NUMERICAL SIMULATION OF LAMINAR FLUID FLOW INSIDE TWO DIMENSIONAL 180º CURVED DUCT

Carlos Alberto de Almeida Vilela, carlosavgeo@yahoo.com
Faculdade de Engenharia Mecânica, Universidade Federal de Goias (UFG)

José Ricardo Figueiredo, jrfigue@fem.unicamp.br
Departamento de Energia, Faculdade de Engenharia Mecânica, Universidade Estadual de Campinas (UNICAMP)

Abstract. *Curved ducts are present in many engineering systems, eg in heat exchangers and central air conditioning equipment. Relevant information for the design of these systems are the pressure drop caused by changes in flow direction and, in more detailed cases, the identification of the recirculation areas. An important characteristic present in the flow with high curvature is the emergence of coriolis forces, and a good numerical scheme should be able to ensure the quality of the results in situations like that. UNIFAES scheme has been shown quite robust for solving fluid flow problems in critical situations and this paper presents its application to flow with high curvature. The proposed case is the two dimensional 180º curved duct with aspect ratio 1.0 and Reynolds number between 100 and 500. Using the finite volume method, three discretization schemes were considered for comparison: UNIFAES, QUICK and 2nd Order Upwind. Numerical results were obtained with an algorithm developed by this first author and with FLUENT software.*

Keywords: *UNIFAES, finite volume method, curved duct flow, numerical simulation*

1. INTRODUCTION

Some fluid flow situations are very common in engineering, which does not mean that this is a simple problem to be solved. Curved ducts can be found in a large number of systems like industrial piping, air conditioning distribution, refrigeration equipment, heat transfer devices, etc., and the importance of knowing all aspects of the flow in this kind of geometry is evident.

Curved ducts are geometrically simple and relatively easy to be molded and constructed. Curved ducts include helically coiled, spiraled, bended ducts with 90º or 180º, wavy pipes, etc. with circular or non circular cross section. Of course, circular cross section should be the most common configuration found in industrial equipment but even in this simplest configuration the flow behavior is not simple. The curved geometry provides to the fluid flow some peculiar characteristics that are consequences of coriolis force that raises from the high curvature. Tsai and Sheu (2006) comment that in curved ducts, centrifugal and viscous instabilities may coexist and interact and these interactions has high influence in the secondary flow. Gauthier *et al* (2001) using laser technique visualization found flow instabilities in small aspect ratio ducts, $1/3<R_c/D<1/40$, when Reynolds number ranges from $130<Re<340$, which is a very low value considering industrial devices.

Today numerical simulation is a very important tool to analyze fluid flow structures due to its versatility, good results, financial costs compared to experimental models and quick engineering results.

2. PROBLEM GEOMETRY, EQUATIONS AND BOUNDARY CONDITIONS

The problem geometry is described in Fig. 1. It consists in an 180º curved duct with a slab cross section. The region of interest is located near the bend where recirculation, flow instabilities and the centrifugal forces are present.

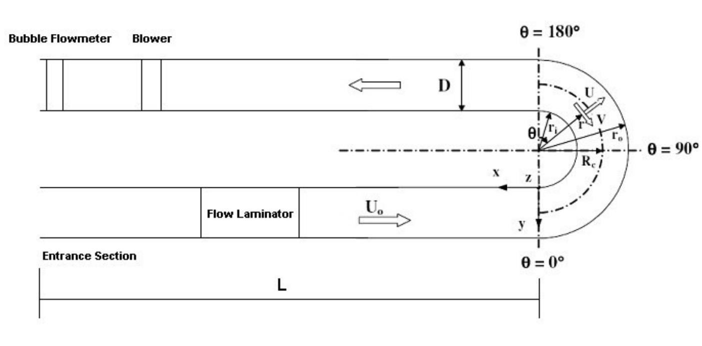


Figure 1. Geometry and boundary conditions.

The dimension L should be large enough to guarantee that inlet and outlet boundary conditions will not disturb the test section. Sugiyama and Hitomi (2005), when studying the flow inside a curved duct with circular cross section, Reynolds number 60000 and aspect ratio relation $R_c/D=2,0$ suggest that $L_{inlet}=100D$ is enough to guarantee fully developed turbulent flow and $L_{outlet}=40D$ could be considered to save computational time. Ali Ali (2008), who covered a wide range of Reynolds number $56700<Re<10^5$ and aspect ratio number $0,65<R_c/D<3,357$ using a rectangular cross section 180^o curved duct using FLUENT code with turbulence models, uses L=4D and L=5D for higher and smaller aspect ratios for numerical computations. Wojtkowiak and Popiel (2000) performed an experimental investigation of the pressure drop in a adiabatic wavy pipe considering L/D=39,7.

