ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

LAMINAR FLOW INSIDE SQUARE CROSS SECTION 180^{O} CURVED DUCT: NUMERICAL SOLUTIONS AND EXPERIMENTAL VISUALIZATION

Carlos Alberto de Almeida Vilela
Universidade Federal de Goias – Av. Universitária, 1488, BL A, Piso 3, Setor Leste Universitário, Goiânia/GO
carlosavgeo@yahoo.com

Abstract. *High curvature flow inside 180^{o} curved duct is numerically studied using the Navier-Stokes equations for laminar flow. 3D model finite volume method with Power-Law spatial discretization scheme was used for numerical calculations and smoke dispersion technique used to experimental visualization in a range of Reynolds and Dean numbers, $100 \leq Re \leq 2500$ ($100 \leq De \leq 2500$) considered to cover the theoretical laminar flow regime range, $Re \leq 2500$. Smoke transport was numerically treated as a specie transport in a flow field previously calculated. As numerical results, pressure coefficient was calculated in the bend, cross section pressure distribution, streamlines and smoke dispersion simulation is presented, also flow patterns in bend cross section at 90^{o} and 180^{o} cutting planes were captured with high velocity camera and show good agreement with numerical solutions.*

Keywords: *180^{o} curved duct flow, high curvature flow, finite volume method, laser-smoke visualization technique.*

1. INTRODUCTION

Curved duct configuration is very common to find in industrial systems, e.g. air conditioner distribution ducts, heat exchangers and water distribution pipe system. Flow patterns in curved duct are quite different than in straight duct due the presence of secondary flows that appears as result of geometric curvature and coriolis forces. Strongest is the duct curvature, strongest will be the coriolis forces and Dean vortices will arise turning the flow to instable, periodic and chaotic state, as discussed by Stephens and Shih (1999), Mondal *et al.* (2007) and Mondal *et al.* (2009). Finding the stability transition point is something that has been search by many authors in a numerical way, Yang and Wang (2003), Wang and Yang (2004), Mondal *et al.* (2007).

Detailed studies of surface curvature and Reynolds number in laminar and turbulent boundary layers and heat transfer coefficients are presented by Bradshaw (1973), Muck *et al* (1985), Hoffman *et al.* (1985), Chung and Hyun (1992). York *et al*. (2008) comments that economically simplifying assumptions considered in turbulence models, k-ε from Launder and Spalding (1972) and k-ω from Wilcox (1988) applied RANS equations are generally insensitive to bending or rotation of the flow. They use 2nd Order Upwind scheme with SIMPLE method to pressure coupling also introduce additional terms in the governing equations to represent the effects of rotational and curvature flow. The effect of curvature on turbulent flow is also observed by Bradshaw (1973), who comments that this effect can significantly affect the field of turbulence influencing the mean field runoff. It is well documented in the literature that convex curvatures tend to stabilize and minimize turbulence levels and concave curvatures destabilize and increase levels of turbulence, Muck *et al*. (1985) and Hoffman *et al*. (1985).

Experimental and numerical studies have been developed through the years in searching for instabilities characterization. The numerical solution quality can be measured by its proximity with experimental observations. A good numerical method will become instable when the physical phenomena become instable as well.

In this particular flow case, the presence of coriolis forces, which are consequences of the strong surfaces curvature, will induce secondary flows in normal planes in and after the bend. These physical disturbances will be notice as numerical instabilities that lead to divergent solutions. It's well known that in general higher order schemes is a good choice to overcome numerical instabilities and Upwind based schemes may smooth the solution.

Goodarzi and Azimian (2003) uses the 1st and 2nd Order Hybrid schemes and QUICK to solve a laminar Re=790, 3D strong curvature flows and the conclusion was that 2nd Order Hybrid scheme is more stable than QUICK, so can be use more aggressive relaxation index. Chung and Hyun (1992) also use a Hybrid scheme for 90º curved duct and obtained good results for a limit Reynolds 2000. Zijlema *et al* (1994) use a Blending scheme, a combination of QUICK and 1st Order Upwind, but they found to be too diffusive.

