Journal of
Polymer & Composites

ISSN: 2321-2810 (Online)
ISSN: 2321-8525 (Print)
Volume 13, Special Issue 5, 2025
DOI (Journal): 10.37591/JoPC

https://journals.stmjournals.com/jopc

Research JoPC

Geometrical, Numerical and Materialistic Parameters used for Studying and Reducing Tube-Side Flow Inconsistency in Tube and Shell Heat Exchangers: An Overview

Kartik Ajugia[1,*], Vinayak H Khatawate[2], Hari Vasudevan[3]

Abstract

A Shell and Tube Heat Exchanger (STHE) exchanges heat between two fluids flowing at different temperatures. More than 60% of the market is occupied by STHE's out of all the different kinds of heat exchangers. Maldistribution is the uneven fluid flow distribution in the tubes of the STHE. The aim of this study is to provide a detailed review on the various methods used by researchers for reducing the maldistribution on tube side of STHE's. With a view to lessen the non-uniformity in the tube side of a STHE, several researchers have employed various strategies. These include the use of baffles placed in the header, nozzle and orifices paced in the tube inlets etc. and a significant amount of reduction in the maldistribution has been observed with the use of these methods. Another way of improving the heat exchanger efficiency is making use of polymeric and composite materials. Some of these material have very high thermal conductivity resulting in drastic improvements of the heat transfer. The problem of maldistribution persist in heat exchangers whether the materials used are conventional or polymers and composites. This study can be further extended to check the feasibility of the various methods used by researchers to reduce the maldistribution at different inlet Reynolds number and different geometrical cases like the nozzle position and the area ratios in sync with high thermal conductivity material usage like the polymer and composites due to their different viscosity compared to conventional materials.

Keywords: *STHE, maldistribution, tube side, header design, uniformity index, polymer, composites*

***Author for Correspondence**
Kartik Ajugia1

[1]Assistant Professor, Department of Mechanical Engineering, SVKM's Dwarkadas J. Sanghvi College of Engineering, Mumbai, Maharashtra, India
[2]Associate Professor, Department of Mechanical Engineering, SVKM's Dwarkadas J. Sanghvi College of Engineering, Mumbai, Maharashtra, India
[3]Professor, Department of Mechanical Engineering, SVKM's Dwarkadas J. Sanghvi College of Engineering, Mumbai, Maharashtra, India

Received Date: January 16, 2025
Accepted Date: April 22, 2025
Published Date: July 18, 2025

Citation: Kartik Ajugia, Vinayak H Khatawate, Hari Vasudevan. Geometrical, Numerical and Materialistic Parameters used for Studying and Reducing Tube-Side Flow Inconsistency in Tube and Shell Heat Exchangers: An Overview. Journal of Polymer & Composites. 2025; 13(Special Issue 5): S603–S621p.

INTRODUCTION

A heat exchanger is an instrument designed to transfer energy, in this case heat, between two or more different fluids, or even between a solid and a fluid. They are used in refrigerators, air conditioners, automobile radiators, and chemical industries for operations like fractionation and distillation.

The STHE (Figure 1) is the most adaptable type of heat exchanger. Major chemical process, oil refineries implement this sort of heat exchanger the most.

With reference to its name, this energy exchanger possesses a collection of passages called tubes enclosed in it.

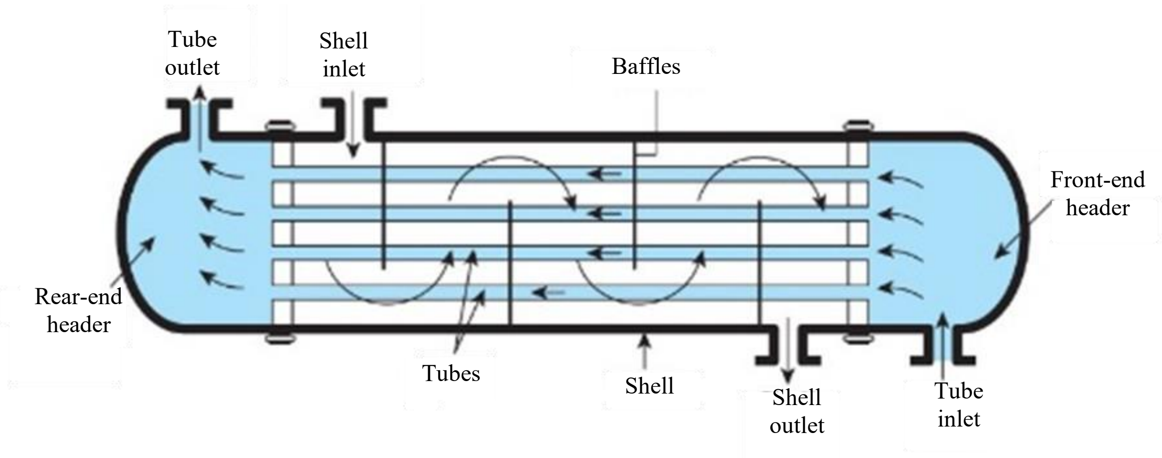


Figure 1. STHE Model.

Fluids two in number with different initial temperatures are circulated via this heat exchanger one through the shell while the other through the tubes. The heat transfer between the two fluids takes place across the tube walls. The fluids on either the shell or tube side may be liquids or gases. For effective heat transfer to take place a larger area is recommended, which implies for the use of many tubes.

The typical shell and tube geometries employ a parallel channel arrangement to fit a lot of heat transmission into a small amount of space. Large capacity equipment can be constructed in a manageable size because to this, however it also causes issues with flow maldistribution.

Since the word mal signifies "defective" or "bad," the definition of "maldistribution" is contingent upon one's definition of distribution. Variability in fluid flow is amongst the major causes of subpar heat exchanger output. Inadequate input outlet port design, header arrangement, distributor orientation, and plate corrugations can all be blamed for this. It seriously impairs and damages the STHE's functionality.

LITERATURE REVIEW

Heat exchangers find many industrial applications like the pharmaceutical, petrochemical, chemical, power plants as stated by B.I. Master et al. and R. Brogan [1, 2].

Peters MS and Timmerhaus KD [3] state that the improvisation in the heat transfer in the STHE is a crucial domain of research as they are amongst the 60% heat exchanger application globally.

The standard design of the STHE as laid down by Tubular Exchanger Manufacturers Association (TEMA) [4] or the Verein Deutscher Ingenieure heat atlas (VDI) [5], are based on the assumption that there is no maldistribution in the tubes of the STHE but practically that is not the case.

There are many causes of maldistribution in exchangers as suggested by Mueller and Chiou (1988) [6] (i) mechanical causes due to the design of headers and inlet ducts influencing the flow distribution, or the effect of manufacturing tolerances affecting the size of the flow passages in compact-type exchangers; (ii) self-induced maldistribution due to the heat transfer process itself as the "freezing" effect in viscous flow coolers or the various types of thermoacoustic oscillations; (iii) two phase (gas-liquid) flows are very difficult to distribute uniformly through the tubes in a tube bundle; (iv) fouling and/or corrosion can affect the flow distribution; (v) increasing inlet velocity or Reynolds number at the inlet nozzle. The velocity distribution near the tube bundle or regenerator core face is greatly influenced by the header and intake duct designs. For heat exchangers with a high NTU (number of transfer units), gross flow maldistribution results in a considerable decrease in efficiency of around 25% for heat exchangers which are cross flow in nature and roughly 7% for applications like condensers.

The flow non uniformity leads to the poor performance of the heat exchanger. There are many geometrical parameters which may lead to the non-uniformity in the flow like the improper design of the inlet or otlet port, configuration of the header and distribution construction as suggested by K. Grijspeerdt, B. Hazarika and D. Vucinic [7].

Maldistribution of flow, which can take place in the header, can cause a change of local heat flow and directly affect the performance of total heat transfer as stated by Choi and Lee [8].

The most common assumption is that that fluid can be evenly dispersed throughout the core and at the exchanger's input on either fluid side. J. K. Lee and S.Y. Lee discovered that the channels' flow rates are generally irregular and, in severe situations, scarcely flow at all, resulting in subpar heat exchange performance [9].

Maldistribution is a function of number of tubes, flow rate and the tube size as found by Mohan et al [10].

Maldistribution and the fouling that frequently occurs in these STHE result to lower production throughput and greater energy utilization. An increase in average wall shear stress is expected to lead to less fouling than in the maldistributed condition inside the tubes. This can be achieved via an even dispersion into the tubes, according to Schab et al. [11].

According to Bejan and Kraus [12], standard heat exchanger design assumes that the flow in a tube bundle of shell and tube heat exchangers is uniformly distributed. However, as Kitto Jr., J.B., and Robertson [13] have noted, flow maldistribution is an inevitable incident in practice that can have serious consequences for the mechanical and thermal performance of heat exchangers.

The header must be designed to produce i) the same mass flux in each tube and ii) little pressure loss in the header because it does not transfer heat. One important source of gross flow maldistribution is the header. In the majority of situations in reality, it is challenging to attain an optimal state in which the flow distribution exhibits minor velocity variation. Shah, R.K., Sekulic, and Hewitt therefore came to the conclusion that, in nearly all of cases, ±5% of the mean flow would be considered to be a uniform distribution. [14, 15].

