Análisis numérico del flujo en ducto con 3 diferentes relaciones de contracción

Numerical analysis of flow in a duct with 3 different relationships of contraction

Oscar A. Morales C.
Universidad Autónoma de Baja California
moraleso97@uabc.edu.mx

Juan G. Barbosa S.
Universidad Autónoma de Baja California
jbarbosas@ipn.mx

José. A. Jiménez B.
Universidad Autónoma de Baja California
jjimenezb@ipn.mx

Antonio Gómez R.
Universidad Autónoma de Baja California
gomez_roa@uabc.edu.mx

Juan A. Paz G.
Universidad Autónoma de Baja California
pazj@uabc.edu.mx

Raúl Vázquez P.
Universidad Autónoma de Baja California
raul.vazquez.prieto@uabc.edu.mx

Resumen

En esta investigación se presentan los resultados numéricos de la influencia de la relación de aspecto y el número de Reynolds en el comportamiento del flujo laminar en un conducto rectangular con escalón (FFS). Se analizan 3 diferentes Reynolds (300, 400 y 500) y 3 diferentes relaciones de contracción (8, 4 y 2.7). El estudio numérico se realiza con el programa desarrollado denominado FLUSS, el código está basado en la técnica de volúmenes finitos y el algoritmo SIMPLE, la validación del código numérico se realiza comparando con los resultados experimentales obtenidos mediante la técnica láser de velocimetría de imágenes de partículas. Los resultados numéricos y experimentales presentan una diferencia máxima del 4%.

Palabras clave: Conducto con contracción, Simulación numérico, Técnica de volúmenes finitos, Vórtice helicoidal, Técnica de PIV.

Abstract.

This research presents the numerical results of the influence of aspect ratio and laminar Reynolds number in the behavior of the flow in a rectangular duct with step (FFS). Three different Reynolds (300, 400, and 500) and 3 different ratios of contraction (8, 4, and 2.7) are analyzed. The numerical study is performed with the developed program called FLUSS, the code is based on the technique of finite volume and SIMPLE algorithm, validation of the numerical code is compared with the experimental results obtained with the laser technique of particle image velocimetry. Numerical and experimental results present a maximum difference of 4%.

Keywords: Duct with contraction, numerical simulation, finite volumes technique, helical Vortex, PIV technique.

Fecha Recepción: Abril 2015 **Fecha Aceptación:** Junio 2015

Introduction

Separation and flow recirculation (also known as EGR) are found in different industrial equipment such as valves and diffusers, blades of turbine. These phenomena primarily generate pressure drop on computers, condition that, under certain circumstances, may not be desired due to losses that arise. However, in some cases the separation and recycling are induced to improve the conditions of heat and mass transfer, such as in the case of combustion chambers and heat exchangers, finned surfaces (1).

A classic example where you can find the separation and recirculation flow is a rectangular duct presenting changes in geometry. These changes are obtained by varying the cross-section to reduce the area of the duct (duct with contraction), to extend the area of the same (duct with expansion) or if they are both cases it's a cavity (2).

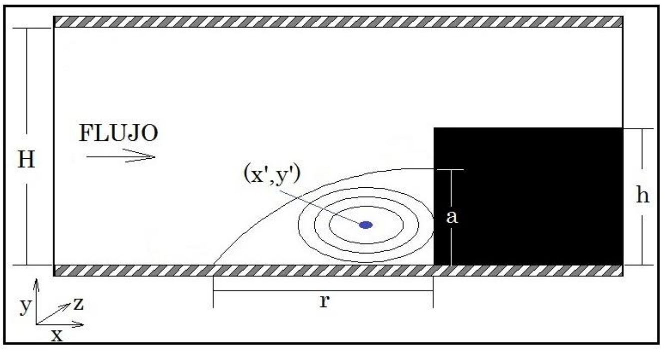
The two last mentioned cases have been widely studied by the researchers, while the flow in ducts with contraction (FFS) has taken relevant interest only in recent decades, by what some details regarding the 3D effects of the structures of this type of duct flow not have been fully analysed, besides this kind of problems are considered type or pattern to validate codes and numerical processes (3). On the other hand, it is important to mention that ducts step apart from being used for the validation of numeric codes are of great importance in the industry, since they are widely used in the food industry, in piping in chemical reactors and processes of polymers (4).

