Archives of Computational Methods in Engineering (2023) 30:1157–1165
https://doi.org/10.1007/s11831-022-09839-z

REVIEW ARTICLE

A Review: CFD Approaches of Plate Heat Exchangers

Ahmad Aboul Khail[1] · **Ali Erişen**[2]

Received: 29 June 2021 / Accepted: 15 October 2022 / Published online: 11 November 2022

Abstract

In this paper, a review of some numerical researches and studies of plate heat exchangers PHEs was carried out according to the methods of computational fluid dynamics CFD. The review includes the most prominent operations of preparing the numerical study such as the calculation domain, methods of forming the computational grid and how to test its independence, the applied boundary conditions, turbulence models used in modelling the flow, the methods used for the solution, and how to validate of the results of the numerical study. New methods that contribute to improving mesh quality have also been reviewed. Recommendations were also made on conducting numerical studies of PHEs.

Abbreviations

CFD	Computational fluid dynamics
PHE	Plate heat exchanger
PHEs	Plate heat exchangers
b	Corrugation depth m
C_f	Friction factor
D_e	Equivalent diameter m
D_h	Hydraulic diameter m
f	Coefficient of friction
j	The average Colburn j-factor
K	Factor equal to (f × Re)
t	Thickness of plate m
P_c	Corrugation pitch m
U_0	The velocity of fluid
u_z	The friction speed m/s
u^+	The ratio of fluid velocity to velocity of friction
y^+	A non-dimensional parameter used to determine the distance of the cell center adjacent to the wall from the wall itself.
ϕ	Surface enlargement factor
Nu	Nusselt Number
Re	Reynolds Number
τ_z	The wall shear stress N/m^2
ρ	Density kg/m^3
ν	Kinematic viscosity m^2/s
SST	Shear stress transport
LMTD	Logarithmic mean temperature difference
ε	Heat exchanger effectiveness
NTU	Number of transfer units

1 Introduction

Heat exchangers are important tools for heat transfer in industrial plants, refrigeration systems, and household appliances. Plate heat exchangers (PHEs) are one of the most important types of heat exchangers, as these heat exchangers are distinguished by their high performance, high compactness, as well as easy maintenance and cleaning. Because of these advantages, PHEs have entered into various industrial, aerospace, and energy production fields. PHEs are also of great importance in cooling aerospace and air engines due to their light weight and compact size advantage. Due to the difficulty in analyzing and studying this type of complex-geometry exchanger analytically, researchers often prefer numerical study, especially with the great development in the field of computers and their increasing potential. It relies on computational fluid dynamics in this matter, which has proven reliable in analyzing different flow types according to their different conditions and complex geometries. Numerical studies usually begin with determining the calculation domain upon which the numerical study will be based. Then the

---

Ahmed Sadik is the second name of the author Ahmad Aboul Khail due to his dual nationality

✉ Ahmad Aboul Khail
khailnuc@gmail.com; ahmed.sadik@hku.edu.tr

Ali Erişen
ali.erisen@ostimteknik.edu.tr

[1] Department of Mechanical Engineering, Hasan Kalyoncu University, 27010 Gaziantep, Turkey

[2] OSTİM Technical University, OSTİM, Yenimahalle, Ankara, Turkey

calculation domain is cut into very small parts to form what is called the computational grid (Mesh) according to which the equations of continuity, momentum, and energy are solved simultaneously according to certain methods and assumptions. In this review, we will summarize the most important steps of the workflow of several numerical studies included in the literature related to the PHE without talking about the results of these studies. The reviewed articles have been selected according to the following criteria: (1) The temporal development of the capabilities of computers was taken into account, and accordingly, chronologically graded articles were selected. (2) Choosing articles belonging to a different publishing house, such as Elsevier and Springer, in addition to important articles published in conferences. (3) Diversity in the topics of articles specialized in the study of plate heat exchangers with chevron plates, the most widespread in the industry. (4) Selecting articles that have studied geometrical modifications on the traditional chevron plates. (5) Selecting articles that have suggested new geometries for plate heat exchangers. Briefly, the review manuscript has been enriched from multiple and varied sources with the aim of reviewing as many as possible numerical study procedures for plate heat exchangers in their various and diverse forms. The aim of this review is to shed light on the special procedures followed in numerical studies of this type of exchangers.

Figure 1 shows the workflow of the numerical study mechanism according to the computational fluid dynamics. In this article, the processes performed in the Pre-processing section of some studies mentioned in the literature will be reviewed.