According to DQ Kern [16], tube-side pressure drop (ΔPtotal) is the total of pressure losses from friction inside the tubes (ΔPt) and pressure losses from rapid expansion and contraction of the tubes (ΔPr), which are accounted for by four velocity heads per pass.

$$\Delta \mathrm{Pt} = \frac{4\ \mathrm{f\ L\ Np}\ \rho\ U^2\ \mathrm{m}}{2\ \mathrm{di}}$$

$$\Delta \mathrm{Pr} = \frac{4\ \mathrm{Np}\ \rho\ U^2\ \mathrm{m}}{2}$$

Thus, $\Delta \mathrm{P\ total} = (\frac{4\ \mathrm{f\ L\ Np}}{\mathrm{di} + 4\ \mathrm{Np}})\ \frac{\rho\ U^2\ \mathrm{m}}{2}$

Where,

L – Tube length
Np - Number of passes
di – Tube inner diameter
f – friction factor, ρ – Fluid density
Um – fluid average velocity in tubes

Because of the unequal flow distribution, the secondary side fluid's internal temperature field is not uniform. This will seriously affect the overall heat transfer characteristic and stable operation, especially for the small and compact STHXs as stated by Stanisław Łopata, and Paweł Ocłoń [17]. Uneven temperature fields will also contribute to thermal stress and tube-sheet cracks.

Schab et al. [11] studied tubeside maldistribution in shell-and-tube heat exchangers numerically. Using computational fluid dynamics (CFD), a parametric STHE model was examined to look into tubeside flow maldistribution. To determine whether SST may produce better simulation results, a comparison between the new standard SST (Shear Stress Transport) model and the conventional k-¬ε model was first conducted. However, the simulation results for the current application were not significantly improved by the higher numerical effort of SST in comparison to k-ε. The fluid flow velocity in a single tube is was found using three models transient SST, mean transient SST and the k-ε model and is resented with blue, red and yellow color respectively in Figure 2.

Numerous investigations were carried out by S. A. Marzouk et al. [18] in an effort to decrease the size and expense of STHE and boost the rate of heat transfer.

A thorough, current, and methodical assessment of the many approaches (active, passive and compound) for improving heat or energy transmission in STHE's was given by the paper. The study is a vital resource for the topic of heat transfer because its conclusions have applications for engineers, researchers, and companies. Heat transfer increase via active, passive, and compound approaches is depicted in Figure 3.

Based on the concept of a maldistribution reduction manifold, Mingkan Zhang et al. [19] created an advanced maldistribution reduction manifold. In order to regulate the fluid flow in every tube for uniform distribution in the manifold, spiral baffles were utilized to reduce non uniformity advancely by creating vortices in the tubes. The sophisticated maldistribution reduction manifold's design is too complex to be manufactured using conventional methods. As demonstrated in Figure 4, CFD models have been created to analyze flow and temperature distributions using the reference, maldistribution reduction, and advanced maldistribution reduction designs. The advanced maldistribution reduction design enabled an important boost in minimizing maldistribution in the manifold.

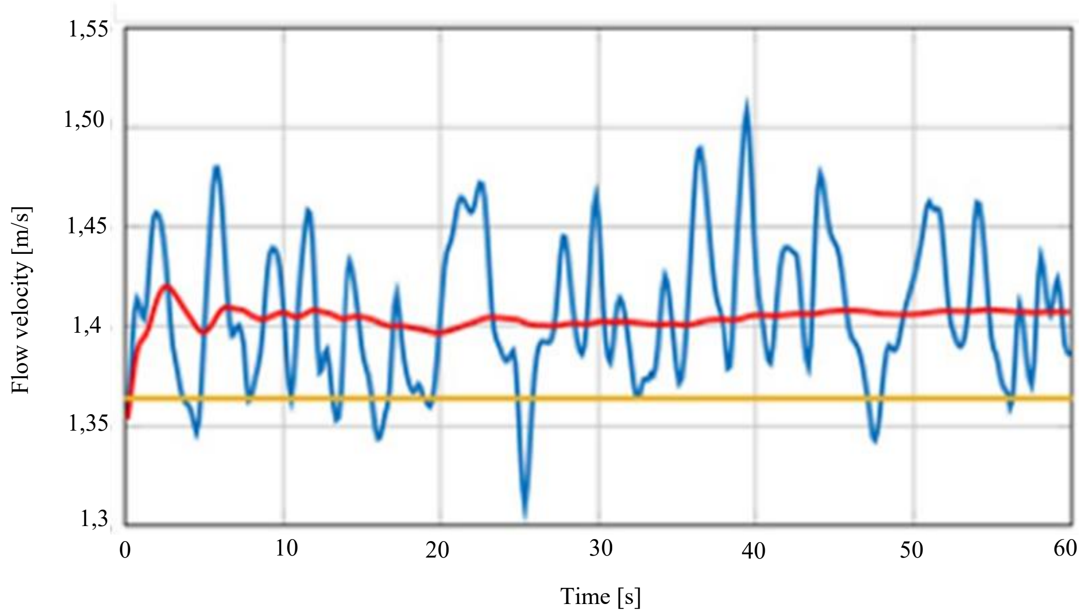


Figure 2. Illustration of difference in tube flow velocity within a single tube using Transient SST (Blue), averaged transient SST (red), and steady-state k − ϵ (yellow) [11].

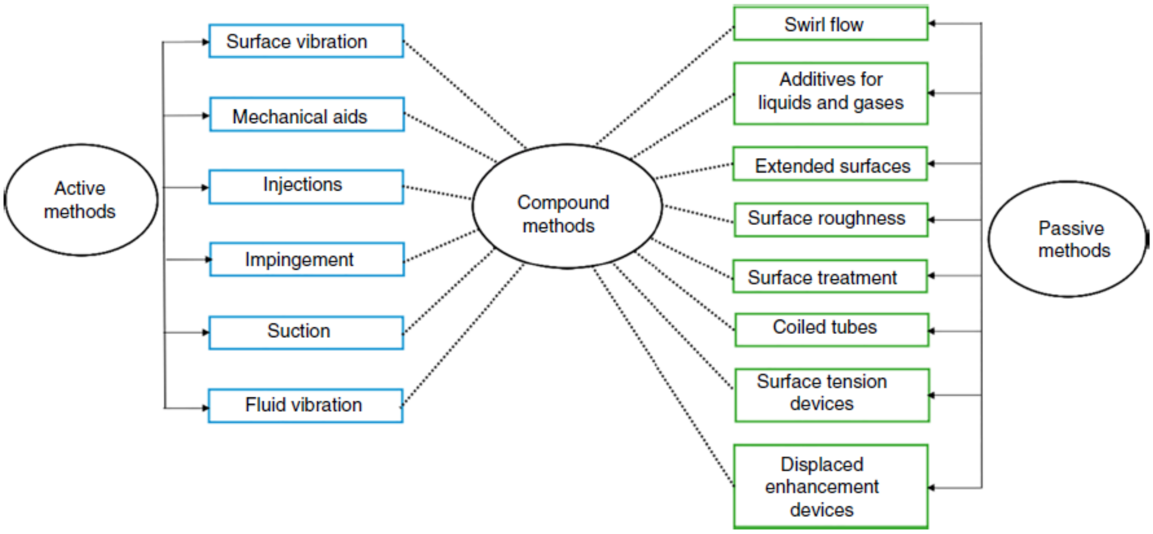


Figure 3. Active, passive and compound methods used [18].

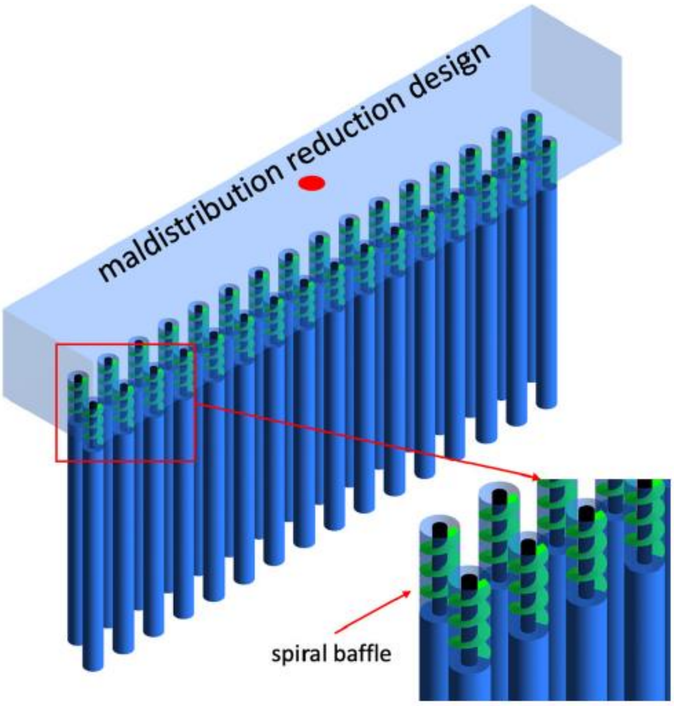


Figure 4. Illustration of the maldistribution reduction design [19].

Grzegorz Ligus et al. [20] researched and presented the liquid flow non uniformity in STHE. This phenomenon negatively impacts drop in pressure and transfer of heat and is the cause of the velocity reduction area. Two visualization techniques were applied to shower knowledge about the liquid distribution in STHE. Particle image velocimetry (PIV) was the first method used in sync with the k- ε model. Both approaches' viability for detecting and evaluating fluid flow abnormalities in STHE was validated by the investigations that were carried out. When compared to experimental approaches, the CFD method's highest inaccuracy in terms of pressure drops and maximum velocities was no more than 7% and 11%, respectively.

According to Richard Schab et al. [21], there are notable differences in the degree of flow maldistribution among the various STHE designs. Consequently, a parametric research was carried out to look at the causes of uneven distribution. For example, the orientation, type, and diameter of the nozzles were determined to be important factors. They came to the conclusion that using a bigger intake nozzle diameter to achieve an area ratio near one is the most effective and straightforward method of continuously reducing tube side flow maldistribution. By increasing average tube velocities, a larger mass flowrate—which is frequently dictated by the entire process—may lessen fouling. However, because there is a greater disparity between the highest and minimum flow velocity values, high mass flowrates reduce flow homogeneity. Inlet flow velocity should only be increased only when flow uniformity is met. Thus far, there has been no discernible relationship between the uniformity index and the inlet angle α_{Nozzle}.