In the present work the following geometrical ratios are considered: The aspect ratio is Rc/D=1,0; the inlet and outlet relative dimensions are. $L_{inlet}/D=4$ and $L_{outlet}/D=20$. The Reynolds number varies between 100 and 500. The Poiseuille fully developed profile is adopted at the inlet, and homogeneous Newman conditions are assumed at the outlet. The Dean number, defined as a relation between the aspect ratio and Reynolds number, for this problem was considered to range from 70.71 to 353.55.

$$De = \mathrm{Re}\sqrt{\frac{D}{2R_c}} \tag{1}$$

To solve fluid flow problems involving curved geometries an interesting procedure is the transformed coordinate idea. It can be done if the governing equations are written in generalized coordinates form and solved in a transformed plane. The governing equations are the incompressible two-dimensional Navier-Stokes and the continuity equations written in generalized coordinates.

Continuity:

$$\frac{\partial(\rho\tilde{u})}{\partial\xi} + \frac{\partial(\rho\tilde{v})}{\partial\eta} = 0 \tag{2}$$

Navier stokes ξ direction

$$\frac{\partial}{\partial\xi}(\rho\tilde{u}u) + \frac{\partial}{\partial\eta}(\rho\tilde{v}u) = \frac{\partial}{\partial\xi}\left[\frac{\mu}{J}\left(q_{11}\frac{\partial u}{\partial\xi} + q_{12}\frac{\partial u}{\partial\eta}\right)\right] + \frac{\partial}{\partial\eta}\left[\frac{\mu}{J}\left(q_{21}\frac{\partial u}{\partial\xi} + q_{22}\frac{\partial u}{\partial\eta}\right)\right] - \left[\frac{\partial}{\partial\xi}(f_{11}p) + \frac{\partial}{\partial\eta}(f_{21}p)\right] + g_x J \tag{3a}$$

Navier stokes η direction

$$\frac{\partial}{\partial\xi}(\rho\tilde{u}v) + \frac{\partial}{\partial\eta}(\rho\tilde{v}v) = \frac{\partial}{\partial\xi}\left[\frac{\mu}{J}\left(q_{11}\frac{\partial v}{\partial\xi} + q_{12}\frac{\partial v}{\partial\eta}\right)\right] + \frac{\partial}{\partial\eta}\left[\frac{\mu}{J}\left(q_{21}\frac{\partial v}{\partial\xi} + q_{22}\frac{\partial v}{\partial\eta}\right)\right] - \left[\frac{\partial}{\partial\xi}(f_{12}p) + \frac{\partial}{\partial\eta}(f_{22}p)\right] + g_y J \tag{3b}$$

where:
ξ, η are generalized coordinates
J is the transform Jacobian
qij are transform coefficients
u,v are cartesian velocities
$\tilde{u}, \tilde{v}$ are covariant base velocities

Equations (2) and (3) are solved for *u, v* and *p* in a rectangular, equally spaced transformed plane ξ, η and the geometric relations between physical and transformed planes are given by Jacobian J and q_{ij} coefficients. The use of generalized coordinates provides some positive aspects such as the transformed plane simplicity and easy to numerically calculate $\partial/\partial\xi$, $\partial/\partial\eta$ terms in flow equations. A simple geometric correspondence between physical and transformed domains and velocities correspondence between cartesian and covariant base vector ($\mathbf{e}_{(1)}\mathbf{e}_{(2)}$) parallel ξ, η coordinate system can be seen in figure 2 where a structured grid is used to physical domain discretization. The key points ABCD are used to grid generation design.

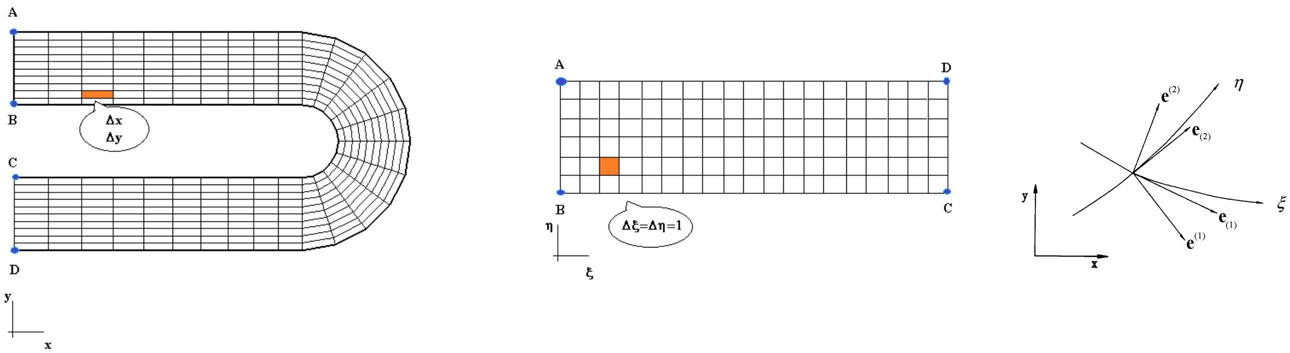


Figure 2. Physical and transformed domain correspondence and covariant base vector

3. NUMERICAL METHOD

The Finite Volume method was used for the numerical discretization of the flow equations.

