COBEM 99
XV Congresso Brasileiro de Engenharia Mecânica
15th Brazilian Congress of Mechanical Engineering
22 - 26 de Novembro de 1999 / November 22 - 26, 1999 Águas de Lindóia, São Paulo,

NUMERICAL FLOW SIMULATION THROUGH SUDDEN GEOMETRY EXPANSIONS

TAVARES, A.[*], DE BORTOLI A.L.[*], GOMES, S.C.P.[†]

[*]Federal University of Rio Grande do Sul
Department of Pure and Applied Mathematics
Bento Gonçalves 9500 – 90501-900 – Porto Alegre – RS – Brazil
E-mail: alex@lula.dmat.furg.br , atavares@mat.ufrgs.br

[†]University of Rio Grande
Department of Mathematics
Av. Itália , Km 8 – 96201-900 – Rio Grande – RS – Brazil
E-mail: dmtscpg@super.furg.br

Abstract. *The present work develops a method for the solution of fluid flow through sudden geometry expansions. The analysis of these phenomena is a very complex work, that is why we are beginning by an easier model as an incompressible, single phase, two-dimensional flow through sudden expansions whose relations varies from 1:3 to 1:20. For this it is used a finite differences method based on an explicit Newton's scheme. A Cartesian grid is used, and it is refined near the expansion (x-direction) and channel walls (y-direction). Tests are carried out for steady flows for Reynolds numbers between 53 and 187 and the results are found to compare well with numerical data found in the literature.*

Key words: *Numerical simulation, Sudden geometry expansions, Newton's method*

1. INTRODUCTION

The rapid evolution of technologies and products creates the necessity of improving the quality processes. Moreover, the development of computational fluid dynamics and the employment of powerful computers allow the faster and more accurate calculation of flow fields through configurations of technical interest. The fluid flow through sudden expansion is present in several industrial processes and in commercial and domestic machines, as refrigerators and air-conditionings. For some of these machines, at expansion region, some critical mechanical and thermodynamical processes appear, and need control instruments to keep the process in equilibrium.

The flow instabilities that appear downstream of an expansion produce pressure losses, vibrations, which originate measuring errors and turn more difficult the control of a process. So, we believe that the numerical analysis of this problem can help us to identify some hydrodynamic phenomena that are very difficult and expensive to identify in a laboratory.

Sudden expansions produce hydrodynamics instabilities (Drazin and Reid, 1981) downstream of step points too. We can note the appearance of vortices on this region. These vortices change their size, shape and location as the Reynolds number increases and as the expansion relationship becomes stronger (Bejan, 1984). The knowledge of these vortices location in time and their size and shape is important to find out more accurately the pressure losses and turbulence effects on this region. Determining these phenomena is very useful for engineering tasks, as control designs. Instruments can be well used, and this can produce a process improvement and cost reduction.

Common methods employed for the solution of fluid flows are the finite differences (Anderson et al., 1984), finite elements, finite volumes (Patankar, 1980) and boundary elements. We intend to use a faster, more accurate, simpler and cheaper method. For incompressible fluid flows it is better to use explicit schemes, because of the small pressure gradient. The finite differences method is an easy and efficient one. Joining the two above factors we chose a Newton's explicit scheme, approaching the governing equations by a forward-time/central-space scheme (George and Zachmann, 1990).

A Cartesian grid is used. The solver contains an automatic in-house mesh generator, that allows an easy changing of the grid configuration. So, the user can fix the code accordingly the accuracy which is wanted. The grid is refined near the expansion point in the x-direction and near the walls in the y-direction, according to the pressure gradient sense.

Results are obtained for Reynolds numbers ranging from 53 to 187, and for expansion relationships between 1:3 and 1:20. They are found to compare well with numerical data found in the literature. Following, the governing equations are presented.

2. GOVERNING EQUATIONS

Dynamical equations of fluid flow can be obtained from the physical principles of conservation of linear momentum and mass for incompressible flows. We will consider here, in detail, only fluid flow in a closed, two-dimensional domain Ω whose boundary Γ is piecewise regular, though the ideas and techniques extend dimensionwise in a natural way.