The flow visualization is carried out using the method of dispersion of smoke in contrast to a light plane generated by a laser source. This method is widely used for visualization of streamlines in the flow field, and these lines of smoke can be injected through small pipes or by hot wire coated with oil. The first technique demand a more sophisticated apparatus to control the smoke input but it has an advantage that is the possibility to input smoke at the same place all the time. The traditional hot wire technique has a peculiar aspect that may become inconvenient to some flow configurations, the smoke generated last only a few seconds, but it has an advantage to generate very thin layers of smoke. Lee and Lee (2001) in their work to visualize the boundary layer in grooved surfaces, use the technique of hot-wire smoke combined with the projection of laser light planes. The method used in this work differs from the other ones by how the smoke is injected to the main stream and the advantage is that the smoke may last longer.

2. PROBLEM GEOMETRY, EQUATIONS AND BOUNDARY CONDITIONS

As long as the flow regime is treated as steady and laminar, the governing equations for isothermal cases are the 3D continuity equation and the Navier-Stokes equations with constant properties.

$$\nabla \cdot (\rho \vec{v}) = 0 \tag{1}$$

$$\nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\mu (\nabla \vec{v})) + \rho \vec{g} \tag{2}$$

The smoke diffusion is treated as a species transport governing by a convection-diffusion equation.

$$\nabla \cdot (\rho \vec{v} Y_i) = \nabla \cdot (\rho D_{i,m} \nabla Y_i) \tag{3}$$

where
Y_i is the transported species,
J_i is the diffusion flux which arises due to concentration gradient,
D_i is the mass diffusion coefficient for species in the mixture.

The problem geometry and numerical boundary conditions are described in Fig. (1) and the test section general dimensions are presented at Tab. (1).

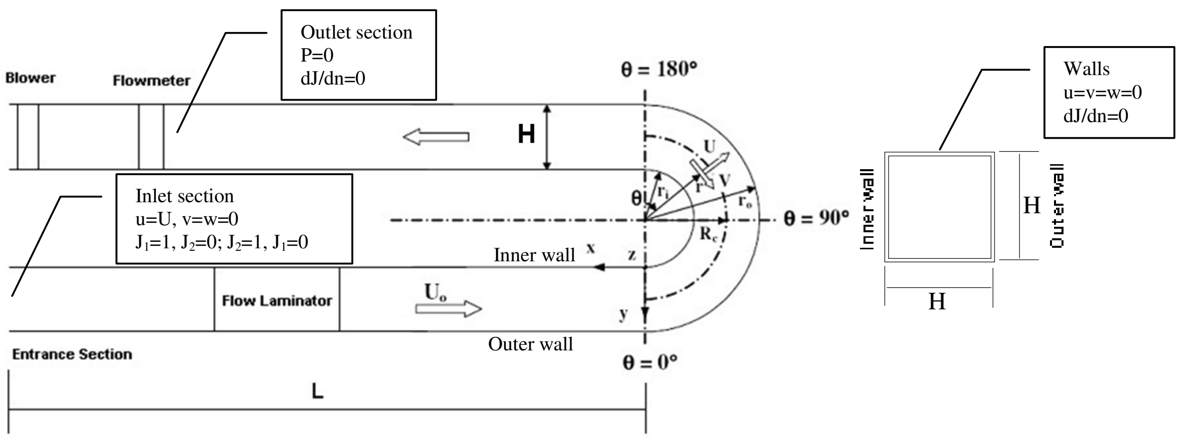


Figure 1. Problem geometry and boundary conditions for U-bend flow problem. Plan and cross section view.

The inlet section for species transport is treated as a check board configuration as seen in Fig. 2a, where one board is a fully injection of specie J_1 (smoke) and the neighbors are fully injection of specie J_2 (air), and so on until complete all cross section.

