WCRG Global Research
WORLD COMET RESEARCH GROUP

International Journal of Modern Research in Science & Engineering
Vol. 2, No. 1, 2019
Doi: 10.67228/30716357/IJMRSE-2019PI3C4Q
PP. 01-15

Original Article

CFD Analysis of Heat Transfer in Compact Heat Exchangers

Dr. Priya Natarajan

Associate Professor, University of Madras, India.

Received: 02-02-2019 Revised: 02-20-2019 Accepted: 02-25-2019 Published: 03-01-2019

ABSTRACT

Compact heat exchangers (CHEs) are widely used in modern thermal systems due to their high heat transfer efficiency, compact size, and reduced material usage. They find applications in aerospace, automotive, power generation, chemical processing, and electronics cooling. With the growing demand for energy-efficient and lightweight systems, advanced designs such as plate-fin, microchannel, and printed circuit heat exchangers have gained attention. This study presents a CFD-based analysis of heat transfer and fluid flow in compact heat exchangers. Using the finite volume method and RANS turbulence models, the research evaluates key parameters such as temperature distribution, velocity fields, pressure drop, and thermal performance under varying operating conditions. Parametric studies examine the influence of Reynolds number, inlet temperature, flow arrangement, and surface enhancements. The results highlight the trade-off between heat transfer efficiency and pressure loss, emphasizing the importance of design optimization. Overall, the study demonstrates that CFD is a powerful tool for improving the performance of compact heat exchangers while reducing development cost and time.

KEYWORDS

Compact Heat Exchanger, Computational Fluid Dynamics, Heat Transfer Enhancement, Pressure Drop, Turbulence Modeling, Thermal Performance.

ISSN: 3071-6357
PP: 01-15.

1. INTRODUCTION

1.1. Background

Heat exchangers comprise a key element in the diverse flow that overall overheat and energy apparatus as they empower the efficient exchange of heat in between liquids with varied temperatures without immediate blendage. They are essential in their use in power generation applications, refrigeration applications, air conditioning applications, chemical processing applications, automotive systems, and renewable energy technologies. Among the host of the heat exchanger designs, traditional shell-and-tube designs have been widely utilized because they are mechanically sound, serviceable and have clearly defined design practices. These conventional heat exchangers are however rather large and massive and hence cannot fit in the modern use where the space, weight and material use are most limited. In addition, high thermal efficiency of shell-and-tube heat exchangers is usually considered a feature of large surface areas thus adding to their size and expenditure. Compact heat exchangers have been used as a viable alternative to such limitations. They are defined by having a very high surface to volume ratio which can exceed 700 m 2 / m 3 that enables a high heat transfer to be attained to be at a much lower volume. This miniature size them especially appealing to be used in applications where size and weight are the most important factors. Small scale heat exchangers are often fitted with enhanced surfaces, e.g. fins or microchannels, to increase the area of heat transfer and fluid mixing, enhancing thermal performance. Because of this usage, they are commonly applied in aerospace systems, automobile heat recovery systems, cryogenics, electronics cooling systems, and high performance energy systems. The increasing trend in the world towards energy conservation and reduction of emissions has only increased the interest on the compact heat exchanger technology. Enhanced thermal efficiency is directly proportional to less fuel usage, less emission of greenhouse gases and better performance of the whole system. Moreover, the drive toward system miniaturization, particularly in electronics and aerospace domains, has posed a high thermal management gadgets as well as the ability to manage great heat consumption whilst sustaining decent pressure difficulties. The excessive pressure losses may lead to substantial pumping power consumption and system efficiency decrease and it is of essence to come up with an ideal balance between heat transfer enhancement and the hydraulic performance. These extremely rigorous and occasionally competing demands require high design and analysis methods than the old empirical methods. Computational Fluid Dynamics (CFD) is thus a very useful tool in the study of the behavior of flow and heat transfer in small scale heat exchangers. CFD facilitates in-depth understanding of the local thermal-hydraulic processes and promotes informed decision making in the design process and it also encourages carrying out of the same research with regard to the CFD analysis and optimization of compact heat exchangers.

1.2. Role of CFD in Heat Exchanger Analysis

Computational Fluid Dynamics (CFD) is now regarded as an essential in the process of analysing, designing and optimising heat exchangers, especially compact heat exchangers with complicated geometries and operating conditions. In contrast to conventional design paradigms which are largely based on a set of correlating data and simple assumptions a CFD can offer a comprehensive, physics-based description of the phenomena of fluid flow and heat transfer. The use of CFD in the heat exchanger analysis can be argued in the following aspects.

ISSN: 3071-6357
PP: 01-15.

