A numerical investigation on heat transfer enhancement and the flow characteristics in a new type plate heat exchanger using helical flow duct

Emad M.S. El-Said, Mohamed Abdulaziz and Mohamed M. Awad

Cogent Engineering (2017), 4: 1396638
A numerical investigation on heat transfer enhancement and the flow characteristics in a new type plate heat exchanger using helical flow duct

Emad M.S. El-Said, Mohamed Abdulaziz and Mohamed M. Awad

Abstract: In this paper, a numerical investigation on heat transfer enhancement and flow characteristics in a new type plate heat exchanger using helical flow duct in a series arrangement in counter flow of water as the test fluid is presented. Computational fluid dynamics package (ANSYS CFX 16.2) is used. Helical plate with different pitch ratios and different flow channel cross section aspect ratio were studied for variation of Reynolds numbers. The results show that the pressure drop is about 34.87–6,269.92 N·m⁻² for hot fluid and 15.06–3,379.42 N·m⁻² overall. The HPHE illustrates evidently high comprehensive performance; effectiveness reach about 67.56 for $Re_h = 2,000$ and $Re_c = 1,400$ and pitch ratio = 0.24 with significant increases in friction factor than other pitch ratios reach 0.51 and decreases to 0.22 for $Re_h = 10,000$ and $Re_c = 7,000$ and pitch ratio = 1.31. Smaller aspect (width-to-height) ratio can enhance heat transfer rate. The effectiveness increases from 17.58 to 67.56 with a reduction in aspect ratio from 1.31 to 0.24 respectively at the same hot and cold fluid simulation conditions.

ABOUT THE AUTHORS

Emad M.S. El-Said is working at Faculty of Engineering, Fayoum University, Fayoum, Egypt. He received his PhD from Tanta University, Egypt in 2014. He has published more than 18 papers in refereed journals and 13 papers in conference proceedings.

Mohamed Abdulaziz is working at SIMTRAN, Germany. He received his PhD from Institute of Ship Technology, University of Duisburg-Essen, Duisburg, Germany in 2017. His research is focused on the numerical simulation of complex systems.

Mohamed M. Awad is working at Faculty of Engineering, Mansoura University, Mansoura, Egypt. He received his PhD from Memorial University of Newfoundland in 2007. His research is focused on the modeling of complex fluid dynamics and heat transfer problems in internal flows. He is the author of 3 book chapters. He is a member of ASME. He was a recipient of the ASME IPTI Award in 2005 and 2006. He won silver medal at 45th International Exhibition of Inventions, Geneva, Swiss, 29 March–2 April 2017.

PUBLIC INTEREST STATEMENT

Heat exchangers performance plays vital role in several industrial applications. Because of low energy sources and high energy costs, there are several efforts to enhance heat exchangers’ efficiency. As a result, it is very important to determine the performance of heat exchange devices on both heat transfer and thermodynamic considerations. In this study, heat transfer enhancement and the flow structure in a new type plate heat exchanger using helical flow duct are investigated numerically. For this purpose, computational fluid dynamics (CFD) package (ANSYS CFX 16.2) is used. It is found that smaller aspect (width-to-height) ratio can enhance the heat transfer rate. The effectiveness increases from 17.58 to 67.56 with reduction in aspect ratio from 1.31 to 0.24 respectively at the same hot and cold fluid simulation conditions.
1. Introduction

Energy saving and recovery are major matters in our world, and a heat exchanger is very useful for energy saving (Tapre & Kaware, 2015). So, the engineering cognizance of the need to increase the heat exchangers’ thermal performance, thereby effecting energy, cost savings, material and as well as a consequential mitigation of environmental degradation, has led to the use and development of several heat transfer enhancement techniques. Various enhancement techniques are identified that can be classified broadly as passive and active techniques. The primary distinguishing feature is that unlike active methods, passive techniques do not require direct input of external power. They generally use surface or geometrical modifications to the flow channel, or incorporate an insert, material, or additional device like treated surfaces, rough surfaces, extended surfaces, swirl flow devices, displaced enhancement devices, coiled tubes, surface tension devices, additives for gases and additives for liquids. Except for extended surfaces, which increase the effective heat transfer surface area, these passive schemes promote higher heat transfer coefficients by altering or disturbing the existing flow behavior. This, however, is accompanied by the pressure drop increase. In the case of active techniques, the addition of external power essentially facilitates the desired flow modification and the concomitant improvement in the heat transfer rate. The use of two or more techniques (passive and/or active) in conjunction constitutes compound enhancement. The effectiveness of any of these methods depends strongly on the heat transfer mode (single-phase free or forced convection, pool boiling, forced convection boiling or condensation, and convective mass transfer), and kind and process application of the heat exchanger (Bejan & Kraus, 2003).