For an incompressible and homogeneous fluid at constant temperature, the principle of conservation of linear momentum in the x and y directions, respectively, yelds

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad (1)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\upsilon\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (2)$$

where t denotes the time; $u(x,y,t)$ and $v(x,y,t)$ are the velocity vector components in the x and y directions, respectively; $p(x,y,t)$ the pressure; ρ the fluid density and υ a nonnegative viscosity coefficient, which is assumed to be constant.

The principle of conservation of mass for an incompressible fluid reduces to the following condition (continuity equation):

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (3)$$

Equations (1)-(3) are called the complete set of Navier-Stokes equations though they are more precisely, the two-dimensional Navier-Stokes equations for compressible flows. They form a system of three differential equations in the three unknown functions $u(x,y,t)$, $v(x,y,t)$ and $p(x,y,t)$.

3. DESCRIPTION OF THE NUMERICAL METHOD

In order to solve numerically the initial-boundary value problem for the Navier-Stokes equations, assume that Γ is a rectangle with vertices (0,0), (a,0), (0,b), (a,b), with a>0 and b>0. As indicated in Fig. (2), the interval [0,a] is now subdivided into n equal parts, each of length $\Delta x=a/n$, and [0,b] is subdivided into m equal parts, each of length $\Delta y=b/m$. Thereby, the region Ω is subdivided naturally into a set of nm rectangular cells of width Δx and height Δy, as shown in Fig. (2). Each cell is labelled by the index pair (i,j), where $i=1,2,...,n$ and $j=1,2,...,m$, and (x_i,y_j) is the upper-right vertex of the cell. The discrete field variables u,v and p are defined at locations of cell (i,j), which are shown in Fig. (3): The horizontal velocity u is defined at the center of each vertical side of the cell; the vertical velocity v is defined at the center of each horizontal side; and the pressure p is defined at the cell center (Greenspan and Casull, 1970).

Denote the time step by Δt. For consistency with current journal literature we will use the notation shown in Fig. (3), and will also denote u,v,p at time $t_k=k\Delta t$, by u^k, v^k, p^k, respectively.

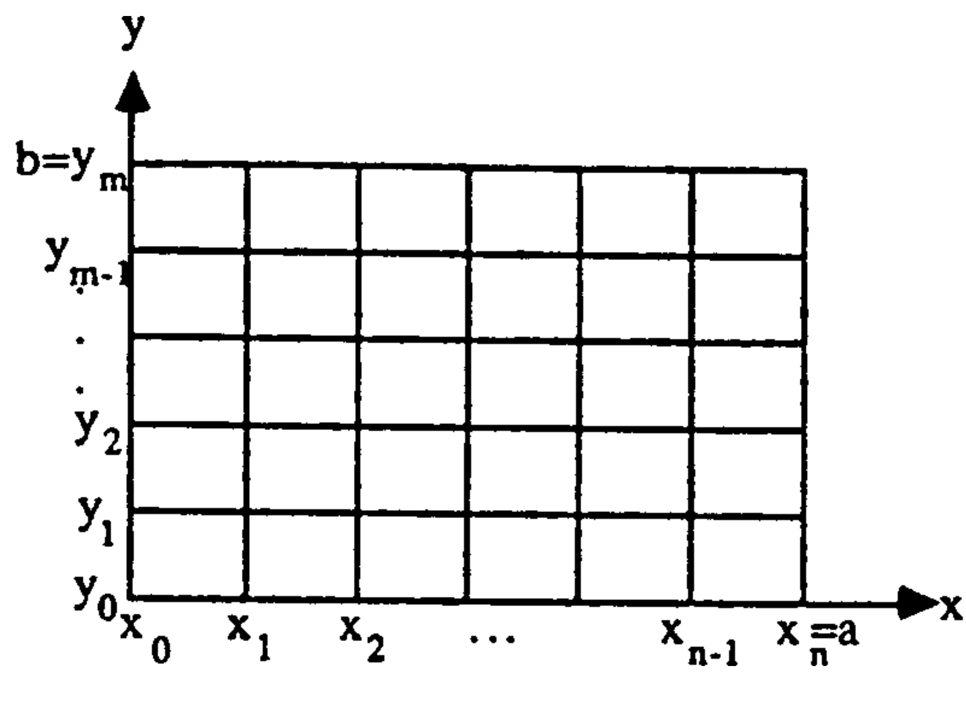

Figure 1 – Cartesian grid subdivided into nm equal parts

Let us show how to approximate the pressure at discrete times t_{k+1}, $k=1,2,...$. At center of the right-vertical side of a cell, approximate the differential Eq. (1) with a corresponding explicit, space centered, finite differences equations as follows:

