20th Australasian Fluid Mechanics Conference
Perth, Australia
5-8 December 2016

Numerical Investigation of Turbulent Flow and Thermal Behaviour in Helical Pipes

O. Joneydi Shariatzadeh[1], N. Nadim[1] and T. T. Chandratilleke[1]

[1] Department of Mechanical Engineering
Curtin University, Western Australia 6845, Australia

Abstract

Fluid flow in helical pipe is associated with a wide range of engineering applications that motivate significant interest for research in this field. Flow in helical pipes has unique behaviour, where both mean flow and boundary layer are influenced by secondary vortices. Numerical modelling of such flow field requires specific considerations to ensure robust and reliable simulation of fluid structures. Choosing a numerical turbulent scheme, in the spectrum of algebraic closures to DNS, is a compromise between resolution of turbulence effects and computational resources. This study applies a selection of URANS, RSM, LES and hybrid turbulence models (DES and SAS) for CFD analysis of flow and heat transfer in a helical pipe. Experimental measurements are utilised as validation benchmark to assess the accuracy of models and refinement level of turbulence scale, essential for this specific application. Wall grid refinements are also examined to highlight essential y+ requirements associated with thermal boundary layer estimations. Turbulence models are evaluated for their accuracy in capturing secondary flow (as the main mean flow feature) and also boundary layer characteristic, which are critical for convective heat transfer. Using FLUENT as a well-trusted commercial code, the validity and performance of turbulence models are investigated and compared to suggest accurate, yet cost-effective model for the flow field affected by centrifugal force.

Introduction

The flow through helical pipes are frequently encountered in industrial applications mainly since they are compact and have a unique thermos-fluid mixing pattern. The helical geometry brings about special flow characteristics which are particularly originated from the centrifugal forces induced by pipe curvature and the fluid momentum associated with continuous flow direction change. This secondary motion develops in the cross section of curved pipes due to the imbalance between inertial and centrifugal forces, that produces spiralling fluid motion through the pipe. These body forces together with turbulence effects acting on the fluid mass create unique flow features that are intrinsically different to those of flow through straight pipes. Such flow features highly promotes fluid mixing with the potential for thermal enhancement while making the thermal boundary layer thinner at the outer pipe wall.

The main characteristics of flow in helical pipes have already been revealed. Yet, characteristics of turbulent flow in helical pipes are relatively less known than that of laminar flow. The effect of secondary and Dean vortices on turbulent flow in helical pipes is of paramount engineering interest whose behaviour is dependent on a wide range of parameters with a key base of thermal and hydrodynamic characteristics.

However, due to limitation of computational resource, numerical modelling of flow in helical pipes has been subject to a wide range of compromise. As a result of work of Chen and Jan [1] and Ciofalo et al. [2] some researchers neglected the influence of pitch on the flow behaviour in helical pipes so that for the economic reasons, reported the result of numerical simulation of zero-pitch helical pipes [3-5]. By imposing symmetry boundary condition at pipe cross-section and periodic boundary condition at inlet-outlet, one half of the cross-section and a small number of grid points were discretised in radial and stream-wise directions respectively leading to massively saving computational costs [4]. The results indicates that the Reynolds Stress Model (RSM) predicts the friction coefficient and Nusselt number slightly better than Shear Stress Transport (SST) k-ω model both compared with Direct Numerical Simulation (DNS). Both models moderately give error in prediction of velocity profile in the transitional range.

More recently, Colombo et al. [6] compared turbulence flow in 3/4 a turn of a helical pipe using k-ε model, SST k-ω model and RSM with applying near wall treatment. The results of dimensional friction coefficient for $5000<Re<50000$ were compared, however, heat transfer characteristics were not investigated. At low-medium Reynolds numbers, $5000<Re<15000$, all the models fail to predict the friction coefficient compared to experimental results. Apparently when the wall function approach is applied, low Reynolds number regions are critical for turbulence modelling. The authors showed that overestimated peaks of the wall shear stress are not observable when the near wall region is resolved.