Plate fin heat exchangers are widely used in automobile, aerospace, cryogenic and chemical industries. The conventional design of plate-type heat exchangers are usually built of thin plates (all prime surfaces). The plates are either smooth or have some form of corrugation, and they are either flat or wound in an exchanger. Plate fin heat exchangers offer several advantages over competing designs; (1) high thermal effectiveness, (2) large heat transfer surface area per unit volume, (3) low weight, (4) multi-stream operation, and (5) true counter (Dewatwal, 2009; Mazumdar & Singh, 2007). But in the other hand, these exchanges cannot accommodate very high pressures, temperatures, or pressure and temperature differences, in addition to difficulty in cleaning of passages, which limits its application to clean and relatively non-corrosive fluids, and difficulty of repair in case of failure or leakage between passages. Therefore, a great number of academic attentions have been overcome the above-mentioned drawbacks of the conventional plate-type heat exchangers, a number of improved methods were proposed. Yin and Ooka (2015) defined the convective heat transfer coefficients of the plate and fin as independent parameters to obtain more precise results. They obtained the Colburn factor and Fanning factor according to the results of computational fluid dynamics (CFD). The researchers considered fin pitch, fin height, fin length, and fin thickness as four design parameters. They adopted a modified number of entropy production units ($N_s$). $N_s$ due to heat transfer ($N_{s,\text{h}}$), $N_s$ due to friction ($N_{s,\text{f}}$), and $N_{s,\text{total}}$ were considered as three objective functions. Finally, they obtained the optimal structural parameters of a water-to-water plate-fin heat exchanger applied to an air-conditioning system using genetic algorithm by single objective optimization and multi-objective optimization. The convective heat transfer coefficients values of the plate and fin were dependent on the proportions of the primary heat transfer surface area (plate) and secondary heat transfer surface area (fin). Villanueva and de Mello (2015) presented pressure drop and heat transfer correlations for finned plate ceramic heat exchangers evaluated using CFD simulations. The researchers used one adequate turbulence model to include transitional Reynolds number range. They included the geometrical parameters effect into the correlations, following the same approach commonly used for offset strip fins heat exchangers. They compared their CFD results to experiments for one particular geometrical configuration for validation purposes. They found that significant heat transfer enhancement produced by a horseshoe vortex formed in the frontal part of the
fins. Piper, Olenberg, Tran, and Kenig (2015) proposed a determination method for the hydraulic diameter, cross-sectional area and heat transfer area for pillow-plate heat exchangers (PPHE) geometries. Their method was based on forming simulations that accurately reproduced the inherently three-dimensional wavy pillow-plate channels by imitating the real manufacturing process of pillow plates. Then, they used the simulation results to develop simple expressions for the geometric design parameters. Peng and Ling (2008) investigated numerically and experimentally heat transfer characteristics and pressure drop over serrated fins in plate-fin heat exchangers (PFHE) at low Reynolds number. At the beginning, the researchers carried out a variety of performance tests of PFHE with a constant air flow rate and six various oil flow rates. The range of Reynolds number was from 10 to 200 at the oil side. They calculated the Colburn factor ($j$) and friction factor ($f$) from the experimental data. Then, they solved governing equations through two steps of increasing complexity to analyze numerically the 3D heat transfer and pressure drop of PFHE. They found that their experimental results agreed well with their corresponding numerical predictions.

There are many attempting to improve heat transfer process in various types of heat exchangers. For example, Zhang, He, and Tao (2009) carried out a 3D numerical simulation of a whole heat exchanger with middle-overlapped helical baffles by using commercial codes of GAMBIT 2.3 and FLEUNT 6.3. First, the researchers presented in detail the computational model and numerical method of the whole heat exchanger with middle-overlapped helical baffles, and adopted parallel computation mode for the simulation of a whole heat exchanger with six cycles of the middle-overlapped helical baffles of $40^\circ$ helical angle on a grid system of 13.5-million cells. Second, they performed the computational model validation by comparing the total pressure drop and average Nusselt number of the whole heat exchanger with experimental data. They obtained reasonably good agreement, and analyzed the reasons causing to the discrepancy. Then, they presented the shell-side fluid pressure and temperature fields of the whole area. At the end, they compared the cycle average Nusselt number of various cycle in the heat exchanger and found that periodic model for one cycle can be used to investigate the pressure drop and heat transfer characteristics for various heat exchanger to save computational source within the accuracy allowed in engineering computation. Eswaraiah and Sarada (2012) used CFD package (ANSYS CFX 11.0) to study numerically the heat transfer characteristics of oil cooler under laminar flow conditions with the variation of Reynolds number ($Re$) on tube side from 250 to 2,400. The working fluid on shell side was water and on tube side was ISOVG46 Turbinol. They did simulation for various oil flow rates. Their simulated results of friction factor and Nusselt number were in good agreement with the available experimental results and with the Sieder and Tate equation for plain tube. Inserts were used to enhance the heat transfer rates on tube side when the heat transfer rates for oil under laminar flow were very low. CFD investigations on the friction factor and Nusselt number of tubes equipped with louvered elliptical backward and forward strip inserts were carried out. They found that the strip inserts use led to higher heat transfer rates over the plain tube. The friction factor was higher for forward insert while the increase in Nusselt number and overall enhancement ratio were higher for backward insert. Mohammed, Hasan, and Wahid (2013) studied numerically the influence of using louvered strip inserts placed in a circular double pipe heat exchanger on the flow and thermal fields utilizing different types of nanofluids. The researchers solved the continuity, momentum and energy equations using the finite volume method (FVM). The bottom and the top walls of the pipe were heated with a uniform heat flux boundary condition. They used two various louvered strip insert arrangements (backward and forward) with a Reynolds number ($Re$) range of 10,000 to 50,000. They investigated the influences of different louvered strip slant angles and pitches. Four various kinds of nanoparticles, Al$_2$O$_3$, CuO, SiO$_2$, and ZnO with various volume fractions in the range of 1 to 4% and various nanoparticle diameters in the range of 20 to 50 nm, dispersed in a base fluid (water) were used. They found that the forward louvered strip arrangement could promote the heat transfer by approximately 367% to 411% at the highest slant angle of $\alpha = 30^\circ$ and lowest pitch of $S = 30$ mm. The maximal skin friction coefficient of the enhanced tube was around 10 times than that of the smooth tube and the performance evaluation criterion (PEC) value was in the range of 1.28–1.56. They found that SiO$_2$ nanofluid had the highest Nusselt number ($Nu$) value, followed by Al$_2$O$_3$, ZnO, and CuO while pure water has the lowest Nusselt number ($Nu$). They found that the Nusselt number ($Nu$) increased with decreasing the...
nanoparticle diameter and it increased slightly with increasing the nanoparticles volume fraction. In addition, there was a slight change in the skin friction coefficient with the variation of nanoparticle diameters of SiO$_2$ nanofluid. Song et al. (2014) analyzed solar incidence angle effect to simulate accurately the heat flux distribution around the absorber tube outer surface. The researchers proposed helical screw-tape inserts to homogenize the absorber tube temperature distribution and improve the thermal efficiency. They established three dimensional periodical models of flow and heat transfer and solved with the heat flux of various transversal angle ($\beta$). They found that $\beta$ effect on the flux distribution was more greatly than longitudinal angle ($\phi$). Transversal angle ($\beta$) of 11.567 mrad increased the heat loss ($Q_{\text{loss}}$) relative change as inlet temperature increased, and also increased the maximum temperature on absorber tube ($T_{\text{max}}$) and the maximum circumferential temperature difference ($\Delta T$). But its influence decreased as Reynolds number ($Re$) increased. Within the range of studied Reynolds number ($Re$), the helical screw-tape inserts of given geometrical parameters decreased greatly the $Q_{\text{loss}}$, $T_{\text{max}}$ and $\Delta T$ that indicated that helical screw-tape inserts was a feasible way to enhance the heat transfer inside the receiver.