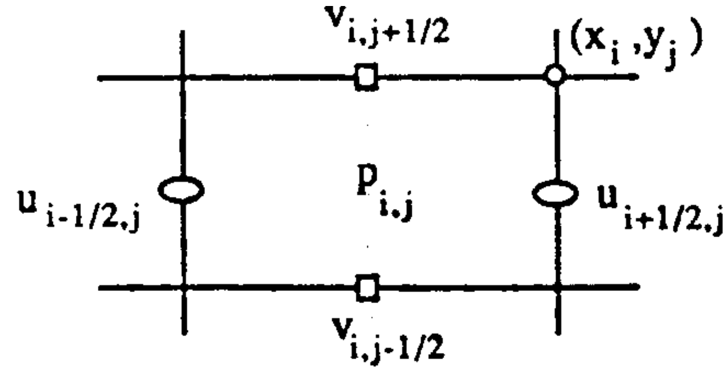

Figure 2 – Grid cell with the discrete field of variables *u, v* and *p*.

$$u^{k+1}_{i+1/2,j} = Fu^{k}_{i+1/2,j} - \frac{\Delta t}{\rho}\frac{p^{k}_{i+1,j} - p^{k}_{i,j}}{\Delta x}, \qquad i=1,2,...,n-1,\ j=1,2,...,m-1 \tag{4}$$

where the finite differences operator *F* is defined by

$$Fu^{k}_{i+1/2,j} = u^{k}_{i+1/2,j} - \Delta t\left[u^{k}_{i+1/2,j}\frac{u^{k}_{i+3/2,j} - u^{k}_{i-1/2,j}}{2\Delta x} + \bar{v}^{k}_{i+1/2,j}\frac{u^{k}_{i+1/2,j+1} - u^{k}_{i+1/2,j-1}}{2\Delta y}\right]$$
$$+ \upsilon\Delta t\left(\frac{u^{k}_{i+3/2,j} - 2u^{k}_{i+1/2,j} + u^{k}_{i-1/2,j}}{(\Delta x)^2} + \frac{u^{k}_{i+1/2,j+1} - 2u^{k}_{i+1/2,j} + u^{k}_{i+1/2,j-1}}{(\Delta y)^2}\right) \tag{5}$$

Similarly, at center of the upper-horizontal side of a cell, a finite difference approximation to the differential Eq. (2) yields

$$v^{k+1}_{i,j+1/2} = Gv^{k}_{i,j+1/2} - \frac{\Delta t}{\rho}\frac{p^{k}_{i,j+1} - p^{k}_{i,j}}{\Delta y}, \tag{6}$$

Note now that although an explicit difference approximation has been used to approximate Eq. (1) and (2), the finite difference formulas (4) and (6) cannot, as yet, be used to determine the new velocity at time t_{k+1} because the pressure field at time t_k is still unknown. In order to determine $p_{i,j}^{k}$ we require that the new velocity field satisfies, in each cell, the finite difference analogue of the incompressibility condition (3), that is

$$\frac{u^{k+1}_{i+1/2,j} - u^{k+1}_{i-1/2,j}}{\Delta x} + \frac{v^{k+1}_{i,j+1/2} - v^{k+1}_{i,j-1/2}}{\Delta y} = 0. \tag{7}$$

Now, if *(i,j)* is a cell which has no sides in common with the boundary, Eq.(5) implies

$$\begin{aligned} u^{k+1}_{i+1/2,j} &= Fu^{k}_{i+1/2,j} - \frac{\Delta t}{\rho}\frac{p^{k}_{i+1,j} - p^{k}_{i,j}}{\Delta x} \\ u^{k+1}_{i-1/2,j} &= Fu^{k}_{i-1/2,j} - \frac{\Delta t}{\rho}\frac{p^{k}_{i,j} - p^{k}_{i-1,j}}{\Delta x} \end{aligned} \tag{8}$$

and similarly, Eq. (7) implies

$$\begin{aligned} v^{k+1}_{i,j+1/2} &= Gv^{k}_{i,j+1/2} - \frac{\Delta t}{\rho}\frac{p^{k}_{i,j+1} - p^{k}_{i,j}}{\Delta y} \\ v^{k+1}_{i,j-1/2} &= Gv^{k}_{i,j-1/2} - \frac{\Delta t}{\rho}\frac{p^{k}_{i,j} - p^{k}_{i,j-1}}{\Delta y} \end{aligned} \tag{9}$$