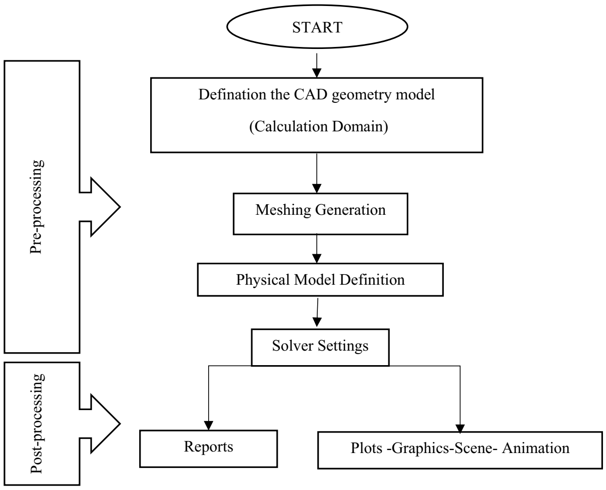


Fig. 1 CFD workflow

2 Calculation Domain

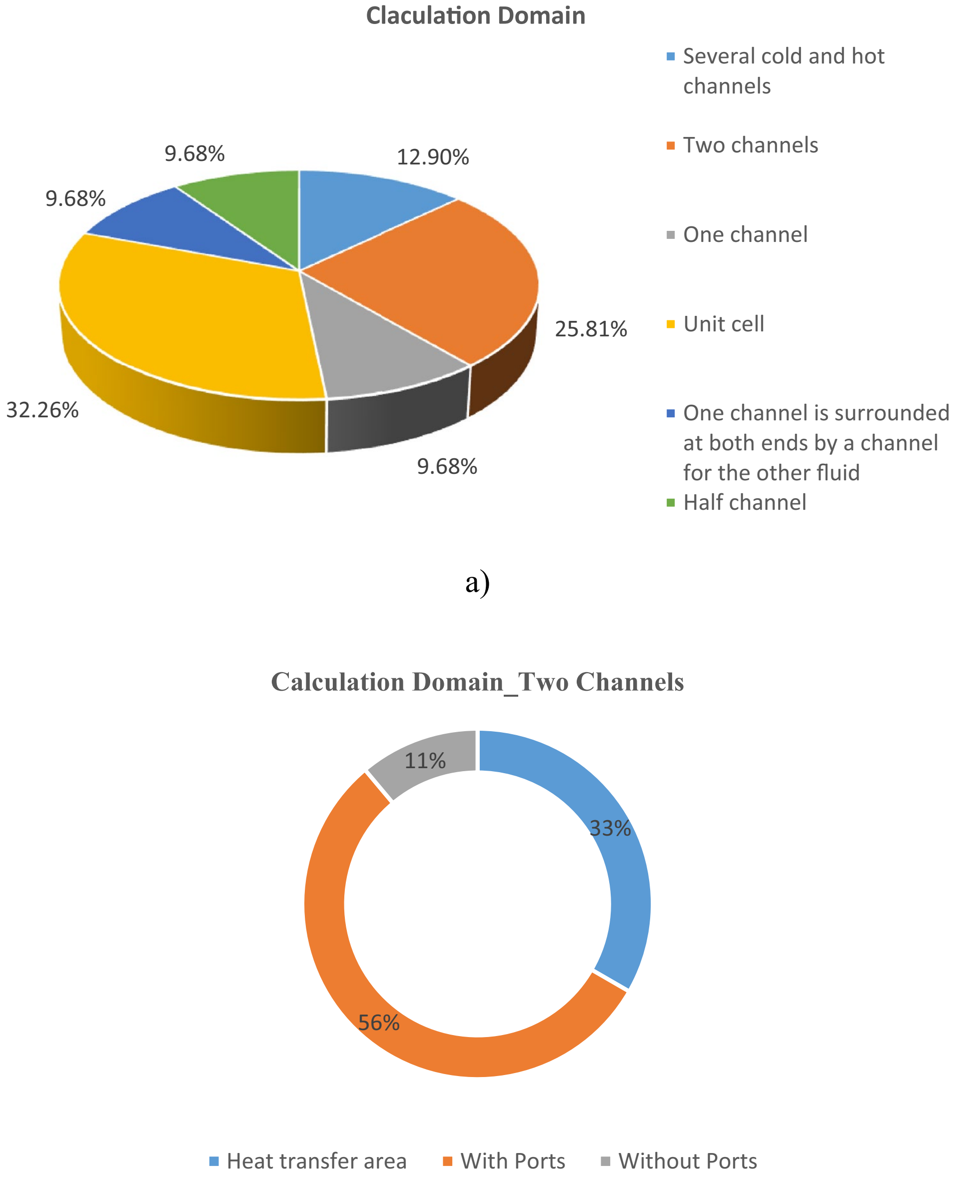
The calculation domain is considered the cornerstone of numerical study and its choice plays a major role in determining the progress of work in subsequent steps. Where some researchers chose several cold and hot channels as a calculation domain for numerical study [1–4], while some chose only two channels (One contains hot fluid and the other cold fluid) [5–11] ([9, 10] only heat transfer area) ([12] without ports). Some researchers chose only one channel containing two plates and a running fluid between them over the entire length of the channel, including the entrance and exit of the fluid [13–15]. Others [16], based on the periodic geometrical properties of the exchanger plate, have chosen a small periodic unite cell that has two hot and cold channels that have been identified as the calculation domain for numerical study. While Carla and others chose seven consecutive unitary cells to achieve his numerical study [17, 18]. Others, on the other hand, have chosen periodic unite cell that contain two plates and a running fluid between them, i.e. one channel [19–21].