Xinyi and Dongsheng (2012) conducted experimental and numerical investigations to study turbulent flow of water and heat transfer characteristics in a rectangular channel with discontinuous crossed ribs and grooves. They investigated the friction factor and overall heat transfer performance in ribbed and ribbed-grooved channels with rib angle of 30°. They found that the overall thermo-hydraulic performance for ribbed-grooved channel was increased by 10–13.6% when compared to ribbed channel. The investigation on the influences of various rib angles and rib pitches on friction factor and heat transfer characteristics in ribbed-grooved channel was carried out using Fluent with shear-stress transport (SST) $k-\omega$ turbulence model. They found that the case for rib angle of 45° had the best overall thermo-hydraulic performance, about 18–36% higher than the case for rib angle of 0°. Also, they analyzed the flow patterns and local heat transfer characteristics for ribbed and ribbed-grooved channels based on their numerical simulation to reveal the heat transfer enhancement mechanism.

Helical rectangular ducts have an important application in industrial processes like in heat exchangers, ventilators, gas turbines, aircraft intakes and centrifugal pumps (Yanase, Mondal, & Kaga, 2005). It is clear that most of the literature mentioned above performed several enhancements in heat transfer process dependent on the disturbance generation in fluid flow in straight tubes or ducts and none considered helical rectangular ducts or flow paths.

Based on the Navier-Stokes equations, Pan, Zhou, and Wang (2014) investigated numerically the pressure drop and heat transfer for oscillating flow in helically coiled tube heat-exchanger. The researchers proposed the correlation of the average friction factor and average Nusselt number considering the frequency and the inlet velocity. The oscillating flow heat transfer problems were affected by several factors. Therefore, they adopted the uniform design method and verified this method feasibility. They used the field synergy principle in order to explain the heat transfer enhancement of oscillating flow in helically coiled tube heat-exchanger. They found that the smaller the volume average field synergy angle in the helically coiled tube, the better the heat transfer rate. Wang, Liu, Wu, and Li (2014) studied an incompressible fully developed laminar flow in a helical rectangular duct having finite pitch and curvature with temperature-dependent viscosity under heating condition. The researchers studied both the cases of one wall heated and four walls heated. The aspect ratio of the rectangular duct was 0.5. They used water as the working fluid and varied the Reynolds number ($Re$) in the range of 100 to 400. The secondary flow with temperature-dependent viscosity was enhanced markedly in comparison to constant viscosity. They found that the friction factor obtained with temperature-dependent viscosity was lower than that of constant viscosity due to the viscosity decrease near the walls, while the convective heat transfer for temperature-dependent viscosity was significantly enhanced owing to the strengthened secondary flow. The viscosity variation effects on the flow resistance and heat transfer were more significant especially for the case of four heated walls. Xing, Zhong, and Zhang (2014) simulated three-dimensional turbulent forced convective heat transfer and its flow characteristics in helical rectangular ducts using SST
The researchers used realizable turbulent model with oil and water as working fluids for two various designs (Spiral (S) and Reverse Annulus (RA) designs) and total of five configurations of the helical ducts but the velocity and temperature profiles were distorted when the temperature-varying viscosity effects were included. Due to the viscosity values increase at the inner points of the curved tube, which reduced the secondary flow influence, the Nusselt number (Nu) obtained when the fluid was cooled with variable viscosity assumption were lower than the constant properties results. Also, they found that the friction factor results were dependent on the viscosity variations in the coil tube cross-section. Using CFD techniques, Sleiti and Naimaster (2009) studied heat transfer and flow optimization in a helical duct of rectangular cross-section used to cool the electric machines stators. The researchers used realizable turbulent model with oil and water as working fluids for two various designs (Spiral (S) and Reverse Annulus (RA) designs) and total of five configurations of the helical duct at small pitch size of 0.00254 m. For the same fluid flow rate, they found that the spiral design provided better heat transfer in terms of lower surface temperatures at the expense of higher pressure drop.