For the numerical simulations two different codes were considered: the program FLOW, developed by, Vilela (2001) and the program FLUENT, a commercial simulation software incorporated by ANSYS. Program FLOW uses generalized coordinate with covariant velocities bases, velocity and pressure with non-staggered grid arrangement and three interpolation schemes: UNIFAES, originally presented by Figueiredo (1997), QUICK and the 2nd Order Upwind, both presented by Leonard (1979 and 1988). FLUENT code were uses QUICK and the 2nd Order Upwind schemes. Both codes uses the SIMPLE technique for the velocity-pressure coupling. The linear system of equations at each iteration is solved using the traditional TDMA (Tri-Diagonal Matrix Algorithm) in FLOW code and an AMG (Algebraic Multi Grid) technique in FLUENT code.

The grid was generated using an algebraic method to provide an organized and regular distribution, as can be seen in detail in figure 3.

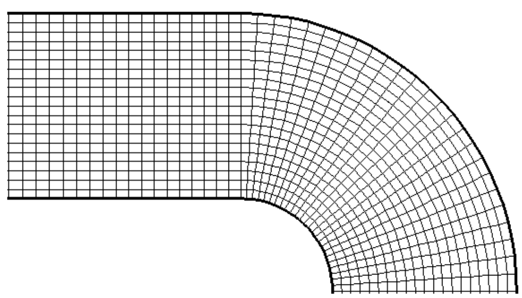


Figure 3. Typical grid arrangement at straight and curved sections

Table 2. Grid configurations used in the numerical simulations with FLOW and FLUENT codes.

	Inlet section	**Curved section**	**Outlet section**	L_{inlet}	L_{outlet}
FLUENT	100 div in **x** direction 150 div in **y** direction	100 div in **θ** direction 150 div in **R** direction	300 div in **x** direction 150 div in **y** direction	0,4	2,0
FLOW grid#1	30 div in **x** direction 90 div in **y** direction	30 div in **θ** direction 90 div in **R** direction	60 div in **x** direction 90 div in **y** direction	0,4	2,0
FLOW grid#2	100 div in **x** direction 150 div in **y** direction	100 div in **θ** direction 150 div in **R** direction	300 div in **x** direction 150 div in **y** direction	0,4	2,0

The three discretization schemes considered here employ up to five nodes on each direction, two in each side of the central node. Indeed, they were all proposed as consequence of the well known limitations of the simpler schemes with three nodes on each direction. For the nodes close to the walls, the larger computation molecules require special considerations since there may be no pair of nodes upwind, as required by the three schemes. UNIFAES has a proper solution for this problem. In the cases of QUICK and Second Order Upwind Schemes, the central differencing scheme was adopted.

4. RESULTS

Figure 4 shows the velocity profiles at three different sections of the bend: at 0° (entrance), at 90° (symmetry) and at 180° (exit), for Reynolds number 500 using FLOW code with UNIFAES, QUICK and 2nd Order Upwind schemes.