Table 1. Geometry specifications

	Experimental domain	Numerical domain
L_{in}	10H	5H fully developed
L_{out}	10H	14H
H	0,1	0,1
R_c	0,1	0,1

Reynolds and Dean numbers are calculated based on duct height as follow. Considering the air physical properties at 20 °C, it results:

$$\mathrm{Re} = \frac{\rho U H}{\mu} \tag{4}$$

$$De = \mathrm{Re}\sqrt{\frac{H}{R_c}} = 1.\mathrm{Re} \tag{5}$$

ISSN 2176-5480

3. NUMERICAL PROCEDURE

All computations were performed with FLUENT software. Some preliminary tests revealed that for the flow conditions, grid arrangement and linear system solver options considered in this work, no convergence were obtained with QUICK and 2nd Order Upwind schemes. No grid refinement analysis was performed because the numerical results seems to be more sensitive to discretization scheme order than to grid refinement. The 1st Order Upwind shown to be very diffusive, smooth the recirculation zones and secondary flows that should appears. For this reason were used the Power Law scheme, that achieved convergence for all Reynolds number considered. The grid configuration and numerical parameters used in the software are presented at Fig. (2) and Tab. (2).

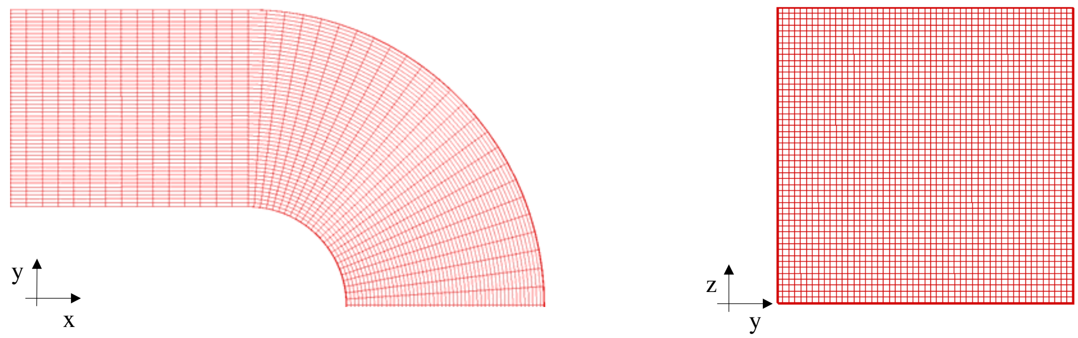


Figure 2. Grid configuration on **xy** and **yz** planes

Table 2. Numerical parameters and grid characteristics

Discretization scheme for momentum equations	Power Law
Discretization scheme for species transport	QUICK
Pressure discretization	Standard
Velocity-pressure coupling	SIMPLEC
Grid density: equally spaced grid refinement	x-y plane : straight inlet section: 200x50 divisions x-y plane : straight outlet section: 600x50 divisions x-y plane : curve section: 100x50 divisions y-z plane : cross section: 50x50 divisions

4. EXPERIMENTAL VISUALIZATION APPARATUS

An experimental apparatus were built with a laser-smoke equipment to flow visualization. A sketch of the apparatus is show at Fig. (3) and a real photograph is show at Fig. (4).