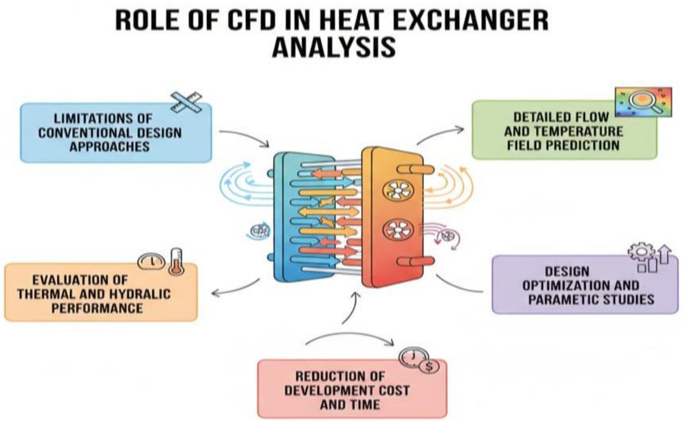

Fig 1 - Role Of CFD In Heat Exchanger Analysis

1.2.1. Limitations of Conventional Design Approaches

The design methods partly rely on one-dimensional analyses and experimentally validated correlations of the heat transfer coefficients and friction factors, to establish the conventional design methods of the heat exchanger. Although these techniques are useful in normal geometries and operating conditions, they do not always convect the localized flow behavior, non-uniformity in temperature field and complicated interplays between fluid flow and surface geometry. In small heat exchangers, in which small hydraulic diameters, high-performance surfaces, and even strong interaction between thermal and hydraulic characteristics are found, these simplifications result in erroneous predictions and even conservative designs.

1.2.2. Detailed Flow and Temperature Field Prediction

CFD can be applied to the solution of three-dimensional velocity, pressure and temperature of the flow in the passages of heat exchanger. Such would be valuable information that can help a designer determine areas of flow separation, recirculation, hot spots, and poor thermal use which cannot be determined through experimental data or modeling. Using the visualization of these local phenomena, CFD would assist in the understanding of the mechanisms of the underlying heat transfer enhancement and pressure losses in small-scale heat exchangers.

1.2.3. Evaluation of Thermal and Hydraulic Performance

Another significant strength related to the CFD is that it is able to assess thermal and hydraulic performance simultaneously. Such parameters as Nusselt number, heat transfer coefficient, pressure drop, friction factor and pumping power can be predicted correctly under the different operating conditions. Compact heat exchangers require this two-fold evaluation, as enhancement of heat transfer in them may be at the price of pressure losses. CFD offers a theoretical model of trade-off analysis to determine the best operating regimes.

1.2.4. Design Optimization and Parametric Studies

The parametric parameter studies that can be done using CFD have the advantage of being able to systematically vary geometric parameters, flow conditions and the material properties without requirement of expensive physical prototypes. Various channel shapes, fin arrangement, improvement

ISSN: 3071-6357
PP: 01-15.

of surface and flow arrangements can be considered effectively. In conjunction with optimization algorithms, CFD can be used to create high-performance designs of heat exchanger that is compact, thermal efficient, and hydraulically efficient.

1.2.5. Reduction of Development Cost and Time

With the reduction of reliance on trial and error experimentation, CFD saves much development time and cost. It enables screening of design at an early stage, quick performance assessment and finding of the possible design problems prior to fabrication. This has led to CFD becoming an inherent element of the contemporary research and industrial product development of heat exchangers, which prompted the selection and usage of the CFD as the main analysis tool in the current study.

1.3. CFD Analysis of Heat Transfer in Compact Heat Exchangers

Computational Fluid Dynamics (CFD) analysis is central to the knowledge and prediction of heat transfer dynamic in small scale heat exchainers where complicated flow systems and augmented surfaces have a severe impact on thermal functionality. The features of compact heat exchangers include small hydraulic diameter, high substitution-volume ratio of the surface area and volume and the proximity of the flow passages and consequently high degree of fluid flow and heat transfer coupling. CFD permits the parallel equation of mass, momentum, and energy which permits representative representation of convective and conductive heat vehicle transfer procedures in both the fluid and the solid phase. CFD through conjugate heat transfer modeling is able to reflect the interplay between fluid convection and heat conduction by using thin metallic walls resulting in a realistic representation of the real working conditions. The benefits of CFD analysis include being able to solve fine structures in temperature and velocity levels in exchanger channels. This enables one to determine both the formation of thermal boundary layer, temperature gradient and existence of local hot spots that have a direct influence on the efficiency of heat transfer. The application of CFD is also used to analyze the effect of the operating parameters which include Reynolds number, inlet temperature, and fluid properties on the rates of heat transfer. With increased flow velocity, CFD results are evident with increased mixing and boundary layer break up culminating into increased heat transfer coefficients and ultimate thermal performance. CFD finds special application in determining the efficiency of surface enhancement methods that are generally applied to small-scale heat exchangers, including fins, corrugations, and microchannels. When these complex geometries are modeled, CFD gives the understanding of the role of the contribution of secondary flows, division, and recirculation regions to the enhancement of heat transfer. Moreover, CFD can be used to compare, in a systematic way, baseline and modified designs and make a quantitative comparison of the increase in heat transfer versus the penalty on pressure drop. In main, CFD analysis provides an adaptable and all-inclusive model to investigate the heat transfer within smaller heat exchanger. It balances between simplified model analyses and expensive experimental studies and permits considered detail in performance evaluation, design enhancement and optimization of the design. Therefore, CFD has turned out to be a key to the further technological evolution of compact heat exchanging systems, and the further improvement of high-efficiency thermal systems.

ISSN: 3071-6357
PP: 01-15.

