NUMERICAL SIMULATION OF 2D FLOW IN CARTESIAN GEOMETRY USING FINITE VOLUMES

Kéteri Poliane Moraes de Oliveira(*)
Keteri19@yahoo.com.br

Luciene Muniz Frigo(*)
luciene@dem.feis.unesp.br

João Batista Aparecido(*)
jbaparec@dem.feis.unesp.br

Sérgio Said Mansur(*)
mansur@dem.feis.unesp.br
(*) Universidade Estadual Paulista – Unesp - Departamento de Engenharia Mecânica. Av. Brasil, 56, Ilha Solteira-SP, CEP-15385-000, Brasil

Abstract. *In the present work the Large-eddy simulation (LES) methodology ha been applied for an analysis two-dimensional flow in a Cartesian geometry that presents an expansion and one contraction. The study is accomplished through a named Fluids developed for incomprehensible flows of Newtonian fluids. The analysis has been done in a growing range of Reynolds numbers, in order to study development of the turbulent structures. The continuity and the Navier-Stokesequations were discretized by the Finite Volume Method. The couplement between velocity and pressure has been done by algorithm SIMPLEC and the QUICK-C was used for the treatment of the advective terms. The results were compared with results from the literature and they presented good agreement*

***Keywords**:* Finite volume method, large-eddy simulation, Incompressible flow

1. Introduction

The flow in a knee duct has been a lot studied in the last years. In spite of its simple geometry, the flow presents high degree complexity being characterized by the presence of boundary layer development and re-circulation zone. Those phenomena gathered in an alone flow, turn the problem particularly appropriated for the validation of numerical codes gone back to the numeric simulation of flow..

The flow in this geometry was investigated by Perng & Street (1991) for number of Reynolds equal to 200 and four velocity profiles were traced. The first profile corresponded to the flow completely developed. In this study, the number of Reynolds, based on the height of the entrance channel, varied from 200, 400 and 1000 and five velocity profiles were traced, the first when the flow was already completely developed. Therefore, the first profiles, were not compared in this study.

In the present work, simulations were accomplished with the aid of a program developed to solve, through numeric solution of the of Navier-Stokes equations for two-dimensional problems in Cartesian geometries, whose details are presented by Campregher (2002). The equations of the movement are discretized in the space by the finite volume method. The couplement between velocity and pressure has been done through the method SIMPLEC, while the QUICK-C is used in the treatment of the advective terms. The main objective in this work is to analyze the turbulent structures formed in an incompressible flow of a Newtonian fluid by the Finite Volume Method, as well as understanding the basic concepts of fluid mechanics in turbulent fluid flows.

2. Governing equations and numeric procedures

In a general way, any isothermal flow, whose number of Mach is inferior to 15, can be mathematically well represented by the equations of conservation of the mass and the Navier-Stokes equations. Under the hypothesis of incompressible two-dimensional flow, with constant physical properties, the following equations are sufficient for the representation of the phenomenon.

Mass conservation equation

$$\frac{\partial u_j}{\partial x_j} = 0 \ , \quad \text{com} \quad j = 1, 2 \tag{1}$$

~ Navier-Stokes equations:

$$\frac{\partial u_j}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 uj}{\partial x_j^2} \text{ ,com } i = 1, 2 \text{ e } j = 1, 2 \tag{2}$$

where ρ represents the density of the fluid; ν the kinematic viscosity, u_i the components of the vectorial velocity and P the pressure.

Except for some cases, where great simplifications can be done, the non lineariaty that characterizes the advective terms of the Navier-Stokes equations impossibility any analytical solution. In this way, it becomes necessary the use of numerical methods, through which the governing equations are discretized in time and space, to supply a approximated solution for the problem. The Finite Volume Method used in this work stands out as one of the methods of space discretization more employed in the numerical simulation of dynamics of the fluids and heat transfer of problems.

For the time discretization, was adopted a totally implicit, where the values of the variables in the whole domain are calculated all at the same time, requiring the resolution of a linear system, to obtain the distribution of the variables in the calculation domain.

For the treatment of the advective terms was used the consistent QUICK, proposed by Hayase et al. (1992), that consists of a derivation of the QUICK of Leonard (1979). The central difference scheme was used to discretize the the diffusive terms of the transport equations. The method SIMPLEC was adopted for the couplement pressure-velocity and, finally, the classic model of Smagorinsky (1963) was used to allow the representation of turbulent flows, through the large-eddy simulation methodology (LES). After a series of preliminary tests, the value of the constant of Smagorinsky (Cs) it was adjusted in 0.18. The geometry of the flow is showed in Fig. (1).