When there is a flow in a rectangular duct with step (FFS) two regions of vortices are distinguished: a prior to the step and another on top of it (5), these regions are caused by the sudden change in the geometry of the duct. The Vortex before the step is defined by its length (r) in the axial direction, its height (a) in the vertical direction and the same central coordinates (x', y') as shown in Figure 1.

In recent years Have Been Conducted studies to analyze the flow separation in step rectangular ducts Considered With duct having aspect ratios greater than ten (RA> 10), esta consideration cases you reduce dimensional analysis, Therefore, it does not have a complete analysis of vortex behavior FORMED against the step along the transverse axis (6).

Of the first studies that addressed the flow in a rectangular duct step is that of Moss and Backer (7), in its results report that the height of the vortex in the center of the pipe is connected with the step height and has a value of 0.6ha 0.7H also they mentioned that the recirculation zone is a structure of three-dimensional flow.

Fig. 1: Nomenclature for describing the vortex front step.



A recent work dealing with the flow duct step is to Largeau and Morieniere (8) in his experimental research used the technique of particle image velocimetry (PIV) to study turbulent flow and three aspect ratios. Among his most important results reported that as RA decreases the length of vortex is formed in front of the step increases, but the coordinate (y ') remains unchanged.

Another work that deals with the problem of flow in rectangular duct step is to Wilhelm and Kleiser (4). The study was conducted through numerical simulation under conditions of laminar flow regime (Reh = 330) and steady state in two dimensions, using the technique of spectral discretization methods. Kleiser Wilhelm and found that the length and height of the vortex before step are almost constant for progressive Reynolds and there is a proportional increase to Reh0.2 Reh0.6 and for (r) and (a) respectively. Conclude that growth rates are a function of Reynolds but geometry.

Another study analyzing the problem of flow separation in a rectangular duct step is to Fiorentini et al. (9). They studied the behavior of vortex PIV technique using turbulent flow when flowing through a conduit with a RA> 10. They conclude that the height of the vortex that is formed before the step does not change if the speed is increased, but the vortex is formed on the step length increases as the Reynolds number increases.

Further investigation which analyzes the behavior of flow in a pipe with step is to Chiang and Sheu (10). This numerical simulation was performed using the technique of finite volume discretization. Rectangular geometric conditions were analyzed through RC = 1.33 and RA = 24. Sheu Chiang and found that the area adjacent to the step acts as a recirculating vortex which moves in a spiral along the axis "z". They also found that the height and length of the recirculation zone in the center plane of the duct are worth respectively 0.33hy 0.63h to Reh = 555.

Finally, one of the most important work is the study of the recirculation zone formed in a line with step is to Stüer et al. (Eleven). In his research they worked with laminar flow (Reh = 330) and a shoulder height of 0.01m to take geometric conditions RC and RA = 4> 10. Visualization technique using flow with bubbles of hydrogen and measuring velocity fields with PTV technique. They conclude that the fluid contained within the recirculation zone is transported parallel to step towards the side walls to continue its trajectory, and report that the height of the vortex in the center of the duct is 0.75h.

Numerical methodology.

In this research a numeric code called FLUSS, based on the finite volume technique suggested by Patankar (12) is performed. Equations for steady laminar flow (Navier-Stokes and continuity) are used which are defined as:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right] \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right] \tag{3}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \tag{4}$$

Through the aforementioned technique equations (1) are discretized to (4) using an arrangement dislocated and nonuniform mesh. The governing equations can be expressed in general terms as follows:

$$div(\rho\phi U)=div(\Gamma grad\phi)+S\phi \tag{5}$$

When integrated (5), over a finite volume control takes the following algebraic form:

$$a_P\phi_P=\sum_{i=1}^{n}a_i\phi_i+b \tag{6}$$

Where i denotes the ratio of the neighboring points, and the term b source represents the variables u, v, w, p. The ai coefficients are calculated as the sum:

$$a_i=D_iA\left(\left|Pe_i\right|\right)+\left[\left|-F_i,0\right|\right] \tag{7}$$

Where: D and F represent the convective and diffusive fluxes respectively, and Pe (Peclet) the relationship between them. The function is calculated with the outline of an inverse-power (12) represented by the following equation:

$$A\left(\left|Pe_i\right|\right)=\max\left[0,(1-0.1\left|Pe_i\right|)^5\right] \tag{8}$$

Coupling the continuity and momentum equations it is performed by SIMPLE (Semi Implicit Pressure Linked Equations) algorithm, solving the tridiagonal system obtained by the approximation of finite volumes by Thomas algorithm (12).

