Review on CFD Analysis of Shell and Tube Type Heat Exchangers

Ankul Kumar[a,*], R.M. Sarviya[b]

[a]PG Scholar, Department of Mechanical Engineering, MANIT, Bhopal (M.P.), 462003, India
[b]Professor, Department of Mechanical Engineering, MANIT, Bhopal (M.P.), 462003, India

Abstract

The purpose of this paper is to study the heat transfer augmentation techniques for Shell and Tube type heat exchanger emphasizing on Computational Fluid Dynamics (CFD) applications. Different features of CFD have been developed and are being used in the design of these categories of heat exchanger that measures its several thermal parameters. Computer-aided-engineering (CAE) software named Ansys Fluent is one of the effective tools of CFD that is employed to accomplish the simulations work of several types of turbulence models of fluid flow within the heat exchangers including velocity-pressure coupling schemes. The accuracy and effectiveness of these simulations along with its realistic solutions are essentially within the acceptable range suggests that CFD is a viable technique to evaluate the performance of several heat exchangers. The present study will provide an aid to the researchers in the selection and design of reliable & novel heat exchangers with efficient thermal performance.

Keywords: CFD; Shell and Tube Heat Exchangers; Simulation; Ansys Fluent.

1. Introduction

In order to ponder regarding various processes or techniques for the enhancement of heat transfer inside the heat exchangers (HEs), we need to know about its fundamental understanding as well. Practically, HEs are one of the most basic but crucial components that is being utilized in every component where the process of heat transfer and fluid flow takes place directly or indirectly whether it be domestic fields, industrial applications, automotive sectors or research & development fields. HEs are steady-flow adiabatic equipments/systems in which two flowing fluids either gases or liquids exchange heat by the virtue of its temperature difference between them without transmitting any amount of heat to the surroundings. In this ongoing study, we'll look at one of the most proficient and multi-purpose heat exchangers i.e., Shell and Tube Heat Exchangers (STHEs). According to Master BI et al. [1], the system of HEs entails two streams of fluid exchanging heat through metal surfaces, one of which flows through tubular elements known as 'tube bundles,' which are further packed inside a cylindrical housing known as 'shells,' and the entire consolidated arrangement is referred to as STHEs. In industries where durability and maintainability are essential, these exchangers have always shown to be a universal alternative. For the purpose of effective designing HEs, the number of heat transfer units (NTU) approach and the Logarithmic Mean Temperature Difference (LMTD) method are usually used. However, these conventional approaches include some shortcomings, such as their tedious and repetitive aspect, inability to save money or time, and the requirement for a realistic model to complete the design process. With all of these obvious reasons in mind, computational fluid dynamics (CFD) proves to be a powerful and appropriate technique during the design phase of HEs due to the use of both affordable and powerful microprocessors. CFD is a type of computer-aided-engineering (CAE) technique that is used to observe fluid flow problems by finding solutions by mathematical equations with proper numerical analysis with the principle of converting the entire system into individual elements termed as grids or cells and implementing the required governing equations in order to find reasonably accurate solutions that involve specific thermal parameters such as temperature and pressure gradients, flow distributions & contours without the requirement of any type of pre experimentation involvement [2, 3].

In light of previous simulation work on STHEs, Q.W.Dong et al. [4] used the periodic unit duct model to forecast the pressure drop of a rod baffle heat exchanger, which is a type of STHE. They came to the conclusion that this model may be utilized to accurately simulate the shell-side characteristics of HEs with longitudinal shell-side fluid flow, resulting in increased dependability and accuracy. They also discovered that at given constant flow velocity, the pressure drop was inversely associated to the baffle pitch. Wang Y et al. [5] also stated that rod baffle type STHEs are the most effective

*Corresponding Author
Email address: ankul.rkaa@gmail.com (Ankul Kumar)

with the least amount of pressure drop when compared to other HEs such as huge round hole plate, plum blossom plate, tiny round hole plate, rectangular hole plate, and ting baffle plates as well. In line with this, D.Ravi et al. [6] proposed that increasing the surface area of the wall between the two fluids and lessening the resistance flow going over the exchangers are two of the most important elements in improving STHE productivity and effectiveness. Furthermore, heat transfer enhancement across HEs has always been a complex issue for investigators all over the world to achieve the best results while investing in the same amount of effort and spending the least amount of time and money.

