In the present paper, along with experimental study, CFD analysis of forced convection in a twisted tube is performed, using the transition SST model which can predict the change of flow regime from laminar through transition to turbulent. The differential governing equations are discretized by the finite volume method. The investigations are conducted for Reynolds numbers ranging from 100 to 50,000 covering laminar, transitional and turbulent regimes, and for three length and three pitch ratios. The predictions are observed to show a good agreement with the measurements and published correlations of other authors. The analysis indicates that the large length ratio and small pitch ratio yields a higher heat transfer rate with relatively low performance penalty. The transition from laminar to turbulent regime is observed between Reynolds numbers of 2500 to 3500 for all cases. For almost all investigated cases the performance factors are greater than unity.

Key words: turbulent forced convection, twisted tube, swirl flow, heat transfer enhancement

Introduction

Heat exchanger is a device facilitating heat transfer between two or more fluids [1]. It is used in a vast number of industrial applications, such as thermal power plants, chemical processing plants, air conditioning equipment, refrigerators, radiator for space vehicles as well as automobiles, extending up to micro-scale applications [2, 3]. Obviously, for improving the efficiency of energy utilization, enhancement of heat transfer in such applications plays an important role. To this goal, two techniques can generally be identified, namely, the active and passive techniques. Active techniques include surface vibration, fluid vibration, etc. for
increasing heat transfer [4]. In passive techniques, the applied methods usually involve a modification of the channel geometry to enhance heat transfer. A special category herein is the use of bluff bodies that increase heat transfer by vortex shedding [5, 6]. Passive methods also include different arrangements, such as impinging jet [7, 8] instead of wall-parallel flow.

A passive technique that attracted considerable attention is the insertion of structures such as twisted tapes that induce a swirling motion. Indeed, industrial heat transfer equipment’s are generally operated in turbulent/swirl flow conditions where their performance in terms of energy transfer rate is high, compared with laminar flow by virtue of the high degree of turbulence in turbulent swirl flow.

Heat transfer characteristics under transitional flow conditions in most of thermal problems are of considerable interest. Predicting transition of laminar regime to turbulent in heat transfer augmentation techniques will be highly useful to design any heat transfer equipment. There is constant thrive in the studies on transition from laminar to turbulent flow.

Many researchers performed surveys on passive enhancement method of this kind, which were very inspiring, such as channel with a cylinder roughness [9] and compound heat transfer enhancement of a convergent-divergent tube with a twisted tape [10]. Swirling flow may increase the heat transfer through the boundary-layer by interruption or thinning and also increase the turbulence levels considerably [11]. Tube with wire coil inserts [12] can generate vortex and also can develop secondary flow.

Tube insert technology – i. e., insertion of tape inside a channel - was used in some previous work. Bhattacharyya et al. [13, 14] and Saha et al. [15] studied, experimentally, center-clearing twisted tapes with artificial rib roughness and achieved considerable improvement. Meng et al. [16] performed experiments on elliptical tube type swirl generator. Some earlier research [17, 18] showed that vortex geometry may affect heat transfer which can be utilized to design heat exchangers. Chen et al. [19] studied experimentally heat transfer augmentation techniques of a dimpled tube. Vicente et al. [20] reported on thermohydraulic performance of helically dimpled tubes for laminar and transition flow. Mengna et al. [21] studied compound heat transfer enhancement of a converging-diverging tube with swirl generator inserts. Sivashanmugam et al. [22] reported on experiments on thermohydraulic characteristics of laminar stream in circular channel fitted with screw type tape. Additional information on similar vortex originator may be acquired from [23-27].

In the present paper, a numerical and experimental study of forced convection in circular twisted tube is presented. Air is used as working medium. At the prevailing relatively small/moderate temperature differentials, constant material properties are assumed. A sketch of the device is presented in fig. 1, where the circular twisted enhancers can be recognized.