The computational domain used is shown in Figure 2 and the different cases studied are indicated in Table I, all results are parameterized according to the step height (h = 0.02m).

Tabla I: Casos analizados.

Caso	Reynolds	Altura de escalón h (m)	Relación de Aspecto (b/h)
1	400	0.01	8.00
2	300 400 500	0.02	4.00
3	400	0.03	2.67

The boundary conditions imposed on the computational domain are:

* Uniform velocity profile at the entrance: $u = \bar{u}$, $v = 0$ y $w = 0$

* Non-slip condition on the walls: $u = 0$, $v = 0$ y $w = 0$

* Fully developed flow condition on departure: $\left.\frac{\partial \phi}{\partial x}\right|_{x_{salida}} = 0$

Where: $\phi = u, v, w, p$.

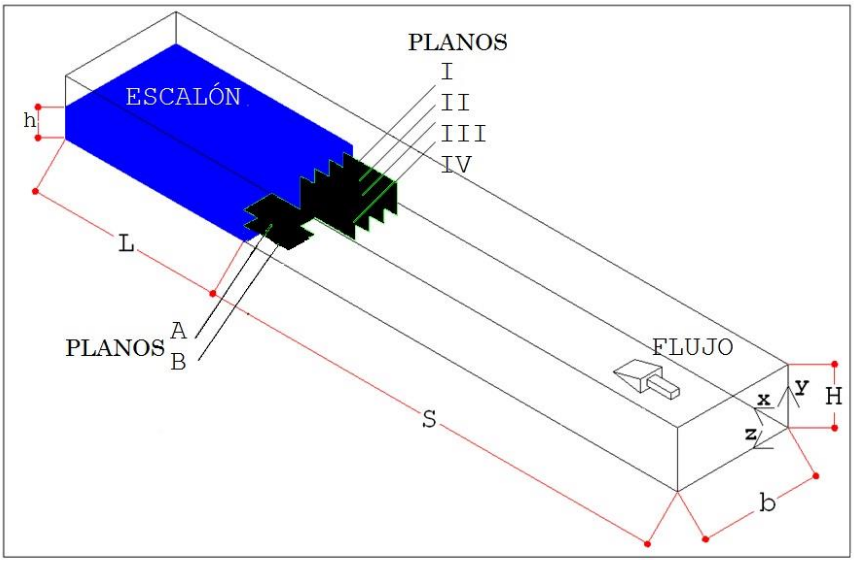


Fig. 2: Dominio computacional.

The grid independence study is performed using the height (a) of the recirculation zone in the center plane as the characteristic value to declare independence, for case 2 with Reynolds 400, the results are shown in Table II. With the results of this study it was decided to work with a mesh of 210: 60: 80 elements in the x, y, z, respectively. The mesh used near the step shown in Figure 3.

Tabla II: Independencia de malla.

Malla [x:y:z]	Altura (a)	Diferencia $\frac{a_{A)} - a_{\phi)}}{a_{A)}}$
A) 210:80:80	0.720h	-------
B) 210:60:80	0.718h	0.3%
C) 210:60:60	0.700h	2.8%
D) 180:80:80	0.695h	3.5%

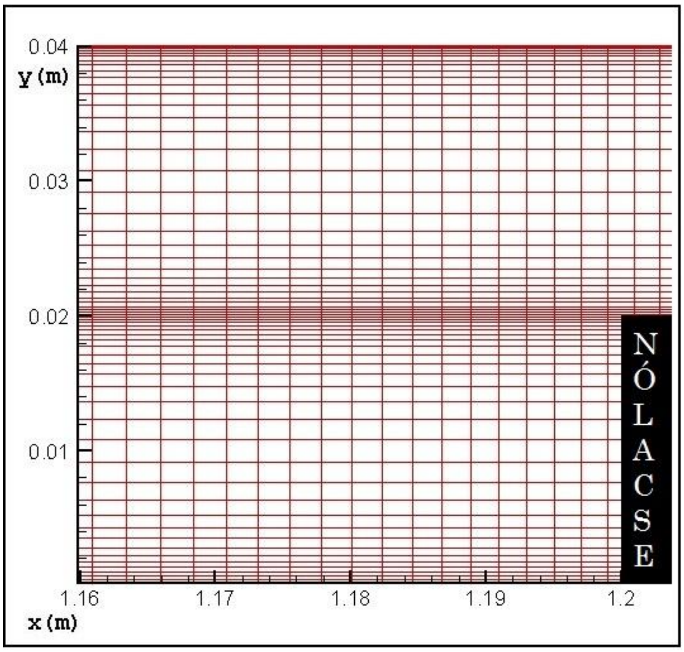


Fig. 3: Malla de dominio computacional.