However, due to a variety of reasons, lack of an extensive numerical study of a long non-zero pitch helical pipe which investigates thermal characteristics using various turbulence models is evident. Additionally, it is argued that choosing periodic and/or symmetry boundary conditions for modelling of such a complex flow is not an accurate method to save computational resource as secondary vortices are present in an asymmetric coil with non-zero pitch. The current paper aims to compare the turbulence models capability to account for the helical pipe thermal boundary layer characteristics on the one hand and mean flow unsteadiness on the other hand. SST k-ω model, RSM, Large Eddy Simulation (LES), Detached Eddy Simulations (DES) and Scale Adaptive Scheme (SAS) were tested and the result of transient simulations have been assessed through comparison with that of experiment [7].

Shortfalls in numerical modelling are more evident in viscous sublayer where an ultra-high resolution mesh is strictly required to predict energy spectrum for a turbulent flow. For this reason, extra attention was placed to boundary layer modelling to examine ability of the models in capturing turbulence scales through viscous sublayer and the area beyond that.

Numerical Modelling

The FLUENT 16.2 code has been used for unsteady numerical simulations in this study. The code uses a finite volume approach, a staggered grid layout and employs SIMPLE algorithm for pressure-velocity decoupling. The pressure and turbulence equations are discretised by PRESTO and second order upwind schemes respectively. Energy equation is discretised using the second order upwind scheme. The second order upwind scheme (central differencing in LES, DES and SAS) was adopted for momentum equation, and the second order implicit scheme for time advancement. The average courant number used in the simulations was maintained at about 1.1 which gives a time step as small as 0.001s and a typical simulation included 12000 time steps. The solutions were iterated 2 times per time step. This enables one to monitor the stability of the solution through transport equations where the convergence is achieved with values smaller than 5×10^{-4} depending on the model used.

Geometry and Boundary Conditions

A schematic representation of the helical pipe with its main geometrical parameters considered in this study is shown in Figure.1 which is geometrically identical to the one used in the experimental tests [7]. The coil diameter, D, and the pipe diameter, d, are 0.25044 m and 0.0125 m respectively. Accordingly, curvature ratio ($\delta = d/D$) is equal to 0.05 where coil pitch is set as 2πb and the inner and outer wall of the pipe are indicated with *I* and *O* respectively.

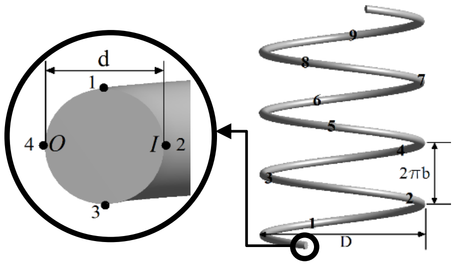

Figure 1. Schematic representation of the helical pipe indicating 4 circumferential thermocouples (left), 9 longitudinal locations (right).

High length of the pipe (3.9 turns) enables one to compare the simulations results with the experimental results where 9 longitudinal thermocouples locations at equal 0.3048 m intervals were installed. At each location, 4 thermocouples were placed around the circumference of the pipe cross-section, each 90 degrees apart, giving a total of 36 point of temperature measurements. The four circumferential locations (at each location) are numbered clock-wise.

The working fluid, water, is assumed to be an incompressible Newtonian fluid with temperature-independent fluid properties. Fluid enters the helical pipe at an inlet temperature of 326.93 K and flows steadily through the coil under transient/turbulent flow conditions (Re~4552). At the inlet, uniform profiles for all the dependent variables are employed and constant velocity and thermal conditions are applied to the inlet of the coil. A uniform heat flux of 48884 w/m^2 is applied throughout the pipe wall which is assumed to have no slip boundary conditions. Turbulence intensity and hydraulic diameter, equal to 5.6% and 0.0125 m respectively, are imposed in the inlet section for all turbulence models which enables one to have a fair comparison between the models. For all the models, the turbulence intensity is used to randomly perturb the instantaneous velocity field at the inlet which accounts for the stochastic components of the inlet flow. Plus, the flow is randomly perturbed using spectral synthesizer, a fluctuating velocity algorithm, for all the turbulence models. This perturbation technique is not employed in the URANS model where the perturbation and the small turbulence scales are dampened. In order to reduce simulation time and mitigate risk of obtaining false results, result of Unsteady Reynolds-Averaged Navier–Stokes (URANS) solution were used for solution initialisation of rest of the turbulence models. The pressure is fixed in the outlet section (pressure-outlet boundary condition).