Recently, Bizhaem and Abbassi (2017) investigated numerically laminar, developing flow of Al2O3-water nanofluid with temperature dependent properties in helical tube at constant wall temperature. The researchers simulated nanofluid flow using two-phase mixture model by control volume method to study convective heat transfer and entropy generation. They compared the numerical results with three test cases including nanofluid forced convection in straight tube, velocity profile in curved tube and Nusselt number in helical tubes. They observed good agreement for all three cases. They discussed in detail and compared the Reynolds number (Re) and nanoparticle volume fraction effects on temperature and flow fields, local and overall heat transfer coefficient, local entropy generation due to heat transfer and viscous dissipation, and the Bejan number (Be) with the base fluid (water). They found that nanofluid could not provide thermal improvement in entrance region. In addition, better heat transfer enhancement and entropy generation reduction could be achieved at low Reynolds number. Fule, Bhanvase, and Sonawane (2017) investigated experimentally heat transfer enhancement using water based CuO nanofluids in the helical coil heat exchanger at laminar flow regime (Re = 812–1,895). The researchers studied the particle loading effect and flow rate on heat transfer coefficient and Nusselt number. They found that the increase in the loading of CuO nanoparticles in base fluid had a significant enhancement in the heat transfer coefficient of nanofluid. Enhancement in heat transfer coefficient was 37.3% as compared to base fluid at 0.1 vol% concentration of CuO nanoparticles in nanofluid, while it was as high as 77.7% at 0.5 vol%. In addition, they observed significant increase in heat transfer coefficient with the increase in the flow rate of the CuO nanofluid. Datta, Yanase, Kouchi, and Shatat (2017) investigated numerically laminar forced convective heat transfer in a helical pipe with circular cross section subjected to wall heating. The researchers used three-dimensional (3D) direct numerical simulations (DNS) comparing with the experimental data obtained by Shatat (Shatat, 2010). They performed their study for three Prandtl numbers (Pr) = 4.02, 7.5 and 8.5 over the wide range of torsion. In 3D steady calculations, they found that the calculated averaged Nusselt number (averaged over the peripheral of the pipe cross section) were in good agreement with the experimental data. Due to the torsion effect on the heat transfer characteristics, the averaged Nusselt number repeated decrease and increase twice as torsion increased from zero for all Reynolds numbers (Re). They found that there existed two minima and two maximums of the averaged Nusselt number. The global maximum of the Nusselt number occurred at torsion parameter (β) ≅ 0.55 and the global minimum at torsion parameter (β) ≅ 0.1. Cancon, Dingbiao, Sa, Yong, and Xu (2017) investigated pressure
drop and heat transfer in helically coiled tube with spherical corrugation (HCTSC) using a three-dimensional numerical simulation. The researchers studied various geometrical parameters of spherical corrugation in helically coiled-tube heat exchangers in order to improve the heat transfer rate. They compared calculated results to experimental tests and existing empirical formulas to study the numerical results validity. The simulation results indicate that the secondary flow induced by the centrifugal force has significant ability to enhance the heat transfer rate, the eddy caused by the corrugation structure destroys the flow boundary layer and increases the turbulence intensity of the flow and strengthens the heat transfer process. With the corrugation height ($H$) increase, friction factor sharply increased 1.01–1.24 times as compared to the smooth helically coiled tube (SHCT), while the enhancement on heat transfer performance was about 1.05–1.7 times. With the corrugation pitch ($P$) increase, the friction factor increased 1.18–1.28 times compared to the SHCT and the augmentation on heat transfer was in the range of 1.37–1.66 times. Under the same condition, the overall heat transfer performance of HCTSC was better than that of SHCT. The PEC value could be up to 1.56. Naik and Vinod (2018) investigated experimentally heat transfer enhancement using non-Newtonian nanofluids in a shell and helical coil heat exchanger. The researchers used three various non-Newtonian nanofluids comprising of Fe$_2$O$_3$, Al$_2$O$_3$ and CuO nanoparticles with the concentration range of 0.2–1.0 wt% in aqueous carboxymethyl cellulose (CMC) base fluid. Water and nanofluid were used on tube side and shell side respectively. They determined overall heat transfer coefficient and shell-side Nusselt number, at various conditions like cold water flow rate (0.5–5 lpm), stirrer speeds (500–1,500 rpm) and shell side fluid (nanofluid) temperature (40–60°C). They found that the Nusselt number increased with increasing nanofluid concentration, shell side fluid temperature, Dean number (coil-side water flow rate), and stirrer speeds. CuO/CMC-based nanofluid had better heat transfer than the other two kinds of fluid (Fe$_2$O$_3$ and Al$_2$O$_3$). The heat transfer performance of non-Newtonian nanofluids was significantly enhanced at higher concentrations of nanofluid, Dean numbers, stirrer speeds and shell-side temperatures. Bhanvase, Sayankar, Kapre, Fule, and Sonawane (2018) investigated experimentally heat transfer enhancement using water based PANI (polyaniline) nanofluid in vertical helically coiled tube heat exchanger. The researchers investigated the Reynolds number ($Re$) effect and PANI nanofibers concentration in nanofluid on heat transfer coefficient in helical coiled heat exchanger. They found that the average heat transfer coefficient increased with an increase in the Reynolds number ($Re$) and volume% of PANI nanofibers in nanofluid. The percentage enhancement in the heat transfer coefficient was 10.52% at 0.1 vol% of PANI nanofibers in nanofluid and was 69.62% for 0.5 vol% of PANI nanofibers. Sharifi, Sabeti, Rafiei, Mohammadi, and Shirazi (2018) employed the CFD technique to study the coiled wire inserts effect on the friction coefficient, Nusselt number and overall efficiency in double pipe heat-exchangers. For this aim, the researchers meshed and simulated some wire coil inserts fitted inside heat-exchangers at different Reynolds numbers with the use of two commercial CFD softwares: Gambit and Fluent. They found that taking the advantage of proper wire coils could improve the Nusselt values to 1.77 times. Also, they proposed proper friction coefficient and Nusselt number correlations for different coiled wire inserts with various geometry arrangements under the laminar flow. These two modified correlations could both be used for non-uniform helical wire insert geometries because their correlations were based on the occupied spaces where helical wires took up inside tubes.