2. LITERATURE SURVEY

2.1. Evolution of Compact Heat Exchangers

The development of compact heat exchangers (CHEs) has been triggered mainly by the necessity of achieving a greater thermal efficiency in minimal volume and weight considerations. There was early plate-fin heat exchanger technological progress which had high surface-area-to-volume ratios and found wide use in cryogenic, aerospace and automotive applications. Convective heat transfer was generally improved with these designs using plain (or offset strip) or louvered fins which are compact. With increasing industrial needs to operate under higher operating pressure and temperatures, the constraints of traditional plate-fin exchangers were realized especially concerning mechanical integrity and risk of leakages. It saw the development of microchannel heat exchangers, with hydraulic diameters less than in the sub millimeter regime, which have much higher heat transfer coefficients because of the increased surface area and a reduced thermal resistance. In more recent times, printed circuit heat exchangers (PCHEs) have also entered the spotlight particularly in the nuclear industry, supercritical CO 2 industry, and chemical industries. PCHEs are also based on chemically etched microchannels welded together by diffusion welding, giving them outstanding structural integrity in harsh environments. Although these improvements have significantly enhanced thermal performance, they have also raised new concerns with regards to augmented pressure drop, mal-distribution of flow, susceptibility to fouling as well as complexity in manufacturing them, which still drives further research.

2.2. Experimental Investigations

The experimental studies have served a very vital purpose in the study of heat transfer and fluid flow behavior in compact heat exchangers. Many researches have been conducted to improve thermal conductivity by making changes to the surface such as louvered fins, offset strip fins, wavy fins, and corrugated or ribbed channels. These geometrical distortions encourage breaking up of the boundary layer, re connector flows, and the secondary flow so that the heat transfer coefficients are greater than with smooth channels. Many experiments to determine the rates of heat transfer, friction factors, and pressure drops have been established through the method of wind tunnel testing, liquid flow loops, infrared thermography, and pressure transducers. According to these experiments, researchers have come up with empirical correlations between Nusselt number and friction factor versus Reynolds number and geometric parameters. Nevertheless, inasmuch as experimental studies prove useful, they tend to be expensive, inflexible to configure and measure at microscale dimensions. Moreover, the derived correlations are generally only valid in the particular geometries, materials, and flow regimes in which the correlations are tested, and do not provide a general indication of applications in new compact heat exchanger designs.

2.3. CFD-Based Studies

Computational Fluid Dynamics (CFD) is coming out as an effective tool in order to supplement and in a few instances to substitute experimental studies in understanding compact heat exchanger studies. CFD can be used to show in great detail velocity, temperature, and pressure fields in complex geometries which are hard to experimentally reach. LDFD Models have been applied in the study of laminar, transitional and turbulent microchannel flow, plate-fin exchanger and printed circuit heat

ISSN: 3071-6357
PP: 01-15.

exchanger through the use of different turbulence models including k - e, k - o and SST models. These experiments have shown that with proper mesh resolution, near-wall treatment and boundary conditions, CFD predictions can be very similar to experimental results. The influence of fin geometry, channel shape, the aspect ratio and arrangement of flow on thermal-hydraulic performance has also been well researched using CFD. Also, CFD has been used to conduct parametric and sensitivity analysis and thus investigate the design variables in a systematic manner without involving a lot of physical prototyping. In spite of these benefits, CFD research tends to be very reliant on model assumptions, numerical schemes, as well as turbulence modeling fidelity especially in low Reynolds number flow regimes and transitional flow regimes typical of compact heat exchangers.

2.4. Identified Research Gaps

Even though much work has been done concerning experimental and numerical analysis of compact heat exchangers, there are a number of research gaps that are yet to be filled. A significant problem is the inability to accurately predict the performance of pressure drops and heat transfer under transitional flow regimes when either of the laminar flow regime model or the fully turbulent flow regime model fails to be able to do so. Other areas that need research are flow maldistribution in multi-channel design particularly when the inlet conditions are non-uniform. Moreover, there are numerous studies devoted to maximizing the heat transfer, and there are less studies that take a holistic approach in which they take thermal performance into account, the pressure drop, the pumping power, and the manufacturability. The combination of the multi-objective optimization methods with the CFD simulations remains a quite new field, especially in the case of more sophisticated geometries (PCHEs and sophisticated microchannel networks, etc.). Also, fouling and thermal stresses as well as long term operational reliability are usually not considered in numerical models. The technology of addressing these gaps is necessary in creating next-generation compact heat exchangers that have higher efficiency, strength, and industry-use capability.

3. METHODOLOGY

3.1. Geometry and Computational Domain

The geometry and the computational domain used to carry out the current study is set up to effectively represent representative unit cell of a compact heat exchanger and be computationally efficient. Small-scale heat exchangers are generally composed of a great number of repeated-periodic flow ways, which are divided by slim metallic walls that transfer heat between the hot and the cold fluid streams. In order to negotiate the crucial thermal and hydraulic processes without the need to model the full exchanger, a unit cell methodology is adopted, where one fluid flow channel and the adjacent solid exception are the object under and around computation. The method is popular in numerical analyses because it maintains the local flow and heat transfer properties with only a huge cut in the amount of computational resources needed. Both the fluid domain and solid domain are defined as is shown in a straight channel possessing a small hydraulic diameter, typical of a microchannel or plate-fin design, and the metallic wall respectively between channels respectively. To reduce the cost of calculation further, the domain lateral faces are subject to symmetry boundary conditions. These planes of symmetry presuppose a periodical repetition of the same flow passages in the transversal direction to do away with the need of explicitly modeling adjacent channels. At the