Now, substitution of Eq. (8) and (9) into (7), yelds

$$\frac{\Delta t}{\rho}\left[\frac{p^{k}_{i+1,j} - 2p^{k}_{i,j} + p^{k}_{i-1,j}}{(\Delta x)^2} + \frac{p^{k}_{i,j+1} - 2p^{k}_{i,j} + p^{k}_{i,j-1}}{(\Delta y)^2}\right] = \frac{Fu^{k}_{i+1/2,j} - Fu^{k}_{i-1/2,j}}{\Delta x} + \frac{Gv^{k}_{i,j+1/2} - Gv^{k}_{i,j-1/2}}{\Delta y}, \tag{10}$$

Let us solve the system (10) with a special form of generalised Newton's Method (Fletcher, 1990), which, in this singular case, can be shown to be convergent (Greenspan and Casull, 1970) for any initial guess of parameter ω ranging between $0<\omega<2$. Specifically, in sweeping the cells consecutively from left to right and from bottom to top, at any interior computational cell, the generalised Newton's formula for $p_{i,j}^{k}$ is

$$\left(p^{k}_{i,j}\right)^{(r+1)} = \left(p^{k}_{i,j}\right)^{(r)} - \frac{\rho\omega}{-2\left[\frac{1}{(\Delta x)^2} + \frac{1}{(\Delta y)^2}\right]} \cdot R^{k}_{i,j} \tag{11}$$

where

$$\begin{aligned} R^{k}_{i,j} = {} & \frac{\left(p^{k}_{i+1,j}\right)^{(r)} - 2\left(p^{k}_{i,j}\right)^{(r)} + \left(p^{k}_{i-1,j}\right)^{(r+1)}}{(\Delta x)^2} + \frac{\left(p^{k}_{i,j+1}\right)^{(r)} - 2\left(p^{k}_{i,j}\right)^{(r)} + \left(p^{k}_{i,j-1}\right)^{(r+1)}}{(\Delta y)^2} \\ & - \frac{Fu^{k}_{i+1/2,j} - Fu^{k}_{i-1/2,j}}{(\Delta t)(\Delta x)} - \frac{Gv^{k}_{i,j+1/2} - Gv^{k}_{i,j-1/2}}{(\Delta t)(\Delta y)} \end{aligned}$$

or

$$\left(p_{i,j}^{k}\right)^{(r+1)} = \left(p_{i,j}^{k}\right)^{(r)} + \left(\delta p_{i,j}^{k}\right)^{(r)} \tag{12}$$

Then, formulas to correct velocities are written as follows

$$\left(u_{i+1/2,j}^{k+1}\right)^{(r+1/2)} = \left(u_{i+1/2,j}^{k+1}\right)^{(r)} + \frac{\Delta t}{\rho \Delta x}\left(\delta p_{i,j}^{k}\right)^{(r)} \tag{13}$$

$$\left(u_{i-1/2,j}^{k+1}\right)^{(r+1)} = \left(u_{i-1/2,j}^{k+1}\right)^{(r+1/2)} - \frac{\Delta t}{\rho \Delta x}\left(\delta p_{i,j}^{k}\right)^{(r)} \tag{14}$$

$$\left(v_{i,j+1/2}^{k+1}\right)^{(r+1/2)} = \left(v_{i,j+1/2}^{k+1}\right)^{(r)} + \frac{\Delta t}{\rho \Delta y}\left(\delta p_{i,j}^{k}\right)^{(r)} \tag{15}$$

$$\left(v_{i,j-1/2}^{k+1}\right)^{(r+1)} = \left(v_{i,j-1/2}^{k+1}\right)^{(r+1/2)} - \frac{\Delta t}{\rho \Delta y}\left(\delta p_{i,j}^{k}\right)^{(r)} \tag{16}$$

In spite of the pressure-velocity coupling used to obtain the pressure equation, it can be associated to Patankar's SIMPLER method. On the other hand, there are some differences nearly related to the discussion of the appropriate interpolation function for velocities (see Patankar, 1980), in the SIMPLER method.