Subsequently, further simulations can be performed, and additional parameters affecting maldistribution can be investigated.

The fluid flow distribution in the 1170 tubes of the heat exchanger with an inlet mass flux of 220 kg/s is shown in Figure 5.

Figure 6 shows the parameter study's findings. In a scatter plot, the most important parameters are displayed. Every single point denotes a distinct combination of parameters (one simulation setting for a heat exchanger). To illustrate the impact of the shown parameter on the uniformity index, a regression line is added in each plot.

Marwa Ben Slimene et al. [22], numerically investigated the heat transfer between glucol and water in a STHE with one shell and six tube passes. The mode of heat transfer between the fluids was conduction and convection. Glycol flowed through the tube as it was the refrigerant while water passed through the shell. It was found that the system of equations closed using the k-ω SST model.

Tubes with different cross section i.e. circular and elliptical with an attack angle of 90° and 0^0 and their combinations were studied by Mohammad Reza Saffarian et al. [23]. A model of a STHE was shown, using tubes with circular cross section STHE-CT&ET90° and tubes with a 90° attack angle elliptical in shape.

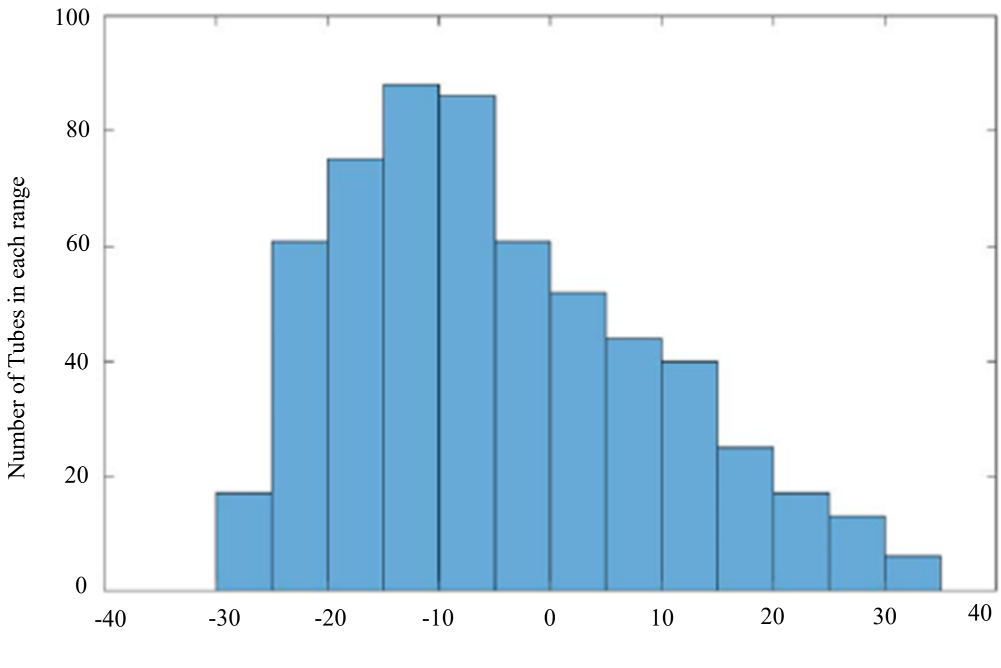


Figure 5. % Deviation for 1170 tubes as compared to mean tube velocity [21].

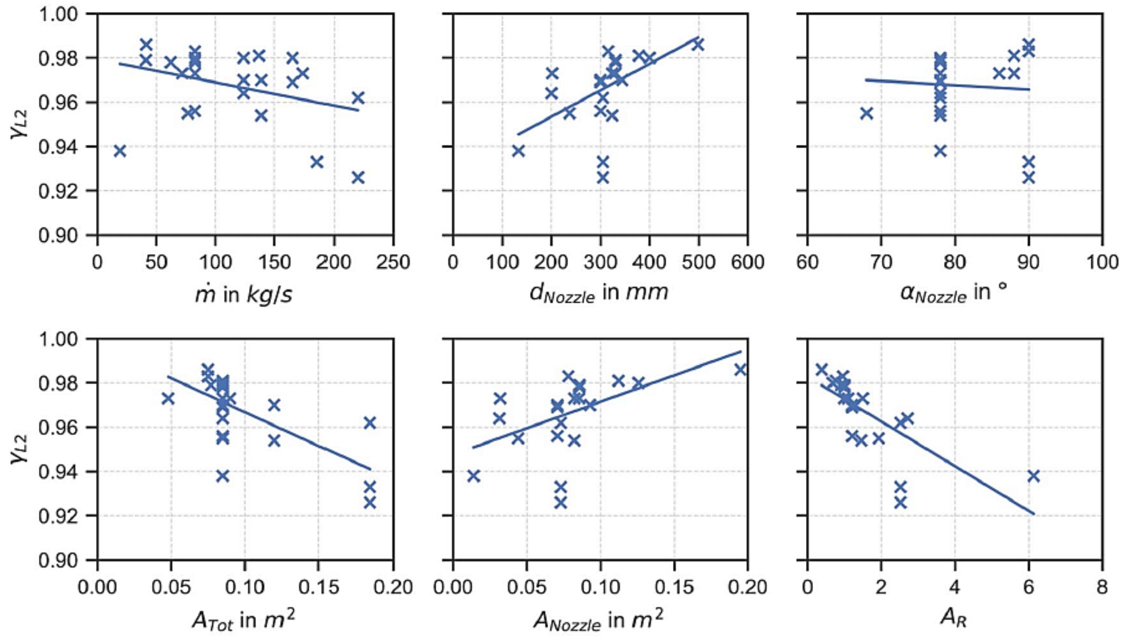
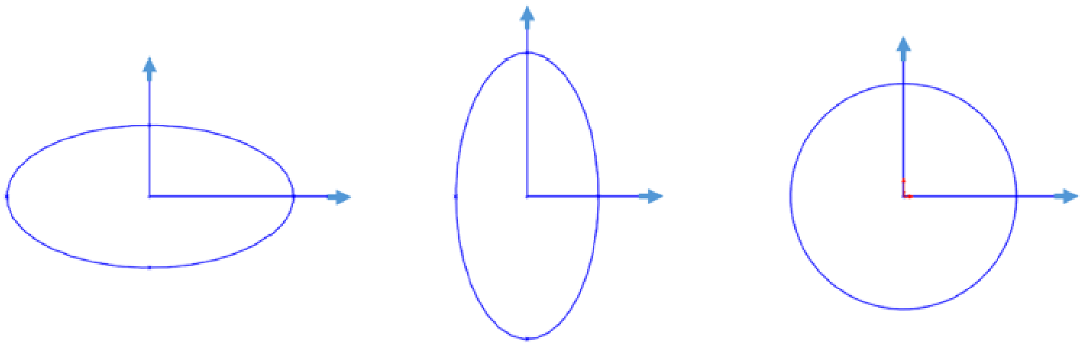


Figure 6. Variation of Uniformity Index with Geometric Parameters [21].



A B C

Figure 7. Tubes with: a) elliptical 0° angle attack; b) elliptical 90° angle attack; c) circular cross section [23].

The heat exchanger with ellipsoidal tubes around the shell and circular shaped tubes in the middle of the shell exhibited the greatest degree of heat transfer while STHEs with circular and elliptical shaped tubes with degrees of attack of 90° and 0° were examined. The various tube configuration and cross-section is as shown in Figure 7. Tubes in the vicinity of the shell surface had a larger impact on heat transfer than those at the center. The velocity contours for the setups used is displayed in Figure 8.

A. Ravinthiran et al. [24] found that this maldistribution of flow in conventional heat exchangers can be rectified by changing the shape of the headers of the heat exchanger. Figure 9. indicates the models of various tubes used. A layer of boundaries with different thicknesses appears across the cross-section when the header's cross-sectional shape is altered to a polygon. This results in a more uniform distribution of flow compared to the header's circular cross-section. Figure shows the models of various setups.

In the study it was shown that when a polygon-shaped header was used it lowered the maximum flow velocity, thereby reducing the maximum heat transmission.

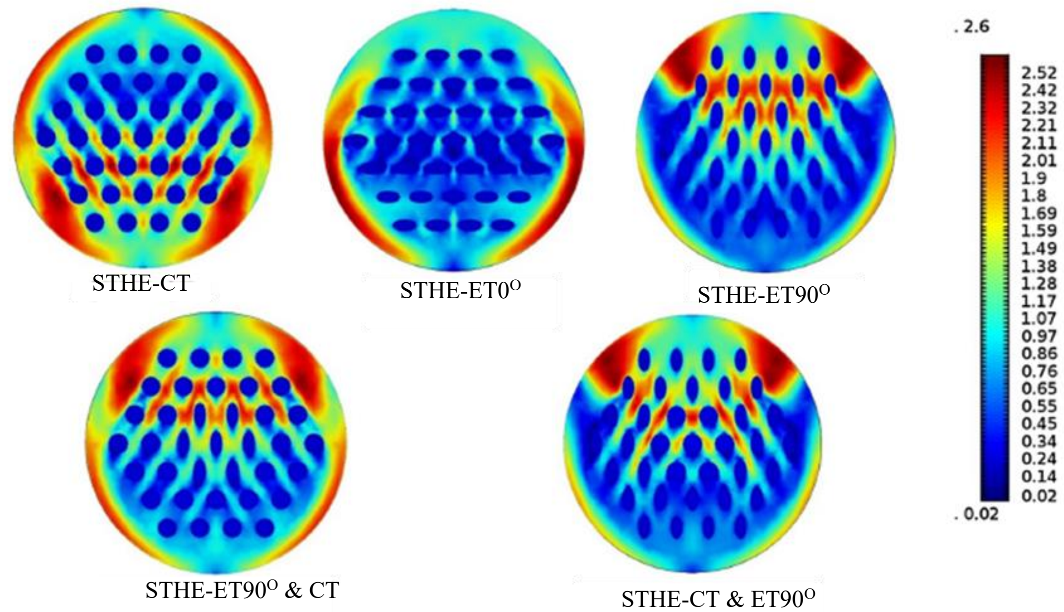


Figure 8. The velocity contour at the cross-section of the shell and tube heat exchanger for the STHE-CT, STHE-ET0°, STHE-ET90°, STHEET90°&CT, STHE-CT&ET90°.