Computational Mesh

The finite volume computational grid is hexahedral and multi-block structured composed of rectangular cells; it is characterized by the parameters A, B and C as shown in figure 2, where A=B. A 12-block grid has been defined aiming to systematically save on computational costs, reduce the aspect ratio in the near wall region and more importantly diminish y^+ fluctuation associated with variation of radial dimension of cells adjacent to the wall (located in the same distance from the wall). The main problem with conventional 5-block grid used in [3-6] is that it results in relatively higher y^+ variations due to double value of A.

For all the selected turbulence models a maximum y^+ value of 1 would ideally meet the grid criteria (in case no wall functions being used). Additionally, LES requires highly refined mesh both in wall parallel plane ($\theta^+ < 2y^+$) and in stream-wise direction ($x^+ < 4y^+$) with a smooth growth ratio in the radial direction no more than 1.03. However, such refinement is not possible with the current computational resources.

The cross section of the computational mesh was resolved by 3200 volumes. In the stream-wise direction, the domain was discretised by 4000 cells, so that the total number of control volumes was N=12800000. Geometric refinement was introduced at the wall, with a growth ratio of ~1.13 in the radial direction. The average value of y^+ (normal to wall) at the wall adjacent cell was 0.95 not exceeding a maximum of 2.8. The viscous sublayer ($y^+ \approx 11$) was resolved by ~3 grid cells.

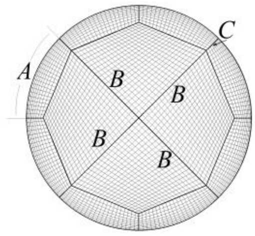

Figure 2. The computational grid used to perform the computations.

Turbulence modelling

The SST k-ω model, developed by Menter [8], has been extensively used by different researchers to predict hydrodynamic and thermal characteristics of turbulent flow in curved ducts. It is formulated to solve the viscous sublayer explicitly, which requires high mesh resolution inside this layer. The turbulence kinetic energy, k, and the specific dissipation rate, ω, are obtained from the following transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + Y_k + S_k \tag{1}$$

And

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + G_\omega + D_\omega + Y_\omega + S_\omega \tag{2}$$

where the first term in the right hand side of the equations represents the effective diffusivities in which eddy viscosity is computed by combining k and ω. G_k and G_ω are the production terms, Y_k and Y_ω account for dissipation terms and S_k and S_ω stand for user-defined source terms. D_ω represents the blending function term. In the near-wall region, D_ω activates the standard k-ω model, whereas away from the surface the function deactivates it and activates the transformed k–ε model.

Some features make the SST k-ω model more accurate for the streamline curvature and adverse pressure gradient flows than other RANS models; the turbulent viscosity is modified to account for transport of the turbulent shear stress; the model incorporates a damped cross-diffusion derivative term in the equation; and the modelling constants are different. However, a Reynolds correction coefficient has been applied to the model near the wall to account for radial velocity gradient towards the wall. Furthermore, a multiplier of the production term (curvature correction) has been used to account for streamline curvature effects in the model.

The next model which is frequently used to account for several effects in curved geometries in a more rigorous manner than (U)RANS models, is RSM. This model has supposedly a great potential to predict turbulence induced by curvature and secondary flow. However, the accuracy of RSM depends on the geometry, grid refinement and closure assumptions employed to model various terms in the exact transport equations for the Reynolds stresses. Diffusion terms in the Reynolds stress transport equations are treated by a simple eddy diffusivity approach, whereas the modelling of pressure-strain and dissipation-rate terms is challenging. Therefore, due to the additional computational cost, using the RSM is not justified in all classes of flow, because RSM might not always yield results that are clearly superior to the one-equation and two-equation models. The exact equation for the transport of the Reynolds stresses, is written as follows:

$$\frac{\partial\left(\rho\overline{u_i'u_j'}\right)}{\partial t}+\frac{\partial\left(\rho u_k\overline{u_i'u_j'}\right)}{\partial x_k}=D_{T,ij}+D_{L,ij}+P_{ij}+G_{ij}+\phi_{ij}+\varepsilon_{ij}+F_{ij}+S \quad (3)$$

where terms presented in the right side of the equation represent turbulent diffusion, molecular diffusion, stress production, buoyancy production, pressure strain, dissipation production by system rotation and user-defined source term respectively. The low-Re stress-omega closure, which is a stress-transport model based on the omega equations and LRR model [9] has been used in this study. This model is able to predict for a wide range of turbulent flows and requires no wall functions. Plus, low Reynolds number modifications applied are similar to the k-ω model.