ISSN: 3071-6357
PP: 01-15.

fluid inlet, uniform velocity and temperature fields are imposed to give fully developed or controlled inlet conditions and the outlet is imposed with a pressure boundary condition that lets the flow naturally develop. Conjugate heat transfer conditions are used in the solid fluid interfaces so that temperature and heat flux are continuous through the wall. This will allow sufficient prediction of both heat conduction in the solid wall and convection in the fluid domain. The computational domain length is chosen in order to provide an adequate growth of thermal and hydrodynamic boundary layers in the flow direction. Special attention is paid to the geometric parameters, including channel width, channel height, wall thickness, and aspect ratio, which have a direct impact on the enhancement of heat transfer and the pressure drop. In general, the selected geometry and computational domain are found to give an acceptable trade-off between physical realism and numerical efficiency and will be useful in the high-resolution CFD-based thermal-hydraulic modeling of compact heat exchangers.

3.2. Governing Equations

The basic equations of conservation, i.e., the mass, momentum, and energy equations, would determine the thermal fluid behaviour of the compact heat exchanger. They are equations that say how fluid flow and heat transfer is transported in the computational domain, and make the basis of the CFD analysis.

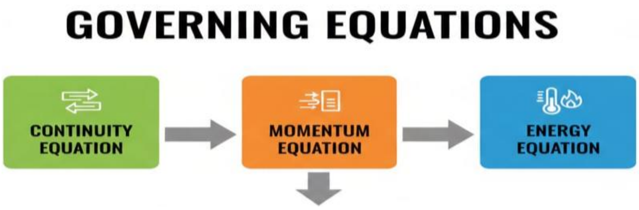

Fig 2 - Governing Equations

3.2.1. Continuity Equation

The continuity equation is the law of mass conservation. In the case of incompressible flow which is usually steady, and in which the heat exchanger typically takes the form of compact units, then this equation makes it such that the mass flow rate entering the domain is the same as the mass flow rate leaving the domain. Physically, it means there is neither any build up or drainage of mass in the fluid regime. The mass flux divergence therefore equals zero meaning that density of fluid is constant and flow field is spatially continuous all along the channel.

3.2.2. Momentum Equation

The equation of momentum is based on Newton second law and it controls the flow of the fluid in the heat exchanger channels. It takes into consideration the balance between the forces of inertia, pressure and viscous forces which act on the fluid. The convective term denotes the transfer of momentum by the movement of fluids whereas pressure gradient term helps in moving the flow through the channel. The viscous diffusion is a term that embodies the effects of fluid viscosity resistance to movement and increases the pressure loss at the direction of movement. The equation is important to predict the velocity distribution, wall shear stress and pressure drop that are important parameters to assess the hydraulic performance of compact heat exchangers.

ISSN: 3071-6357
PP: 01-15.

3.2.3. Energy Equation

The equation of energy explains the conservation of thermal power in the area of fluid. It takes into consideration both transport of heat through heat movement (convection) and heat diffusion through gradients in temperature (conduction). The product of fluid density and specific heat holds the capacity of the fluid to store heat whereas the thermal conduction term is used to govern the speed at which the fluid gains heat by diffusion. This equation allows to predict the temperature distributions in the channel and at the fluid-solid interface that is the foundation to assess the coefficients in terms of heat transfer and overall heat exchanger thermal efficiency. Collectively, the governing equations establish a full mathematical model of the simulation of overall systems of phenomena of fluid flow and heat transfer associated with compact heat exchangers.

3.3. Turbulence Modelling

In the case of turbulent flow in the compact heat exchanger channels, the standard k -ε turbulence model is used due to its strength, numerically stable, and relatively low-calculation cost. The turbulent flows are identified as random and chaotic variations in velocity that have a high magnitude in increasing momentum and heat transfer in comparison to the laminar flows. These fluctuations are directly solvable by direct numerical simulation, which is computationally prohibitive in practice in small-scale heat exchangers with small hydraulic diameter and narrow flow passages. As such, k-epsilon model of turbulence is extensively employed to give an average measure of the turbulence impact. The k- e model is founded on the resolution of two extra transport equations. The equation of state of the first order controls the turbulent kinetic energy that is the power possessed in velocity variations of the flow. This amount is an indication of the strength of turbulence and is caused by the gradients of velocities in the flow and dissipated by the forces of viscosity. The second equation is used to control the rate of dissipation of turbulence which is the rate at which turbulent kinetic energy is changed into thermal energy by the viscous action at smaller scales. A combination of these two variables leads to the calculation of the turbulent or eddy viscosity that amplifies the effective diffusion of the momentum and heat in the governing equations. Physically, the turbulent viscosity gives an explanation to the greater mixing brought about by the turbulent eddies in such a way that, the model has the capability of better forecasting higher rates of momentum and heat transfer than laminar flow formulations. The typical k -e model is based upon isotropic turbulence, and works effectively with fully-developed turbulent flows in internal channels, and is therefore useful in many internal compact heat exchanger applications with moderate and high Reynolds numbers. Wall functions are used to typically bridge the near-wall region without extreme fine mesh resolution and this also increases computational efficiency. Despite the fact that the standard k - - model has one or two weaknesses in its capacity to correctly describe flow separation and the transitional regimes, the balance of the factor between character and efficiency renders it an apt solution in the initial phase of design analysis and parametric investigation of the turbulent heat transfer in compact heat exchangers.