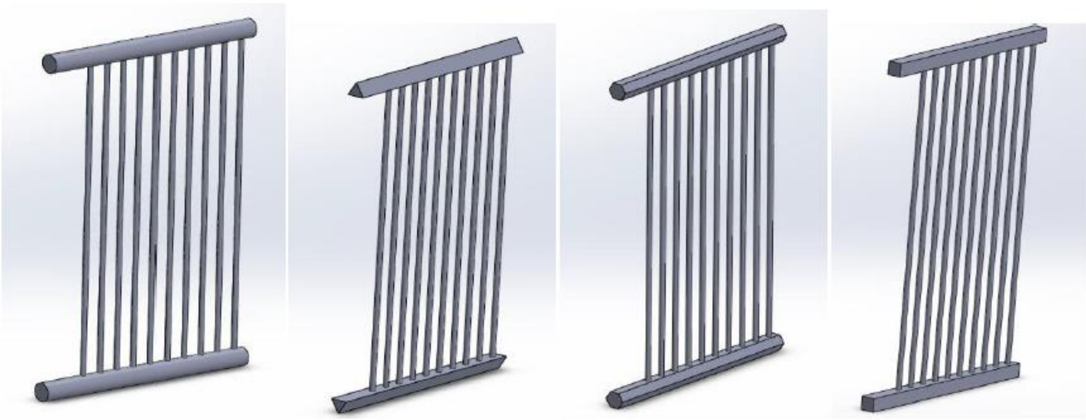


Figure 9. Models of various tubes [24].

In order to develop and build a shell and tube heat exchanger for laboratory usage, B. C. Chukwudi and M. B. Ogunedo [25] conducted a study. To accomplish this, care was given to both mechanical and thermal factors. The Bell Delaware approach was used to finish the thermal design. When developing the heat exchanger, it was expected that there would be no phase change.

Myoung Il Kim et al. [26] used numerical analysis to investigate a number of header designs in order to attain a uniform distribution of gas phase flow in the shell-and-tube heat exchanger header. The different geometries included length, the quantity of outlet tubes, and the shape and location of the intake nozzle. Utilizing the k-ε turbulence model, simulations were conducted. To assess the consistency of the velocity distribution throughout all of the header's outlets, standard deviation was employed. Therefore, by analysing the numerical findings, it became possible for demonstrating the patterns of flow in the header. While it decreased with gas flow rate, the homogeneity of flow dispersion rose with header length. Additionally, the ideal location and form of the nozzle inlet might be suggested for a consistent distribution of a header that is 1.3 meters long; this same header is employed for the heat exchange of the commercially feasible allyl chloride process.

S.A.M. Said et al. [27] found that the flow maldistribution within the tubes occurs due to the development of vena-contracta at the inlet to the tubes. In this study, two techniques for decreasing the flow nonuniformity in the header and tube arrangement were proposed and numerically determined. The approaches used during the study are displayed in Figure 10. Firstly, an orifice was introduced to reduce the tube inlet. Figure 11. shows the geometry used in the study.

According to the current study's findings, flow maldistribution can be decreased by about 12 times when employing the orifice technique. On the other hand, the heat exchanger's pressure loss is 7.8% more than in the case without orifices. The second method involves adding a nozzle to enhance the actual tube inlet. According to the study's findings, flow mal-distribution can be avoided by adopting a nozzle technique by about 7.5 times. However, when compared to the scenario without nozzles, the heat exchanger's pressure drop drops by 9.8%.

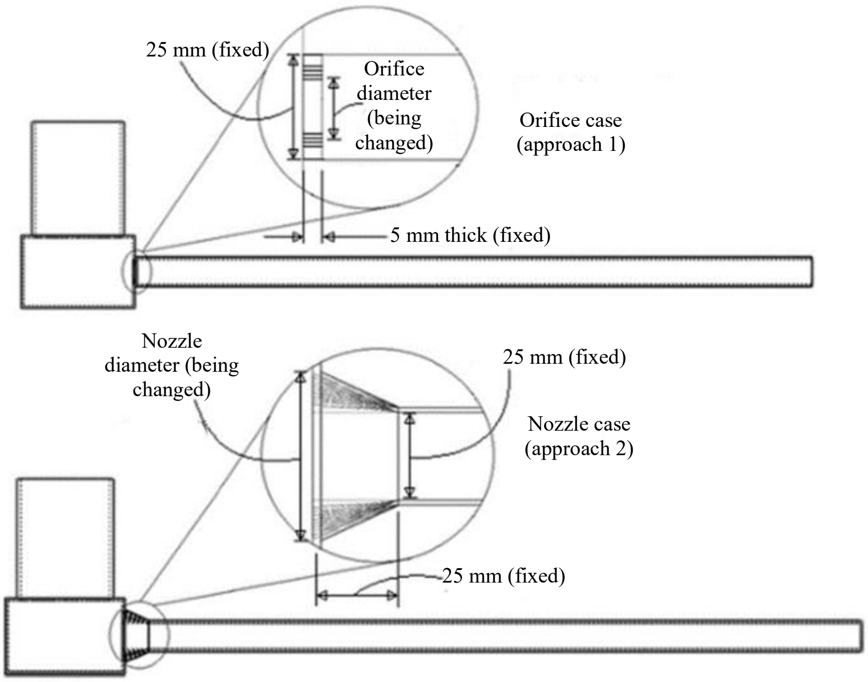


Figure 10. Methods for lowering flow mal-distribution [25].

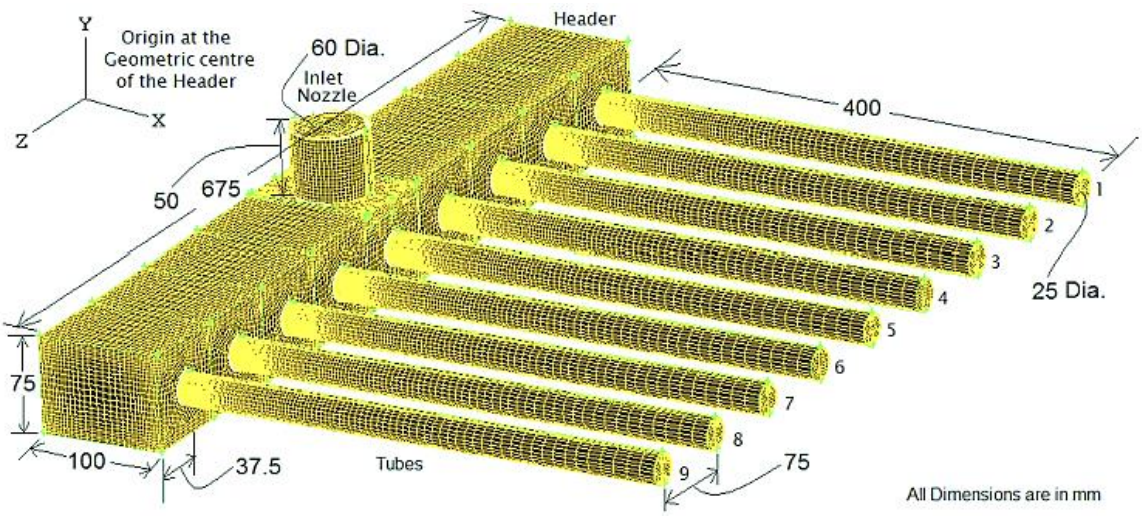


Figure 11. Geometry of the problem [25].

A. Ravinthiran et al. [28] corrected the heat exchangers' uneven flow distribution that degrades performance. The distribution of flow between the flow header's branch pipes was examined for two distinct geometrical flow types (circular and square). CFD software is used for the analysis of the flow distribution. The values derived from the examination of the square and circular header tubes are compared with the laboratory experimental results of Osakabae [29]. The results conclude that the header shape will always impact the boiler's overall performance. By selecting the best shapes, the Heat Exchanger's performance may be improved while fuel consumption can be decreased.

Kai Wang et al. [30] optimized the porous plate experimentally and numerically to improve the flow distribution in the tube side. To verify the fact that a baffle which is porous in nature increases the uniformity in tube side of a STHE, an experimental investigation was conducted initially. Figure 12 shows the layout of the apparatus used.

The findings demonstrated that the baffle can significantly reduce maldistribution. They subsequently optimized the porous baffle shape numerically. The spread of circular holes in the baffle and the shape of baffle were decided after the use of numerical methods. Measuring the flow field with the optimized baffle allowed the numerical optimization to be validated. The results demonstrated that, despite some discrepancies between the results obtained from experiments and numerically, the flow distribution utilizing the optimized model was significantly better than that of the proto-type porous baffle.

Air cooled heat exchangers with rectangular shaped headers was evaluated for flow maldistribution by M.A. Habib et al. [31]. A 3-D computational technique was used to find out how nozzle diameter and number impacted the flow uniformity within the tubes. The mass flow rate standard deviations are used to display the results. It was proved from the study that the maldistribution increased with nozzle diameter. For every 25% decrease in diameter, the standard deviation rises by 25%. Figure 13 shows the impact of nozzle diameter on the mass flux along with static pressure standard deviations. An increase in nozzle count modifies the uneven distribution of flow. By adding four nozzles instead of two, the standard variation of the mass flow rate inside the tubes is reduced by 62.5%. The impact of nozzle diameter on the static pressure and total pressure loss standard deviations is shown in Table 1.