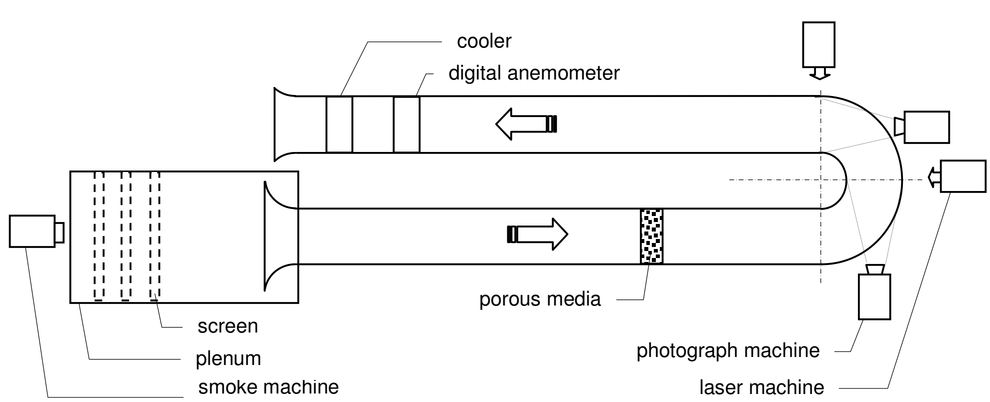


Figure 3. Experimental arrangement used to flow visualization.

ISSN 2176-5480

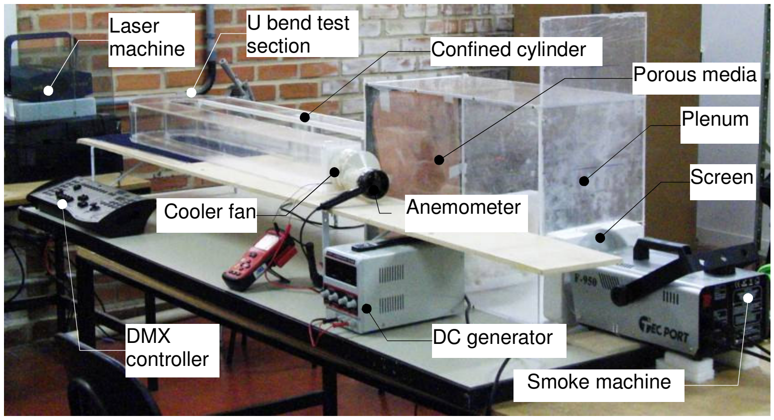


Figure 4. Workbench arrangement for flow visualization on U Duct and cylinder confined configurations

The smoke generator machine used was a PORT-TEC brand, F-950 model. The smoke is stored in a 300x300x150 mm plenum divided by three nylon screens located at inlet straight section. The laser machine used was a Optilux brand, OP-3022 Laser Aerial Device model that works together with a DMX Controller Table, EXELL brand, model DMX-512 Controller Operator II. A high resolution camera CASIO brand, model EXILIM EX-F1, was also used to capture the images. The flow was measured with a digital anemometer Instrutherm brand, model TAD-500 assembled on the output section of the circuit and a fan (cooler) 12V DC, was controlled by a DC generator Instrutherm brand, model Power Supply FA-3005.

5. RESULTS

Table 3 presents the results obtained by the experimental flow visualization and numerical simulation in cross section planes located at 90 º and 180 º. In the figures on the left side is the inner surface of the duct section and the right side is the outer surface of the duct section. It is observed in the results presented that there is a good agreement between the experimental and numerical dispersion of smoke.

For the case of low Reynolds number, Re <500, is possible to see the typical formation of two great recirculation zones located in the plane 90º, but this structure is maintained until the plane 180º just in Reynolds number 100. Observing the table is easy to see that the flow profile in the plane 180° becomes more unstable that the flow in the plane at 90°, and this is due to the effects of Coriolis forces.