The alternative to the URANS model and RSM is LES in which the large-scale field is directly computed from the solution of the filtered Navier-Stokes equations, and the small-scale stresses are modelled. The main challenge in LES of wall-bounded flows is that the largest scales in the turbulent spectrum near the wall are geometrically very small and need excessive spatial and temporal refinement. On top of that, unlike URANS, the grid must be refined in wall parallel plane in addition to the wall normal direction. For these reasons, the computational cost involved with LES is normally orders of magnitudes higher than that for unsteady RANS calculations.

The Wall-Adapting Local Eddy-Viscosity (WALE) model proposed by Nicoud and Ducros [10], as the most balanced LES model in such cases, is used in this study. Ma et al. [11] found out that the results of WALE model agree well with experimental results in a closed curved duct. In the WALE model the eddy viscosity is modelled by:

$$\mu_t=\rho\left[\min\left(0.4187d,0.325V^{1/3}\right)\right]^2\frac{\left(S_{ij}^dS_{ij}^d\right)^{1.5}}{\left(\bar{S}_{ij}\bar{S}_{ij}\right)^{2.5}+\left(S_{ij}^dS_{ij}^d\right)^{1.25}} \quad (4)$$

where d represents the distance to the closest wall, $V^{1/3}$ accounts for the local grid scale which is computed according to the volume of a computational cell and S_{ij}^d is defined as bellow:

$$S_{ij}^d=\frac{1}{2}\left[\left(\frac{\partial\bar{u}_i}{\partial x_j}\right)^2+\left(\frac{\partial\bar{u}_j}{\partial x_i}\right)^2\right]-\frac{1}{3}\delta_{ij}\left(\frac{\partial\bar{u}_k}{\partial x_k}\right)^2 \quad (5)$$

The current study is an attempt to investigate how applicable LES is in prediction of heat transfer and hydrodynamics characteristics of flow in helical pipes, compared with URANS models. When it comes to LES of turbulent flows in curved passages, there is a huge gap in the literature. Nevertheless, it is always questionable that whether additional computational resource required for LES will add to reasonable URANS accuracy or not. For this reason, scale-resolving simulation options such as DES and SAS models are identified as relatively cost-effective alternatives.

Hybrid turbulence models in this study use curvature correction modification method. The DES model employs the transient RANS model (SST k-ω model) in the boundary layer, while applies the LES treatment to the core turbulent region where large turbulence scales and secondary vortices play a dominant role resulting the DES models recover LES-like subgrid models.

The SAS model, is an improved URANS formulation, which allows the resolution of the turbulent spectrum in unstable flow conditions. The model behaves similar to DES models but without an explicit influence of the grid spacing on the URANS mode of the model which allows for a safer passage from URANS to LES, especially for complex flow applications where generation of high quality computational grids for the detached flow regions is prohibitive. At the same time, the model provides standard RANS capabilities in stable flow regions.

Results and Discussions

The numerical results of single phase convective heat transfer in the turbulent regime are presented to compare accuracy of selected models in capturing turbulence scales and thermal characteristics using experimental validations. The temperature at the four circumferential locations obtained by the modelling and the corresponding experimental tests are separately plotted along the coil and accuracy of the models are examined. Following that, thermal characteristics and flow patterns (dimensionless helicity function) are correlated and discussed.