Table 1. The Nozzle Diameter's Impact [31].

Diameter of Nozzle mm	Standard Deviation in the mass flow rates, kg/s	Standard Deviation in The Static Pressure, kPa	Total Pressure Loss ΔP, kPa
69	0.0504	221.1	0.836
80.5	0.0448	169.5	0.92
92	0.0407	150.6	0.9

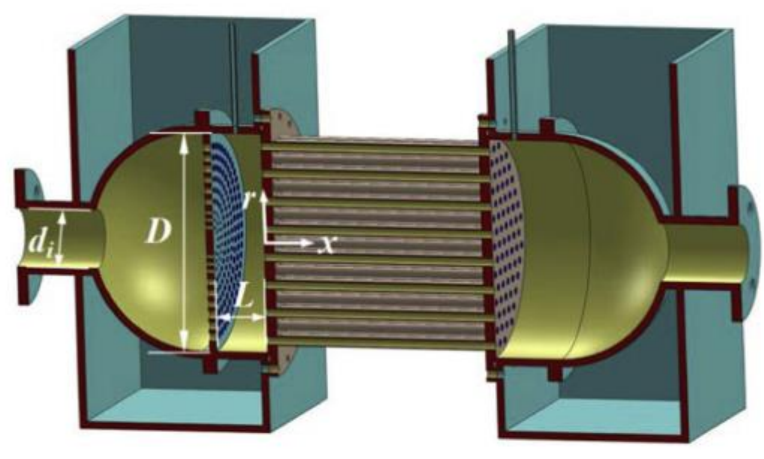


Figure 12. Experimental apparatus's schematic diagram [30].

The results of the effect of the number of nozzles on the standard deviations of the mass flow rates in the tubes and the pressure distribution inside the header are shown in Table 2 and Figure 14. It is evident that the nozzles are dispersed uniformly across the header. The image illustrates how the number of nozzles has a major effect on the maldistribution. Each nozzle's velocity values decrease as the number of nozzles rises because the flow rate is dispersed among them. Consequently, the pressure inside the header and the flow inside the tubes are more constant. Table 2 shows that a 62.5% reduction in the standard deviation occurs when the number of nozzles is increased from two to four.

Gayatri Kuchi et al. [32] installed a baffle plate in the header portion with a view to improve the header layout. A discernible reduction in the flow maldistribution in the tubes was seen upon the implementation of the baffle plate. The uniformity of flow was also examined using the tubes' internal catalytic bed. This configuration resulted in a notable reduction in flow maldistribution.

Figure 15 displays the STHE's examined 3-D tube configuration.

Table 2. Effect of number of Nozzle [31].

Number of Nozzles	Standard Deviation in the mass flow rates, kg/s	Standard Deviation in The Static Pressure, kPa	Total Pressure Loss ΔP, kPa
2	0.0407	150.6	0.9
3	0.0232	76.8	0.920
4	0.0165	45.3	0.903

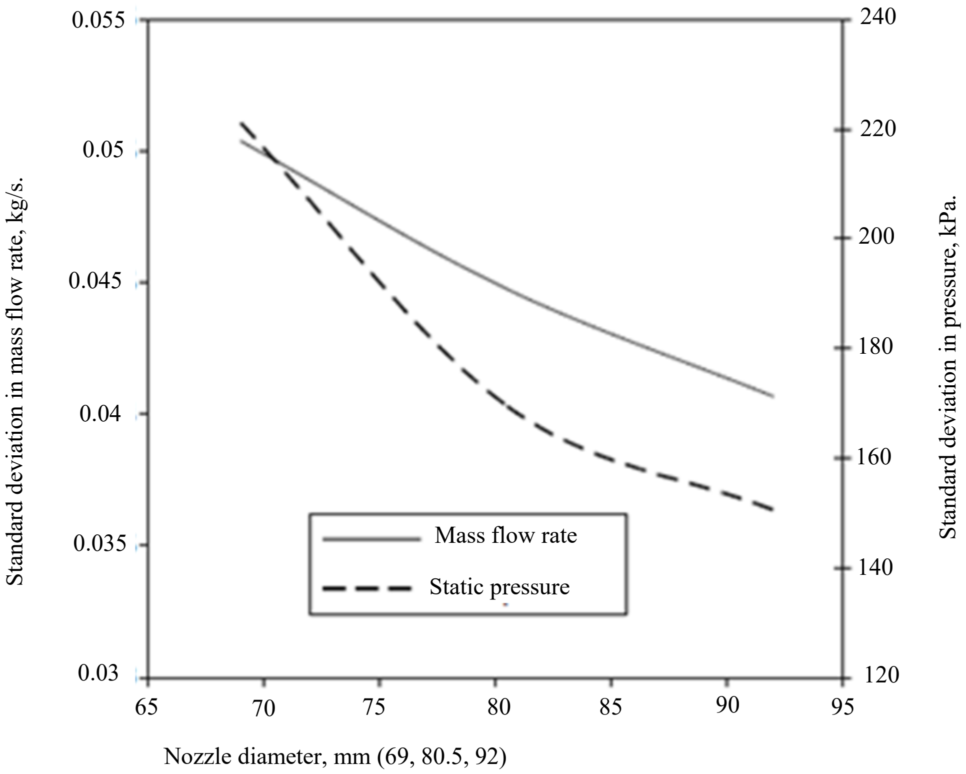


Figure 13. Effect of nozzle diameter on the mass flow rate and static pressure standard deviations [31].

The flow distribution in the tubes is highly irregular, according to the baseline design's initial analysis. It is therefore advised to enhance the inlet manifold arrangement. Utilizing this information, a baffle is built and placed in the inlet manifold section. A 90 mm diameter and 5 mm thick baffle plate has been made from the model. It is mounted in the conventional header to optimize the inlet manifold design and features three distinct types of holes drilled in it. Midway between the header core and the inflow tube is the ideal location for the perforated grid, according to studies [33]. In Figure 16, the baffle plate model is displayed.

The flow homogeneity was successfully improved by the upgraded header with the baffle added. Optimizing the baffle configuration involves taking into consideration the header configuration and fluid flow condition, which have an impact on the baffle configuration. Another solution to the problem of flow maldistribution is the development of a catalytic bed zone inside the tubes along their whole length, which has culminated in a remarkably uniform flow distribution. This results in uniform flow and redistribution of the flow in the inlet manifold due to the high resistance of the pebble bed zone.

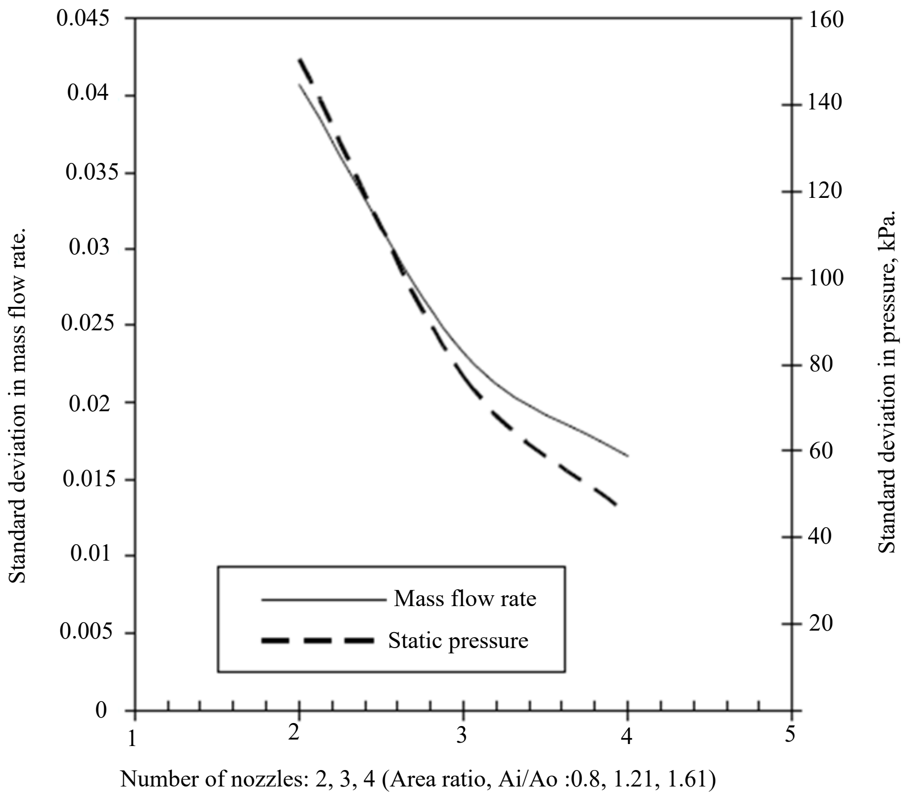


Figure 14. Effect of the nozzle count on the mass flow rate and static pressure standard deviations [31].