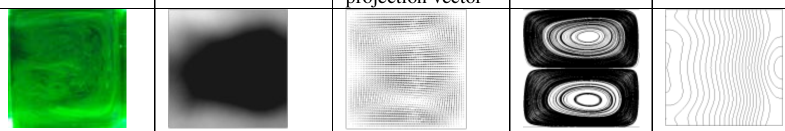
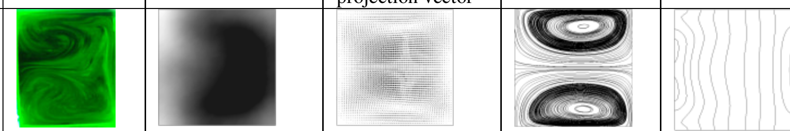
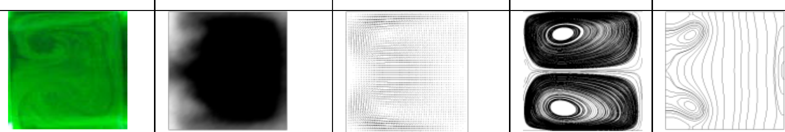
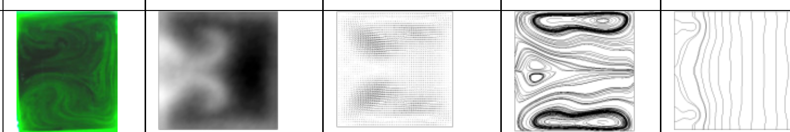
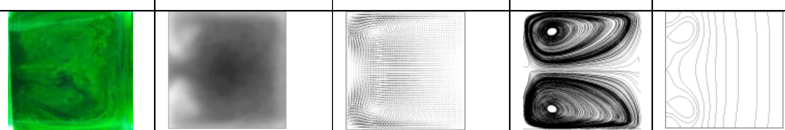
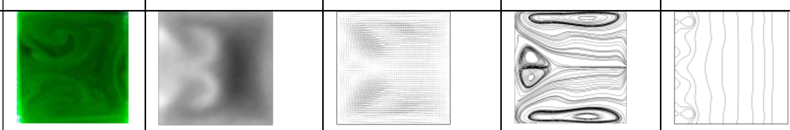
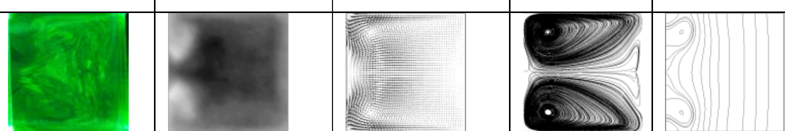
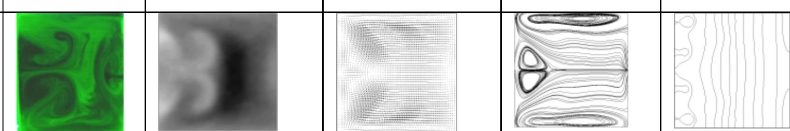
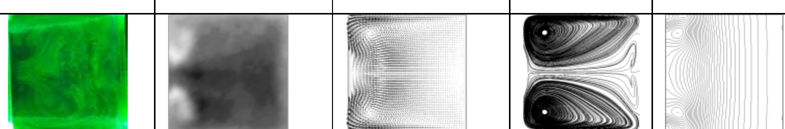
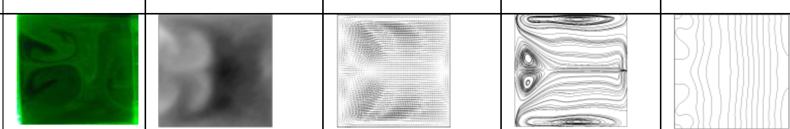
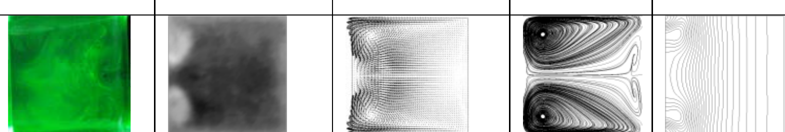
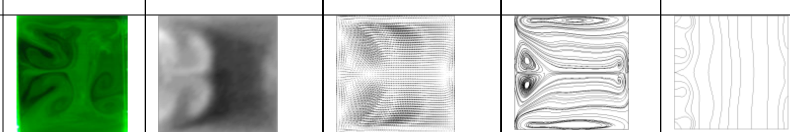
Experimentally it was observed that the flow is stable for cases Reynolds less than 1500. For cases above this value, the formation of the secondary recirculation zone outer surface of the duct cross section have an oscillating motion up and down turning the flow to be unstable. This is evident in the case of Reynolds number 2500 in the plan 180º.

