inside core. **Wasewar K L et al. [27]** simulated CFD Code FLUENT 6.1, segregated solver, $k-\epsilon$ turbulence model, GAMBIT v2.1 meshing software, structured mesh tri-tetra and a grid of 2,000,000 cells to simulate plate heat exchanger for fluid flow maldistribution. They found that maximum velocity of flow occurs at the center and it decreases sideways. Incorporating a large header height, they were able to decrease the maldistribution by 70%. Further, **Zhang [28]** employed CFD code FLUENT, Pressure-Velocity Coupling Scheme- SIMPLE, GAMBIT meshing scheme with coarse tetrahedral and hexahedral 160,837 cells to simulate plate-fin heat exchanger for fluid flow maldistribution. They studied three different channel pitches and found that channel pitch with large dimensions was more responsible for maldistribution. With the help of simulation analysis, researchers have approximated tendency of heat exchangers i.e. Shown in Summary of simulation technique and outcomes in Table no.1 below.

Table no. 1 Summary of simulation technique and outcomes.

S no.	Author	Simulation technique used	Significant Outcomes
1.	Khan Sarfaraz et al.[6]	CFD technique (SIMPLEC,PISO)	Simulation result Varies from 2-10% with experimental study.
2.	Ozden Ender et al. [7]	CFD pack. (compared with bell Delaware method)	Total heat transfer rate was below 2%.
3.	Thirumarimurugan M.et al [8]	MATLAB program(compared with experimental values)	Result obtained were very close to experimental values
4.	Das D.H et. al [9]	CFD package (Fluent 6.3.16)	Carried out the effect. Overall heat transfer coff.
5.	Kumar R et al. [11]	ANN(2Dmodel)	Deterministic model accuracy was 75.3% whereas intelligent model accuracy was 72.6%.
6.	Breesch H et al. [12]	3D simulation model TRNSYS/COMIS	Hybrid systems enhance the overall efficiency.

7.	Bansal V et al. [14]	FLUENT 6.3	Justified the combination of EAHE system with evaporative cooler
8.	Zheng Z et al. [21]	MATLAB	Simulation solution that larger center distance of the tube legs i.e. 100–200 mm was more apposite to ensure greater heat flow and reduce the probability of heat short-circuit occurrence.

III CONCLUSION

Traditional methods used for the design and development of Heat Exchangers are largely fatiguing and high in price today's era. Simulation has emerged as a cost effective alternative and highly optimum package. The simulation generally shows satisfactory results with comparison to the experimental modeling results ranging from 2% to 10% usually while in some exceptional cases vary up to 30% to 36% in case of large deviations[6]. It makes simulation essential part of all design processes because of prototyping elimination, which is expensive as well hectic. As per review simulation is essential and commendable and draws following remarks in favor of simulation:-

- Identify the imperative constraints that performance measures are most sensitive to modal and the inter-relationships among them.
- CFD fluent6.3.16 and 6.3 has shown exemplary results for comparative studies and researches[9,14]
- MATLAB simulation of experiential data is a convenient and most precise tool used by most of the researchers for analysis of data [8, 21].
- K-ε turbulence model has been most widely employed for heat exchanger design optimization [6].
- By eliminating error in model Develop well designed and powerful systems/modal and reduce system development time

REFERENCES

1. Navarro H.A & Cabezas – Gomez L.C. “Effective –NTU computation with a mathematical model for cross-flow heat exchangers ,Brazilian journal of chemical engineering ,Vol.24,no.4,ISSN 0104 -6632 , 2007 , pp 509-521
2. Module 8, Heat Exchangers, Lecture 35 <<http://nptel.ac.in/courses/103103032/module8/lec35/1.html> > (accessed on 24dec, 2016)
3. https://en.wikipedia.org/wiki/Logarithmic_mean_temperature_difference > (accessed on 24dec, 2016)
4. <http://www.solidworks.in/sw/products/simulation/thermal-analysis.htm> >2016 Dassault Systems Solid Works Corporation (accessed on 24 Dec, 2016)
5. Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011
6. Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan, CFD Applications in Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2012, pp. 1-12
7. Ender Ozden, Ilker Tari ,Shell side CFD analysis of a small shell-and-tube heat exchanger, Energy Conversion and Management, 51 (2010), pp. 1004–1014