In the above literature review, most studies presented to enhance heat transfer, or to reduce power consumption, or to increase cost-efficiency, or to simplify manufacturing and assembling processes with a few attempts to introduce a new design solves most or all of these challenges at the time. Unfortunately, most of these studies have positive effects for some parameters and negative for the others.

As a result, the novelty of this paper is the introducing a promising type of heat exchanger based on helical plate (HPHE) in order to enhance the thermal-hydraulic characteristics and performance. Due to curvature of the flow duct centrifugal force, secondary flows or vortices are generated which promotes higher heat transfer coefficient and leads to relatively more compact heat exchangers. A three dimensional simulation model of flow and heat transfer in the fluids channels of a whole HPHE with different helical plate pitch ratios and different flow channel cross section aspect ratio will be
established in detail Numerical simulation using CFD for the whole heat exchanger with 9 turns for helical plate will be conducted at a series of cases with different design and operation conditions.

2. HPHE geometry and problem formulation
The Helical plate heat exchanger with nine helix turns is shown in Figures 1 and 2. The hot fluid flows in the helical channel with the series arrangement in counter with cold fluid. The heat transfer...
process occurs through a helical copper plate with thickness 1 mm. These plates are repeated in the $x$-direction, with a pitch $P$, and height $h$. Here, the dimensionless geometric parameter; pitch ratio $= P/h$ and aspect ratio $= w/h$ were used in the numerical study. The hydraulic diameter $D_h$ was used as the characteristic flow channel diameter.

2.1. Numerical domain and grid generation
The whole HPHE with 9 helix turns was modeled as the numerical simulation domain. Commercial software (ANSYS CFX 16.2) was used with a structural hexahedral grid of a total number of nodes in the range of 197,324 to 237,888 using the multi-zone meshing approach as shown in Figure 3. The grid spacing is non-uniform, being concentrated near the interfaces between the solid walls and fluid phase because of the heat transfer and frictions in that region.

2.2. Grid independence test
To check the validity and accuracy of the numerical results, a grid-independent test was performed. Computations were carried out using four different grid sizes: 220,000–250,000 grid nodes. Figure 4 shows the outlet humid carrier gas density difference with different grid nodes for air at a water bed temperature $= 298$ K. As can be seen the density difference profile in results obtained from using 215,000 to 224,000 node grid size is very small increasing. Therefore, a grid size of 217,120 nodes was chosen for all of the cases in this study in order to utilize the computation time efficiently without compromising on the accuracy of the computation.

2.3. Governing equations and solution assumptions
The problem investigated is a three-dimensional steady state turbulent flow through a helical flow channel fitted with plain tube using the governing equations for the mass, momentum, and energy conservations, and for $k$ and $\varepsilon$ turbulence model. The following assumptions were employed:

Figure 3. Typical hexahedral mesh used to discretize the HPHE numerical domain with pitch ratio $= 0.24$. 
(1) The heat transfer and fluid flow is time-independent (steady-state), three-dimensional, and incompressible.
(2) Phase changes and heat transfer by radiation and natural convection are neglected.
(3) All the thermo-physical properties of the solid are assumed to be constants.