The low order discretization schemes or even QUICK showed no satisfactory results in the calculation of the flow in curved duct. With the low-order schemes, the numerical convergence was achieved, however the secondary recirculation were suppressed due to numerical diffusion and with higher-order schemes like QUICK, the secondary recirculation were revealed, but the levels were not achieved satisfactory convergence due to numerical oscillations.

Goodarzi and Azimian (2003) uses the schemes Hybrid 1st and 2nd Order and QUICK, for a 3D solution with Reynolds number 790 and commented that the 2nd Order scheme is more stable than the QUICK, and because of that is possible to use more aggressive relaxation coefficients although both present solutions close. The Power-Law scheme was presented the best results regarding the convergence and calculation of secondary recirculation zones for all values of Reynolds number considered in this work.

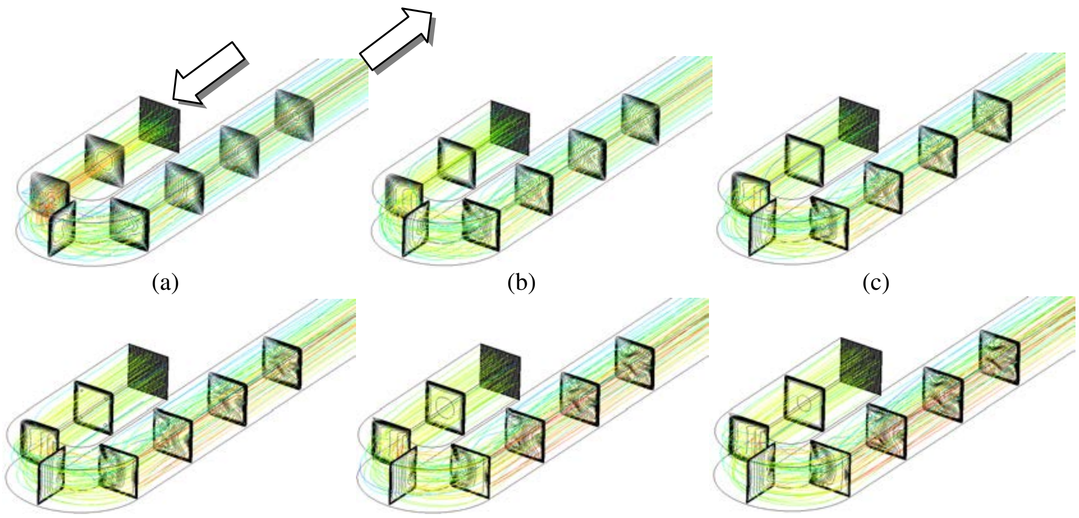
ISSN 2176-5480

Table 3. Experimental and numerical results.

	90º					180º				
Flow Conditions	Smoke visualization	Numerical smoke dispersion	Velocity tangential projection vector	Plane streamlines	Pressure distribution	Smoke visualization	Numerical smoke dispersion	Velocity tangential projection vector	Plane streamlines	Pressure distribution
Re=100 De=100										
Re=500 De=500										
Re=1000 De=1000										
Re=1500 De=1500										
Re=2000 De=2000										
Re=2500 De=2500										

ISSN 2176-5480

Figure 5 shows the distribution of streamlines along the curved duct and the iso-velocity patterns in different cross section planes.



(a) (b) (c) (d) (e) (f)

Figure 5. 3D streamlines and cross section iso-velocity. Re=100(a), 500(b), 1000(c), 1500(d), 2000(e), 2500(f)

6. CONCLUSION

The flow visualization method by smoke dispersion seems to be appropriate to this flow configuration. It is visible the consistency between the experimental and numerical qualitative results. A note should be make about this method is that no streamlines are formed as in traditional methods such as hot wire or smoke injection with small tubes, but a non-homogeneous structure is formed where smoke in lamellae can be seen. Because of this effect is that it becomes possible to flow visualization even after the test section, the air-smoke mixture does not become homogeneous due to the low diffusion of mass of smoke.