**Experimental**

Figure 2 demonstrates the schematic figure of the experimental set-up. The working medium, atmospheric air, is sucked by a 7.0 kW blower, which flows through a PVC pipe and finally a rotamer. The latter is used to measure the flow rate (range: 120-520 LPH, error: 0.0442%). A bypass valve is installed between the blower and the rotamer to control the flow rate. For measuring the pressure drop across the tube, two pressure taps are placed at inlet and outlet of the test section. A U-tube manometer (range: 1-150 mm Hg) is, then, used.
to measure the pressure drop. A total of 28 thermocouples are fitted on the tube outside wall by brazing at 7 equidistant axial locations, each position containing 4 thermocouples equidistantly distributed along the circumference. The inner wall temperatures are obtained from the measured heat-flow rate and the thermal resistance of the wall, assuming 1-D heat conduction across the wall. The test section is electrically heated up by nichrome heater wire providing essentially uniform wall heat flux. Porcelain bead types of insulation are used on nichrome heater wire, with no direct interaction of the nichrome heater wire with the tube wall. A 1.5 KW variac is used in which the variations of voltage and current are 10.0-30.0 V and 1.0-3.0 A, respectively (error: 0.072% in the current and voltage compared to the nominal value). In order to minimize the heat loss, the tube is insulated using glass-wool layers. During the experiments, the heat loss to surroundings has been quantified. The heat equilibrium test has shown that the heat absorbed by air to be 2-3% lower than the heat supplied by the electrical winding. The uncertainty in Reynolds number, friction factor and Nusselt number are estimated as ±4.17%, ±6.66%, and ±7.42%, respectively, following the procedure of Bhattacharyya et al. [13].

Carrying out the experiments, the set-up is, first, allowed to avail steady-state by providing sufficient time, which was about two hours. When the standard deviations of the temperature and mass-flow rate were less than 0.03 °C and 0.011 kg/s, respectively, a steady-state was assumed to be reached.

A circular tube having 22 mm inside diameter and length of 2.0 m is used for the experiment purpose. The tube as depicted in fig. 1 is made out of brass. For validation purpose the experiment is executed in a smooth tube and then for twisted tubes for three different length ratios ($H = l/L = 0.2, 0.25$ and $0.30$), and three different pitch ratios ($S = p/d = 0.22, 0.36$, and $0.45$).

Modelling

Steady-state, incompressible flow of a Newtonian fluid with constant material properties and negligible buoyancy are assumed. Obviously, the radiative heat transfer does either not play a role in the present application at low temperatures with a practically non-
participating medium [28]. The turbulence is modelled within RANS applying a turbulent viscosity model, in particular, the transitional SST model of turbulence of Menter [see 29], which was successfully used, previously, in various applications in fluids and thermal engineering [30-34]. Oclon et al. [32] used this model for heat transfer predictions in conjunction with the prediction of fouling in fin-and-tube heat exchanger, whereas Kadiyala and Chattopadhyay [34] applied the model for studying the effects of cooling jets on moving surface. The computational modelling is performed based on the general-purpose CFD solver ANSYS Fluent [29]. For a detailed description of the model and for the model equations, the reader may refer to [29].

At the inlet boundary, all dependent variables are prescribed. For the turbulence quantities, a turbulence intensity of 5% and a macro-length scale equal to the tube diameter is assumed to derive the boundary conditions. This may be seen inappropriate for low Reynolds numbers that indicate a laminar flow. However Abraham et al. [31] showed that the applied transitional SST model can cope with this situation, predicting a rapid drop of the viscosity to the molecular value, if the flow is laminar, so that the results are not affected. At the outlet, a zero-gradient boundary condition applied for all convective-diffusively transported variables, along with a constant static pressure. At walls, the no-slip boundary conditions apply, where the wall temperature is prescribed. As far as the near-wall turbulence is concerned, no wall-functions [29] are used, as the applied turbulence model can cope with low Reynolds number, transitional turbulence.

The velocity-pressure coupling is treated by the Semi Implicit Pressure Linked Equations (SIMPLE) scheme [29]. For discretizing the convective terms, a second order accurate upwind scheme [29] is used. The discretized equations are then solved iteratively in double precision solver. The convergence criteria for continuity, momentum and energy are set at $10^{-4}$, $10^{-5}$, and $10^{-7}$, respectively. The convergence criterion for the four turbulence quantities was fixed at $10^{-4}$.

Data reduction

The bulk fluid temperature, $T_b$, is calculated as the arithmetic average of the fluid and inlet, $T_i$, outlet, $T_o$, temperatures. The wall heat flux, $q_w$, is deduced from the difference in the energy flow rates of the inflowing and outflowing fluids. The heat transfer coefficient, $h$, [1] is obtained from:

$$h = \frac{q_w}{T_w - T_b}$$

where the wall temperature, $T_w$, is obtained as the arithmetic average of the wall temperatures measured by the 28 thermocouples. The Nusselt number is obtained from:

$$\text{Nu} = \frac{hD}{k}$$

where $k$ denotes the fluid thermal conductivity and $D$ the diameter of the plain tube. The Darcy friction factor, $f$, is evaluated from:

$$f = \frac{\Delta p}{\frac{1}{2} \rho U^2}$$
In the previous equation, \( \Delta p \) denotes the pressure drop across the tube, \( U \) – the axial bulk velocity for the plain tube, \( \rho \) – the density, and \( L \) – the tube length. The Reynolds number is also defined based on \( D \) and \( U \). The air material properties are evaluated at the bulk fluid temperature, \( T_b \).