3.4. Boundary Conditions

Boundary conditions are necessary in order to recreate the fluid flow and heat transfer properties of the compact heat exchanger precisely. Physically realistic and well-established boundary conditions are used in the current research to guarantee the existence of numerical stability and

ISSN: 3071-6357
PP: 01-15.

significant prediction of thermal and hydraulic performance. On inlet boundary of the computational domain, a uniform velocity boundary condition is given. This means that the fluid will be introduced into the channel at a uniform velocity profile throughout the inlet cross-section, and this means that the flowing fluid is well controlled and pumped or blown by a pump or blower before it reaches the heat exchanger at the upstream side. The inlet velocity uniformity make the analysis much easier, but it still allows one to simply follow the behavior of the flow development, pressure drop, and heat transfer through the channel length. Besides velocity, an inlet temperature constraint is provided at the inlet boundary which has to be constant. The condition is when a fluid enters the heat exchanger at a fixed and constant thermal condition that is common in experimental and industrial application of heat exchangers. The thermal driving potential between the fluid and the solid wall can be definitively determined by correcting the inlet temperature which allows temperature increase or decrease along the flow direction to be accurately assessed and the heat transfer coefficients determined. On the outlet boundary, pressure outlet is applied. This boundary condition defines a reference static pressure, commonly atmospheric pressure or gauge zero pressure that can be used by the fluid to leave the computational domain naturally. The pressure outlet condition is especially appropriate to internal flow simulations as it allows the flow rate and the velocity profile to vary due to the upstream conditions and pressure losses inside the channel. The boundary condition no-slip is used in all solid walls of the fluid domain implying that the fluid velocity at the wall surface is zero because of the visuosity. In addition, there is a demand of conjugate heat transfer conditions on the fluid solid interfaces. This offers consistency in the temperature and heat flux across the wall; which permits model processes of both heat conduction in the solid material and convection in the fluid to be modeled simultaneously. Taken together, these boundary conditions offer a realistic simulation of the operating condition of small heat exchangers and allow confident thermal-hydraulic analysis by CFD.

3.5. Numerical Solution Procedure

The finite volume method is used to numerically solve the governing equations and is commonly used in computational fluid dynamics because it is said to have high conservation properties and three-dimensional complex geometries. The method embraces the use of a finite number of control volumes that discretize the computational domain upon which the mass, momentum, energy, and turbulence governing equations are then integrated over the control volume. This guarantees that mass, momentum, and energy are strictly conserved over the whole domain, and this is particularly essential to the correct prediction of heat transfer and pressure drop in miniature heat exchangers. In order to increase the accuracy of the solution, the second-order discretization schemes are used to discretize convective and diffusive terms spatially. Second-order schemes are much more resistant to numerical diffusion than first-order schemes and give a better representation of velocity and temperature gradients, particularly in the wake of solid walls where thermal and hydrodynamic boundary layers form. Normally, pressure-velocity interaction is addressed through the use of an iterative algorithm, which maintains uniform pressure-velocity interaction through the solution sequence. A steady-state, iterative solver has been used to get the solution. The discrete algebraic equations are solved at each iteration till the convergence is reached. Convergence is evaluated on the basis of the residual values as well as the physical monitoring parameters. The residuals, or the imbalance in the discrete form of the governing equations, must meet a high parameter of ten to the

ISSN: 3071-6357
PP: 01-15.

power of six requirements of continuity, momentum, energy and turbulence equations. This is so that the errors in the numbers are reduced to a minimum and the solution has achieved a stable point. Along with residual monitoring, such key integrals as the pressure drop, outlet mean temperatures, and the heat transfer rate are also monitored as the iterations proceed. It is verified that convergence is achieved when the studied parameters have an insignificant change as additional iterations are made. The solution procedure is also used to consider grid independence and numerical stability. Initial simulations are done using increasingly more refined meshes so that the results of the simulation do not depend on the grid resolution. On the whole, the solution procedure based on numbers adopted offers the sound and precise model of predicting the combined thermal and fluid flow behavior in small-scale heat exchanger designs.