(a) At top wall

(b) At inner wall

(c) At bottom wall

(d) At outer wall

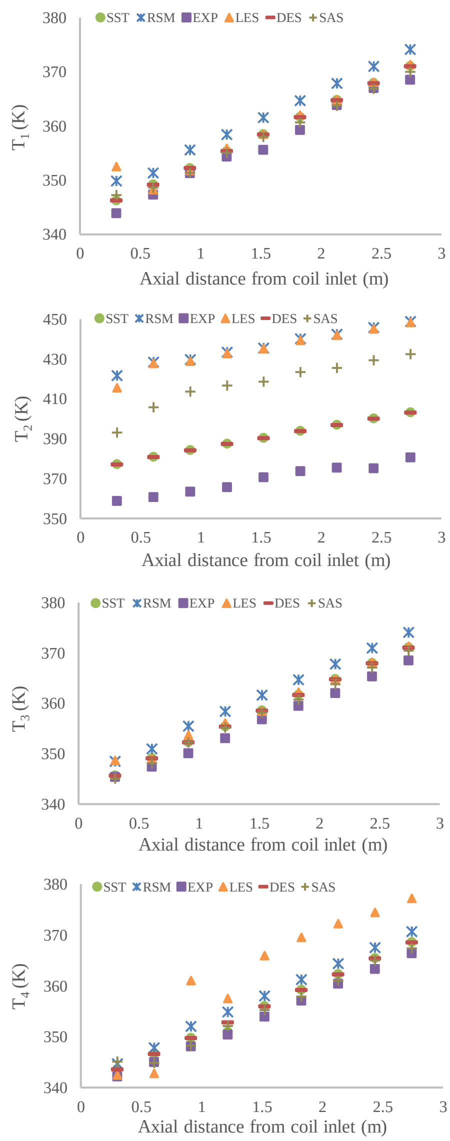

Figure 3.Wall temperature in in the four circumferential locations.

Figure 3 shows temperature as a function of axial distance from coil inlet in the four circumferential locations. This figure compares ability of the models to reproduce the wall temperature at the 4 sides of the coil. Both boundary layer and mean flow characteristics are influenced by unique pattern of secondary flow which implies unevenness to cross sectional flow pattern. Interestingly, the LES model, which directly resolves large eddies and models small eddies, and the RSM overestimate wall temperature, while the results of the SST model are relatively more consistent with the results of the experiment. This simply means temperature and velocity gradients are poorly estimated at wall. The scale-resolving simulation models, behave differently when it comes to prediction of thermal conditions at the tube wall. The DES model gives almost the same results as the URANS does. However, the results of SAS model shows lowest deviation from the experimental values except at the inner wall where the model largely overestimates the inner wall temperature. Generally, inner wall temperature is highest in heated curved passages due to influence of secondary flow mechanism. The discrepancy between the result of numerical modelling and experimental measurement in inner and outer wall (T_2 and T_4) is relatively higher than top and bottom wall (T_1 and T_3). This is attributed to the effect of secondary flow separation in Inner-Outer direction which is absent in the vertical direction. This could be explained based on the mechanics of secondary flow vortex generation. In capturing the helix-like fluid motion of secondary flow a dimensionless helicity function is defined as:

$$H^{+} = \left[u\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + v\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + w\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \right] \left(\frac{D_{hydraulic}}{U_{inlet}^{2}}\right) \tag{6}$$

A typical flow profile in terms of helicity contours in the helical pipe cross section at exit obtained by different turbulence models is shown in figure 4. The DES and SST models give valid prediction of the profile. However, the LES and, to some extent, the RSM flow patterns suggest that the incorrect prediction of Dean vortex structure near the outer wall is more likely to cause poor estimation of wall thermal characteristics which is closely dependent on the low quality of boundary layer mesh. Moreover, study of pressure gradient profile along the outer wall boundary shows that there is a strong affiliation between the outer wall fluid pressure profile and the hydrodynamic instability induced by secondary flow [12]. LES models, are unable to resolve small turbulence scales especially in boundary layer, unless the models meet the LES meshing criteria. The huge discrepancy shown in figure 3, is associated with inability of the LES model to resolve smaller eddies in the viscous sublayer region responsible for heat transfer between the wall (especially the inner wall) and mean flow area which brings about incorrect simulation of Dean vortices.