To more closely represent the physical reality within the PHE, some researchers have chosen as a calculation domain for numerical study an entire channel (two plates and a fluid between them), and two halves of the other fluid channel surrounded the entire channel. In this case also, some of them chose the entire plate of the exchanger with the inlets, outlets of fluids [22], and others chose a periodic unite cell [23]. Dora [24] adopted for his numerical study a calculation domain consisting of four plates of type H-theta, forming a hot channel in the middle and two cold channels at the ends.

To improve the performance of coolants in aero-engines, Doo, in his numerical study of flow in a new plate exchanger, used a calculation domain consisting of a unit cell compound of two plates and fluid between them [25, 26]. The same unit cell was used as a calculation domain for another numerical study of a plate heat exchanger with different surface corrugations for use in an advanced inter-cooled-cycle gas turbine engine [27, 28].

Fernandes [29] and Kanaris [30] selected a half channel consisting of a fluid and two halves of two plates in order to take advantage of the condition of symmetry and the regular distribution of flow within the channel.

As indicated previously, choosing the calculation domain for numerical study sometimes leads to some restriction in other procedures that must be chosen. For example, when choosing the entire channel, it is freely possible to choose the opposite or parallel flow of fluids within the channels, but when choosing a small repeat cell, the cross-flow of the fluid is often chosen for the difficulty of applying the opposite flow in this case (like [23]).



a)

b)

Fig. 2 A statistic for the type of calculation domain used according to the articles in the review

Figure 2 shows a statistic for the type of account domain used according to the articles in the review.

3 Mesh

The formation of the computational grid (Mesh) for any geometrical shape, in general, is related to the shape geometry and the physical conditions in addition to the capabilities of the computer through which the grid formation process is conducted. For modelling in computational fluid dynamics, the fluid is represented by drawing a geometric shape that expresses the volume occupied by the fluid in the studied calculation domain. In CFD solvers, there are generally four ways to form the Mesh: Automatic, Tetrahedrons, Hex Dominant, Sweep, and Multi-Zone. As a result of the complex geometry of flow channels within the PHEs and the presence of narrow, sloping, and curved pathways, most researchers have adopted the unstructured tetrahedron mesh [1, 2, 4–10, 13, 14, 16, 19–22]. However, some researchers like Doo [16] and Patil [31] chose to generate the mesh as a structured hexahedral grid. There are two algorithms for generating tetrahedral elements in the grid: Patch Conforming and Patch Independent (Fluent 14.5 Release Meshing Method). In the first algorithm, the grid creation process begins from the faces and edges and then the volume. As for the second algorithm, it starts first by generating the grid in volume and then projected on the edges and faces. Some researchers, such as

Al-Zahrani [1], have simultaneously adopted the generation of the Mesh by the two methods together. Dora used the Patch dependent method to generate the tetrahedron mesh [24]. In order to increase the accuracy of the calculations near the walls in the calculation domain, the number of elements in this area is usually increased by making inflation of the computational cells near the wall [19]. The calculation of the height of the first layer of inflation layers near the wall is based on the non-dimensional parameter y^+. In order to define this parameter, one must first refer to the components of the boundary layer of the fluid near the wall. The turbulence boundary layer of the fluid generally consists of several different sub-layers. Turbulent flow along a wall consists of four regions, which can be considered characterized by the distance from the wall. The very thin layer next to the wall where viscous effects predominate is the viscous (or laminar or linear or wall) sublayer. The velocity profile in this sublayer is almost linear and the flow is streamlined. Next to the viscous sublayer is the buffer layer where turbulent effects become evident, but the flow is still dominated by viscous effects. Above the buffer layer is the transition (or overlap) layer, also called the inertial sublayer, where turbulent effects are much more important but still not dominant. Above this is the outer (or turbulent) layer in the rest of the flow, where turbulent effects dominate molecular diffusion (viscous) effects [32]. The y^+ parameter is defined as a non-dimensional parameter that determines the dominant flow pattern in the computational cells near the wall and thus determines which sub-layer within the boundary layer is resolved [33]. It is given by the following relationship:

$$y^+ = \frac{u_\tau . y}{\nu} \tag{1}$$

where y is the distance between the wall and the center of the adjacent cell, u_τ is the friction speed and it is given by the following relationship:

$$u_\tau = \sqrt{\frac{\tau_{wall}}{\rho}} \tag{2}$$

where τ_{wall} is the wall shear stress it is given by the following relationship:

$$\tau_{wall} = 0.5.C_f.\rho.U_0^2 \tag{3}$$

Using the non-dimensional parameter u^+, which is defined by the following relationship:

$$u^+ = \frac{U}{u_\tau} \tag{4}$$

For CFD, the most important layers are the closest to the wall and next to it, meaning the viscous sublayer and the inertial sublayer. The computational grid analysis of these layers requires different inputs that must be compatible with the different turbulence models. Accordingly, the values of y^+ are as follows (Fluent, 2005):

- At the viscous sublayer, y^+ is in the range: $y^+ < 5$.
- At the buffer region, y^+ is in the range: $5 < y^+ < 30$.
- At the log-law region, y^+ is in the range: $30 < y^+ < 300$.