**Mass conservation equation**
\[
\nabla \cdot (\rho \vec{V}) = 0
\]

**Momentum conservation equation**
\[
\nabla [\nabla \cdot (\rho \vec{V})] = -\nabla p - \frac{2}{3} \nabla [\mu (\nabla \cdot \vec{V})] + \nabla \cdot [\mu (\nabla \cdot \vec{V})] + \nabla \cdot [\mu (\nabla \cdot \vec{V})]
\]

**Energy conservation equation**
\[
\rho c_p \nabla \cdot \vec{V} = \nabla \cdot [k(\nabla T)] + \nabla \cdot \nabla p + \psi
\]

**Turbulence model**
\[
\nabla \cdot (\rho k \vec{V}) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) (\nabla \cdot k) \right] + G_k - \rho \varepsilon
\]

The hydraulic diameter \( D_h \) is defined as:
\[
D_h = \frac{2wh}{w + h}
\]
The friction factor $f$ is calculated as

$$f = \frac{2D_n \Delta p}{\rho u^2} \tag{6}$$

The heat exchanger effectiveness $\varepsilon$ is calculated as

$$\varepsilon = \frac{\dot{m}_h c_{p,h} (T_{h,i} - T_{h,o})}{\min mc_p (T_{h,i} - T_{c,i})} = \frac{\dot{m}_c c_{p,c} (T_{c,o} - T_{c,i})}{\min mc_p (T_{h,i} - T_{c,i})} \tag{7}$$

2.4. Numerical method

The above mentioned equations were solved with the commercial software ANSYS CFX 16.2. The realizable $k - \varepsilon$ model is adopted because it can provide improved predictions of near-wall flows and flows with high streamline curvature (Yin, Wang, Cheng, & Gu, 2013). Solution sequential algorithm (segregated solver algorithm) with settings including implicit formulation, steady (time-independent) calculation, SIMPLE as the pressure-velocity coupling method, and second-order upwind scheme for energy and momentum equations was selected for simulation.

2.5. Boundary and initial conditions

The inlet boundary and initial conditions of hot and cold fluid are axial velocity and outlet boundary condition is fixed average static pressure equal to the standard atmospheric pressure. The inner and outer surface of the HPHE is adiabatic (isolated). All blocks are starting with water. The hot and cold fluids have inlet temperatures of 400 and 300 K for all simulations. The thermo-physical properties of the fluid, materials, and the numerical values of velocities and pitch ratios which were used in a number of the simulations are given in Tables 1 and 2.

Table 1. Thermo-physical properties of materials

<table>
<thead>
<tr>
<th>Materials</th>
<th>Density (kg·m$^{-3}$)</th>
<th>Specific heat (J·kg$^{-1}$·K$^{-1}$)</th>
<th>Thermal conductivity (W·m$^{-1}$·K$^{-1}$)</th>
<th>Viscosity (Pa·s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>998.2</td>
<td>4,182</td>
<td>0.6</td>
<td>0.001003</td>
</tr>
<tr>
<td>Copper</td>
<td>8,940</td>
<td>386</td>
<td>385</td>
<td>–</td>
</tr>
</tbody>
</table>

Table 2. Numerical values of the parameters used for simulations

<table>
<thead>
<tr>
<th>Pitch ratio</th>
<th>Velocity (m·s$^{-1}$)</th>
<th>Hot fluid</th>
<th>Cold fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.24</td>
<td>0.1</td>
<td>0.2</td>
<td>0.5</td>
</tr>
<tr>
<td>0.67</td>
<td>0.3</td>
<td>0.2</td>
<td>0.3</td>
</tr>
<tr>
<td>1.31</td>
<td>0.36</td>
<td>0.24</td>
<td>0.35</td>
</tr>
<tr>
<td>0.24</td>
<td>0.5</td>
<td>0.37</td>
<td>0.19</td>
</tr>
<tr>
<td>0.67</td>
<td>0.4</td>
<td>0.27</td>
<td>0.13</td>
</tr>
<tr>
<td>1.31</td>
<td>0.52</td>
<td>0.25</td>
<td>0.17</td>
</tr>
<tr>
<td>0.24</td>
<td>0.65</td>
<td>0.47</td>
<td>0.37</td>
</tr>
<tr>
<td>0.67</td>
<td>0.52</td>
<td>0.28</td>
<td>0.19</td>
</tr>
<tr>
<td>1.31</td>
<td>0.41</td>
<td>0.35</td>
<td>0.24</td>
</tr>
</tbody>
</table>
3. Results and discussions

3.1. Model results validation

To ensure the reliability of the numerical simulation methodology used in this study, the numerical results were compared with the correlation proposed by Guo, Feng, and Chen (2001) for the friction factor for turbulent flow in a helical duct as follows:

\[ f = 2.552Re^{-0.15} \left( \frac{D_h}{d} \right)^{0.51} \]  

As shown in Figure 5, the numerical simulation results are in good agreement with the experimental results, within maximum deviation 13%, which can be attributed to the discrepancies between the numerical model and experimental model. Therefore, the numerical methods adopted in this study for pressure drop predictions were judged to be reliable. In addition, an experimental replication of HPHE is currently underway to validate model assumptions and predictions.