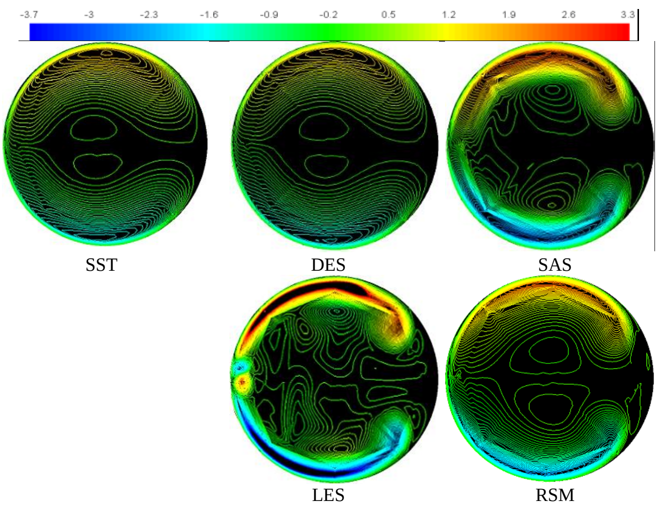

Figure 4. Helicity contours at pipe exit obtained from different models.

Conclusions

The current study is an attempt to assess the validity and applicability of URANS, LES, RSM and Hybrid models for prediction of heat transfer in helical pipes as an extreme case of internal flow in a curved geometry. Wall temperature values obtained from various turbulence schemes are evaluated against the corresponding experimentally measured values. The results show that even using a mesh with $y^{+}_{ave} < 1$ and $y^{+}_{max} = 2.85$, the LES model is unable to correctly estimate flow and thermal behaviour in boundary layer where boundary mesh is not able to resolve smallest turbulence scales. This shortfall is not solely limited to wall heat transfer and any wall-induced structure in mean flow (e.g. Dean vortices) could be poorly estimated. Providing a turbulence energy spectrum would be a means to comprehend the refinement level of LES turbulence scale toward the wall where LES results show a high deviation from the experimental values.

References

[1] Chen, W.H. & Jan, R., The characteristics of laminar flow in helical circular pipe, *J. Fluid Mech.*,**244**, 1992, 241–256.

[2] Ciofalo, M., Di Liberto, M., & Marotta, G., On the influence of curvature and torsion on turbulence in helically coiled pipes, *31st UIT Heat Transfer Conf. J. of Physics*, Italy, 2013.

[3] Castiglia, F., et al., Modelling flow & heat transfer in helically coiled pipes. Part 3: Assessment of turbulence models, parametrical study & proposed correlations for fully turbulent flow in the case of zero pitch, *ENEA*, Italy, 2010.

[4] Di Piazza, I. & Ciofalo, M., Numerical prediction of turbulent flow and heat transfer in helically coiled pipes., *Int. J. Therm. Sci.*,**49**, 2010, 653–663.

[5] Di Piazza, I. & Ciofalo, M., Transition to turbulence in toroidal pipes, *J. Fluid Mech.*,**687**, 2011, 72-117.

[6] Colombo, M., Cammi, A., & Ricotti, M.E. Assessment of different turbulence models in helically coiled pipes through comparison with experimental data, *Proc. 20th Int. Conf. Nuc. Eng.*,ASME, Anaheim, California, USA, 2012.

[7] Owhadi, A., *Boiling in self-induced radial acceleration fields*, Chemical Engineering, Oklahoma State University: Ann Arbor, Michigan, 1966,

[8] Menter,F.R.,Two-equation eddy-viscosity turbulence models for engineering applications. *AIAAJ.*,**32**, 1994, 269-289.

[9] Wilcox, D.C., *Turbulence modeling for CFD.*, DCW Industries, Inc., La Canada, California, 1998.

[10] Nicoud, F. & Ducros, F., Subgrid-scale stress modelling based on the square of the velocity gradient tensor, *Flow, Turbulence and Combustion*, **62**, 1999, 183-200.

[11] Ma, J., Wang, F., & Tang, X., Comparison of several subgrid-scale models for large-eddy simulation of turbulent flows in water turbine, *The 4th Int. Symposium on Fluid Machinery and Fluid Engineering*, Beijing, China, 2008.

[12] Chandratilleke, T.T., Nadim, N., & Narayanaswamy, R., Vortex structure-based analysis of laminar flow behaviour and thermal characteristics in curved ducts, *Int. J. Therm. Sci.*,**59**, 2012, 75-86.