Vaclav has developed a new method for the Mesh generation based on the dynamic Mesh method with the aim of improving the quality of the Mesh used for modelling operations in numerical studies of PHEs [34, 35]. This method has been adopted in other numerical studies of this type of heat exchanger [36, 37]. Because of the problems in the quality of the Mesh that arise when forming the computational grid at the contact points of the PHE plates, Luan has proposed a new way to improve the quality of the Mesh by placing an appropriate clearance between the two plates at the contact points [38]. This study showed that the ratio of clearance to the equivalent diameter is appropriate at a value equal to 0.02. To ensure the independence of the results of the numerical study from the computational grid, a so-called test of the Mesh is usually performed, which takes several forms. Some researchers tracked the average outlet temperature of the fluids [1, 2, 8] or the heat transfer rate [5] with the grid size change. Some tracked other parameters, such as pressure drop, with the change in y^+ [21]. Others tracked the change of Nusselt number (or heat transfer coefficient) and the coefficient of friction with the change in the size of the grid [3, 19, 20, 23]. To check the independence of the Mesh, Dora [24] monitored the temperature of the hot water outlet with the change of the computational grid skewness associated with the change in the Mesh size.

To perform the grid test, Fernandes [17] tracked the change of the parameter K with the change in the grid size and the grid size was approved for a change in the parameter K less than one percent. Zhang [39] tracked both the Nusselt number and the friction coefficient. Fernandez in other research [29] tracked the fluid mean velocity to perform the grid test, and the grid with the lowest size corresponding to a change in the fluid mean velocity was chosen at 0.02%. It is important to note here that in the grid test, the percentage of change in the size of the grid must be acceptable in comparison with the change of the monitored parameter in order to adopt the final grid size for the numerical study. For example, the change in the grid for the Zhang study [23] was around 26%, compared to a change in pressure drop and heat transfer coefficient of 0.89% and 1.24%, respectively.

4 Physical Model Definition

4.1 Boundary Conditions

The choice of boundary conditions applied to the calculation domain plays an important role in the numerical study. To the extent that the conditions applied are real and close to reality, the results are logical and credible. But sometimes resorting to some assumptions and simplification in order to facilitate study and save time in the solution. Conditions generally must meet the requirements of the equations of continuity, momentum conservation, and energy conservation. Fluid speeds or mass flows in addition to temperatures are often given at entry, and pressure of fluids at the exit [1, 2, 5, 8]. As for the boundary condition for the outer surfaces of the calculation domain, some articles have adopted the condition of constant temperature [13, 30], and some others have adopted the condition of constant heat flux [21], while others have adopted the condition of no heat exchange (isolation) [10]. In some researches, it was based on the application of periodic boundary conditions on external surfaces, especially when selecting a cold channel, for example, surrounded by two halves of two hot channels as a calculation domain for numerical study [23]. In others, especially where the calculation domain was a single cell, it depended on the application of periodic boundary conditions to the lateral surfaces of the calculation domain [21].

As for the other main walls in the calculation domain, are applied to them Stationary and no-slip boundary conditions [1]. Because of the periodicity of the flow in the width of the channel, Carla [17] imposed a symmetry condition on the lateral levels (xy) of the calculation domain. In Fernandez's study [29], the boundary condition for the upper and lower plates was chosen in two ways: the first is constant heat flow and the second is a variable heat flow that is simplified so that it changes linearly.

4.2 Turbulence Model

Generally, in PHEs, flow turbulence occurs when Reynolds numbers are relatively low ($Re > 400$) [40]. Since Reynolds number is usually calculated by a distinct length, some references take this length as the equivalent diameter of the channel D_e, others take it as the hydraulic diameter of the channel D_h. For traditional chevron sheets, they are given by the following relationships [41]:

$$D_e \approx 2b/\phi \tag{5}$$

where b is the corrugation depth of the plate [42].

$$D_h = D_e/\phi \tag{6}$$

where ϕ is the surface enlargement factor: the ratio of developed length to projected length.

