NUMERICAL INVESTIGATION OF LAMINAR FLOW IN SQUARE CURVED DUCT WITH 90° BEND

A. Ćoćić[1], **I. Guranov**[2], **M. Lečić**[3]
[1,2,3] Faculty of Mechanical Engineering, The University of Belgrade,
Kraljice Marije 16, 11120 Belgrade 35
e-mail: acocic@mas.bg.ac.rs, i.guranov@gmail.com, mlecic@mas.bg.ac.rs

Abstract. Numerical solution of incompressible laminar flow in square curved duct with 90° bend is performed in order to better understand the flow phenomena present in this type of flow. Simulation of flow was done in OpenFOAM, an open-source CFD software. Essentially, OpenFOAM is large C++ library which can be used to create application for solution of various problems in continuum mechanics. Besides object-oriented approach, the main advantage of OpenFOAM is that it is extendable, i.e. users can add their own applications and utilities. The solution for the flow investigated in this paper is for Reynolds number $Re = 790$. We used experimental results from literature as a validation tool for our simulation. The results of simulation showed very good agreement with experimental results. They also showed that centrifugal force convects the quickly moving fluid particles towards the outer wall. As a result, the axial velocity has peaks in curved region, and secondary flow with various number of vortices is also present. Development of flow in vertical straight section after the bend is also analyzed, where presence of swirl is detected.

1. Introduction

Fluid flow in curved ducts are practically inevitable thing in most devices and systems in mechanical engineering. We could only mentioned a few of them: turbomachines, ducts for air ventilation, heat exchangers, aircraft intakes, etc. Their practical importance motivated considerable research effort in the past decades, and it's still a motive today. In curved ducts, centrifugal and viscous (Tollmien-Schlichting) instabilities may exist and interact strongly [1]. The resulting nonlinear interaction between these two type of instabilities may cause the flow to evolve to exhibit turbulence. Advancing of knowledge about this three-dimensional curved flow is, thus, of fundamental importance. Nowadays, a large computer power, even on ordinary PC, enables efficient numerical simulations of flow in various engineering devices or systems. Results of simulations help us to better understand the flow phenomena by predicting flow structure and values of significant flow quantities.

We can say that main flow characterics in curved channels or pipes is formation of so called secondary flow. This was first pointed out by Eustice, [2] in his experimental work with flow in curved pipes. Main reason for formation of secondary flow is presence of centrifugal force and it has great influence on primary flow development. Also, centrifugal force causes that velocity is skewed towards the outer wall. This experimentally observed phenomenon was analytically confirmed by Dean, [3]. Presence of secondary flow increases dissipation of fluid mechanical energy, which implies a larger pressure drop. Also, wall shear stress distribution along the wall is not uniform, in contrast with straight channel or pipe.

Advance of PIV experimental equipment and techniques in past years enables visualization of these secondary flows. Jarrahi et al., [4] studied process of mixing by secondary flow in a developing laminar pulsating flow through a circular curved pipe. They noticed different secondary flow patterns during an oscillation period due to competition among the centrifugal, inertial, and viscous forces.

Besides experimental and analytical studies of these kind of flows, numerous numerical investigations are performed. Some relevant papers in that spirit are Patankar et al., [5], Humphrey, [6], Soh & Berger, [7], Tsai [8]. In papers by Soh [9] and Bara [10] studies of flow development in curved ducts with square cross-sections have been addressed on the determination of critical Dean number, above which the formation and disintegration of secondary flow can lead to multiple vortex pair solutions.