EXPERIMENTAL METHODOLOGY.

To validate the numerical code an experimental analysis is performed in the wind tunnel of the Laboratory for Thermal Engineering and Applied Hydraulics (LABINTHAP) of the National Polytechnic Institute, which is shown in Figure 4.

The dimensions used are the same as those imposed on the computational domain. Wind tunnel has a flared entrance which allows the uniform flow in the input section, this device was built to the specifications of the ANSI Standard 210-85 (13) in the tunnel also has a base for supporting the injector smoke placed at a distance of 1.25m from the test area in order that this device does not influence the flow behavior, it eventually has a processing part for connecting the wind tunnel with the fan (Otto mark 5 pallets) and decrease the pressure loss in the tunnel piece transformation was constructed with specifications Barlow et al. (14). Environmental conditions (temperature [° C], air pressure [Pa], humidity [%]) are monitored with the meteorological measurement system Digiquartz®.

To measure the velocity field in the laser technique test particle image velocimetry (PIV) is used. The system features a high resolution CCD camera (1600 x 1186 pixels) Kodak Megaplus ES 1.0, a twin Nd: YAG high energy (400mJ) and a CPU that contains the system data acquisition FlowManager.

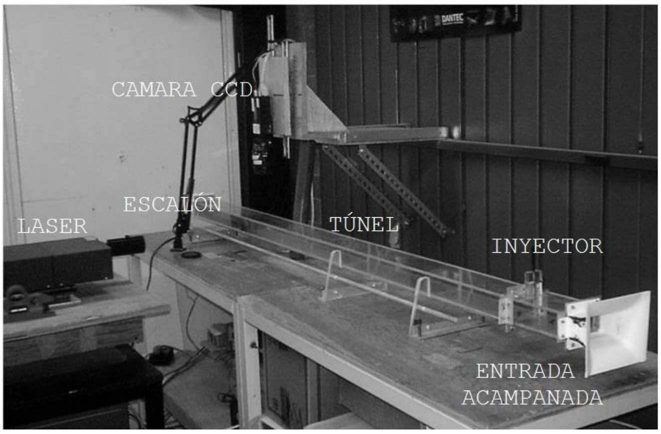


Fig. 4: Túnel de viento del LABINTHAP.

The tracer particles used in the measurement of the velocity field are obtained with a smoke generator of TEKNOVA® mark, which utilizes mineral oil of density 854kg / m3 and yields particles with average diameter of 10 microns. Smoke is introduced into the tunnel through a nozzle LABINTHAP built in, which has an outer diameter of 0.016my has 40 holes of diameter 0.0008m uniformly distributed in the injector.

RESULTS.

To analyze the recirculation zone 6 facing the step planes are studied in different positions as specified in Table III and shown in Figure 2. To characterize the vortex height, length and coordinates of the center of the vortex is measured: vertical axis (y ') and long axis (x') in all planes, as shown in Figure 1.

To determine the ability of the numeric code FLUSS reproduce the behavior of the flow in a rectangular duct with abrupt change of section, the results obtained by means of it, are compared with results obtained experimentally in this investigation. The data compared are the height (a), and the coordinates of the center of the vortex (x ', y') in the central plane of the duct to the geometric conditions of RA and RC = 4 = 2 for a Reynolds 400.

Tabla III: Planos analizados.

Plano	Posición
I	z = 0.125b
II	z = 0.250b
III	z = 0.375b
IV	z = 0.500b
A	y = 1.0h
B	y = 0.5h

The results obtained with the experimental program FLUSS and are summarized in Table IV and in Figures 5A and 5B is the same. In the first figure (5A) the velocity field obtained with the program developed FLUSS shown, while in the second (5B) the velocity field obtained experimentally is shown.

Tabla IV: Validación de código numérico FLUSS.

Parámetro	Experimento PIV	Simulación FLUSS	$\frac{\phi_{\exp} - \phi_{FLUSS}}{\phi_{\exp}}$
(a)	0.74h	0.72h	2.7%
(x’)	0.43h	0.42h	2.3%
(y’)	0.31h	0.32h	3.1%

From the results shown in Table IV is concluded that the numerical code developed FLUSS is a reliable tool to solve the proposed problem, because the difference between the results obtained by FLUSS program and experimental study are not greater than 4%.