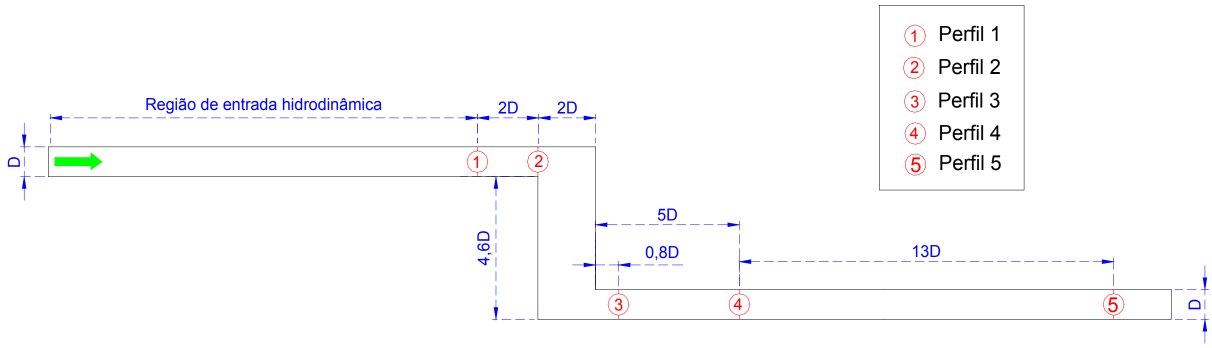


Figure 1 Geometry and dimensions of a knee where the flow was analyzed

The studies presented were for Reynolds numbers of 200, 400 and 1000, in a regular mesh of 440x112 and later with a more refined mesh of 564x200, respectively. The Reynolds numbers was based on the height of the step and in the uniform velocity of the flow in the entrance channel. Non slipping conditions were imposed at the superior and inferior walls and condition of Neumann was imposed in the exit of the calculation domain.

The results adopted as reference are from Perng (1991). In that article the author subdivided the domain in three sub-domains: entrance area, expansion area and area of contraction of the flow, as illustrated in Fig (2). Each domain was discretized in 66x18, 34x66 and 194x18 meshes, respectively.

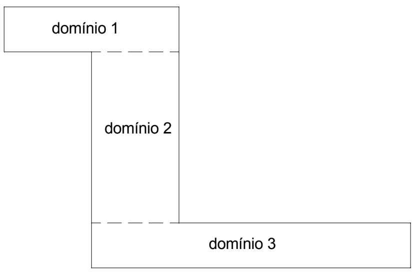


Figure 2 – Domain of the flow divided in 3 sub-domains.

Perng (1991) analyzed four profiles of velocity located in different transversal sections, as indicated in the Figure (3).

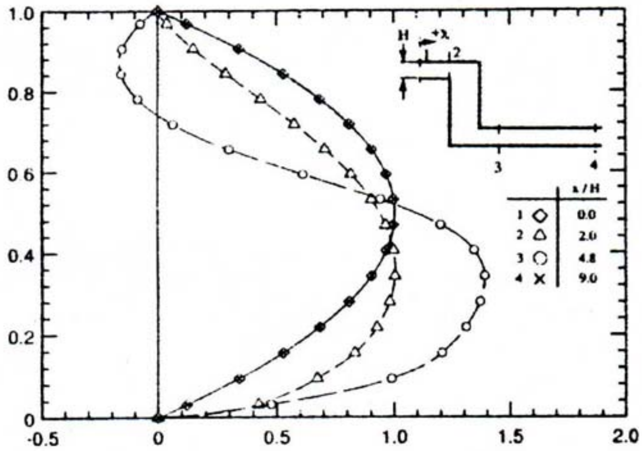


Figure 3 – Profiles of velocity in the four sections analyzed by Perng.