To evaluate the effect of heat transfer enhancement under given pumping power, the thermohydraulic performance factor, \( \eta \), [35-37] is used, where the subscript 0 refers to the smooth tube:

\[
\eta = \frac{\text{Nu}}{\text{Nu}_0} \left( \frac{f}{f_0} \right)^{0.33}
\]

\( \text{(4)} \)

Results and Discussion

An unstructured grid with non-uniform grid density distribution was created. A grid independence study was performed, where the basic parameters such as the Nusselt number and friction factor were obtained and compared on three different grids, the results of which are shown in tab. 1, for the case with \( \text{Re} = 10000, H = 0.30, S = 0.22 \). The Grid 1 consisting of 777235 nodes, which seems to provide sufficient accuracy, is used for the present computations. Different views of the surface grid are provided in fig. 3.

Table 1. Grid independence study (Re = 10000, H = 0.30, S = 0.22)

<table>
<thead>
<tr>
<th>Grid</th>
<th>Total number of nodes</th>
<th>Nusselt number</th>
<th>( f )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>777235</td>
<td>49.14</td>
<td>0.046</td>
</tr>
<tr>
<td>Grid 2</td>
<td>814887</td>
<td>49.12</td>
<td>0.047</td>
</tr>
<tr>
<td>Grid 3</td>
<td>1000112</td>
<td>49.12</td>
<td>0.047</td>
</tr>
</tbody>
</table>

Figures 4(a) and 4(b) show the assessment for the Nusselt number and friction penalties, for a plain tube. Here, the results are compared with the empirical correlations by Dittrus-Boelter [1] for the Nusselt number, and with the correlation of Blasius [1] for the friction factor in the turbulent region (\( \text{Re} > 5000 \)), whereas, in the laminar region (\( \text{Re} < 2000 \)), the analytical expressions known for the laminar flow (\( \text{Nu} = 3.66 \) and \( f = 64/\text{Re} \)) are used for the comparison. On the whole, the relative errors are quite small, indicating that there is a wonderful agreement between the investigational data and those forecasted by the standard correlations for the plain tube. The results prove the reliability of the experimental and numerical procedures for predicting fluid-flow and heat transfer characteristics.

The numerically (num.) and experimentally (exp.) obtained Nusselt numbers for the tube with twisted geometry, for certain values of the length ratio, \( H (H = l/L) \), and the pitch
ratio, $S (S = p/D)$, are presented in fig. 5, as function of Reynolds number (see fig. 1, for the definitions of $H$ and $S$).

As expected, Nusselt number increases with increasing Reynolds number for all cases. In fig. 5(a), cases with different $S = 0.22$, 0.36, and 0.45 are compared while $H = 0.3$ remains constant. One can observe that the small values of $S$ lead to increased Nusselt numbers. Laminar, turbulent and transitional regions are also indicated in the figure, which show different characteristics. Figure 5(b) compares the effect of $H$ by considering its three values (0.2, 0.25, and 0.3) while keeping $S = 0.22$ constant. It is observed that larger values of $H$ lead to higher Nusselt numbers and this effect, fig. 5(b), is even stronger than the one observed for decreasing $S$, fig. 5(a). The minimum and maximum increases of the Nusselt number are observed for the smallest $S$ and largest $H$ ($H = 0.2$, $S = 0.45$) and the largest $H$ and smallest $S$ ($H = 0.3$, $S = 0.22$), respectively. These cases are compared with the results for the plain tube in fig. 5(c). One can see the case $H = 0.2$, $S = 0.45$ provides only a marginal increase of the Nusselt number compared to the plain tube, while for the case $H = 0.3$, $S = 0.22$ a remarkable increase, which is around 50%, is observed. One can also see that the agreement between the predictions and experiments is very good, for all considered cases, fig. 5.