3.6. Solution Flowchart

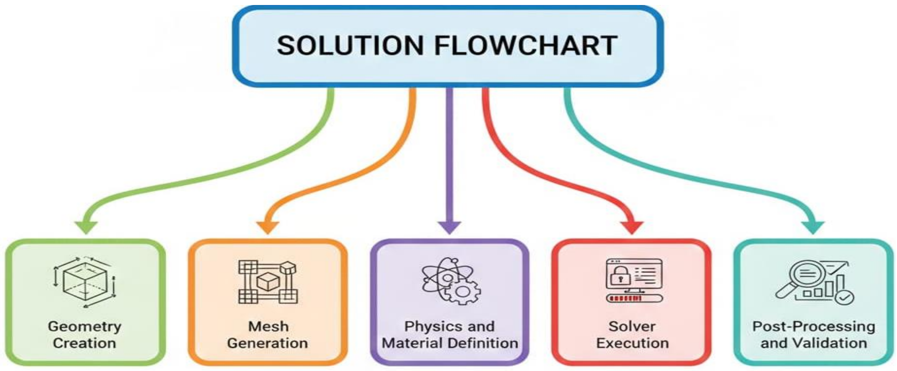

Fig 3 - Solution Flowchart

3.6.1. Geometry Creation

Creation Numerical solution process starts with the creation of geometry containing the physical setup of the compact heat exchanger consisting of a computer-aided design environment. Based on preset geometric parameters, (channel dimensions, wall thickness and length), a representative unit cell is built up of fluid passages and separating solid walls. Caution is exercised to make sure there is geometric accuracy and consistency with the assumptions of symmetry and periodicity that have been assumed in the study.

3.6.2. Mesh Generation

After finalizing the geometry, mesh generation (discretization) is carried out to solve the control volumes to a finite number. The structured mesh or a hybrid mesh is normally used to effectively reflect the movement and thermal gradients in the small channels. Near the solid walls the mesh resolution is required to be finer and in the core flow region the mesh resolution is coarser, implying that computational cost is reduced.

3.6.3. Physics and Material Definition

During this, pertinent physical models used in defining fluid flow and heat transfer are established. These are choice of the laminar or turbulent flow models, activation of the energy equation and specification of conjugate heat transfer. Both fluid and solid domains are given material properties

ISSN: 3071-6357
PP: 01-15.

in terms of density, viscosity, specific heat and thermal conductivity on the basis of operating conditions.

3.6.4. Boundary Condition Assignment

A proper set of boundary conditions is used on the computational domain to simulate realistic operating conditions. These are the specification of inlet velocity and temperature, outlet pressure, no-slip wall conditions and heat conduction at fluid, solid interfaces. Where possible symmetry conditions are also enforced to use less computation.

3.6.5. Solver Execution

The numerical solver is run once all the physical models and conditioning conditions have been established. The solver sequentially converts the governing equations i.e. equations of state, into the finite volume form and then solves these equations to convergence as indicated by the magnitude of the residues and variables that are monitored. The stability of a solution and its convergence behavior are continually observed at this point.

3.6.6. Post-Processing and Validation

The final step is post-processing of the converged solution to obtain meaningful solutions in form of velocity fields, temperature contours, pressure drop, and heat transfer coefficients. The analysis of these results to test the thermal and hydraulic performance of the compact heat exchanger is carried out. The accuracy and reliability of the results of the simulation is checked by comparison with the numerical predictions with the available experimental data, or by comparison with the existing literature correlations.

4. RESULTS AND DISCUSSION

4.1. Temperature Distribution

The obtained temperatures of the CFD simulations give significant knowledge regarding the thermal performance of the small size heat exchanger. It shows smooth and close temperature gradient in the direction of flow, and proves that the process of heat transfer between the fluid mass and the solid wall takes place effectively in the entire length of the channel. The steepness of the temperature gradient at the region of inlet is relatively large because of the significant difference of the temperature of both the fluid entering it, and the heated or cooled wall surface. The temperature fluctuates with the fluid as it travels downstream since the transfer of thermal energy is continuous resulting in a more advanced thermal field. This mode of behavior implies that the geometry and the boundary conditions applied in the selected geometries facilitate efficient and stable heat transfer without localized and hot spots or heat dead zones. Of special importance is the effect of Reynolds number on the distribution of temperature. At reduced Reynolds numbers, the flow is loosely mixed, causing thicker thermal boundary layers and comparatively low rates of convective heat transfer. Therefore, the changes of temperature within the channel are less abrupt, and the fluid might not make full use of available heat transfer surface. As the Reynolds number on the other hand increases, more fluid inertia and turbulence intensity translates to more mixing in the channel. This enhanced mixing can break up the thermal boundary layer close to the wall, decrease thermal resistance and also enhance more even

ISSN: 3071-6357
PP: 01-15.

distribution of temperature throughout the channel cross-section. Consequently, Reynolds numbers increase so do heat transfer coefficients and intensity in the temperature changes across the flow direction. Also, the CFD results indicate that the contours of temperature are uniform along the width of the channel, and supports the inclusion of symmetry boundary conditions in the computational domain. This was also a symptom of numerical stability and convergence of the solution since there were no sudden temperature changes. Comprehensively, the given temperature field distribution is also validated regarding the fact that with the increase in the Reynolds number, the efficiency of convectional heat transfer is notably high, and it makes sense that flow processes and heat exchanger operation are closely interconnected within compact heat exchangers. The findings have been valid in the literature regarding the heat transfer theory and give assurance on the quality of the numerical model in conducting parametric and optimization studies.