Some of the literature relied on the equivalent diameter, while others relied on the hydraulic diameter for basic calculations [1, 5]. In general, CFD solvers provide three basic approaches to analyzing turbulent flow: DNS (**D**irect **N**umerical **S**imulation), LES (**L**arge **E**ddy **S**imulation), RANS (**R**eynolds **A**veraged **N**avier- Stokes **S**imulation). The first method is very expensive and impractical for the analysis of industrial flows. Whereas the second method is less expensive than the first method and requires large computational resources and directly solves some turbulences. The third method is the most used for industrial flows and is used to solve most levels of turbulence and several models for turbulence are available (Fluent 15.0 Release Turbulence Modeling). Lee has used the second method (LES) in an unsteady numerical study to the characterization of flow in PHE [21]. While the third method (RANS) was adopted in many of the numerical studies of the PHE, for example, the turbulence model SST k–ω was used by Zhang [23], Jiang [19, 20], Giurgiu [13], Khail [43], Han [44], and Kanaris [30]. Where SST k–ω combines the features of k–ω and k–ε in the vicinity of the wall [45], as well as good performance in areas where flow separation and adverse pressure gradients occur [46]. The realizable k–ε model with scalable wall function was adopted by Al zahrani [1], and the standard k–ε turbulence- Standard wall function was adopted by Gürel [5], Tiwari [8], and the renormalization-group (RNG) k–ε was adopted by Luan [9], Wang [10], Jain [22], Zhang [39] and the realizable k–ε model was adopted by Tsai [6], the Realizable k–ε model with non-equilibrium wall functions was adopted by Gherasim [12], and Li [15]. In Fig. 3, a statistic of the types of disorder models used in the numerical analysis according to the articles included in the review.

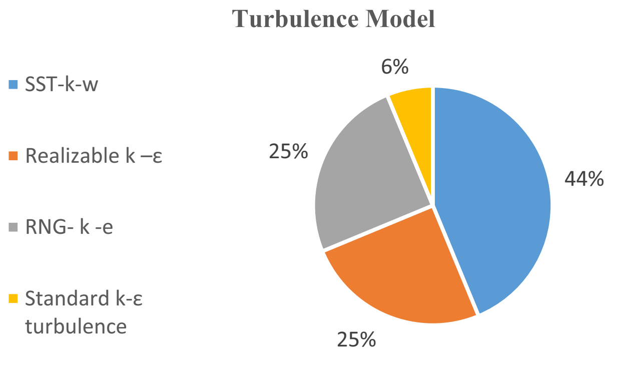


Fig. 3 A statistic of the types of turbulence models used in the numerical analysis according to the articles included in the review

4.3 Methods

In order to perform flow characteristics analysis in computational fluid dynamics, it is necessary to solve the continuity, momentum, and energy equations. To solve these partial differential equations, CFD solvers provide several algorithms for resolve these equations one of them is "SIMPLE" (**S**emi-**I**mplicit **M**ethod for **P**ressure-**L**inked **E**quations). This algorithm was adopted to solve the pressure–velocity coupling with the second-order upwind scheme for the spatial discretization [1]. While Zhang [39] adopted for spatial discretization on the standard method for pressure and the second-order upwind schemes for the momentum and energy equations.

Among the algorithms that have been adopted also in some literature [16, 47] is SIMPLEC (**SIMPLE-C**onsistent) algorithm. The difference between the previous two algorithms lies in the face flux correction since the use of the modified correction equation has demonstrated acceleration of convergence in problems where the pressure–velocity coupling is the main deterrent to obtaining a solution (ANSYS Fluent Theory Guide Release 15.0).

As for convergence criteria, they are usually taken 10^{-3}–10^{-5} for continuity and speed, while for energy is taken 10^{-5}–10^{-6}.

5 Validation

The issue of validating the results is one of the most important steps in the numerical study, which is considered a basic criterion for its success or failure. The results of numerical studies are usually validated in either analytical or experimental methods. In general, the difference in comparison of results should not exceed 10%. It is also possible to validate the results of the numerical study by comparing it with the results of a study, published in the literature, experimental or numerical of the geometrical model corresponding to the geometrical model that was studied in the numerical study. For example, Al-Zahrani [1], to validate the results of the numerical study of a flow model in a PHE according to modified Chevron plates, compared it (Nu Number) with the results of experiments and numerical studies conducted on traditional Chevron plates [48–51]. Gürel [5] suggested the lung model be placed on the heat exchanger plates instead of the chevron plates in order to increase heat exchange and to validate the results of the numerical study of the lung model, he compared it (Heat transfer rate) with the experimental results of a Chevron-type PHE [52]. Jiang [20] conducted a numerical study of a modified capsule type of PHEs with the aim of increasing energy efficiency [19, 20]. To validate the results of the numerical study, he compared the results (Nusselt number and friction coefficient) with previously published experimental results for a Chevron-type PHE [53, 54]. Zhang [23] to validate his numerical study of the capsule type of PHEs, conducted an experiment that included only checking the convergence of the numerical and experimental pressure drop values [23]. Tsai [6] validates his numerical study results of PHE by comparing it with the experimental study in the laboratory. The difference between the results was about 20%. Zhang [14] conducted a numerical study of the flow fields between two corrugated plates at different values of the chevron angle. He validated the results of his study by comparing the fluid flow fields with their counterparts in previously published experimental studies [55]. Han [4] conducted a numerical and experimental study of a Chevron-type PHE. To validate the results of the numerical study, he compared the results of the numerical study (the average temperature of the outlet of hot and cold water in addition to the pressure drop of hot and cold water) with the corresponding results in the experimental study. The result of the comparison with regard to temperature was the maximum difference of 2 degrees. As for the pressure drop of the fluids, the maximum difference was 35%. This great difference was justified as a result of the choice of the entry area of the fluid, which allowed the resistance to be weak in addition to some imbalance in the procedures of experiment and numerical study. Doo [16] studied numerically the effect of longitudinal heat conduction in PHEs, and to validate the results he compared the results of the numerical study (friction coefficient and Colburn factor) with the corresponding results from previously published studies [48, 56, 57]. As a result of the comparison, the maximum difference was found to be 20%. Tiwari [8] chose the exit temperatures for both the cold and hot fluid as parameters compared to the results of his numerical and experimental studies of PHE with the change in the volumetric flow rate of water at the entrance. The comparison was shown that the maximum deviation was 3.75%. Dora [24] validated the results of his numerical study by comparing it (heat transfer amount) with the results of an experimental study in the literature [58]. To validate the results of the numerical study of laminar flow in PHE, Carla compared the K-parameter of the numerical study with his counterparts resulting from the analytical study of laminar flow [59] previously studied in the literature. In the numerical study of a new type of PHE, Zhang [39] compared his numerical results with his experimental results the maximum deviation was about 22.5%. In Li Zhang's numerical study of several types of corrugation profiles of PHEs, the maximum deviation of the numerical results (friction coefficient and factor j) was about 14% when compared with experimental results [56] in the literature. Ahmad [11] validated the results of a numerical study (Nusselt Number) for flow in a plate heat exchanger with new plate geometry using a novel analytical method [11]. This method is based on both LMTD and ε-NTU heat exchangers analysis methods. The results showed the convergence of the analytical and numerical results with an error that did not exceed 3.6%.