3.2. Pressure drop

Figures 6 and 7 present pressure drop in hot and cold fluid, respectively, for different pitch ratio against various Reynolds numbers. It can be observed from these figures that as the Reynolds number of the flow increases, the pressure drop across the fluid flow channel also increases for all pitch ratios. Also, the increasing in pitch ratio decreases the pressure drop. The trend pressure drop is the same for both channels. For hot channel, the minimum value of pressure drop obtained is 34.87 N·m⁻² at \( Re = 2,000 \), aspect ratio = 0.67 and pitch ratio = 1.31 and the maximum obtained is 6,296.92 N·m⁻² at \( Re = 10,000 \), aspect ratio = 0.12 and pitch ratio = 0.24. For cold channel, the minimum value of pressure drop obtained is 15.06 N·m⁻² at \( Re = 1,400 \), aspect ratio = 0.67 and pitch ratio = 1.31 and the maximum obtained is 3,379.42 N·m⁻² at \( Re = 7,000 \), aspect ratio = 0.12 and pitch ratio = 0.24. So based on this observation, it can be concluded that as the pitch ratio increases and Reynolds number decreases, the pressure drops offered by the hot and cold fluid flow channels increases.
Figures 8 and 9 present pressure distributions of the hot and cold fluid flow channels in $x$-$y$ plane for two simulation conditions; pitch ratio = 0.24, aspect ratio = 0.12 and 1.31 with $Re_h = 10,000$ and $Re_c = 7000$. 
3.3. Friction factor

Theoretically, as pressure drop for a flow increases, it results in a decrease in the friction factor values. According to the pressure loss obtained by CFD, the friction factor \( f \) can be calculated using Equation (6). Figures 10 and 11 present friction factor in hot and cold fluid, respectively for different pitch ratio against various Reynolds numbers. It can be observed from these figures that as the Reynolds number of the flow increases, the friction factor across the fluid flow channel decreases for all pitch ratios. Also, the increasing in pitch ratio decreases the friction factor. The trend friction factor approximately is same for both channels. For hot channel, the minimum value of friction factor obtained is 0.22 at \( Re = 10,000 \), aspect ratio = 0.67 and pitch ratio = 1.31 and the maximum obtained is 0.51 at \( Re = 2,000 \), aspect ratio = 0.12 and pitch ratio = 0.24. For cold channel, the minimum value of friction factor obtained is 0.24 at \( Re = 7000 \) and pitch ratio = 1.31 and the maximum obtained is 0.51 N·m\(^{-2}\) at \( Re = 1,400 \), aspect ratio = 0.12 and pitch ratio = 0.24. So based on this observation, it can be concluded that as the pitch ratio and Reynolds number increase, the friction factor offered by the hot and cold fluid flow channels decreases.

3.4. Velocity distributions

Figures 12 and 13 present the visible computational results of velocity distributions of the hot and cold fluid flow channels in \( x-z \) plane section for two simulation conditions; pitch ratio = 0.24 and 1.31 with \( Re_h = 10,000 \) and \( Re_c = 7,000 \) at \( y = 0.31 \) m. Maximum velocity in hot and cold flow channel on inclined surface appears between the upward and downward flow paths.
3.5. Heat exchanger effectiveness
According to the pressure loss obtained by CFD, the heat exchanger effectiveness ($\varepsilon$) can be calculated using Equation (7). The variations in HPHE heat exchanger effectiveness ($\varepsilon$) as a function of the pitch ratio with different Reynolds number is shown through Figure 14. For constant pitch ratio, it is shown that the increasing of Reynolds number for both fluids; the heat exchanger effectiveness ($\varepsilon$) will decrease. In addition, the increasing in pitch ratio decreases HPHE effectiveness. The heat exchanger effectiveness is increased up to 67.56 for $Re_h = 2,000$ and $Re_c = 1,400$ and pitch ratio = 0.24.

3.6. Temperature distributions
Figures 15 and 16 present the visible computational results of temperature distributions of the hot and cold fluid flow channels in $x$–$z$ plane section for two simulation conditions; pitch ratio = 0.24 and 1.31 with $Re_h = 10,000$ and $Re_c = 7,000$ at $y = 0.31$ m. Maximum temperature in hot flow channel on
inclined surface appears between the upward and downward flow paths. While on the contrary, maximum temperature in cold flow channel on inclined surface appears at the maximum upward and minimum downward flow paths.
Figure 12. Velocity distributions in x–z plane section at y = 0.031 m for Pitch ratio = 0.24, Re_h = 10,000 and Re_c = 7,000.

Figure 13. Velocity distributions in x–z plane section at y = 0.031 m for pitch ratio = 0.24, Re_h = 10,000 and Re_c = 7,000.

Figure 14. Variations in HPHE effectiveness (ε) as a function of Reynolds number and pitch ratio.
3.7. Helical plate in comparison with flat plate heat exchangers

In this section a comparison of helical plate with flat plate heat exchangers is presented according to thermal, hydraulic and thermodynamic parameters such as heat exchanger effectiveness, pressure drop and entropy generations. Figures 19 and 20 present pressure drop for helical plate and flat plate heat exchangers in both hot and cold fluids, respectively for aspect = 0.12 against various Reynolds numbers. It can be observed that at the same flow Reynolds number for both helical plate and flat plate heat exchanger, the pressure drop across the fluid flow channel of helical plate heat exchanger is more than in flat plate heat exchanger by about 1.1 to 1.13 on average. Figure 21 shows that an improvement in effectiveness (\(\varepsilon\)) of the helical plate heat exchanger over flat plate heat exchanger by about 1.2 to 1.9 on average.
Figure 17. Temperature distributions for pitch ratio = 0.24, $Re_h = 10,000$ and $Re_c = 7,000$.

Figure 18. Temperature distributions for pitch ratio = 1.31, $Re_h = 10,000$ and $Re_c = 7,000$. 
Figure 19. Pressure drop for helical and straight flow channel in hot fluid against various Reynolds number at aspect ratio = 0.12.