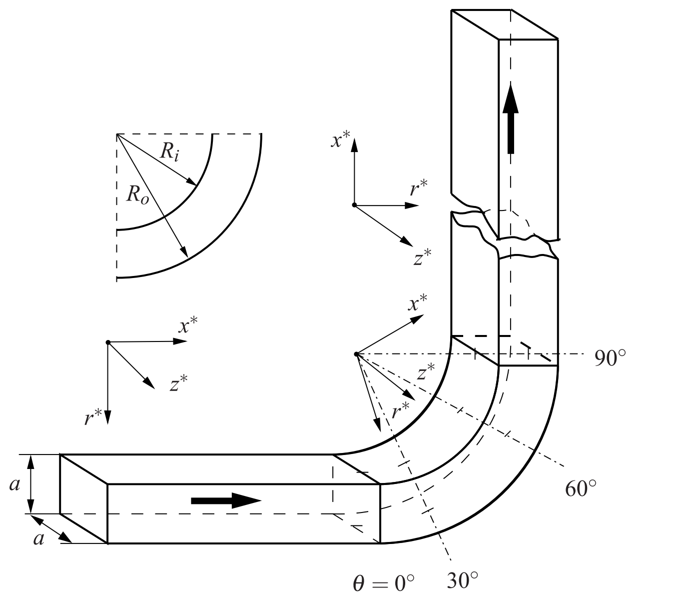
In this paper, we made a contribution to numerical studies of flow in curved duct. An open-source CFD software named OpenFOAM is used. OpenFOAM is essentially a large collection of C++ libraries which can be used to create application for solution of various problems in continuum mechanics. There are numerous advantages of open source software over commercial ones. First one is, of course, the fact that code is open; it is extendable and user can create their own applications and utilities, in contrast to "black-box" principle of commercial codes. Secondly, open-source principle implies formation of community in which people can freely communicate, cooperate and help each other in solving particular problems. They are not limited by software licenses, inability to adopt the software to their needs, etc. On the other hand, most open-source software have a lack of documentation, but on global scale every user of software give a small contribution to development of software and documentation.

The fact that OpenFOAM is written in C++ has big advantage over CFD codes written in procedural programming languages like FORTRAN and C. Object oriented approach in programming involves abstraction, inheritance and polymorphism. That enables implementation of complicated mathematical and physical models in code to be similar to high-level mathematical expressions, [11].

2. Problem description and governing equations

In this paper we simulated the flow which was studied experimentally by Humphrey, [6]. The sketch of duct is shown in Fig. 1. The flow configuration is a 90° bend of mean radius 92 mm attached to the end of the 1.8 m long squared channel with cross-section of 40×40 mm. Experiments in [6] were done by LDA technique and the measurements of all velocity field are available at three cross-sections before the bend, and also in three cross sections in the bend itself. These experimental results is a validation tool for our simulation. On the Fig. 1 characteristic coordinate systems are shown. Angle θ is measured along the bend, and coordinate $x^* = 0$ corresponds to the cross-section with $\theta = 0^\circ$. When we say $x^* = -2.5$ we are thinking on cross-section at the distance $L = 2.5 \cdot 40 = 100$ mm before bend.

Reynolds number in experiment, based on bulk velocity $V = 1.98 \cdot 10^{-2}$ m/s and hydraulic diameter $D_H = 40$ mm was $\mathrm{Re} = 790$. The Dean number of the flow was $\mathrm{De} = \mathrm{Re}(\frac{1}{2}d/R_c) = 368$, where $R_c = 0.5(R_i + R_o)$ is the mean radius of curvature.



Geometry: $a = 40\,\mathrm{mm}$, $R_i = 72\,\mathrm{mm}$, $R_o = 112\,\mathrm{mm}$

Non-dimensional coordinates: $z^* = \dfrac{z}{a/2}$, $\quad r^* = \dfrac{r - R_i}{R_o - R_o}$, $\quad x^* = \frac{x}{a}$

Figure 1. Sketch of computational domain, dimensions and non-dimensional coordinates.

2.1. Governing equations

The flow in channel and bend is incompressible, laminar and steady and it is described with continuity and momentum equations, which are, under that conditions

$$\nabla \cdot \vec{U} = 0 \tag{1}$$

$$\vec{U} \cdot \nabla \vec{U} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \vec{U}, \tag{2}$$

where P is generalized pressure. Form of continuity equation (1) for incompressible fluid enables that convection term in momentum equation can be written in following form: $\nabla \cdot \vec{U}\vec{U}$, where $\vec{U}\vec{U}$ is second-order tensor, whose component is $u_i u_j$. Constant density of fluid enables that we define kinematic pressure $p^* = P/\rho$. Now the momentum equation have the form

$$\nabla \cdot \vec{U}\vec{U} = -\nabla p^* + \nabla \cdot (\nu \nabla \vec{U}) \tag{3}$$

This form of momentum equation is called *strong conservation* form, which is suitable for discretisation by finite-volume method.