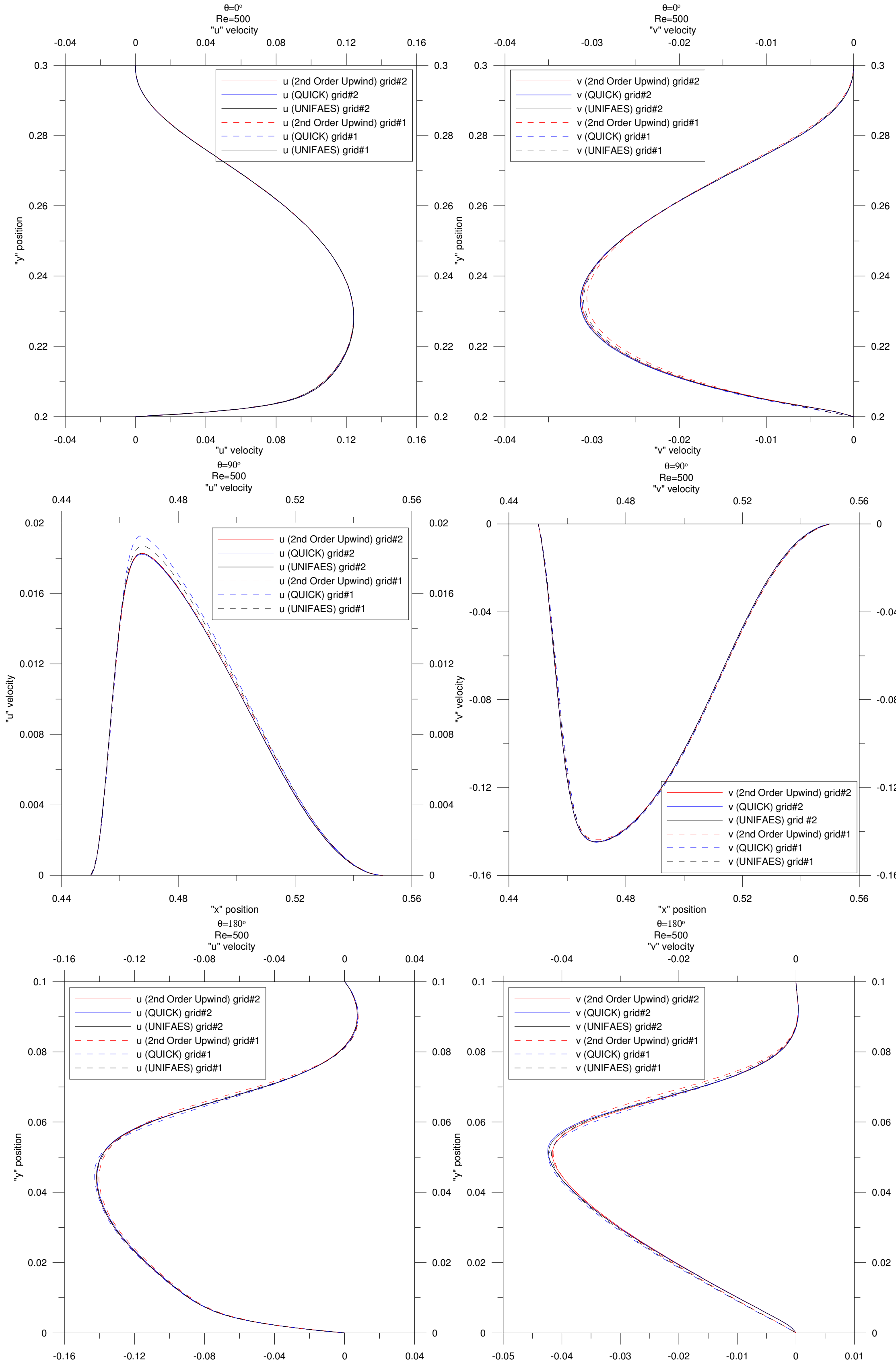


Figure 4. Velocity profile at 0°, 90° and 180°. Re=500.

Figure 5 shows the streamlines and pressure distribution for various Reynolds number using FLOW and FLUENT codes.