For cases of low Reynolds number, special care should be taken relative to the presence of natural convection phenomena that can arise in the flow, because the smoke temperature is higher than the air temperature and this can affect the flow structure. For the present study, the minimization of this effect was achieved from the moment the plenum section and stabilization tests were thermally insulated and laboratory air conditioning system of the where the experiment was performed, was switched off.

Regarding the numerical solutions, as already pointed out by several authors, the low-order schemes can achieve the numerical convergence but fail to reveal the structures of secondary recirculation, and higher order schemes can reveal these structures but are very unstable. The schemes identified as the most suitable for this type of flow are the Hybrid and Power-Law.

7. REFERENCES

Bradshaw, P., 1973, ‘‘The Effects of Streamline Curvature on Turbulent Flow’’, Gardograph N^o. 169.

Chung, J.H. and Hyun, J.M., 1992, “Convective Heat Transfer in the Developing Flow Region of a Square Duct With Strong Curvature”, International Journal of Heat Mass Transfer, Vol. 35, N^o. 10, pp. 2537-2550.

Goodarzi, M. and Azimian, A.R., 2003, “Calculation of Complicated Flows, Using First and Second-Order Schemes”, Ije Transactions A: Basics, Vol. 16, N^o. 3, pp. 269

Hoffman, P.H., Muck, K.C. and Bradshaw, P., 1985, ‘‘The Effect of Concave Surface Curvature on Turbulent Boundary Layers’’, Journal of Fluid Mechanics, Vol. 161, pp. 371-403.

Launder, B.E. and Spalding, D.B., 1972, “Lectures in Mathematical Models of Turbulence”, Academic Press, London.

Lee, S.J. and Lee, S.H., 2001, “Flow Field Analysis of a Turbulent Boundary Layer Over a Riblet Surface”, Experiments in Fluids, Vol. 30, pp. 153-166.

Mark A. Stephens, M.A., and. Shih, T.I., 1999, “Flow And Heat Transfer In A Smooth U-Duct With And Without Rotation”, Journal Of Propulsion And Power, Vol. 15, No. 2.

Mondal, R.N., Kaga, Y, Hyakutake, T., Yanase, S., 2007, “Bifurcation Diagram for Two-Dimensional Steady Flow and Unsteady Solutions in a Curved Square Duct”, Fluid Dynamics Research, Vol. 39, pp.413–446.

ISSN 2176-5480

Mondal, R.N., Islam, S. And Hossain, T., 2009, "Flow Instability Through a Curved Duct With Square Cross Section, J. Bangladesh Math. Soc, Vol. 29, pp. 71-85.

Muck, K.C., Hoffman, P.H. and Bradshaw, P., 1985, ''The Effect of Convex Surface Curvature on Turbulent Boundary Layers'', Journal of Fluid Mechanics, Vol. 161, pp. 347-69.

Wang, L, Yang, T., 2004, "Bifurcation and Stability of Forced Convection in Curved Ducts of Square Cross-Section", International Journal of Heat and Mass Transfer, Vol. 47, pp. 2971–2987.

Wilcox, D.C., 1988, ''Reassessment of the Scale-Determining Equation for Advanced Turbulence Models'', AIAA Journal, Vol. 26, pp. 1299-310.

Yang, T., Wang, T., 2003, "Bifurcation and Stability of Combined Free and Forced Convection In Rotating Curved Ducts of Square Cross-Section", International Journal of Heat and Mass Transfer, Vol. 46, pp. 613–629.

Zijlema, M., Segal, A., Wesseling, P., 1994, "Finite Volume Computation of 2D Incompressible Turbulent Flows in General Coordinates on Staggered Grids", Report 94-24, ISSN 0922-5641, Faculty of Technical Mathematics and Informatics, Delft, The Netherlands.

8. RESPONSIBILITY NOTICE

The author is the only responsible for the printed material included in this paper.