Figure 20. Pressure drop for helical and straight flow channel in cold fluid against various Reynolds number at aspect ratio = 0.12.
4. Conclusions

In this paper, a three dimensional simulation model of flow and heat transfer in the fluids channels of a whole HPHE with different helical plate pitch and aspect ratios are established numerically by a CFD commercial code in a series of cases with different design and operation conditions. The main conclusions are summarized:

(1) The present numerical simulation had been compared and a good agreement with experimental data trend had been obtained from another published work.

(2) The pitch ratio and aspect (width-to-height) ratio variation can enhance heat transfer rate and reduce the pressure drop.

(3) For hot channel, the minimum value of pressure drop obtained is 34.87 N·m⁻² and the maximum obtained is 6,296.92 N·m⁻² for cold channel, the minimum value of pressure drop obtained is 15.06 N·m⁻² and the maximum obtained is 3,379.42 N·m⁻².

(4) For hot channel, the minimum value of friction factor obtained is 0.22 and the maximum obtained is 0.51. For cold channel, the minimum value of friction factor obtained is 0.24 and the maximum obtained is 0.51 N·m⁻².

(5) Maximum velocity in hot and cold flow channel on inclined surface appears between the upward and downward flow paths.

(6) The heat exchanger effectiveness is increased up to 67.56 for \( Re_h = 2,000 \) and \( Re_c = 1,400 \) and pitch ratio = 0.24.

(7) There is a noticeable improvement in helical plate over flat plate heat exchangers in thermal and thermodynamic parameters with the exception of a slight increase in pressure drop.
The present work is expected to provide some insights to the understanding and optimization of flow and convective heat transfer mechanisms of the helical heat exchanger.

Nomenclature

Latin symbols

- \( c_p \): specific heat, J·kg\(^{-1}\)·K\(^{-1}\)
- \( D \): diameter, m
- \( D_h \): helical coil diameter, m
- \( F \): friction factor, dimensionless
- \( G_g \): rate of generation, kg·m\(^{-1}\)·s\(^{-3}\)
- \( H \): plate height, mm
- \( K \): thermal conductivity, W·m\(^{-1}\)·K\(^{-1}\)
- \( l \): flow path length, m
- \( m \): mass flow rate, kg·s\(^{-1}\)
- \( N_s \): number of entropy generation units, dimensionless
- \( P \): helical pitch, mm
- \( P \): pressure, N·m\(^{-2}\)
- \( \Delta p \): pressure drop, N·m\(^{-2}\)
- \( Q \): heat quantity, W
- \( Re \): Reynolds number \( \equiv \frac{\rho u_D D}{\mu} \), dimensionless
- \( S \): entropy, W·K\(^{-1}\)
- \( T \): temperature, K
- \( u, v, w \): velocity in \( x, y, z \) respectively, m·s\(^{-1}\)
- \( V \): volume, m\(^3\)
- \( w \): channel width, mm

Greek letters

- \( \beta \): thermal expansion coefficient, K\(^{-1}\)
- \( \varepsilon \): heat exchanger effectiveness, dimensionless or turbulent dissipation rate, m\(^2\)·s\(^{-3}\)
- \( \phi \): source term, Equation (3), W·m\(^{-1}\)
- \( \mu M \): dynamic viscosity, kg·m\(^{-1}\)·s\(^{-1}\)
- \( \rho \): density, kg·m\(^{-3}\)

Subscripts

- \( c \): cold
- \( g \): generation
- \( h \): hot or hydraulic
- \( i \): inlet
- \( \text{min} \): minimum
- \( o \): outlet
Acronyms and abbreviations

CFD: Computational fluid dynamics
HPHE: Helical plate heat exchanger
RNG: Renormalization group

Acknowledgements

The third author, Mohamed M. Awad, would like to thank Lesly Hernandez, Journal Editorial Office on behalf of Cogent Engineering, Taylor & Francis Group for waiving the Article Processing Charges (APC) of US$250 for Egypt, which is one of the group B countries in the Research4Life program (http://www.research4life.org/eligibility/) to support sustainable open access publishing for this contribution because he is the Guest editor for an open access special issue entitled: “Recent advances in Enhanced Heat Transfer and Engineering Applications”, Cogent Engineering Journal, Taylor & Francis Group.

Funding

The authors received no direct funding for this research.

Author details

Emad M.S. El-Said1
E-mail: emadsoaad@fayoum.edu.eg
Mohamed Abdulaziz2
E-mail: info@simtran.de
Mohamed M. Awad3
E-mails: m_m_awad@mans.edu.eg, awadm3@gmail.com

1 Faculty of Engineering, Mechanical Engineering Department, Fayoum University, Fayoum, Egypt.
2 SIMTRAN, Eningen unter Achalm, Reutlingen, Baden-Württemberg, Germany.
3 Faculty of Engineering, Mechanical Power Engineering Department, Mansoura University, Mansoura 35516, Egypt.

Citation information

Cite this article as: A numerical investigation on heat transfer enhancement and the flow characteristics in a new type plate heat exchanger using helical flow duct. Emad M.S. El-Said, Mohamed Abdulaziz & Mohamed M. Awad, Cogent Engineering (2017), 4: 1396638.

Cover image

Source: Authors

References