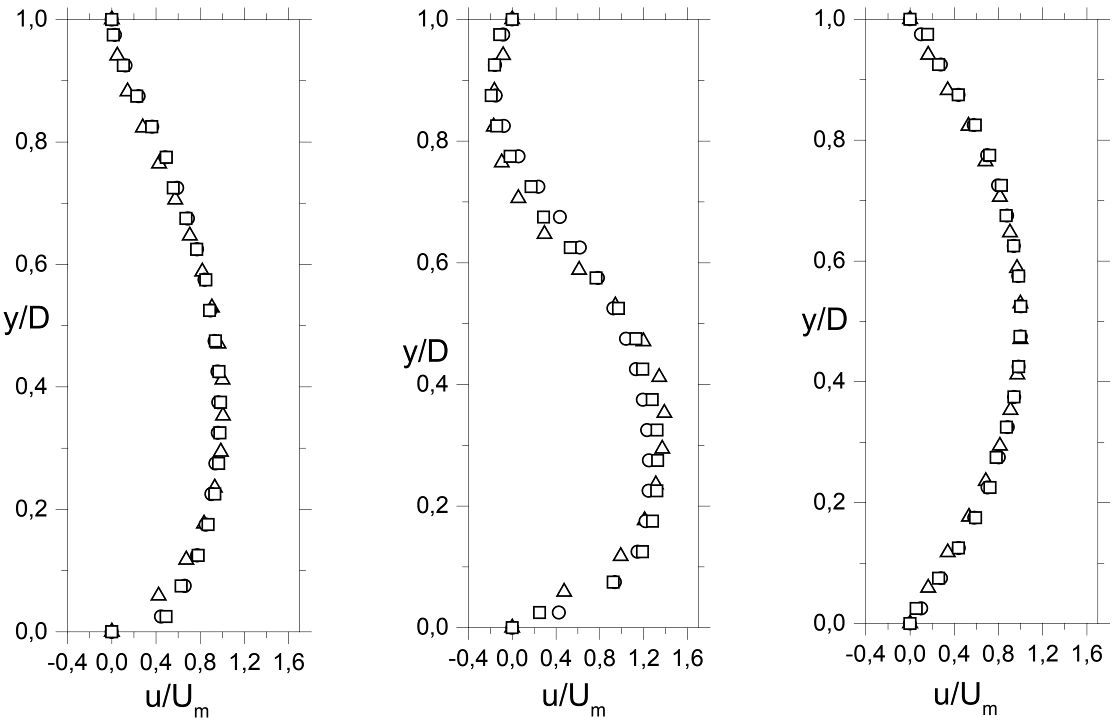
The geometry and the dimensions of the flow border were defined in agreement with the information contained in the article of Perng (1991). The fltow was defined in an only domain with a mesh of 440 x 112. The size of the mesh was also defined in according to dimensions of the meshes determined by Perng (1991).

For comparative analysis, 110 numerical probes were placed in five coincident sections with the locations of the case studied by Perng, that used 72 probes. The probes were placed in the middle of the face of a mesh element, where the velocity is very well defined.

The borders of the flow were defined as solids and an uniform profile of velocity was imposed in the entrance. In the flow analyzed by Perng (1991), the profile of velocity in the first traversal section was already completely developed. To obtain compatibility in order to compare the results, the length of the area of hydrodynamic entrance was calculated, so, the profile of velocity was completely developed to arrive in the traversal section of comparison.