2. Several Domains of CFD Applications

The review of several features of STHEs, including the turbulence type model, maldistribution of fluid flow, impacts of fouling, criterion of pressure drop and its thermal analysis has been investigated with the proper selection of all simulation codes. CFD 2000, PHOENIX, ADINA, FIDAP, FLUENT, CFX, STAR CD, and others are some of the effective commercial CFD codes that are being used in different types of numerical simulations of STHEs for the prediction of several dependent thermal parameters and to analyze the heat transfer rate, according to Ozden E et al. [7].

2.1. Turbulence type model

Turbulence is caused by internal disturbances & instabilities in the fluid flow. Turbulence flows include a diverse range of length, time scales including velocity, and resolving all of them significantly enhances simulation cost. As a result, numerous turbulence models with distinct degrees of resolution have been established. Large Eddy Simulation (LES) and Reynolds Average Navier Stokes (RANS) are two of them. RANS models based on flow characteristics include the standard k-ε model, k-ε RNG model, Realizable k-ε, k-ω, and RSM (Reynolds Stress Model) [8]. As there is no universal criterion for defining a turbulence model, it is advisable to explore with a variety of turbulence models for relevant simulations. Ender Ozden et al. [7] used the Spalart-Allmaras [9] and two alternative k-ε turbulence models, Standard and Realizable, in their research. Only one turbulence equation is computed in the Spalart-Allmaras turbulence model, making it a very cost-effective model in Ansys fluent.

2.2. Maldistribution of fluid flow

According to Grijspeerdt K et al. [10], the non-uniform behavior of fluid flow causes STHEs/HEs to operate poorly and in an undesirable manner. This could be due to poor inlet or outlet port design, plate corrugations, header layout, or distributor assembly. Kim MI et al. [11] studied the characteristics of the flow pattern of STHE using different header types and discovered that the A-type design, which has the lowest flow rate and the smallest header dimension of length, is unsatisfactory for uniform flow distribution along the lengthwise cross section, whereas the C-type design yields the best output. Furthermore, flow uniformity increases with increased header length and decreases with a steady decrease in fluid flow rate. If the positioning and shape of the inlet nozzle are modified, the pattern of fluid flow near the header section that we get as an output after specific simulation work can be changed as well. If the number of baffles employed is low, it's probable that some recirculation sections will occur on the back side. With a larger number of baffles, the heat transfer rate improves too [7].

2.3. Impacts of fouling

By-products are formed when working fluids either gases or liquids disintegrate due to excess temperatures or interactions with the walls of STHEs, accumulating on the walls and lowering the heat transfer rate with a lower overall heat transfer coefficient. This undesired process is called as 'fouling' in HEs is a common occurrence in the food processing industry, and it demands regular maintenance in order to keep the HEs free of this unpleasant occurrence.

Moreover, Emani S et al. [12] explained that asphaltenes are the main cause of crude oil type fouling in STHEs and used a CFD approach to investigate the effect of shear stress as well as surface roughness on the shell side of that asphaltenes in STHEs. To determine the effect of various arising forces on deposition, a Lagrangian-based discrete-phase model was used, and it was discovered that if the shear stress applied to the wall is increased until pressure equals 0.04 Pa and velocity equals about 1 m/s, the mass of the asphaltene particles that are accumulating is substantially reduced.

Bayat M et al. [13] used the CFD technique to assess the rate of fouling in a crude oil pre heater in an industry in the year of 2012. During this procedure, they considered that the crude oil contained three components: salt, asphaltene, and petroleum into it. The 'species transport modal' was used to simulate the mixing and transportation of certain chemical species. They also observed that the coefficient of molecular diffusion is critical for accurately simulating the behavior of crude oil employed in industrial STHEs. Binary diffusion coefficients were divided into five categories for this purpose. Finally, they reached to the conclusion that CFD is sufficient for predicting fouling rate in both the beginning and developing stages of fouling.

Employing the finite volume approach, Wang Y et al. [14] investigated the various effects that fouling phenomena could have on the shell side of STHEs in his continuing research work. They noticed that fluid 'vortex flow' and 'velocity gradient' are two key aspects that activate fluid flow characteristics and, as a result, lead to particulate type fouling. The fouling effect will increase as the effects of the two causes outlined above increase. One of these places is Shell Inlet. When it comes to crystallization fouling, temperature is the primary factor that causes even fouling formation across the HE tube surface.