4.2. Pressure Drop Characteristics

The features of pressure drops, which are based on the CFD results give important information on the hydraulic performance of the compact heat exchanger. The findings indicate that pressure drop will rise nonlinearly towards an increase in flow velocity, which is congruent with the foundational theory of fluid flow (in internal channels). Viscous effects dominate at lower velocities and the pressure drop is nearly proportional to the flow rate as is the case with laminar flow resistance. Inertial effects increase with flow velocity and as the flow approaches the transitional or the turbent regimes, the pressure losses sharply increase with channel length. This nonlinear nature maintains the high optimism of the pumping power requirements on the nature of operation. Surface enhancement procedures which may include the use of corrugations or micro-scale surface shows and/or fins also increase the pressure drop behavior. Although these surface modifications are very successful in interrupting the surface layer, and improving the heat transfer between the flow, it makes flows more resistant by adding more wetted surfaces and geometric impediments in the flow. The CFD findings reveal topographical surfaces lead to an increase in wall shear stress and development of secondary flows and recirculation areas that adds to higher losses in friction. Consequently pressure drop is much larger than with smooth channel configurations at the same Reynolds number. A trade-off between thermal enhancement and hydraulic penalty is also seen clearly based on the simulation. Even though a flow velocity increase and surface modifications result in better heat transfer performance, they also cause an increment in pressure drop and hence more pumping power is needed. This interaction explains why both thermal and hydraulic requirements have to be taken into consideration when developing compact heat exchangers. The contours of pressures generated in the CFD analysis reveal that the pressure decreases in a smooth and monotonic fashion on the direction of flow, which indicates stable flow action and numerical convergence. Generally the pressure drop properties underscore the need to design excellently balanced heat transfer augmentation with tolerable frictional losses to determine an energy efficient operation of the heat exchanger.

4.3. Heat Transfer Performance

The Nusselt number of the compact heat exchanger is used as indicative of its heat transfer between the conductive and convective heat transfer during the calculation of the ratio of the conductive heat transfer and the convective heat transfer, based on the fluid and solid layers. The CFD outcomes reveal that one of the key factors in determining thermal performance is the Nusselt number that is largely reliant on the Reynolds number, which means that the flow dynamics are primarily influential. There are also the weaker mixing and thicker thermal boundary layers and consequent lower Nusselt numbers at lower Reynolds number. With an increase in Reynolds number, fluid inertia and turbulence intensity increase and this gives a greater disruption of the thermal boundary layer, hence, sharply raising the rate of convective heat transfer. The trend is made in line with classical theory of heat transfer in internal flows. The geometry of the surface also has very significant impact on the heat transfer performance. The irregularities in the surface, like fins, corrugations or micro-scale structures, enhance the separation of flows, reattachment or both which enhances local heat transfer coefficients. It can be seen with the CFD simulations that the mechanisms can be effectively used by optimized surface geometries to promote heat transfer without excessive thermal resistance. The optimized designs over the baseline smooth-channel designs exhibit an increase in the average Nusselt number between about 25 and 40 percent, depending on the operating Reynolds number, and on geometric parameters. Such improvement is the result of better exploitation of the heat transfer area availed and more homogeneous temperature across the flow passages. The findings, however, also show that the extent of transfer heat increase also decreases after a Reynolds number, and this implies the occurrence of diminishing returns at extremely high flow rates. This trend is explained by the fact that a further increase in the turbulence intensity, though, can only make only slight improvements in heat transfer once the thermal boundary layer disruption is already significant. The spatial distribution of the Nusselt number between the channel length is higher towards it inlet region which is a strong thermal gradient region before gradually stabilizing further downstream. Altogether, the analysis of the heat transfer performance and optimization of the Reynolds number and surface geometry allow showing that in the compact heat exchanger designs, significant thermal improvements may be obtained with an acceptable hydraulic performance when careful attention is paid to the optimization of the Reynolds number and the surface geometry.

4.4. Model Validation

Another important process in this research that enhances the accuracy and reliability of the undertaken CFD simulations is model validation. The numerical values obtained of the important thermal-hydraulic parameters such as Nusselt number and pressure drop are systematically established with the well-established experimental correlations and the literature published data of compact heat exchangers. The given comparative analysis enables one to evaluate the quality of the modelling of the real physical behaviour in terms of the adopted numerical models, boundary conditions, and solution procedures. The validation outcome shows that CFD forecast is in close-following behavior of the trends in experimental researches within the examined range of Reynolds numbers. On a quantitative basis, it is evidenced that the difference between the numerical forecast and experimental correlations is found to be within a range of error of ±8 percent in both parameters which are heat transfer and pressure drop. This type of consensus has been found to be usual with CFD-based studies of compact heat exchangers, since experimental measurements, material characteristics, and geometric tolerances are uncertain terms, intrinsically. The small differences may be explained by simplifications in the computational geometry, assumptions of similarity of inlet conditions and the constraints of the turbulence modeling, especially in transitional flow regimes. The uniformity in the verification of the CFD findings and experimental results proves the appropriateness of the chosen turbulence model, mesh resolution and numerical schemes. It also illustrates that the conjugate heat transfer strategy is useful in the way it captures the mutual interaction between the fluid and solid conduction phenomenon. Besides, quantitative comparison, qualitative agreement is also realized in terms of flow patterns, temperature distributions and pressure gradients, which also enhances the validity of the numerical model. In general, the validation exercise creates trust in the capability of the CFD framework used on this research to be predicted. This validated model can thus be depended on to make parametric analysis, performance comparisons of varying surface geometries and parametric optimization studies in the future in order to achieve better thermal efficiency as well as reduction of hydraulic penalties in small heat exchangers.