We need to add the following boundary condition to the equations in order to solve them for particular problem. For chosen coordinate system we can define components u_r, u_θ and u_z of vector $\vec{U}$. Now the boundary condition are:

$$x_H = -10 \text{ (inlet plane)}: \; u = \text{fully developed duct flow}, \; v = w = 0,$$

$$
\begin{aligned}
&\text{walls} && : u = v = w = 0 \\
&x_H = 10 \text{ (exit plane)} && : \partial_\theta u_\theta = \partial_\theta u_r = \partial_\theta u_z = 0
\end{aligned} \qquad (4)
$$

for velocity field, and

$$
\begin{aligned}
&x_H = -10 \text{ (inlet plane)} && : \partial_\theta p^* = 0 \\
&\text{walls} && : \partial_n p^* = 0 \\
&x_H = 10 \text{ (exit plane)} && : p^* = p_0^*
\end{aligned} \qquad (5)
$$

for pressure field. Index n in wall boundary condition for pressure designates a direction perpendicular to wall and p_0^* is specified, constant value of kinematic pressure in exit plane. With this boundary conditions it is possible to solve the system of equations (1)-(2), using numerical methods. It is done is next section.

3. Set-up of the problem and solution procedure in OpenFOAM

OpenFOAM, as mentioned before, is first and foremost a C++ library. It is divided into a set of precompiled libraries that are dynamically linked during compilation of the solvers and utilities. Libraries such as those for physical models are supplied as source code so that users may conveniently add their own models to the libraries, [12], [13].

At the top-level of OpenFOAM code are the solvers, each designed to solve a specific problem in computational continuum mechanics. Each solver is based on finite-volume method of discretisation, which consists of three steps:

- *spatial discretisation* where solution domain is defined by a set of points that fill and bound a region of space when connected - generation of numerical mesh. In that space points, face, cells and connection between them is defined.
- *equation discretisation* where system of algebraic equation is defined in terms of discrete quantities defined at specific locations in the domain. Starting equation is partial differential equation that characterize the problem.
- *temporal discretisation* where time is divided into a finite number of time intervals (steps) - for unsteady problems

Geometry in this problem is rather simple, and OpenFOAM application `blockMesh` is used for mesh generation. This geometry also implies that mesh is orthogonal, so we don't need to introduce some non-orthogonal correctors in discretised equations. The mesh is shown in Fig. 2.

In our problem we have laminar, stationary and steady flow of incompressible fluid, and the solver designed for that type of flow is `simpleFoam`. Object-orientated approach enables the representation of the equations in code in their natural language. So, the momentum equation in `simpleFoam` solver for laminar flow is represented as

```
fvm::div(phi, U) - fvm::laplacian(nu, U) == - fvc::grad(p)
```

The syntax of the equation is straight-forward and clear; `fvm`, FiniteVolumeMethod designates implicit method of discretization of equation terms, while `fvc`, FiniteVolumeCalculus designates explicit method.

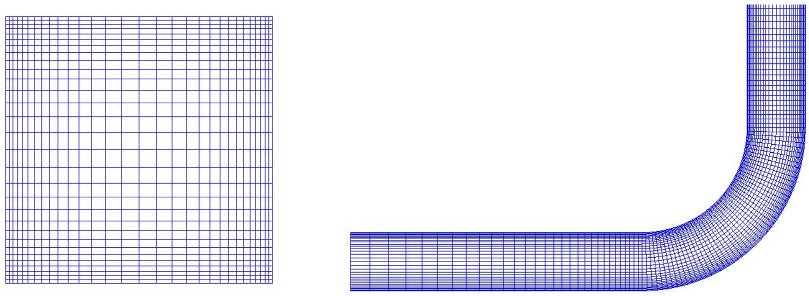


Figure 2. Numerical mesh for the problem.

Finite volume discretisation of each term is formulated by first integrating the term over a cell volume V. Most spatial derivative terms are then converted to integrals over the cell surface S bounding the volume using Gauss-Ostrogradski theorem

$$\int_v \nabla \star \phi \, \mathrm{d}V = \oint_s \vec{n} \star \phi \, \mathrm{d}A, \tag{6}$$

where ϕ represents any tensor field and the star notation $\star$ represents any tensor product, i.e. inner, outer and cross and the respective derivatives: divergence $\nabla \cdot \phi$, gradient $\nabla \phi$ and $\nabla \times \phi$. Volume and surface integrals are then linearized by some differencing scheme. Result of discretisation is set of algebraic equations for each cell and it can be expressed in matrix form as

$$[A]\{x\} = \{b\} \tag{7}$$

where $[A]$ is a square matrix, $\{x\}$ is the column vector of dependent variable and $\{b\}$ is the so-called "source vector", [13].