Using general purpose CFD code ANSYS CFX, Oclon P et al. [15] computationally examined the effects of fouling on the inner side of the tube surface to forecast the flow of flue gases inside a narrow passage made up of six fins within a cross-flow fin-and-tube type HE. They observed that the fouling on the surface was approximately 1 mm deep. Furthermore, as fouling accumulated on the fin's wall and its connected tube, the temperature of the fin and its connected tube increased, slowing down the heat transfer process. After doing all of the necessary calculations, they came to the conclusion that once the deposition of silicate scale across the tube begins, the flue gas temperature at the outlet region might climb by 122 K to 163 K, causing the tube to fracture.

In 2013, Movassag SZ et al. [16] highlighted that the segmental baffles kind of STHE design is not as effective as it should be since it promotes dead zones locations where fouling occurs, as well as lower the HE thermal efficiency & performance. The major priority of this research was to lower the rate of fouling and manage the pressure drop. After examining all forms of possible STHE configurations, they concluded that helical type baffles function significantly better than conventional segmental baffles, resulting in lower fouling probability.

The constant action of fouling over the surfaces decreased the thermodynamic as well as hydrodynamic efficiency of crude oil exchangers, according to Yang J [17]. To gain a better knowledge of the induction time of fouling, he introduced a framework that uses a CFD method and includes numerous operating circumstances to monitor the influence of two fouling mechanisms, chemical and precipitation fouling, as well as anticipate the deposition rate [18–21].

2.4. Criterion of pressure drop

Heat transfer augmentation in STHEs is always coupled with a certain amount of pressure drop/loss inside the system, which should be within a reasonable range for improved thermal efficiency along with its performance. The pressure drop across the HEs is primarily affected by the design of the core or matrix and fluid flow distribution components including headers, ports, ducts, nozzles, and others [22].

Wang Q et al. [23] used a CFD approach to evaluate STHEs with segmental baffles to a combined multiple shell-pass shell-and-tube heat exchanger (CMSP-SHTX) with continuous helical baffles in the outer shell to maximize heat transfer performance. During their investigation, they found that CMSP-SHTX has a 13% lower pressure drop than SG-SHTX for the same uniform mass flow rate and heat transfer rate.

Mohammadi K et al. [24] used FLUENT CFD codes to analyze the influence of baffle cut with its orientation, as well as the viscosity of the working fluid, on the heat transfer performance and overall pressure drop of STHEs with 660 tubes, according to Tubular Exchanger Manufactures Association (TEMA) standards in both laminar and turbulent regions, while ignoring leakage flow. They demonstrated that baffle orientation has a substantial impact on shell-side pressure drop, with horizontal baffle orientation outperforming vertical baffle orientation in the inlet and exit zones of STHEs. The vertical baffle orientation, on the other hand, can be used in the intermediate baffle spacing zones.

In the context of the research, Mohammadi K et al. [25] conducted additional simulation work in 2009, this time including leakage flow, using 76 plane tubes standardized by TEMA. Additionally, they evaluated the impact of viscosity with a Prandtl number (Pr) ranging from 0.7 to 206 for two distinct baffle orientations with three different fluids: water, air, and engine oil. A horizontal baffle exchanger has a 20% higher heat transfer coefficient and a 250% higher pressure drop than a vertical baffle exchanger, according to CFD data.

However, Shahril SM et al. [26] reported that the average percentage increase in overall heat transfer rate per overall pressure drop of a shell-and-double concentric tube heat exchanger (SDCTHEX) with an inner tube diameter of 8/12 mm/mm is 343% higher than that of traditional STHEs. They showed that the SDCTHEX has excellent heat transfer effectiveness with minimal pressure drop, which is also desirable, during the simulation study.

2.5. Thermal analysis

Since it is well known, the main intention of HE is to promote heat transmission between fluids, resulting in lower input energy requirements and increased process efficiency in terms of both production and economics. In order to review the thermal properties of HEs from the perspectives of 'impact of physical features on heat transfer rate' and 'thermal coefficients,' researchers conducted CFD analysis and simulation work.

A fundamental method for assessing an exchanger's performance is to analyze thermal parameters such as Nusselt number (Nu), Dean's number (De), Pr, Friction Factor(f), and Colburn Factor and their contribution to the overall heat transfer coefficient of HEs [27, 28].