3. Results

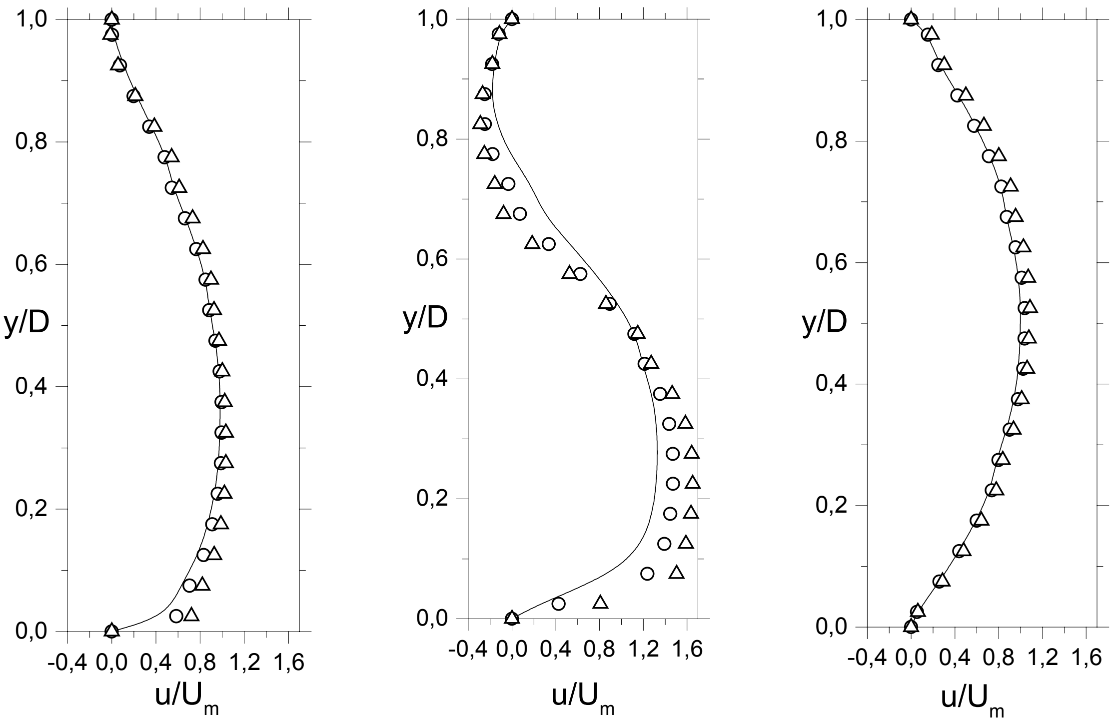
The results will be presented in form of graphs and illustrations followed by comments. The Figure (4) shows the profiles velocity 2, 3 and 5 obtained in FLUIDS and those obtained by Perng (1991). We won't show the profiles 1 and 4 since the profile 1 is completely developed as well as of the author and the author's profile 4 was already completely developed, while the profile 4 of this study was not still completely developed, then the profile 5 was already completely developed and it was compared with the author's profile 4.



(a) x=0 (b) x=2,8D (c) x=20D

Figure 4: Profile of velocity u, Re= 200: Δ Perng & Street (1991); ○ Present work (440x112); □ Present work (564x200)

In the first table it was compared the results of Perng & Street (1991), first with the compatible mesh with the one of the article and later with a refined mesh. It can be noticed that there was a small improvement in the results with the refinement of the mesh.



(a) x=0 (b) x=2,8D (c) x=20D

Figure 5. Profile of velocity u, — RE 200; ○ RE 400; Δ RE 1000

The graphs of the Figure (5) show the behavior of the profiles of velocity 2, 3 and 5 for Reynolds equal to 200, 400 and 1000 respectively for a mesh of 564x200.

4. Discussions and conclusions

One of the found difficulties went understand the conditions of flow of the reference article, the author doesn't explain clearly in the text that the first analyzed profile is about a completely developed profile and he doesn't also say if he considered profile was a uniform profile at the entrance. This aspect hinders the reproduction of the flow in the same conditions of the article, because FLUIDS was not still implemented for non uniform profiles at the entrance section

Another aspect that hindered the comparison of the results went to normalization of the velocity profiles adopted by Perng (1991). It is subtended by the results presented graphically by that author that all the velocity profiles were normalized in relation to larger speed of the first profile, in this way, that normalization was also used in the results obtained in FLUIDS, but it is a questionable point in the used normalization.

In agreement with the results shown through the illustrations it can be noticed that the program worked well with an equivalent mesh the one of Perng (1991) and it improved with the refinement of the mesh, passing of a mesh of 440x112 for a mesh of 564x200 volumes. For Reynolds 400 and 1000 the program behaved as the expected.

5. Acknowledgement

The autors acknowledge Fapesp and Fundunesp grants

6. Bibliographical references

NETO, A. S., 2002*Apostila de Turbulência nos Fluidos Aplicada*, Faculdade de Engenharia Mecânica, Universidade Federal de Uberlândia, Uberlândia-MG.

CAMPREGHER, JR R., 2002, S*imulação Numérica de Escoamentos Transicionais e Turbentos ao Redor de Geometrias Cartesianas*, Dissertação de mestrado, Universidade Estadual Paulista, Ilha Solteira, SP, Brasil.

PERNG, C. Y.; STREET, R. L. 1991*, A Coupled Multigrid-Domain-Splitting Technique for Simulating Incompressible Flows in Geometrically Complex Domains,* International Journal for Numerical Methods in Fluids, vol. 13, pp 269-286.

7. Responsibility notice

The authors are the only responsible for the printed material included in this paper.