For velocity, we prescribed fully developed profile at inlet from previous simulation for straight channel, non-slip conditions at the walls and zero gradient at the outlet. For pressure, we prescribed zero gradient at inlet and walls, and fixed value $p^* = 0\,\mathrm{m}^2/\mathrm{s}^2$ (gage pressure) at the outlet. Outlet was placed long enough after the bend so that we have developed, unidirectional flow at the outlet. We used upwind differencing scheme for convective therm, and central differencing scheme for gradient and laplacian therm in equation (3). For solution of Navier-Stokes equation (3) characteristic thing is that there is no independent equation for pressure, whose gradients contribute to each of three momentum equations, [14]. These difficulties are overcomed by use of numerical procedure called SIMPLE algorithm, [15], which is implemented in OpenFOAM. Preconditioned conjugate gradient methods are used for iterative solutions for system of linear equations.

Three different meshes were used. First mesh had 36000 cells, second 72000 cells and third had 200000 cells. In order to achieve faster convergence results from first mesh were prescribed to mesh with greater density by use of `mapFields` application, and after that results from mesh with 72000 cells were prescribed to the mesh with 200000 cells. At the end, simulation on mesh with 300000 cells gave the same results as the simulation on mesh with 200000 cells, and solution independence of mesh resolution is obtained. We used non-uniform mesh, by increasing the grading near the wall, in order to capture small vortices which are formed in the corners of the inner and the end walls.

4. Numerical results

From the simulation, we can see that flow is symmetric over the r, θ plane, which is expected, and it's also confirmed by experiments. It can be seen the movement of the fluid away from the inner radius wall towards the outer radius wall in bend. This movement progresses throughout the bend and it's accompanied by secondary motion directed towards the side walls and along the outer-radius wall and towards the symmetry plane along the inner-radius.

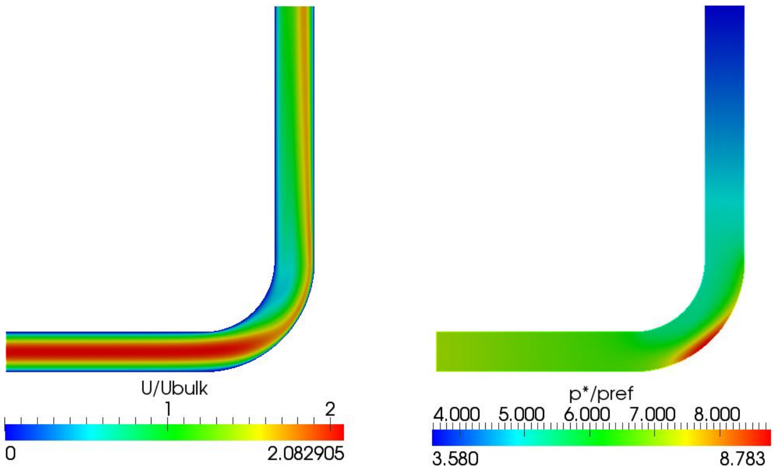


Figure 3. Velocity and pressure magnitude in middle plane of the channel - $z^* = 0$.

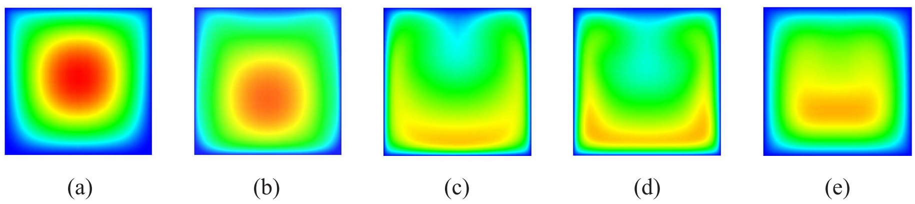


Figure 4. Contours of velocity magnitude in various cross-sections: (a) $\theta = 0°$; (b) $\theta = 30°$; (c) $\theta = 60°$; (d) $\theta = 90°$; (e) $x^* = 12.5$ after cross-section with $\theta = 90°$

Also, upstream of the curved duct, the flow is accelerated in regions near the inner-radius wall due to the favorable longitudinal pressure gradient. Conversely, the decelerated flow is observed in regions near the outer-radius wall because of the developed adverse pressure gradient downstream of $\theta = 0°$.

The comparison between experimental results in [6] and results of numerical simulations for axial velocity in (in direction of curvilinear coordinate x^*) is shown in Fig. 5. Results of simulation have very good agreement with experimental results, and all trends are well captured. We may argue that at cross-section of the bend for $\theta = 60°$ that agreement is not good. Also, with comparing measured results of velocity for constant r^* along the coordinate z^* similar trend is visible for cross-section around $\theta = 60°$. Some explanation for that remains unclear. One of possible explanations can be that error of measurement results in that section are too high.