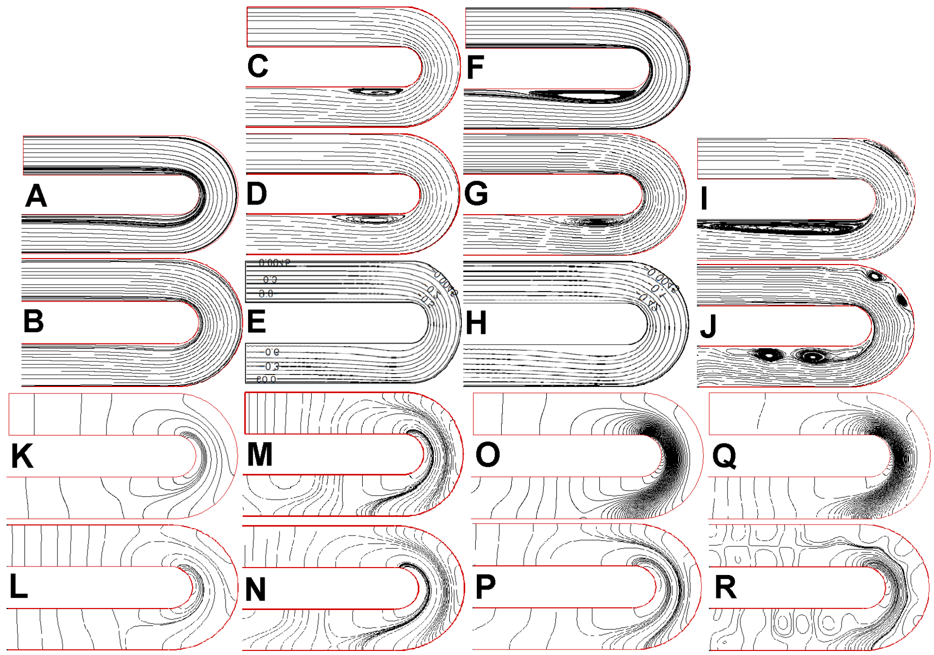


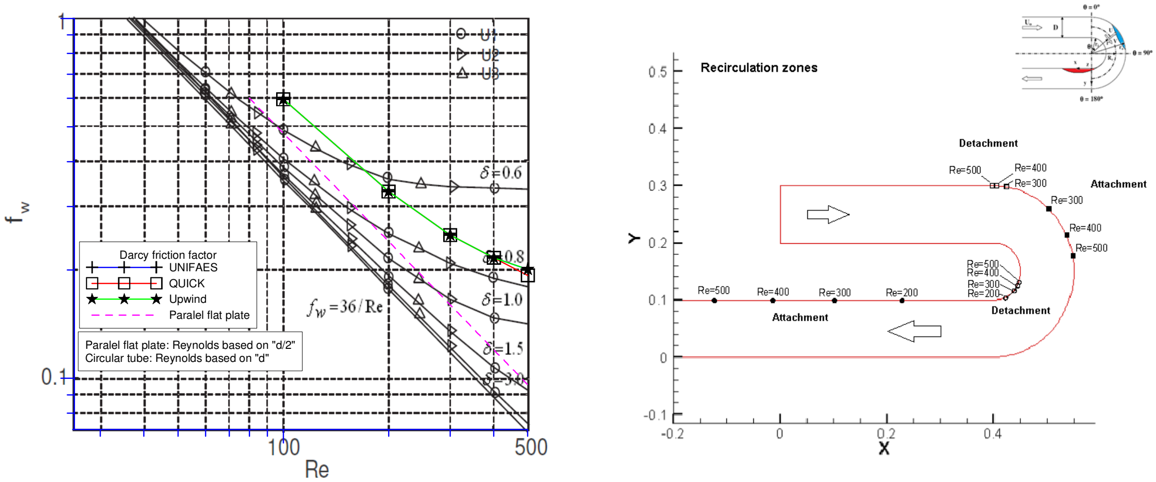
Figure 5. Streamlines and pressure distribution. Streamlines: Re=100 (A)Flow code UNIFAES scheme, (B) FLUENT code QUICK scheme, Re=200 (C) Flow code UNIFAES scheme, (D) FLUENT code QUICK scheme, (E) Yambangwai (2008), Re=300 (F) Flow code UNIFAES scheme, (G) FLUENT code QUICK scheme, (H) Yambangwai (2008), Re=500 (I)Flow code UNIFAES scheme, (J) FLUENT code QUICK scheme, Pressure distribution: Re=100 (K)Flow code UNIFAES scheme, (L) FLUENT code QUICK scheme, Re=200 (M) Flow code UNIFAES scheme, (N) FLUENT code QUICK scheme, Re=300 (O) Flow code UNIFAES scheme, (P) FLUENT code QUICK scheme, Re=500 (Q)Flow code UNIFAES scheme, (R) FLUENT code QUICK scheme,

Two important parameters about curved ducts are the friction factor and the recirculation mapping. Wojtkowiak and Popiel (2000) suggested that the Darcy friction factor f_w, should also be applied for wavy pipes.

$$f_w = \frac{(2\Delta p)}{\left(\frac{\rho U^2 L}{d}\right)} \tag{4}$$

where Δp is the pressure losses, p_2-p_1.

Analogously, this friction factor was computed for the present configuration with the results obtained by numerical simulations with the FLOW code. Figure 6a, shows the friction factor dependence with Reynolds number for the three discretization schemes and figure 6b shows the flow separation and reattachment points.



(a) (b)

Figure 6. (a) Friction factor *(Adapted from Yambangwai,2008),* (b) Recirculation mapping.

In order to evaluate and compare numerical stability between UNIFAES, QUICK and 2nd Order Upwind schemes, the convergence history provides a panoramic view along numerical solutions. Figure 7 shows the convergence history for "u" velocity component and pressure obtained with FLOW code for all performed cases. Velocity component "v" shows similar behavior to velocity component "u".