Jayakumar JS et al. [28] employed helical pipe followed by a STHE in their 2007 investigation. Nu and De were used to represent thermal characteristics, with the presumption that they were solely temperature dependent. The helical pipe was first put through three tests to determine uniform features at ambient and fluid average temperatures, as well as other temperature-dependent properties. The MATLAB mathematical tool was used to obtain equations through a regression operation for modelling. They noted that constant parameters such as specific heat, thermal conductivity, density, and others change with temperature, therefore assuming they are constant throughout a greater temperature range produces correct and validated results. Heat transfer rate was improved with increased heat transfer coefficient as De was increased, with temperature dependent properties generating the nearest results and having adequate agreement with experimental outcomes.

Since its beginning, CFD has been an outstanding tool for examining thermal properties in the context of STHE design optimization. It has been used to investigate different modifications, compare results, and present the most possible and acceptable combination of variables in order to achieve the best results [29, 30].

In order to compare and obtain better knowledge of two different forms of HEs, Wang Q et al. [23] used CFD simulation with many related turbulence models. According to a comparison of the data acquired from simulation of the two distinct models, the overall heat transfer rate of CMSP-STHX is 5.6% greater than the SG-STHX under similar mass and heat transfer rates with the same pressure drop in the shell side.

The impact of baffle spacing, cut, and shell diameter on heat transmission and pressure drop was explored using a CFD simulation in a STHE by Ozden E et al. [7]. The performance of various baffles and turbulence models was investigated as well. The Bell Delaware method and the Kern method were compared in order to determine the best turbulence model [3], and it was discovered that the Kern method consistently lowers the heat transfer coefficient, whereas the Bell Delaware [31] method lowers the heat transfer rate of variable spaced baffles, demonstrating strong agreement with the heat transfer characteristics of equally spaced baffles, and that a 25% reduction in baffle cut enhances the rate of heat transfer across it.

Raj VA et al. [32] investigated a STHE based on the PCM Module for cooling. The exchanger was built in a modular format, with multiple modules stacked on top of one another with air spacers between each pair. For a single type module and two successive air spacers, transient and steady state CFD simulations were used to examine the pressure drop and temperature fluctuations of the working fluid. The obtained results were critical in identifying the PCM's solidification properties and confirming the suitability of the geometrical arrangements chosen. Air spacers were shown to improve heat transfer rate significantly, lowering solidification time by 50-75 percent, with a stronger effect at lower air stream speeds [33–38].

3. Conclusions

The ongoing study is a thorough review of shell and tube heat exchangers (STHEs), including all aspects of their thermal performance, suitable design selection criteria, and other influencing thermal characteristics using a CFD technique. As traditional methods for designing and constructing heat exchangers are primarily complex and expensive, CFD is unquestionably a cost-effective strategy that provides a swift response and avoids the need for a physical or actual prototype. For general-purpose CFD analysis of a variety of heat exchangers, readily accessible commercial CFD software can suffice. These are versatile enough to handle any form of analysis assignment, from the most basic phase of fluid flow behavior prediction to full HE designs using a number of turbulence models. The simulations we run produce results that are in good accord with the experimental studies. The sizes of the tubes and shells, the number of tubes, the pitch, and the baffle angles of STHEs are all key determinants that must be examined by researchers for future study.

The following are some of the key findings from this detailed review and analysis study that may be useful for further research in this field:

- The application of a computational fluid dynamics (CFD) technique for the entire analysis of any sort of HE, including STHEs, is considerably superior to the use of expensive and time-consuming experiments to improve the overall thermal behavior of exchangers used in industries or laboratories.

- The general purpose commercial CFD codes should be used depending on the required result so that no time or money is wasted.

- When compared to all other models and traditional methods (Kern method) for the suitable design of HEs, the Bell-Dellware turbulence model is one of the best simulation approaches of CFD approach. Also, because some recirculation sections are constructed with a small number of baffles across the tubes, it is advised that in order to improve the thermal properties of the HEs/STHEs, a larger number of baffles with suitable orientation is desired.

- For improved performance of the related equipment of exchangers, uniform behaviour of thermal fluid flow distribution pattern is required. The proper placement and shape of STHEs inlet nozzles should also be taken into account for a homogeneous flow distribution.

- The CFD model with 'slice contours' and 'particle tracking' features that is being used to predict the fouling rate of deposition across the STHEs is quite advantageous in terms of protecting the shell side or tube side from any damage or fractures caused by excessive temperature increment is quite advantageous.

- Helical type organized baffles are preferred over others because they produce a smoother fluid flow with a lower pressure drop and a lesser possibility of fouling, as well as a higher heat transfer rate.