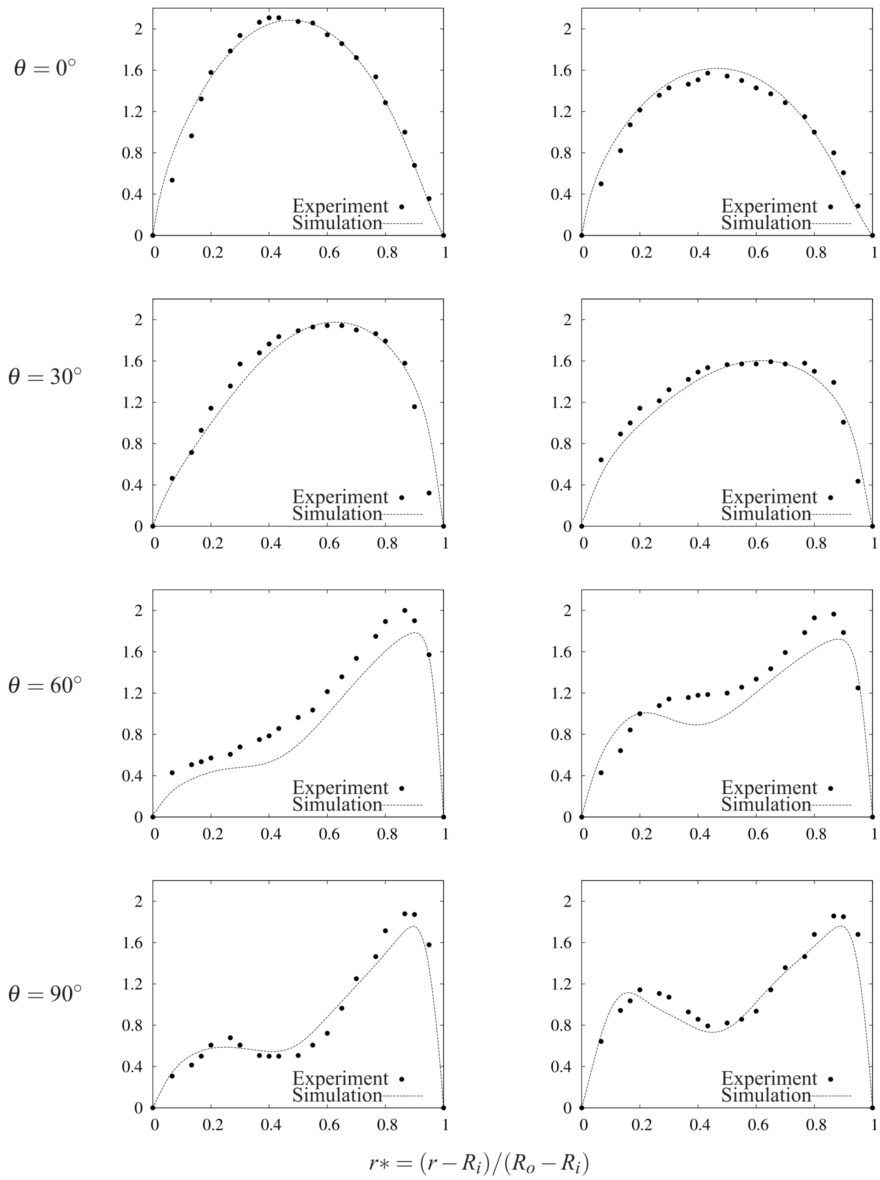


Figure 5. Comparison between measured and computed velocity profiles. Ordinate corresponds to non-dimensional velocity U/V; left column $z^* = 0.5$, right column $z^* = 0$.

From Fig. 5 the shifting of axial velocity peak towards the outer wall is also clearly visible. This is happening because of centrifugal effect. This degree of velocity skewness increases with the increasing turning angle of the flow and such a skewed axial velocity can persist very far downstream. Our simulation showed that distance needed to obtain fully developed profile after the bend is $L \approx 50D_H$.