5. CONCLUSION

This paper shows clearly that the concept of Computational Fluid Dynamics (CFD) is a powerful tool when it comes to the analysis and optimization of compact heat exchangers. Through the development of a complex numerical structure, which accepts the use of proper governing equations, turbulence modeling, and boundary conditions, together with numerical solution processes, the current study gives a comprehensive understanding of the thermal and hydraulic interactions between the compact heat exchanger velocities. The CFD method can be used to visualize and quantify velocity fields, temperature distributions, pressure changes, and heat transfer features, and this can otherwise be challenging and expensive to measure experimentally alone. The quantitative data affirms the hypothesis that CFD can be used to predict the trends of basic performance such as enhancements of heat transfer and pressure drop behavior across a series of operating conditions with accuracy. The calculated and the standard experimentally known correlations of Nusselt number and pressure drop nearly match with each other, and deviations are acceptable. This justification explains the effectiveness of the taken numerical models and proves the fact that CFD can be safely adopted as a predictive design and analysis instrument of compact heat exchangers. Among the key observations that have been made on the basis of this study is the strong influence of surface enhancement techniques on the thermal performance. Adjusted surface geometries are effectively employed to interrupt thermal boundary layers, to enhance fluid mixing and to increase the convective heat transfer coefficients with significant improvements in heat transfer overall. These advantages come, however, at the price of a higher frictional loss, and higher pressure drop and pumping power demand. These findings emphasize that the thermal performance cannot be analyzed separately and always it is necessary to take into account in combination with hydraulic performance. The research paper underlines the need to attain the optimum ratio between heat loss gains and pressure loss in the development of miniature heat exchangers. Although an increase in Reynolds number and high-surface enhances the thermal capacity, high pressure drops eventually offset this by raising the cost of operation. Hence, optimization of design should be in a comprehensive manner, working together to include the thermal and hydraulic losses besides the total energy efficiency. It could be said that future research can further expand upon the current study and designate multi-objective optimization methods to find optimal geometric and operating parameters in a systematic manner. Also, transient thermal analysis was possible to investigate start-up behavior, dynamic operation, and effects of

ISSN: 3071-6357
PP: 01-15.

thermal fatigue. The combination of novel turbulence models and machine learning-based optimization systems is also one of potential paths of creating the next-generation compact heat exchangers with improved performance and reliability.

REFERENCES

[1] R. K. Shah and D. P. Sekulić, Fundamentals of Heat Exchanger Design, Hoboken, NJ, USA: Wiley, 2003.
[2] A. L. London and W. M. Kays, Compact Heat Exchangers, 3rd ed., New York, NY, USA: McGraw-Hill, 1984.
[3] S. Kakac, H. Liu, and A. Pramuanjaroenkij, Heat Exchangers: Selection, Rating, and Thermal Design, 3rd ed., Boca Raton, FL, USA: CRC Press, 2012.
[4] P. D. McDonald, "Printed circuit heat exchangers," Applied Thermal Engineering, vol. 20, no. 4, pp. 471–482, 2000.
[5] J. G. Kim and J. M. Lee, "Thermal–hydraulic performance of microchannel heat exchangers," International Journal of Heat and Mass Transfer, vol. 47, no. 6–7, pp. 1369–1381, 2004.
[6] B. Webb and N. Kim, Principles of Enhanced Heat Transfer, 2nd ed., New York, NY, USA: Taylor & Francis, 2005.
[7] M. Manglik and A. E. Bergles, "Heat transfer and pressure drop correlations for the rectangular offset strip fin compact heat exchanger," Experimental Thermal and Fluid Science, vol. 10, no. 2, pp. 171–180, 1995.
[8] T. A. Ameel, R. O. Warrington, R. A. Wegeng, and M. K. Drost, "Experimental investigation of turbulent flow in microchannels," International Journal of Heat and Mass Transfer, vol. 40, no. 5, pp. 1043–1054, 1997.
[9] S. G. Kandlikar, "High flux heat removal with microchannels—A roadmap of challenges and opportunities," Heat Transfer Engineering, vol. 26, no. 8, pp. 5–14, 2005.
[10] Y. Zhang, Z. Guo, and J. Li, "CFD simulation and experimental validation of plate-fin heat exchangers," Applied Thermal Engineering, vol. 31, no. 17–18, pp. 3378–3385, 2011.
[11] A. Bejan, Convection Heat Transfer, 4th ed., Hoboken, NJ, USA: Wiley, 2013.
[12] J. M. P. Quesada, L. Cheng, and G. Ribatski, "Flow boiling heat transfer in microchannels—A review," International Journal of Heat and Mass Transfer, vol. 55, no. 1–3, pp. 645–662, 2012.
[13] H. Peng, G. Ding, and Y. Hu, "Numerical investigation of flow and heat transfer characteristics in corrugated microchannels," Applied Thermal Engineering, vol. 29, no. 17–18, pp. 3402–3410, 2009.
[14] S. Patankar, Numerical Heat Transfer and Fluid Flow, Washington, DC, USA: Hemisphere Publishing, 1980.
[15] X. Li and A. Kleinstreuer, "Thermal performance and pressure drop correlations for microchannel heat sinks," International Journal of Heat and Mass Transfer, vol. 53, no. 23–24, pp. 5023–5034, 2010.