4. INITIAL AND BOUNDARY CONDITIONS

At the initial time $t_0=0$, the initial conditions

$$\begin{aligned} u(x,y,0) &= u_0(x,y), \\ v(x,y,0) &= v_0(x,y), \end{aligned} \qquad (x,y)\in \Omega \tag{17}$$

imply

$$u_{i+1/2,j}^{0} = u_0\left(x_i, y_{j-1/2}\right) \tag{18}$$

$$v_{i,j+1/2}^{0} = v_0\left(x_{i-1/2}, y_j\right) \qquad i=1,2,\ldots,n; \qquad j=1,2,\ldots,m \tag{19}$$

Since no initial conditions have been given for pressure, $p_{i,j}^{0}$ must be approximated from the differential equations.

If the discrete velocity field is known at time step t_k, from

$$\begin{aligned} u(x,y,t) &= u_\Gamma(x,y,t), \\ v(x,y,t) &= v_\Gamma(x,y,t), \end{aligned} \qquad (x,y) \in \Gamma,\ t\geq 0, \tag{20}$$

one can set tangential velocity boundary conditions at time t_k and normal velocity boundary conditions at time t_{k+1}. As indicated in Fig (3), since the discrete u velocities are not located at the top and at the bottom boundaries, boundary conditions on the tangential velocity are taken to be

$$\frac{u^k_{i+\frac{1}{2},0} + u^k_{i+\frac{1}{2},1}}{2} = u_\Gamma\left(x_i, 0, t_k\right), \qquad i=1,2,...,n-1 \tag{21}$$

$$\frac{u^k_{i+\frac{1}{2},m} + u^k_{i+\frac{1}{2},m+1}}{2} = u_\Gamma\left(x_i, b, t_k\right) \tag{22}$$

The normal velocity boundary conditions at time t_{k+1} are given by

$$\begin{aligned} u^{k+1}_{\frac{1}{2},j} &= u_\Gamma\left(0, y_{j-\frac{1}{2}}, t_{k+1}\right) \\ u^{k+1}_{n+\frac{1}{2},j} &= u_\Gamma\left(a, y_{j-\frac{1}{2}}, t_{k+1}\right) \end{aligned} \qquad j=1,2,...,m \tag{23}$$

$$\begin{aligned} v^{k+1}_{i,\frac{1}{2}} &= v_\Gamma\left(x_{i-\frac{1}{2}}, 0, t_{k+1}\right) \\ v^{k+1}_{i,m+\frac{1}{2}} &= v_\Gamma\left(x_{i-\frac{1}{2}}, b, t_{k+1}\right) \end{aligned} \qquad i=1,2,...,n \tag{24}$$

5. NUMERICAL RESULTS

In the following, numerical results for a two-dimensional channel with a sudden expansion are presented. Expansion relationships of 1:3, 1:5, 1:20 for Reynolds numbers 53, 80, and 187 are shown in Fig.(4)-(9), as follows. A grid containing 65 x 51 mesh points is used (see Fig. (3)).

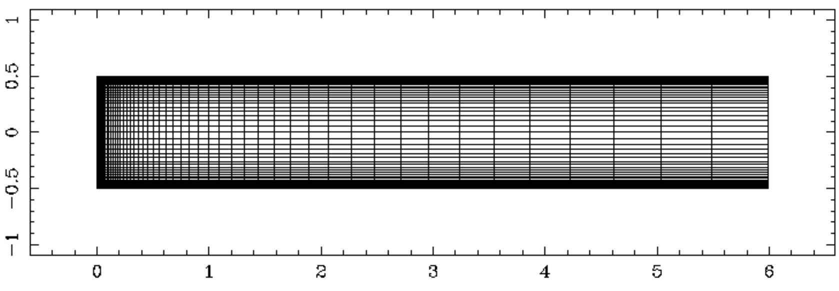

Figure 3 – Cartesian grid for sudden expansion, 65x51 points

Figure (4-a) displays the streamlines for the incompressible flow through an expansion with a 1:3 diametrical relationship for Reynolds number Re=53. Results are in good agreement with the ones presented in the literature (Battaglia et al., 1997). Note that in Fig. (4-a) it is shown some lines concentrated in the proximity of superior vortex; however these vortices are symmetric as shown in Fig. (4-b).