Figure 6 shows the Num. and Exp. obtained variations of the friction factor with the Reynolds number. It can be viewed that the friction factor decreases with increasing Reynolds number, for all cases. Figures 6(a) and 6(b) demonstrate the effect of the pitch ratio (varying $S$ with constant $H$) and length ratio (varying $H$ with constant $S$), respectively. One can see that
the friction factor increases with decreasing $S$ and increasing $H$. The minimum and maximum increase of the friction factor is observed for the smallest $S$ and largest $H$ ($H = 0.2, S = 0.45$) and the largest $H$ and smallest $S$ ($H = 0.3, S = 0.22$), respectively. These cases are compared with the results for the plain tube in fig. 6(c). One can see the case $H = 0.2, S = 0.45$ has a considerable increase compared to the plain tube, while for the case $H = 0.3, S = 0.22$, only a marginal further increase is observed, except for low Reynolds numbers. One can also see that the agreement between the predictions and experiments is very good for all cases, fig. 6.

Distribution of the dimensionless velocity magnitude (non-dimensionalized by the inlet velocity) is shown in fig. 7, for the case with $Re = 10000, H = 0.3$, and $S = 0.22$. Figure 7(a) shows the distribution in a longitudinal plane, whereas cross-sectional planes along the length of the tube are displayed in fig. 7(b). It can be witnessed that twisted geometry has solid impact over the flow field. In the twisted parts, the velocity in the central regions increases considerably due to the displacement effect caused by the twist structure acting like roughness, whereas an additional contribution results by the generated swirl component.

Figures 8 shows the distribution of dimensionless temperature based on the wall, $T_w$, and inlet, $T_{in}$, temperatures, $(T - T_{in})/(T_w - T_{in})$, for $Re = 10000$ for the twisted tube with $H = 0.3$ and $S = 0.22$. Figure 8(a) shows the distribution in a longitudinal plane and fig. 8(b) shows distributions in cross-sectional planes along the length of the tube. One can see from the figure that there is a change in the temperature field throughout the tube. This shows that
the turbulator (twisted geometry) can offer a significant effect on the temperature field, since it can induce better mixing of the fluid between the boundary layer and the core flow.

In general, the thermal effectiveness of heat exchangers is evaluated based on the enhancement of convective heat transfer coefficient and reduction of power consumption to pump the fluid through the device. In the present study, the thermal performance is assessed by comparing the convective heat transfer coefficients of the enhanced and reference heat transfer surfaces for the same pumping power.

Figure 9 shows the variations of the thermal performance factor with Reynolds number for the twisted tube. It can be distinguished that the thermal performance factor is more than unity for most of the time with varying Reynolds number. At higher Reynolds numbers (Re = 35000-50000) for the case of \( H = 0.2 \) and \( S = 0.45 \), the thermal performance factor decreases to a value less than 1.0. If \( \eta \geq 1.0 \), this indicates that the heat-flow rate for the tube with twisted geometry is higher than that for the plain tube at the same pumping power. Figure 9 also shows that the thermal performance factor of the tube with twisted geometry increases with an increase of length ratio \( (H = l/L) \) and decrease in pitch ratio \( (S = p/d) \). The highest thermal performance factor is obtained for tube with \( H = 0.3 \) and \( S = 0.22 \). The lowest thermal performance factor is obtained for the tube with \( H = 0.2 \) and \( S = 0.45 \).

For a better overview, the experimentally and numerically obtained maximum \((H = 0.3, S = 0.22)\) and minimum \((H = 0.2, S = 0.45)\) performance factors are presented,
again, in fig. 10. It can again be observed that a certain improvement in the thermal performance is obtained for almost all configurations, within the range 1.0-2.0, approximately.

Conclusions

In this work, the heat transfer and fluid-flow characteristics of turbulent flow through a heat exchanger mounted with circular twisted tube were investigated experimentally and numerically. It is observed that the twisted geometry enhances heat transfer considerably. Heat transfer also increases with an increase of length ratio and decrease in pitch ratio. The twisted geometry with the large length ratio and small pitch ratio leads to the highest heat transfer augmentation, where the Nusselt number is higher by approximately 50% on the average, compared with that of the plain tube. The friction factor decreases as the Reynolds number increases, but increases with an increase in the length ratio and decrease in pitch ratio. In spite of the high friction penalty, the case with \( H = 0.3 \) and \( S = 0.22 \) yields the maximum thermal performance factor, with a value of approximately 2.0, on the average.

References


Paper revised: December 15, 2018 Published by the Vinča Institute of Nuclear Sciences, Belgrade, Serbia.
Paper accepted: January 24, 2019 This is an open access article distributed under the CC BY-NC-ND 4.0 terms and conditions.