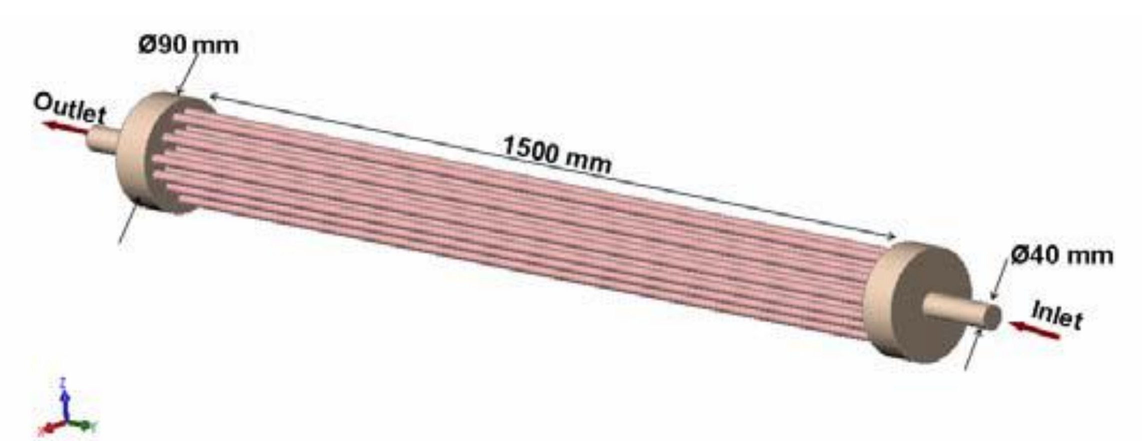


Figure 15. Heat exchanger geometry used [32].

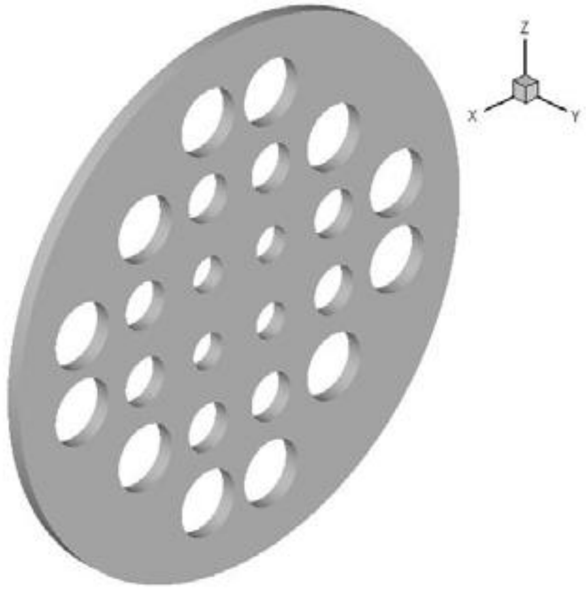


Figure 16. Baffle plate [32].

K. Mohammadi and M.R. Malayeri [34] devised mathematical models to predict the number of tubes that have been impacted due to maldistribution and their extent. The models were based on maximum possible velocity deviation. The resulting model's validity has been validated by comparison with related research that uses computational fluid dynamics (CFD). Additionally, compared to the basic model, the optimized model's heat transmission per tube row was 7.80% higher. This was primarily because decreased flow non-uniformity in each tube resulted in a 7.81% increase in the surface heat transfer coefficient.

The optimization iterations were sufficiently few to be directly applied to the real design process, which was another significant discovery. It has been verified that the suggested approach can be incorporated to increase the efficiency of different exchangers with parallel headers.

The many variables affecting the fluid's behavior makes a STHE's internal circulation complex. Based to studies conducted by O. Labbadlia et al., the flow distribution significantly affects the operation of fluidic devices like shell and tube heat exchangers [35]. The unequal flow distribution renders the method less effective. Figure 17 shows the effects of tube designs on the flow distribution using CFD simulations. Four sorts of designs are looked at in this study. The findings from the literature and the results that were given agree quite well. The results show that the tube layout has an important impact on the flow distribution. O. Labbadlia and coworkers found that the 60° flow distribution is 21% more uniform than the 90° conventional setup. The 45° layout has a more uniform pressure distribution than the other versions.

Arun Muley, T.S. Ravigururajan, and Rohitha Paruchuri [36] examined how the headers' velocity spread and pressure drop impacted the efficacy of heat transfer. According to the study, performance can be enhanced by reducing maldistribution by header geometry changes. Figure 18 illustrates the two header types that were taken into consideration. The velocities of their entering flows varied from 0.8373 mm/sec to 2.344 mm/sec. As header length increased, the homogeneity of the flow distribution improved; but, as flow rate boosted it declined. In the case of a conical header, the flow maldistribution decreased and the static pressure was approximately identical for all tubes as the header length went to 1500mm. The results revealed that, in comparison to a cylindrical header, a conical header lowers flow maldistribution.

Nasir Hayat et al. looked at the application of Computational Fluid Dynamics (CFD) in heat exchanger applications [37]. The simulations included a variety of turbulence models from general-purpose commercial CFD applications, comprising standard, realizable, k-ε, and others, as well as velocity-pressure coupling schemes like SIMPLE and SIMPLEC.

The fact that the majority of the solutions from these simulations were within the permitted range demonstrated that CFD is a useful technique for forecasting the performance and behavior of a broad range of heat exchangers. They discovered that the CFD was used to several research domains, such as fluid flow maldistribution. These simulations have produced solutions that are mainly accurate and hence CFD can be used in a number of heat exchanger applications.

Mayur et al. [38] assessed basic geometrical shapes as headers for consistent distribution of liquid flow utilizing computational fluid dynamics (CFD) simulations. Different headers like the spherical, conical, pyramidal, cylindrical and rectangular headers were assessed, as seen in Figure 19. By contrasting numerical simulation predictions with experimental findings for a cylindrical header, the study's computational methodology has been verified. It has been investigated how flow distribution is affected by header volume, flow rate, outlet diameter, and pressure imbalance at outlets.

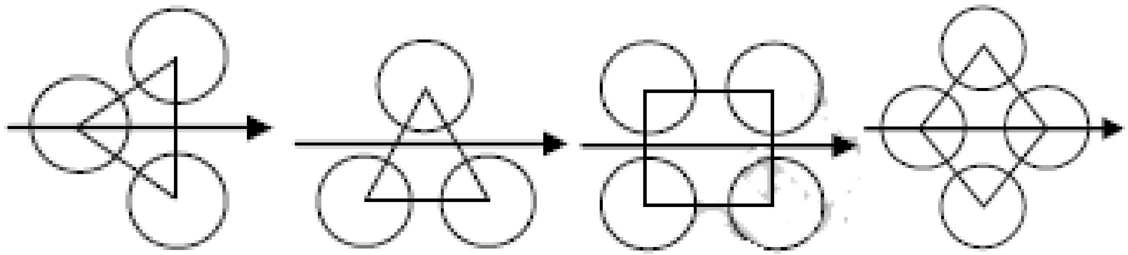
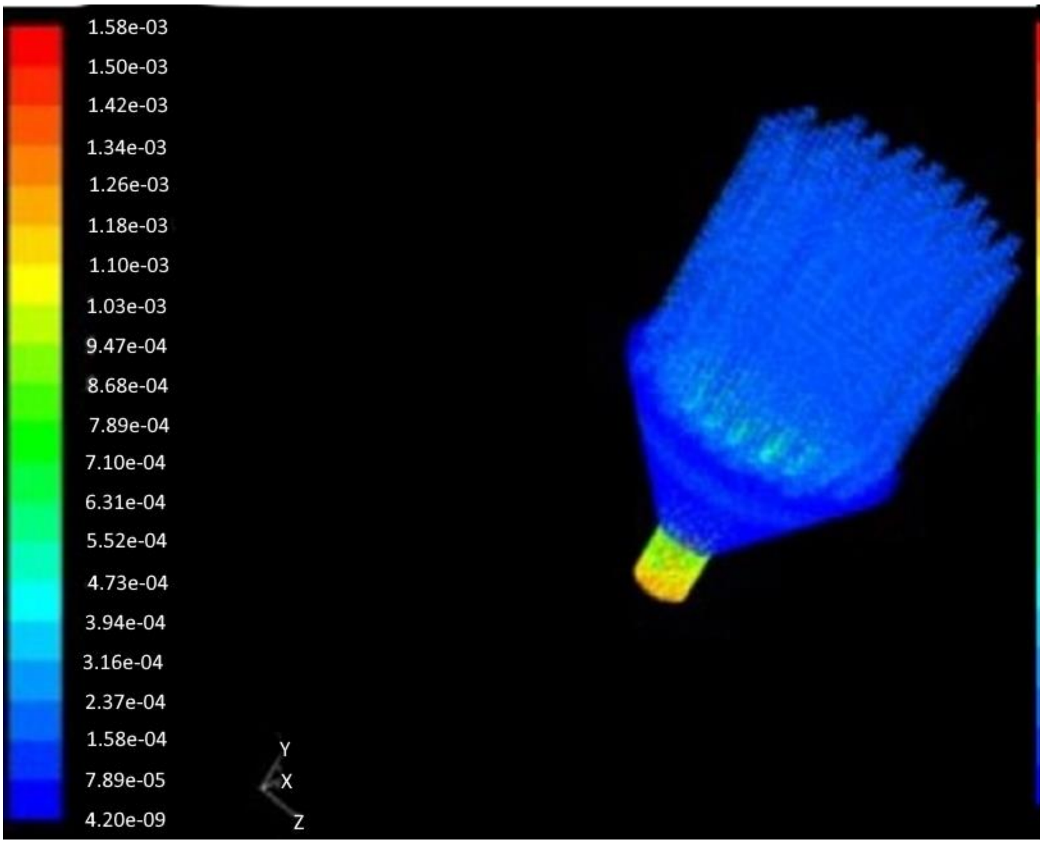


Figure 17. Different tubes arrangements [35].