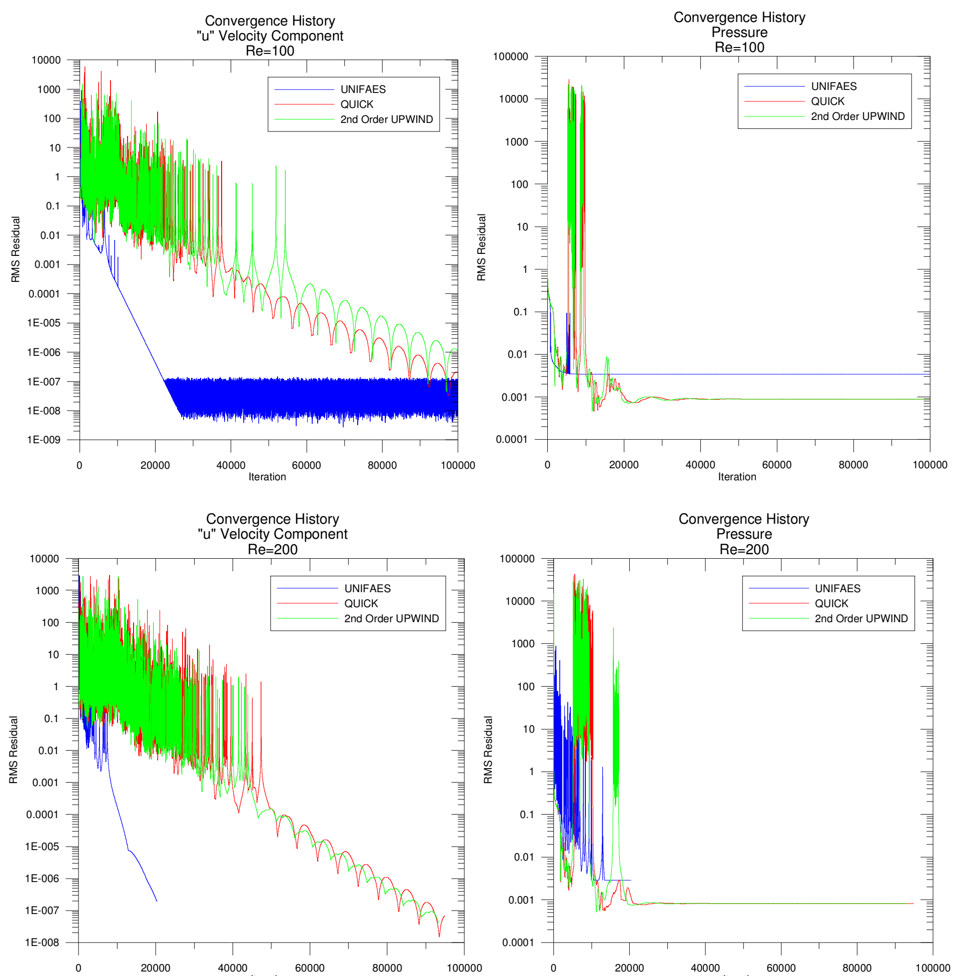


Figure 7. Convergence history - FLOW code.

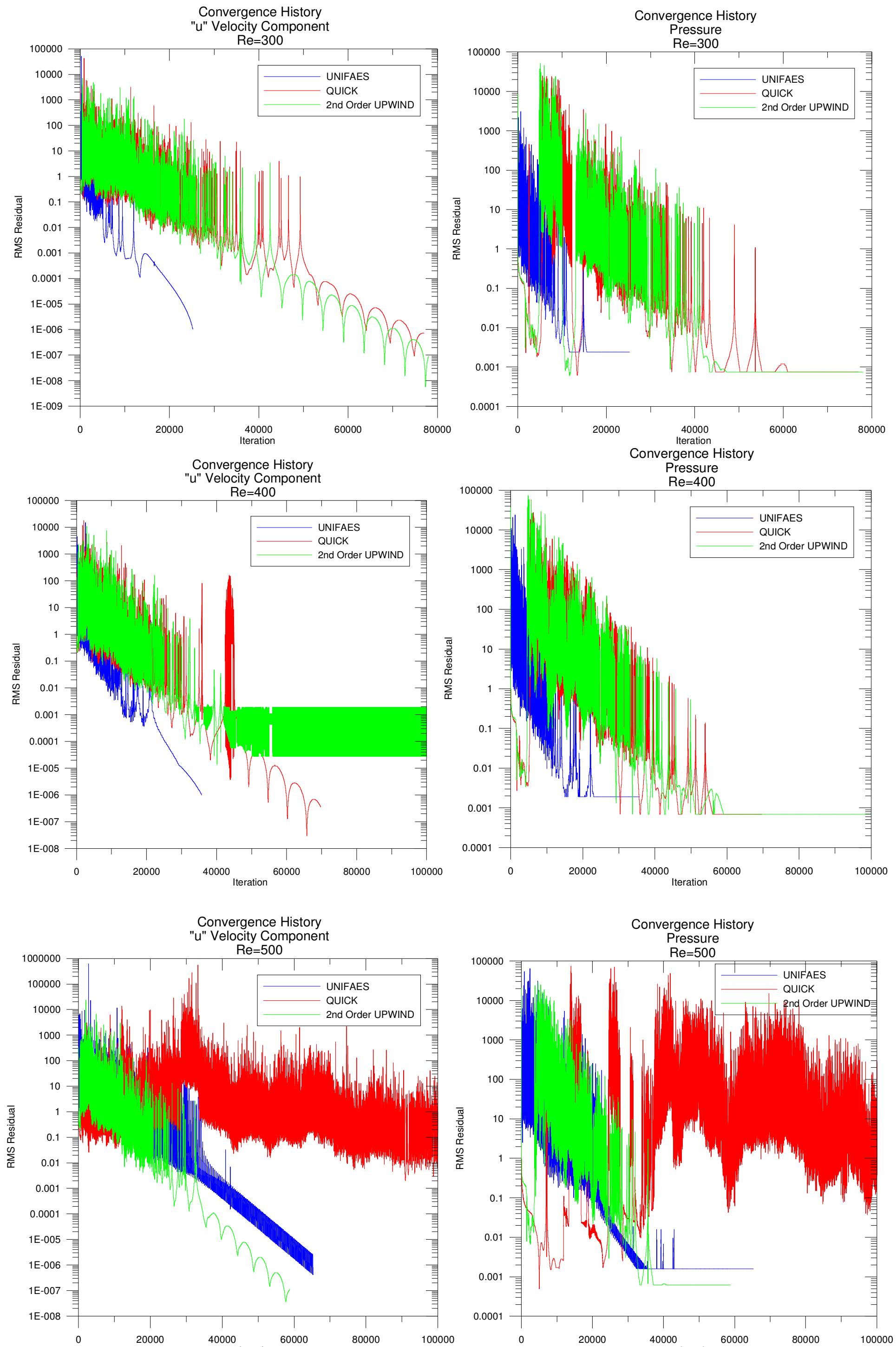
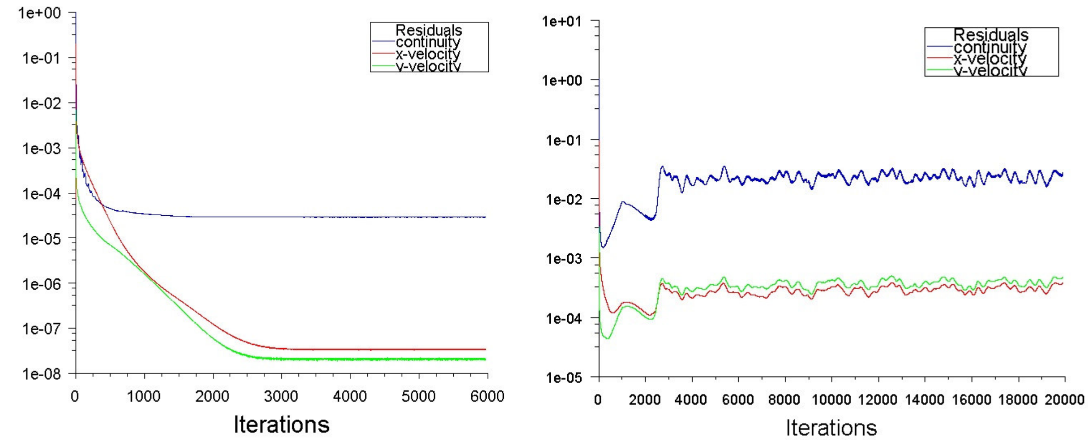