Figures (5-a) and (5-b) show the streamlines for the same diametrical relationship, but at Reynolds 80. These numbers were chosen to compare obtained curves with the results of

Battaglia et al. (1997). They developed numerical simulations like the ones present in this work. Selected Reynolds numbers are critical, since the size and shape of the vortices become asymmetrical. The following results are also in agreement with the experiments of *Goldstein et al.* (1970).

Figures (6-a) and (6-b) present the streamlines for flow when the Reynolds number increases to 187. You can note that a new vortex appear, increasing the instabilities downstream of the expansion.

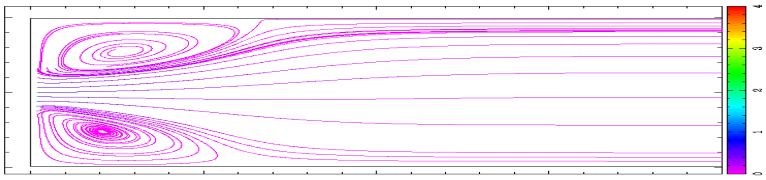

Figure 4-a – Streamlines for 1:3 expansion relationship, Re=53

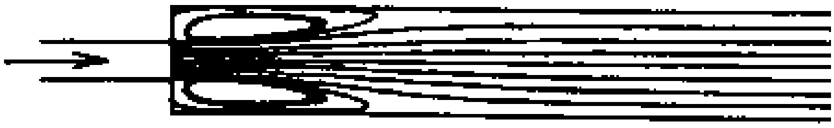

Figure 4-b – Streamlines for 1:3 expansion relationship, Re = 53[15]

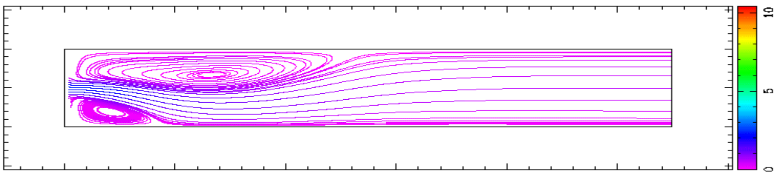

Figure 5-a – Streamlines for 1:3 expansion relationship, Re=80

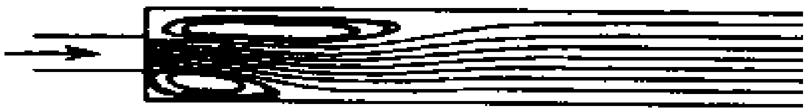

Figure 5-b – Streamlines for 1:3 expansion relationship, Re = 80

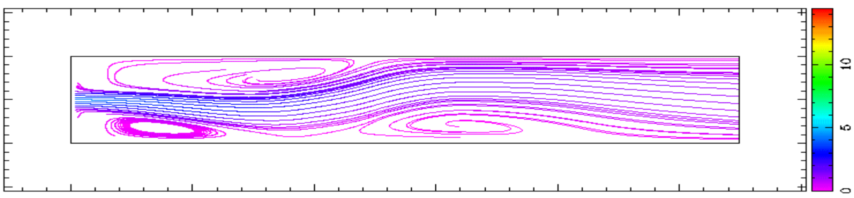

Figure 6-a – Streamlines for 1:3 expansion relationship, Re=187

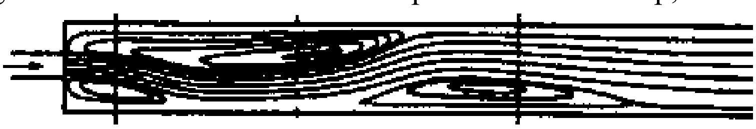

Figure 6-b – Streamlines for 1:3 expansion relationship, Re=187

Seeing the above figures, the reader can note the Reynolds Number increasing effect. As it increases, flow instabilities extend along the channel, with the appearance of the new vortices.

Figure (7) shows the flow simulation for 1:5 expansion relationship and Reynolds number 53. As the Reynolds number and the expansion relationship increase, the instabilities increase too. A greater number of vortices appear and the time interval to reach the steady state also increases. So, we simulated the unsteady incompressible flow through sudden expansion with a diametrical relationship 1:20 at time of 30 seconds, as is shown in the Fig. (8), for Re=200.