8. M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (5): 2008, pp.548-552.
9. Apu Roy, D.H. Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, National Institute Of Technology, 2011.
10. R. W. Tapre, Dr.Jayant P. Kaware. "Review on Heat Transfer in Spiral Heat Exchanger", International Journal of Scientific and Research Publications, Volume 5, Issue 6, June 2015 pp. - 2250-3153
11. Kumar R, Kaushik SC, Garg SN. Heating and cooling potential of an earth-to air heat exchanger using artificial neural network. Renew Energy 2006; 31:pp.1139–55.
12. Breesch H, Bossaer A. Janssens. Passive cooling in a low-energy office building. Sol Energy 2005; 79(6):pp. 682–96.
13. Khalajzadeh V, Farmahini-Farahani M, Heidarinejad G. A novel integrated system of ground heat exchanger and indirect evaporative cooler. Energy Build 2012; 49:pp.604–10.
14. Bansal V, Mishra R, Agrawal GD, Mathur J. Performance analysis of integrated earth-air-tunnel-evaporative cooling system in hot and dry climate. Energy Build 2012; 47: pp. 525–32.
15. Bansal V, Mishra R, Agrawal GD, Mathur J. Performance evaluation and economic analysis of integrated earth-air-tunnel heat exchanger-evaporative cooling system. Energy Build 2012; 55:pp.102–8.
16. Bansal V, Misra R, Agrawal GD, Mathur J. Derating factor for evaluating thermal performance of earth tunnel air heat exchanger: a transient CFD analysis. Appl. Energy 2013; 102:pp.418–26.
17. Bansal V, Mishra R, Agrawal GD, Mathur J. Transient effect of soil thermal conductivity and duration of operation on performance of earth air tunnel heat exchanger. Appl. Energy 2013; 103:pp.1–11.
18. Kujawa T, Nowak W, Stachel A. Heat-flow characteristics of one-hole and two-hole systems for winning geothermal heat. Appl. Energy 2003; 74:pp.21–31.
19. Magraner T, Montero A, Quilis S, Urchueguia JF. Comparison between design and actual energy performance of a HVAC-ground coupled heat pump system in cooling and heating operation. Energy Build 2010; 42:pp.1394–401.
20. Cocchi S, Castellucci S, Tucci A. Modeling of an air conditioning system with geothermal heat pump for a residential building. Math Probl. Eng. 2013; 2013:pp.1–6.
21. Zheng Z, Wang W, Ji C. A study on the thermal performance of vertical U-tube ground heat exchanger. Energy Procedia 2011; 12: pp. 906–14.
22. Congedo PM, Colangelo G, Starace G. CFD simulations of horizontal ground heat exchangers: a comparison among different configurations. Appl. Thermal Eng. 2012; 33-34:pp. 24–32.
23. Zeng H, Diano N, Fang Z. Heat transfer analysis of boreholes in vertical ground heat exchangers. Int J Heat Mass Transfer 2003; 46:pp.4467–81.
24. Grijspeerdt K, Hazarika B, Vucinic D. Application of computational fluid dynamics to model the hydrodynamics of plate heat exchangers for milk processing. J Fluid Eng. 2003; 57(3):pp.237–42.
25. Zhang Z, Li YZ. CFD simulation on inlet configuration of plate-fin heat exchanger. Cryogenics 2003; 43(12):pp.673–8.
26. Wen J, Li YZ, Zhou A, Zhang K. An experimental and numerical investigation of flow patterns in the entrance of plate-fin heat exchanger. J Heat Mass Transfer 2006; 49(9–10): pp. 1667–78.
27. Wasewar KL, Hargunani S, Atluri P, Kumar N. CFD simulation of flow distribution in the header of plate-fin heat exchangers. Chem Eng. Techno 2007; 30(10):1340–6.
28. Zhang LZ. Flow maldistribution and thermal performance deterioration in a cross-flow air to air heat exchanger with plate-fin cores. Int J Heat Mass Transfer 2009; 52:pp.4500–9.