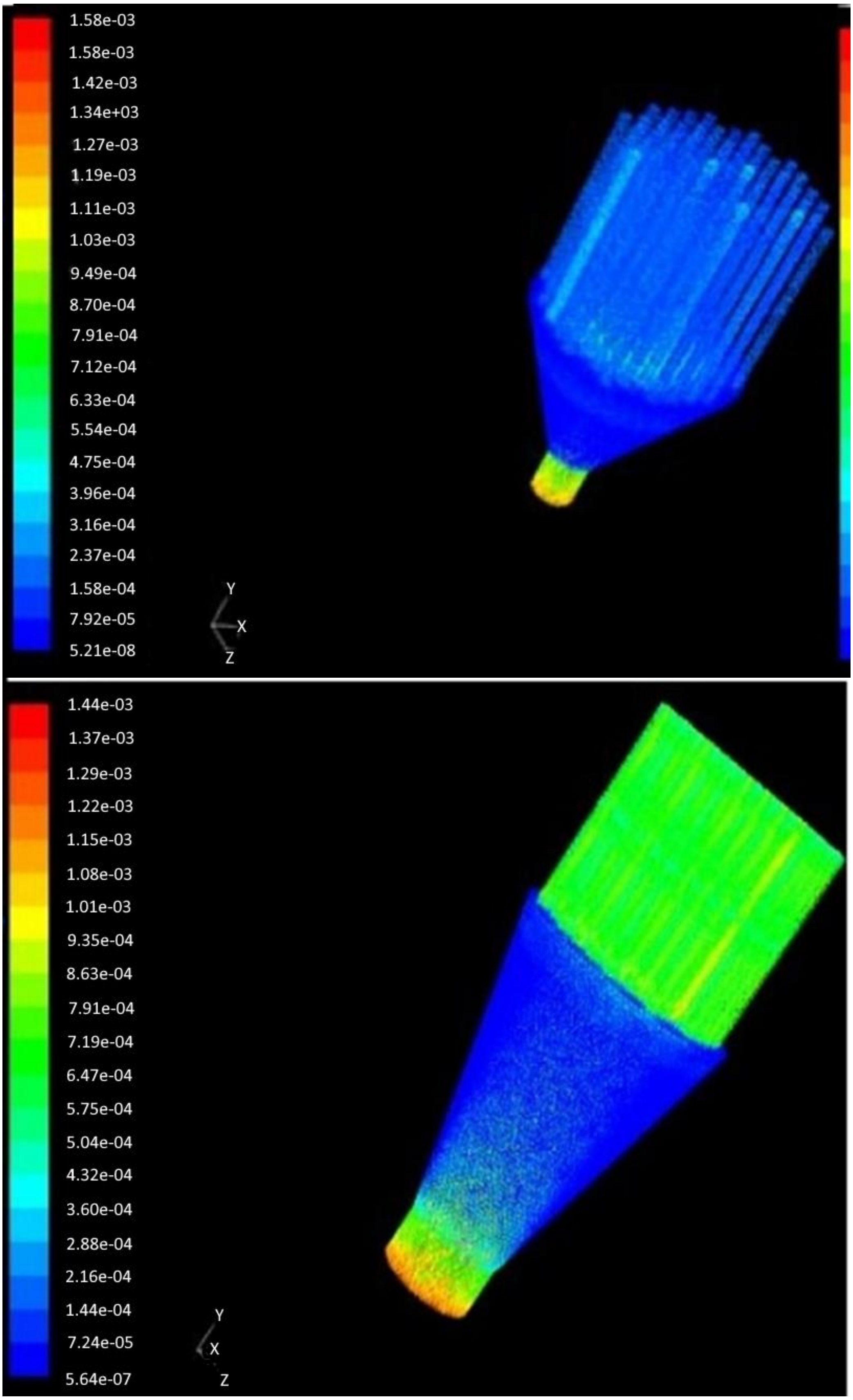


Figure 18. Velocity Distribution inside the conical header with flow rate of Re = 1500. *(*a) 700mm header length, (b) 900mm header length, (c) 1500mm header length [36].

The performance of the heads of various geometric shapes does not differ much at low flow rates. The spherical header is shown to be superior at larger flow rates. It was pointed out that having outlets with different cross-sectional areas might guarantee an even dispersion of flow. In Figure 20, the various outlets utilized for this purpose are shown.

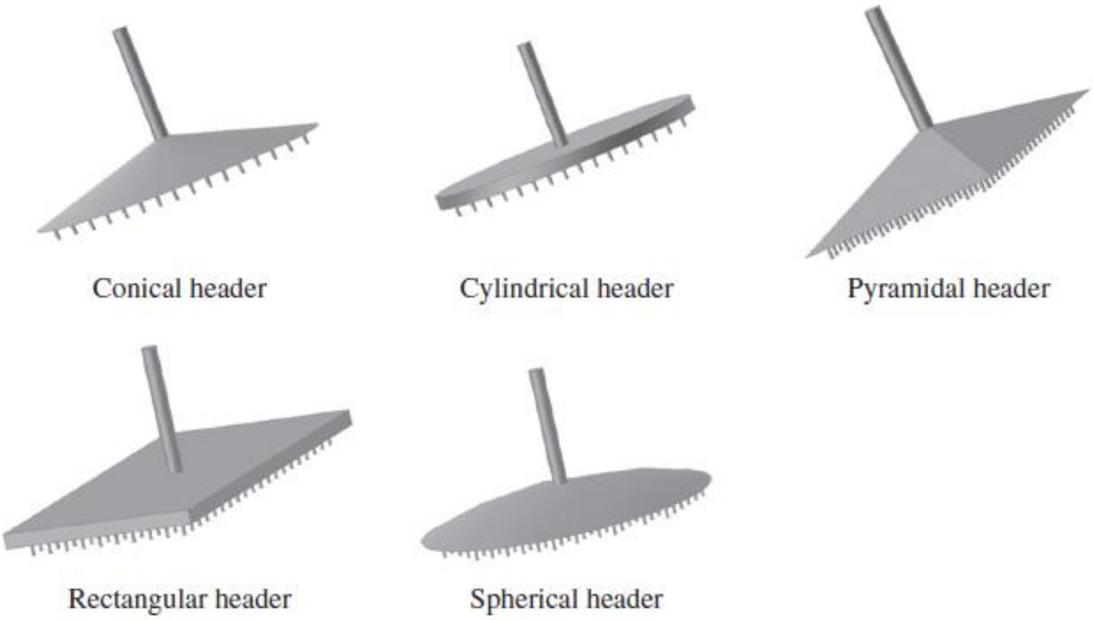


Figure 19. Different types of headers [38].

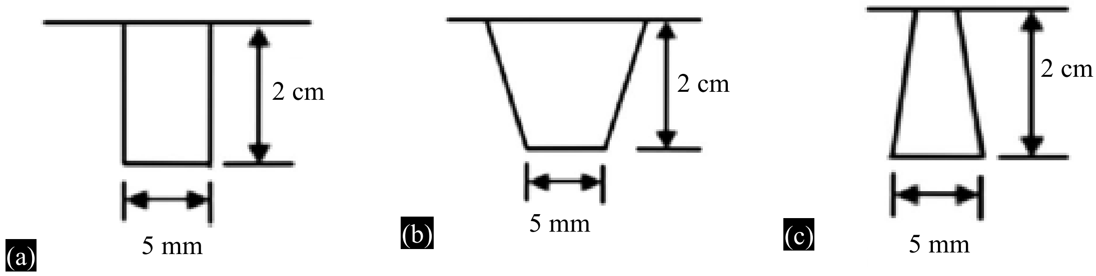


Figure 20. Changes to the outlets' shapes include (a) the original outlet, (b) an outlet that resembles an inverted cone's frustum, and (c) an outlet is similar to a cone's frustum [38].

The study's findings offer helpful information pertinent to header design issues. Figure 21 shows the percentage of flow non – uniformity decreased when cylindrical headers with modified outlet shape was used as compared to cylindrical header without outlet modification.

The use of polymers and composite materials in the heat exchangers have increased the heat transfer rate multiple times as compared to conventional materials. The use of PCM (phase change material)/graphite matrix has a significant effect in the time taken for melting and the phase change heat transfer. The heat transfer rate increased to about 35 % in using PCM/graphite matrix as compared to the use of pure paraffin. Also the time taken for melting was found to decrease by 92% as compared to the conventional case as stated by M.Y. Yazici et al. [39]

CONCLUSION

The methodologies used by various researchers in order to reduce the maldistribution were used and proven fruitful. The methods used include various geometrical modification like increasing the number of nozzles, modifying inlet diameter, use of baffle plates, modifying the inlet of the tubes with use of nozzle and orifices etc.

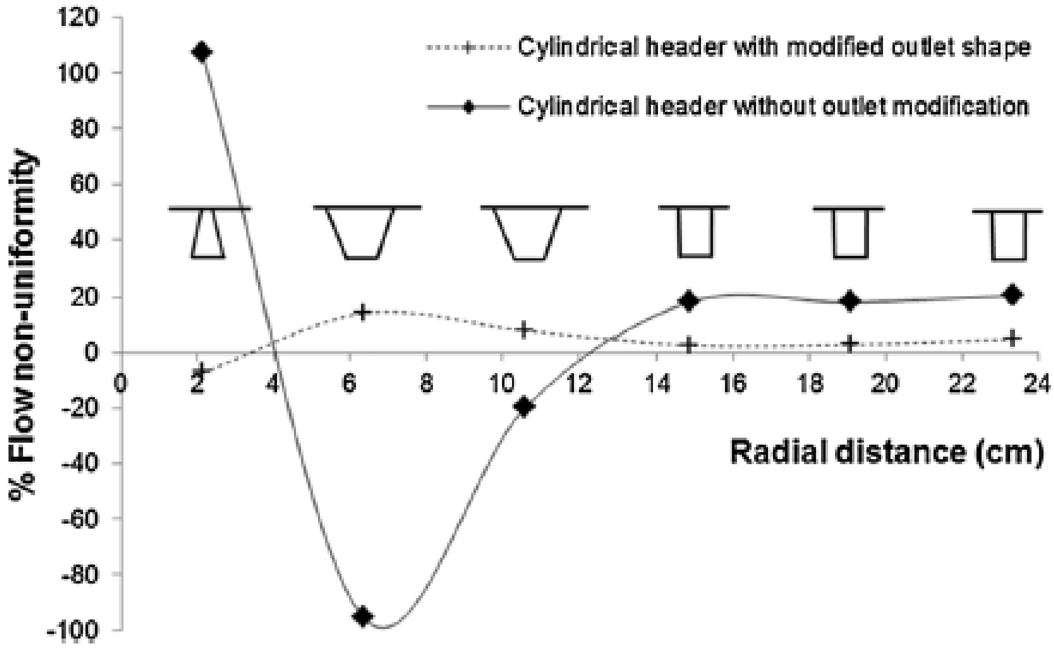


Figure 21. The original and modified cylindrical headers' flow distributions [38].