- The pressure drop and performance of STHEs are influenced by the distinct orientations (vertical/horizontal) of baffles spacing, which may be easily analyzed and compared using CFD models.

4. Conflicts of Interest

The authors state that they have no known competing financial or personal interests that would have influenced the findings of this study.

References

[1] B. I. Master, K. S. Chunangad, A. Boxma, D. Kral, P. Stehlik, Most frequently used heat exchangers from pioneering research to worldwide applications, Heat Transfer Engineering 27 (6) (2006) 4–11.

[2] V. Kumar, S. Saini, M. Sharma, K. Nigam, Pressure drop and heat transfer study in tube-in-tube helical heat exchanger, Chemical Engineering Science 61 (13) (2006) 4403–4416.

[3] D. Q. Kern, Process heat transfer, Tech. rep. (1950).

[4] Q. Dong, Y. Wang, M. Liu, Numerical and experimental investigation of shellside characteristics for rodbaffle heat exchanger, Applied thermal engineering 28 (7) (2008) 651–660.

[5] Y. Wang, Q. Dong, M. Liu, Characteristics of fluid flow and heat transfer in shellside of heat exchangers with longitudinal flow of shellside fluid with different supporting structures, in: Challenges of Power Engineering and Environment, Springer, 2007, pp. 474–479.

[6] D. Ravi, P. P. RK, A study on analysis of cfd analysis of heat transfer enhancement in shell by using passive technique, Journal of Xi'an University of Architecture & Technology (2021) 404–418.

[7] E. Ozden, I. Tari, Shell side cfd analysis of a small shell-and-tube heat exchanger, Energy Conversion and Management 51 (5) (2010) 1004–1014.

[8] D. S. Patel, R. R. Parmar, V. M. Prajapati, Cfd analysis of shell and tube heat exchangers-a review, International Research Journal of Engineering and Technology 17.

[9] S. R. Allmaras, F. T. Johnson, Modifications and clarifications for the implementation of the spalart-allmaras turbulence model, in: Seventh international conference on computational fluid dynamics (ICCFD7), Vol. 1902, Big Island, HI, 2012.

[10] K. Grijspeerdt, B. Hazarika, D. Vucinic, Application of computational fluid dynamics to model the hydrodynamics of plate heat exchangers for milk processing, Journal of food engineering 57 (3) (2003) 237–242.

[11] M. I. Kim, Y. Lee, B. W. Kim, D. H. Lee, W. S. Song, Cfd modeling of shell-and-tube heat exchanger header for uniform distribution among tubes, Korean Journal of Chemical Engineering 26 (2) (2009) 359–363.

[12] S. Emani, M. Ramasamy, K. Z. K. Shaari, Cfd modelling of shell-side asphaltenes deposition in a shell and tube heat exchanger, in: AIP Conference Proceedings, Vol. 1859, AIP Publishing LLC, 2017, p. 020118.

[13] M. Bayat, J. Aminian, M. Bazmi, S. Shahhosseini, K. Sharifi, Cfd modeling of fouling in crude oil pre-heaters, Energy Conversion and Management 64 (2012) 344–350.

[14] Y. Wang, Q. Lu, Y. Zheng, B. Yuan, H. Tao, A cfd-based analysis on trends of heat exchanger fouling, in: 2012 Asia-Pacific Power and Energy Engineering Conference, IEEE, 2012, pp. 1–4.

[15] P. Ocłoń, S. Łopata, M. Nowak, A. C. Benim, Numerical study on the effect of inner tube fouling on the thermal performance of high-temperature fin-and-tube heat exchanger, Progress in Computational Fluid Dynamics, An International Journal 15 (5) (2015) 290–306.

[16] S. Z. Movassag, F. N. Taher, K. Razmi, R. T. Azar, Tube bundle replacement for segmental and helical shell and tube heat exchangers: Performance comparison and fouling investigation on the shell side, Applied Thermal Engineering 51 (1-2) (2013) 1162–1169.

[17] J. Yang, Computational fluid dynamics studies on the induction period of crude oil fouling in a heat exchanger tube, International Journal of Heat and Mass Transfer 159 (2020) 120129.

[18] Z. Tajudina, J. A. Martinez-Minuesaa, E. Diaz-Bejaranoa, I. Valkova, P. Orzlowskia, F. Colettib, S. Macchietto, G. F. Hewitta, Experiment analysis and baseline hydraulic characterisation of hipor, a high pressure crude oil fouling rig, CHEMICAL ENGINEERING 43.