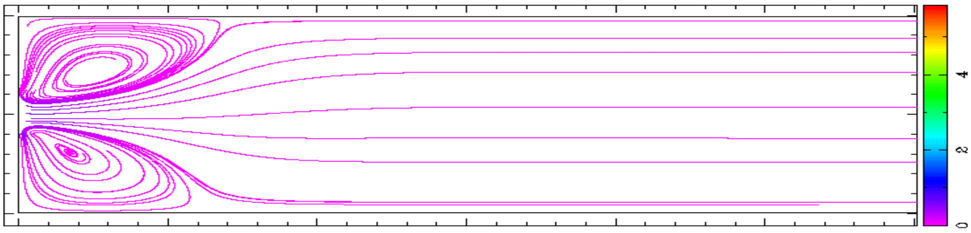

Figure 7 – Streamlines for 1:5 expansion relationship, Re=53

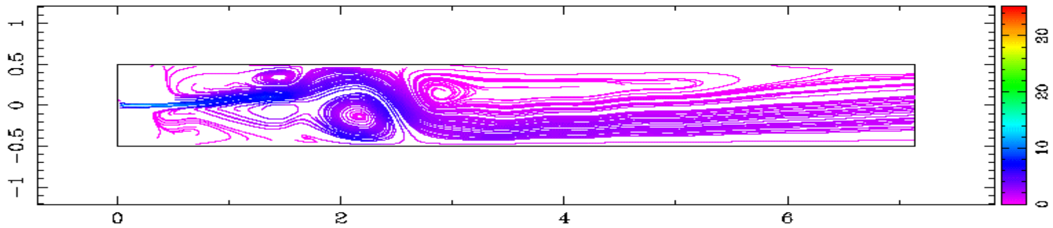

Figure 8 – Streamlines for 1:20 expansion relationship, Re=200

6. CONCLUSIONS

Numerical results were obtained for flows in a two-dimensional channel with a sudden symmetric geometry expansion. A finite differences and the Newton's explicit method were used in order to obtain our results in an efficient way. The critical Reynolds number was determined for various expansion ratios. It was shown that the critical Reynolds number decreased with the increasing of the expansion ratio. The results of numerical simulations were found to be in agreement with numerical/experimental results found in the literature.

Obtained results show us some particularities of a flow separation and reattachment downstream of a sudden expansion. For low Reynolds numbers (under 53) appear two symmetrical (toroidal) vortices. As the Reynolds number increases, the vortices become asymmetric. For Reynolds numbers over 187, new vortices appear downstream of the expansion point; the flow reattachment happens then at larger distances from the expansion. A similar flow behaviour occurs with the increasing of the expansion ratio. For greater expansion relationships the effect is the same as the Reynolds number increases. For an expansion ratio of 1:20, the steady state just occurs after a transition process. The transition time is about 30 seconds.

It is the author's opinion that the obtained results are encouraging, however much work must still be done in order to obtain a suitable method to analyse the complete complex expansion behaviour in a refrigerator machine valve.

REFERENCES

Anderson, D. A., Tannehil, J. C., Pletcher, R. H., Computational Fluid Mechanics and Heat Transfer, McGraw-Hill, New York, 1984.

Battaglia, F., Simon, J., Kulkarni, A., Merkle, C., Bifurcation of Low Reynolds Number Flows in Symmetric Channels, AIAA Journal, Vol. 35, Pennsylvania, January 1997.

Bejan, A., Convection Heat Transfer, John Wiley & Sons, 1984.

Drazin, P. G., Reid, W. H., Hydrodynamic Stability, Binghannton, New York, 1981.

Fletcher, C. A. J., Computational Techniques for Fluid Dynamics, Vol. 1, Fundamental and General Techniques, Springer-Verlag, Sydney, 1990.

George, K., Zachmann, D., Nonlinear Convection Equations and Newton-Like Methods, American Mathematical Society, Computational Solution of Nonlinear Systems of Equations, Lectures in Applied Mathematics, Vol. 26, 1990.

Goldstein, R. J., Eriksen, V. L., Olson, R. M., Eckert, E. R. G., Laminar Separation, Reattachment and Transition of the Flow over a Downstream Facing Step, Transactions of the ASME, Minneapolis, 1970.

Greenspan, D., Casull, V., Numerical Analysis to Applied Mathematics Science and Engineering, Addison-Wesley Publishing Co. Inc., 1960.

Patankar, S.V., Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1980.