The use of composite and polymeric materials also improves the heat transfer characteristics of heat exchanger. However it also leads to the opening of the future scope of research as the methods used as stated above were applicable to a specific geometrical configuration and for specific boundary conditions and conventional materials like copper and stainless steel. However the use and reliability of these methods different geometrical, boundary conditions and the effect of the material usage weren't done by researchers which is the possible scope of research for future innovators.

Declaration of Interest

Regarding the publishing of this work, the authors confirm that they have no problems of interest.

REFERENCES

1. Master BI, Chunangad KS, Boxma A, et al. Most frequently used heat exchangers from pioneering research to worldwide applications. Heat Transf Eng. 2006;27(6):4–11. doi:10.1080/01457630600671960.
2. Brogan R. Shell and tube heat exchangers. Inc; 2013.
3. Peters MS. In: Timmerhaus KD, West RE, editors. *Plant design and economics for chemical engineers*. 5th ed. Boston: McGraw-Hill; 2004.
4. Tubular Exchanger Manufacturers Association. *Standards of The Tubular Exchanger Manufacturers Association*. 9th ed. Tarrytown, NY: TEMA; 2007.
5. Verein Deutscher Ingenieure, editor. *VDI Heat Atlas*. 2nd ed. Berlin Heidelberg: Springer-Verlag; 2010.
6. Mueller AC, Chiou JP. Review of various types of flow maldistribution in heat exchangers. Heat Transf Eng. 1988;9(2):36–50. doi:10.1080/01457638808939664.
7. Grijspeerdt K, Hazarika B, Vucinic D. Application of computational fluid dynamics to model the hydrodynamics of plate heat exchangers for milk processing. J Food Eng. 2003;57(3):237–42. doi:10.1016/S0260-8774(02)00303-5.
8. Kim MI, Lee Y, Kim BW, et al. CFD modeling of shell-and-tube heat exchanger header for uniform distribution among tubes. Korean J Chem Eng. 2009;26(2):359–63. doi:10.1007/s11814-009-0060-7.
9. Lee JK, Lee SY. Distribution of two-phase annular flow at header–channel junctions. Exp Therm Fluid Sci. 2004;28:217–22. doi:10.1016/S0894-1777(03)00042-6.

10. Mohan G, Rao BP, Das SK, Pandiyan S, Rajalakshmi N, Dhathathreyan K. Analysis of flow maldistribution of fuel and oxidant in a PEMFC. J Energy Resour Technol. 2004;126(4):262–70. doi:10.1115/1.1789519.
11. Schab R, Kutschabsky A, Unz S, et al. Numerical investigation of tubeside maldistribution in shell-and-tube heat exchangers. Heat Mass Transf. 2023. doi:10.1007/s00231-023-03385-5.
12. Bejan A, Kraus AD. *Heat Transfer Handbook*. Hoboken: John Wiley & Sons; 2003.
13. Kitto JB Jr, Robertson JM. Effects of maldistribution of flow on heat transfer equipment performance. Heat Transf Eng. 1989;10(1):18–25. doi:10.1080/01457638908939688.
14. Shah RK, Sekulic DP. *Fundamentals of Heat Exchanger Design*. Hoboken: John Wiley & Sons; 2003.
15. Hewitt GF. *Handbook of Heat Exchanger Design*. New York: Begell House Inc.; 2008.
16. Kern DQ. *Heat Transfer*. New York: McGraw-Hill Book Company; 1965.
17. Łopata S, Ocłoń P. Numerical study of the effect of fouling on local heat transfer conditions in a high-temperature fin-and-tube heat exchanger. Energy. 2015;92:100–16. doi:10.1016/j.energy.2015.03.048.
18. Marzouk SA, Abou Al-Sood MM, El-Said EM, Younes MM, El-Fakharany MK. A comprehensive review of methods of heat transfer enhancement in shell and tube heat exchangers. J Therm Anal Calorim. 2023;148(15):7539–78. doi:10.1007/s10973-023-12265-3.
19. Zhang M, Yang CM, Li K, Nawaz K. Reducing the flow maldistribution in heat exchangers through a novel polymer manifold: numerical evaluation. Energies. 2023;16(20):7120. doi:10.3390/en16207120.
20. Ligus G, Wasilewski M, Kołodziej S, Zając D. CFD and PIV investigation of a liquid flow maldistribution across a tube bundle in the shell-and-tube heat exchanger with segmental baffles. Energies. 2020;13(19):5150. doi:10.3390/en13195150.
21. Schab R, Dorau T, Unz S, Beckmann M. Parameter study of geometrically induced flow maldistribution in shell and tube heat exchangers. J Therm Sci Eng Appl. 2023;14(10):101002. doi:10.1115/1.4053633.
22. Slimene MB, Poncet S, Bessrour J, Kallel F. Numerical investigation of the flow dynamics and heat transfer in a rectangular shell-and-tube heat exchanger. Case Stud Therm Eng. 2022;32:101873. doi:10.1016/j.csite.2022.101873.
23. Saffarian MR, Fazelpour F, Sham M. Numerical study of shell and tube heat exchanger with different cross-section tubes and combined tubes. Int J Energy Environ Eng. 2019;10:33–46. doi:10.1007/s40095-019-0297-9.
24. Ravinthiran A, Babu T, Raj GT, Siddaarth PY, Kamatchi KS. Flow variations and comparisons of heat exchanger headers with different cross sections. In: *AIP Conf Proc*. 2020;2283(1). doi:10.1063/5.0029256.
25. Chukwudi BC, Ogunedo MB. Design and construction of a shell and tube heat exchanger. J Mech Eng Elixir. 2018;118.
26. Kim MI, Lee Y, Kim BW, Lee DH, Song WS. CFD modeling of shell-and-tube heat exchanger header for uniform distribution among tubes. Korean J Chem Eng. 2009;26:359–63.
27. Said SAM, Ben-Mansour R, Habib MA, Siddiqui MU. Reducing the flow mal-distribution in a heat exchanger. Comput Fluids. 2015;107:1–10. doi:10.1016/j.compfluid.2014.09.012.
28. Ravinthiran A, Sankar N, Rajan PV, Dutt P. A comparative study and flow analysis of multiple branch pipe flow header used in tube heat exchangers. Indian J Sci Technol. 2016. doi:10.17485/ijst/2016/v9i48/108486.
29. Osakabe M, Hamada T, Horiki S. Water flow distribution in horizontal header contaminated with bubbles. Int J Multiph Flow. 1999;25(5):827–40.
30. Wang K, Tu XC, Bae CH, Kim HB. Optimal design of porous baffle to improve the flow distribution in the tube-side inlet of a shell and tube heat exchanger. Int J Heat Mass Transf. 2014;80:865–72. doi:10.1016/j.ijheatmasstransfer.2014.09.076.

31. Habib MA, Ben-Mansour R, Said SAM, Al-Qahtani MS, Al-Bagawi JJ, Al-Mansour KM. Evaluation of flow maldistribution in air-cooled heat exchangers. Comput Fluids. 2009;38(3):677–90. doi:10.1016/j.compfluid.2008.07.004.
32. Kuchi G, Ponyavin V, Chen Y, Sherman S, Hechanova AE. Flow distribution on the tube side of a high temperature heat exchanger and chemical decomposer. In: *Proc ASME Int Mech Eng Congr Expo*. 2007;8:903–10. doi:10.1115/IMECE2007-42652.
33. Zhang Z, Li Y. CFD simulation on inlet configuration of plate-fin heat exchangers. Cryogenics. 2003;43(12):673–8. doi:10.1016/S0011-2275(03)00179-6.
34. Mohammadi K, Malayeri MR. Parametric study of gross flow maldistribution in a single-pass shell and tube heat exchanger in turbulent regime. Int J Heat Fluid Flow. 2013;44:14–27. doi:10.1016/j.ijheatfluidflow.2013.02.010.
35. Labbadlia O, Laribi B, Chetti B, Hendrick P. Numerical study of the influence of tube arrangement on the flow distribution in the header of shell and tube heat exchangers. Appl Therm Eng. 2017;126:315–21. doi:10.1007/s40095-019-0297-9.
36. Paruchuri R, Ravigururajan TS, Muley A. Comparative CFD analysis of maldistribution in a shell and tube heat exchanger with conical and cylindrical headers. In: *Proc ASME Int Mech Eng Congr Expo*. 2010;5:143–50. doi:10.1115/IMECE2010-40946.
37. Bhutta MMA, Hayat N, Bashir MH, Khan AR, Ahmad KN, Khan S. CFD applications in various heat exchangers design: a review. Appl Therm Eng. 2012;32:1–12. doi:10.1016/j.applthermaleng.2011.09.001.
38. Darekar M, Singh KK, Shenoy KT, Kundu GK, Rao H, Ghosh S. Numerical simulations to evaluate basic geometrical shapes as headers for equal liquid flow distribution. Asia-Pac J Chem Eng. 2014;9:707–17. doi:10.1002/apj.1814.
39. Yazici MY, Saglam M, Aydin O, Avci M. Thermal energy storage performance of PCM/graphite matrix composite in a tube-in-shell geometry. Therm Sci Eng Prog. 2021;23:100915. doi:10.1016/j.tsep.2021.100915.