Figure 7 *(cont)*. Convergence history - FLOW code.

It can be seen that for all cases UNIFAES shows to have a smooth behavior and faster convergence. Figure 8, shows a typical history convergence graphic for FLUENT code results when convergence is achieved or not.



(a) (b)

Figure 8. Convergence history - FLUENT code. (a) Re=100, (b) Re=500

FLUENT code user manual, like many others commercial codes, consider that the solution is converged if the RMS monitor error goes under a specified tolerance or if this same monitor does not have considerable changes in time. Usually the last case happens when the solution is oscillating. For 100≤Re≤400 the convergence history are very similar to presented in Fig. 8a and only for Re=500 oscillating convergence appears just like Fig. 8b.

4. CONCLUSION

Despite having simple geometry and boundary conditions, the U bend duct problem has shown to be a peculiar numerical problem to be solved. An extra effort was spent to achieve the numerical convergence, even when using a commercial code as FLUENT. Numerical instabilities were observed with FLOW and FLUENT codes, when the Reynolds number approach 500.

However, flow instabilities were observed experimentally by Gauthier *et al* (2001) when Reynolds number approaches 300. Therefore, the instability observed at higher Reynolds numbers is not surprising, and can not be considered as numerical instability, since it may be the numerical expression of a physical instability.

Three interpolation schemes were used in this comparative paper: QUICK, 2nd Order Upwind and UNIFAES. In FLOW code two refinement grid levels were considered. For grid#2, the fine one, all schemes give visually coincident results. For grid#1, the coarse one, the results are somewhat different for the three schemes, especially at 90^o and 180^o. The coarser results with UNIFAES are shown to be closer to the finer solution than QUICK and 2nd Order Upwind, indicating the superior performance of UNIFAES.

The greater difference found among the schemes was related to the convergence history. At Reynolds 100 up to 400, the mass residual and the pressure convergence histories for all schemes are very close, but looking to velocities components it's easy to see that there are some important differences between UNIFAES and the other schemes behavior. All schemes shows an initial state of apparent numerical instability characterized by the high amplitude in convergence monitor, but UNIFAES shows to early achieve a second stage in convergence characterized by a smooth behavior. Even when UNIFAES still remains oscillating with high amplitude, for Reynolds 100, it´s happens when the monitor is in a range of 10^{-8} RMS. When UNIFAES convergence history suddenly stops, behavior observed in Reynolds 200 up to 500, it suddenly drops to a level under 10^{-15} RMS error.

Two typical convergence histories are presented for FLUENT. They show that if the numerical convergence is smooth, there is an initial stage of high convergence rate followed by a stagnation stage where the residual does not drop, even up to 10^4 iterations,. On the other side if the numerical convergence is not smooth, the convergence monitor seems instable and the results seem to be very distant from those found by FLOW code. Again in FLUENT at this situation the residual does not drop but remains at very high RMS error.