[19] C. Shen, Y. Wang, Z. Tang, Y. Yao, Y. Huang, X. Wang, Experimental study on the interaction between particulate fouling and precipitation fouling in the fouling process on heat transfer tubes, International Journal of Heat and Mass Transfer 138 (2019) 1238–1250.

[20] G. McGarvey, C. Turner, Synergism models for heat exchanger fouling mitigation (1999).

[21] A. Watkinson, D. Wilson, Chemical reaction fouling: A review, Experimental Thermal and Fluid Science 14 (4) (1997) 361–374.

[22] R. K. Shah, D. P. Sekulic, Fundamentals of heat exchanger design, John Wiley & Sons, 2003.

[23] Q. Wang, Q. Chen, G. Chen, M. Zeng, Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles, International Journal of Heat and Mass Transfer 52 (5-6) (2009) 1214–1222.

[24] W. Heidemann, K. Mohammadi, H. Mueller-Steinhagen, Numerical investigation of the effect of baffle orientation and baffle cut on heat transfer and pressure drop of a shell and tube heat exchanger, in: International Heat Transfer Conference 13, Begel House Inc., 2006.

[25] K. Mohammadi, W. Heidemann, H. Müller-Steinhagen, Numerical investigation of the effect of baffle orientation on heat transfer and pressure drop in a shell and tube heat exchanger with leakage flows, Heat transfer engineering 30 (14) (2009) 1123–1135.

[26] S. Shahril, G. Quadir, N. Amin, I. A. Badruddin, Thermo hydraulic performance analysis of a shell-and-double concentric tube heat exchanger using cfd, International Journal of Heat and Mass Transfer 105 (2017) 781–798.

[27] A. G. Kanaris, A. A. Mouza, S. V. Paras, Flow and heat transfer prediction in a corrugated plate heat exchanger using a cfd code, Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology 29 (8) (2006) 923–930.

[28] J. Jayakumar, S. Mahajani, J. Mandal, P. Vijayan, R. Bhoi, Experimental and cfd estimation of heat transfer in helically coiled heat exchangers, Chemical engineering research and design 86 (3) (2008) 221–232.

[29] S. Knudsen, G. Morrison, M. Behnia, S. Furbo, Analysis of the flow structure and heat transfer in a vertical mantle heat exchanger, Solar energy 78 (2) (2005) 281–289.

[30] K. Foli, T. Okabe, M. Olhofer, Y. Jin, B. Sendhoff, Optimization of micro heat exchanger: Cfd, analytical approach and multi-objective evolutionary algorithms, International Journal of Heat and Mass Transfer 49 (5-6) (2006) 1090–1099.

[31] K. Bell, S. Kakac, A. Bergles, F. Mayinger, Delaware method for shell side design, heat exchanger thermal-hydraulic fundamentals and design, Kakac, AE Bergles, and F.

[32] V. A. A. Raj, R. Velraj, Heat transfer and pressure drop studies on a pcm-heat exchanger module for free cooling applications, International Journal of Thermal Sciences 50 (8) (2011) 1573–1582.

[33] P. Lamberg, R. Lehtiniemi, A.-M. Henell, Numerical and experimental investigation of melting and freezing processes in phase change material storage, International Journal of Thermal Sciences 43 (3) (2004) 277–287.

[34] D. Beasley, C. Ramanarayanan, H. Torab, Thermal response of a packed bed of spheres containing a phase-change material, International journal of energy research 13 (3) (1989) 253–265.

[35] M. Rajagopal, R. Dinesh Babu, V. Antony Aroul Raj, R. Velraj, Investigation on phase change material-based flat plate heat exchanger modules for free cooling applications in energy-efficient buildings, Advances in Building Energy Research 11 (2) (2017) 282–304.

[36] T. Muthuvelan, K. Panchabikesan, R. Munisamy, K. M. Nibhanupudi, V. Ramalingam, Experimental investigation of free cooling using phase change material-filled air heat exchanger for energy efficiency in buildings, Advances in Building Energy Research 12 (2) (2018) 139–149.

[37] A. Ousegui, B. Marcos, M. Havet, Inverse method to estimate air flow rate during free cooling using pcm-air heat exchanger, Applied Thermal Engineering 146 (2019) 432–439.

[38] S. Nada, W. Alshaer, R. Saleh, Experimental investigation of pcm transient performance in free cooling of the fresh air of air conditioning systems, Journal of Building Engineering 29 (2020) 101153.