6 Conclusion and Recommendations

This article discusses a review of the methodology of some numerical studies related to PHEs included in the literature. The articles presented in the review were selected from different time periods that keep pace with the development of CFD solvers and the capabilities of computers in terms of processing speed and capacity of RAM. This tremendous technical development in computer technology reflected positively on the procedures of numerical studies. Where instead of studying a single unit cell from the exchanger as a calculation domain and thanks to the development in the capabilities of computers the exchanger as a whole has become numerically taught which led to more comprehensive and more realistic results. Furthermore, in this case, included all areas of heat exchange and flow distribution in addition to fluids ports. This does not mean that the choice of a single unite cell only for numerical study reduces the value of the study, but on the contrary, thanks to the advanced options available in CFD solvers, where the number of elements can be greatly increased and benefit from periodic boundary conditions to investigate the thermal and hydraulic characteristics of flow in the heat transfer area more accurately and especially near the wall, where analysis of this area requires a large intensification of the number of computational cells to ensure the quality of the analysis. Through a review conducted on the numerical studies of PHEs, it can be concluded that:

- There is no single method with specific options approved to study PHEs numerically.
- The choice of the calculation domain for the numerical study is related to several considerations, including exchanger geometry, the profile of plate corrugation, type of flow within the exchanger, the purpose of the study, and most importantly the capabilities of the computer on which the study will be conducted.
- The generation of the Mesh requires, first of all, a clean geometrical body, and then the use of a method to create the Mesh corresponds to the chosen calculation domain, taking into account the choice of shapes for the elements that correspond to the place in which they are located.
- When performing a grid test, it should be noted that the percentage of change in the size of the grid must be acceptable with the change in the monitored parameter.
- The boundary conditions should be chosen appropriately for the purpose of the study, especially on the external surfaces of the calculation domain, taking into account the possibility of using periodic boundary conditions and the state of symmetry to save on the cost of the calculation.
- There is no single turbulence model approved for all numerical studies, but it differs from one study to another according to the aim of the study and achieving stability in the solution. This also applies to the choice of methods used in the solution.
- Validation of the results of the numerical study is done either by doing experiments that confirm the validity of the results of the numerical study or compare it with the results in the literature for numerical or experimental studies of the same studied model. It should be noted here that when the maximum deviation when comparing the results is more than 10%, the expected reasons for this should be explained.

It can be recommended during the numerical study to pay attention to the following:

- Attention when dealing with the chevron angle or the relationships associated with it, as the angle of the chevron in some literature is taken with the horizon while in others it is taken with the plumb.
- Also, in some literature, the hydraulic diameter is used in calculations, while in others the equivalent diameter is adopted.
- When studying a new geometry of the exchanger plate, the turbulence model is adopted which achieves stability during solution and gives results closer to experimental values.
- Adopting some of the new methods mentioned in the literature in creating the Mesh like that it takes clearance with an appropriate distance between the connection points of the plates and use of the dynamic Mesh to create similar grids, and improving the quality of the Mesh.

Acknowledgements This research was supported by scientific research projects coordination unit at KIRIKKALE University under the Number 2019/062.

Declarations

Conflict of interest On behalf of all authors, the corresponding author states that there is no conflict of interest.