In the curved duct flow problem, there are two major aspects that can have great influence in numerical convergence: the coriolis force that appears with the high curvature flow and the mesh distortion. Related to numerical mesh, Xu and Zhang (1999) and Peric (1990) have noticed that SIMPLE and derivatives algorithms have convergence problems when the physical mesh becomes geometrically distant from the reference rectangle volume. This could be an important contribution to difficulties found in convergence history, but as the SIMPLE and others parameters were the same to for all cases, the difference between one and other is the discretization scheme. Based on that information, we can say that UNIFAES seems to have a more robust behavior to overcome this geometric issue. Vilela and Figueiredo (2002) presented some results that already pointed to this direction when compared various discretization schemes solving convective-diffusive problems using orthogonal and non-orthogonal meshes.

UNIFAES and 2nd Order Upwind converged for the highest Reynolds number Re=500 but UNIFAES shows to be more stable and achieved convergence even using a coarser grid (20 div in y direction). Also, convergence with UNIFAES could easily be guaranteed by a minor adjustment of the relaxation factor, while QUICK and 2nd Order Upwind could not achieved convergence by this means.

5. REFERENCES

Ali Ali, A. E. E., 2008, “Numerical Simulation of Turbulent Flow in Curved Ducts and Diffusers”, Thesis Submitted in Partial Fulfillment of Requirements for the Master Degree in Mechanical Power Engineering B.Sc. of Mechanical Power Engineering, Mansoura University, Faculty of Engineering Mechanical Power Engineering Dept.

Das, K., Alam, M. M., Razzaque, M. M., 2007, “Turbulent Regime Friction Factors For U-Type Wavy Tubes”, Proceedings of the International Conference on Mechanical Engineering 2007 (ICME2007), pp. 29- 31, Dhaka, Bangladesh.

Figueiredo, J. R., 1997, “A Unified Finite-Volume Finite-Differencing Exponential-Type Scheme for Convective-Diffusive Fluid Transport Equations”, Journal of Brazilian Society of Mechanical Sciences, vol. 19, n. 03, pp. 371-391.

Gauthier, G., Gondret, P., Thomé, H., Rabaud, M., 2001, “Centrifugal Instabilities in a Curved Rectangular Duct of Small Aspect Ratio”, Physics of Fluids, Vol. 13, N. 10, pp. 2831-2834.

H. Xu and C. Zhang, 1998, “Study of the Effect of the Non-Orthogonality for Non-Staggered Grids - The Theory”, International Journal for Numerical Methods in Fluids, vol. 28, pp. 1265-1280.

H. Xu and C. Zhang, 1999, “Study of the Effect of the Non-Orthogonality for Non-Staggered Grids - The Results”, International Journal for Numerical Methods in Fluids, vol. 29, pp. 625-644.

Leonard, B. P., 1979, “A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation”, Computational Methods Applied to Mechanical Engineering, Vol. 19, pp. 59-98.

Leonard, B. P., 1988, “Simple High-Accuracy Resolution Program for Convective Modelling of Discontinuities”, International Journal for Numerical Methods in Fluids, Vol. 8, pp. 1291-1318.

M. Peric, 1990, “Analisys of Pressure-Velocity Coupling on Nonorthogonal Grids”, Numerical Heat Transfer, Part B, vol. 17, pp. 63-82.

Tsai, S. F., Sheu, T. W. H., 2006, “Numerical Exploration of Flow Topology and Vortex Stability in a Curved Duct”, International Journal for Numerical Methods in Engineering, Published online in Wiley InterScience (www.interscience.wiley.com), John Wiley & Sons.

Vilela, C. A. A., 2001, “Simulação de Escoamento em Geometrias Complexas Utilizando o Método dos Volumes Finitos com o Esquema UNIFAES de Discretização”, Tese de dou torado apresentada em 2001 no Departamento de Energia da Faculdade de Engenharia Mecânica da Universidade Estadual de Campinas.

Vilela, C. A. A., Figueiredo, J. R., 2002, “Avaliação de Esquemas de Discretização Numérica para Equação de Transporte Convectivo-Difusivo em Coordenadas Generalizadas”, II Congresso Nacional de Engenharia Mecânica (CONEM), João Pessoa – PB – Brasil.

Wojtkowiak, J., Popiel, C.O., 2000, “Effect of Cooling on Pressure Losses in U-Type Wavy Pipe Flow”, Int. Comnt HeatMass Transfer, Vol. 27, N. 2, pp. 169-177.

Yambangwai, D., 2008, “Finite Volume Method For The Prediction Of Incompressible Flow Through A Bounded Domain With Specified Pressure As Boundary Condition”, A Doctor of Philosophy in Applied Mathematics Thesis, Suranaree University of Technology Academic.