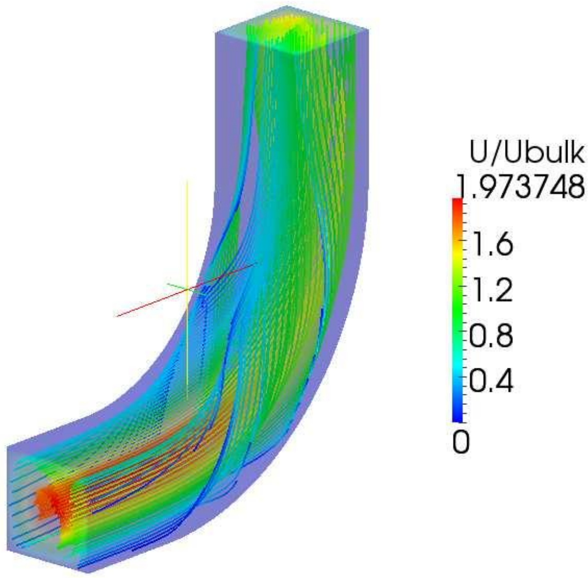


Figure 6. Some streamlines in the bend. It it visible that fluid particles are exhibited to swirling motion.

It is also very interesting to show the streamlines in such flow. Figure 6 shows some streamlines in 3D in the bend, and the presence of swirling motion is clearly visible. That swirling flow, like skewness of velocity profile, is maintained far downstream.

5. Concluding remarks

In this paper we made a numerical simulation of laminar flow of incompressible fluid in a curved duct with 90° bend in OpenFOAM. OpenFOAM is open-source CFD software, written in C++, and it's main advantage is that users can create their own applications and utilities. Results of numerical simulation show very good agreement with experimental results given in [6], which was used as validation tool for numerical results. They also give us a deeper insight in complex flow structure which is formed in bend. Also, effect of centrifugal force on flow structure is also spreading after the bend itself. We came to the conclusion that it takes $L \approx 50D_H$ after to bend to have fully developed velocity profile.

References

[1] Bennett J., Hall P., Smith F.T. (1991) The strong nonlinear interaction of Tollmien-Schlichting waves and Taylor-Görtler vortices in curved channel flow, *J. Fluid Mech.*, Vol. 223, pp. 475-495

[2] Eustice J. (1911) Experiments on stream-line motion in curved pipes, *Proc. R. Soc. Lond. Ser. A*, Vol. 85, pp. 119-131
[3] Dean W.R. (1928), The stream-line motion of fluid in a curved pipe, *Philos. Mag.*, Vol. 30, pp. 673-693
[4] Jarrahi M., Castelain C., Peerhossaini H. (2010) Secondary flow patterns and mixing in laminar pulsating flow through a curved pipe, *Experiments in Fluids*, Online first, published: 27 November 2010, DOI: 10.1007/s00348-010-1012-z
[5] Patankar S.V., Pratap V.S., Spalding D.B. (1974) Prediction of laminar flow and heat transfer in helically coiled pipes, *J. Fluid Mech.*, Vol. 62, 1974, pp. 539-551
[6] Humhprey J.A.C, Taylor A.M.K, Whitelaw J.H. (1977), Laminar flow in a square duct of strong curvature, *Journal of Fluid Mechanics*, Vol. 83, pp. 509-527.
[7] Soh W.Y., Berger S.A. (1984) Laminar entrance flow in a curved pipe, *J. Fluid Mech.*, Vol. 148,
[8] Tsai S.F., Sheu T.W.H. (2007), Numerical exploration of flow topology and vortex stability in a curved duct, *International Journal for Numerical Methods in Engineering*, Volume 71, Issue 5, pp. 505-629
[9] Soh W.Y.(1988) Developing fluid flow in a curved duct of square cross-section and its fully developed dual solutions, *J. Fluid Mech.*, Vol. 188, pp. 337-361
[10] Bara B., Nandakumar K., Masliyah J.H. (1992), An experimental and numerical study of the Dean problem: flow development towards two-dimensional multiple solutions, *J. Fluid Mech.*, Vol. 244, pp. 339-376
[11] Weller, H.G.; Tabor G.; Jasak, H. and Fureby, C. (1998) A Tensorial Approach to CFD using Object Orientated Techniques, *Computers in Physics*, Vol. 12 No. 6, pp 620 - 631
[12] OpenFOAM User Guide, 2010, http://www.openfoam.com/docs/
[13] OpenFOAM Programming Guide, 2010, http://www.openfoam.com/docs/
[14] Ferziger J.H., Perić (2002) Computational Methods for Fluid Dynamics, 3rd Edition, *Springer*, 2002
[15] Patankar S.V. and Spalding D. B. (1972) A Calculation Procedure for Heat, Mass and Momentum Transfer in 3-Dimensional Parabolic Flows, *Int. J. Heat Mass Transfer*, Vol. 15, pp. 1787-1806.