References

1. Islam MS, Xu F, Saha SC (2020) Thermal performance investigation in a novel corrugated plate heat exchanger. Int J Heat Mass Transf 148:119095

2. Alzahran S, Islam M, Saha S (2019) A thermo-hydraulic characteristics investigation in corrugated plate heat exchanger. Energy Procedia 160:597–605
3. Hu Z et al (2017) Full-scale research on heat transfer and pressure drop of high flux plate heat exchanger. Appl Therm Eng 118:585–592
4. Han X-H et al (2010) A numerical and experimental study of chevron, corrugated-plate heat exchangers. Int Commun Heat Mass Transf 37(8):1008–1014
5. Gürel B et al (2020) Investigation on flow and heat transfer of compact brazed plate heat exchanger with lung pattern. Appl Therm Eng 175:115309
6. Tsai Y-C, Liu F-B, Shen P-T (2009) Investigations of the pressure drop and flow distribution in a chevron-type plate heat exchanger. Int Commun Heat Mass Transf 36(6):574–578
7. Sarraf K, Launay S, Tadrist L (2015) Complex 3D-flow analysis and corrugation angle effect in plate heat exchangers. Int J Therm Sci 94:126–138
8. Tiwari AK et al (2014) Numerical investigation of heat transfer and fluid flow in plate heat exchanger using nanofluids. Int J Therm Sci 85:93–103
9. Luan Z-J et al (2008) Flow resistance and heat transfer characteristics of a new-type plate heat exchanger. J Hydrodyn 20(4):524–529
10. Wang Y-N et al (2017) A study on 3D numerical model for plate heat exchanger. Procedia Eng 174:188–194
11. Khail AA, Erişen A (2021) Improvement of plate heat exchanger performance using a new plate geometry. Arab J Sci Eng 46(3):2877–2889
12. Gherasim I, Galanis N, Nguyen CT (2011) Effects of smooth longitudinal passages and port configuration on the flow and thermal fields in a plate heat exchanger. Appl Therm Eng 31(17–18):4113–4124
13. Giurgiu O, Plesa A, Socaciu L (2016) Plate heat exchangers–flow analysis through mini channels. Energy Procedia 85:244–251
14. Zhang G, Tian M, Zhou S (2006) Simulation and analysis of flow pattern in cross-corrugated plate heat exchangers. J Hydrodyn B 18(5):547–551
15. Li W et al (2013) Numerical and experimental analysis of composite fouling in corrugated plate heat exchangers. Int J Heat Mass Transf 63:351–360
16. Doo J et al (2012) Theoretical prediction of longitudinal heat conduction effect in cross-corrugated heat exchanger. Int J Heat Mass Transf 55(15–16):4129–4138
17. Fernandes CS et al (2007) Laminar flow in chevron-type plate heat exchangers: CFD analysis of tortuosity, shape factor and friction factor. Chem Eng Process 46(9):825–833
18. Fernandes CS et al (2008) Heat transfer to Newtonian and non-Newtonian fluids in cross-corrugated chevron-type plate heat exchangers: numerical approach. In: 18th international congress of chemical and process engineering
19. Jiang C et al (2019) Influence of capsule length and width on heat transfer in capsule-type plate heat exchangers. Adv Mech Eng 11(12):1687814019895742
20. Jiang C et al (2020) Heat transfer enhancement of plate heat exchangers with symmetrically distributed capsules to generate counter-rotating vortices. Int J Heat Mass Transf 151:119455
21. Lee J, Lee K-S (2015) Friction and Colburn factor correlations and shape optimization of chevron-type plate heat exchangers. Appl Therm Eng 89:62–69
22. Jain S, Joshi A, Bansal P (2007) A new approach to numerical simulation of small sized plate heat exchangers with chevron plates
23. Zhang Y et al (2016) Numerical study on heat transfer enhancement in capsule-type plate heat exchangers. Appl Therm Eng 108:1237–1242
24. Dora N, Mendu SS, Raju B (2020) Comparative study of conjugate heat transfer characteristics in chevron corrugated plate heat exchanger using CuO nanocoolant. Arab J Sci Eng 45:8955
25. Doo J et al (2012) An investigation of cross-corrugated heat exchanger primary surfaces for advanced intercooled-cycle aero engines (Part-I: Novel geometry of primary surface). Int J Heat Mass Transf 55(19–20):5256–5267
26. Doo J et al (2013) An investigation of cross-corrugated heat exchanger primary surfaces for advanced intercooled-cycle aero engines (Part-II: Design optimization of primary surface). Int J Heat Mass Transf 61:138–148
27. Kim M et al (2013) Numerical study on the cross-corrugated primary surface heat exchanger having asymmetric cross-sectional profiles for advanced intercooled-cycle aero engines. Int J Heat Mass Transf 66:139–153
28. Lee JM et al (2015) Characterizations of aerothermal performance of novel cross-corrugated plate heat exchangers for advanced cycle aero-engines. Int J Heat Mass Transf 85:166–180
29. Fernandes CS et al (2005) Simulation of stirred yoghurt processing in plate heat exchangers. J Food Eng 69(3):281–290
30. Kanaris A, Mouza A, Paras S (2009) Optimal design of a plate heat exchanger with undulated surfaces. Int J Therm Sci 48(6):1184–1195
31. Patil V, Manjunath H, Kusammanavar B (2013) Validation of plate heat exchanger design using CFD. Int J Mech Eng Robot Res 2(4):222–230
32. Cengel YA (2010) Fluid mechanics. Tata McGraw-Hill Education, New York
33. Ariff M, Salim SM, Cheah SC (2009) Wall y+ approach for dealing with turbulent flow over a surface mounted cube: Part 1-Low Reynolds Number. In: Seventh international conference on CFD in the minerals and process industries, Australia
34. Dvorak V (2016) Advanced methods of modelling and design of plate heat exchangers. DEStech Transactions on Environment, Energy and Earth Sciences
35. Dvořák V, Novosád J (2015) Influence of mesh quality and density on numerical calculation of heat exchanger with undulation in herringbone pattern. In: 19th international conference on circuits, systems, communications and computers, Zakynthos, Greece
36. Dvořák V, Vít T (2017) Evaluation of cae methods used for plate heat exchanger design. Energy Procedia 111:141–150
37. Dvořák V (2017) Optimization of wave pattern of corrugation on plates of heat exchanger. In: MATEC web of conferences. EDP Sciences
38. Luan H et al (2016) Numerical simulation of plate heat exchanger with improved mesh performance
39. Zhang C et al (2017) Experimental and numerical investigation on the exergy and entransy performance of a novel plate heat exchanger. Exp Heat Transf 30(2):162–177
40. Gherasim I, Galanis N, Nguyen CT (2011) Heat transfer and fluid flow in a plate heat exchanger. Part II Assessment of laminar and two-equation turbulent models. Int J Therm Sci 50(8):1499–1511
41. Kakaç S, Liu H, Pramuanjaroenkij A (2012) Heat exchangers: selection, rating, and thermal design. CRC Press, Boca Raton
42. Sundén B, Manglik RM (2007) Plate heat exchangers: design, applications and performance, vol 11. Wit Press, Southampton
43. A.A. Khail, A. Erişen, Heat transfer and performance enhancement investigation of novel plate heat exchanger, Thermal Science and Engineering Progress, 101368 (2022),
44. Han W et al (2011) Numerical simulation and optimization of single-phase turbulent flow in chevron-type plate heat exchanger with sinusoidal corrugations. HVAC&R Res 17(2):186–197
45. Kanaris AG, Mouza AA, Paras SV (2006) Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code. Chem Eng Technol 29(8):923–930

46. Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J 32(8):1598–1605
47. Zhang L, Che D (2011) Influence of corrugation profile on the thermalhydraulic performance of cross-corrugated plates. Numer Heat Transf A 59(4):267–296
48. Okada K et al (1972) Design and heat transfer characteristics of new plate heat exchanger. Heat Transf Jpn Res 1(1):90–95
49. Muley A, Manglik RM (1997) Experimental study of turbulent flow heat transfer and pressure drop in a plate heat exchanger with chevron plates. Univ. of Cincinnati, OH
50. Thonon B (1995) Design method for plate evaporators and condensers. In: BHR Group Conference Series Publication. Mechanical Engineering Publications Limited
51. Lee J, Lee K-S (2014) Flow characteristics and thermal performance in chevron type plate heat exchangers. Int J Heat Mass Transf 78:699–706
52. Kılıç B (2013) Experimental investigation of effects to heat transfer of plate geometry with dynamic and thermal parameters in the plate heat exchangers. Ph. D. Thesis, Süleyman Demirel University, The Graduate School of Natural
53. Khan T et al (2010) Experimental investigation of single phase convective heat transfer coefficient in a corrugated plate heat exchanger for multiple plate configurations. Appl Therm Eng 30(8–9):1058–1065
54. Khan TS, Khan MS, Ayub ZH (2017) Single-phase flow pressure drop analysis in a plate heat exchanger. Heat Transfer Eng 38(2):256–264
55. Zhao Z (2001) Effects of the corrugated inclination angle on heat transfer and resistance performances of plate heat exchangers. Petro-chem Equip 5:2–3
56. Focke W, Zachariades J, Olivier I (1985) The effect of the corrugation inclination angle on the thermohydraulic performance of plate heat exchangers. Int J Heat Mass Transf 28(8):1469–1479
57. Adams JC (2004) Advanced heat transfer surfaces for gas turbine heat exchangers. Oxford University, Oxford
58. Khairul MA et al (2014) Heat transfer performance and exergy analyses of a corrugated plate heat exchanger using metal oxide nanofluids. Int Commun Heat Mass Transf 50:8–14
59. Ding J, Manglik R (1996) Analytical solutions for laminar fully developed flows in double-sine shaped ducts. Heat Mass Transf 31(4):269–277