The experimental results reported in Table IV have an uncertainty value of approximately 0.2%. This percentage was calculated from the average value of the measurements obtained from a series of 90 images (15).

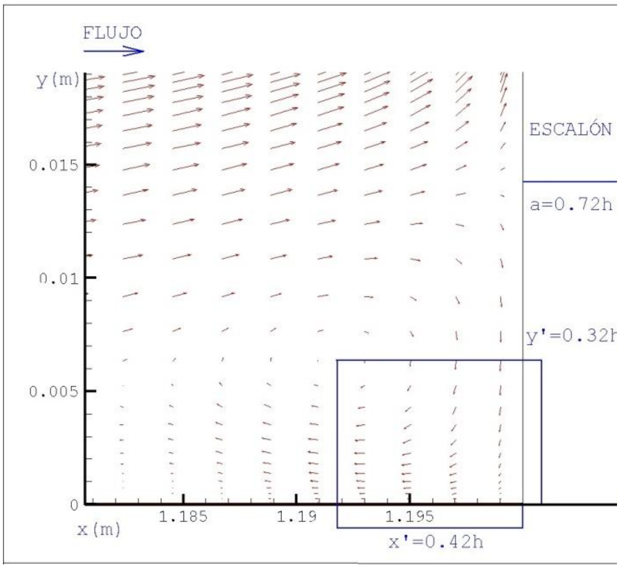


Fig. 5A: Campo de velocidad obtenido con FLUSS (Re_h=400).

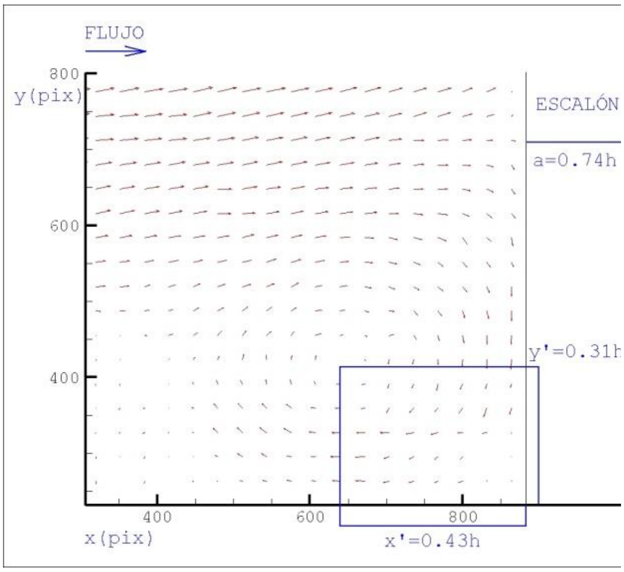


Fig. 5B: Campo de velocidad obtenido con PIV (Re_h=400).

The results of the characterization of the main recirculation zone with the numerical code obtained FLUSS for air flow in a rectangular duct with RA = 4 and Re = 400 are shown in Table V, all results are parameterized according to the step height. In Figures 6A and 6B the first and fourth planes respectively distributed along the axis "z", while appreciated in Figures 7A and 7B the 'x-z planes distributed along axis "y ".

Tabla V: Resultados numéricos en conducto con RA=4 y Re_h=400.

Plano	a/h	r/h	x'/h	y'/h
I (z=0.125b)	0.66	1.41	0.23	0.34
II (z=0.250b)	0.72	1.53	0.35	0.30
III (z=0.375b)	0.72	1.77	0.41	0.31
IV (z=0.500b)	0.72	1.90	0.42	0.32

The shape of the vortex in the central plane is well defined quasi-circular shape typical of a simple vortex as specified Freymuth (16), but close to the wall (in the I) the vortex has elongated shape and non-uniform, this change position and shape along the axis "z" of the recirculation zone indicates that it disappears close to the wall and flow content in the recirculation zone will pass over the step to continue its trajectory outside the duct. The change in shape of the recirculation zone is the growth of the boundary layer on the wall (viscous effects). It can also be noted that the length of the recirculation zone in the plane IV is 25% higher than in the I, and height of the area in the central plane is 8% higher than in the plane near the wall, so the vortex disappears after point located at z = 0.125b.

A and B respectively, plans to Reh = 400 are shown in Figures 7A and 7B. In these figures it can be seen the development of the boundary layer at the side wall, and as a new recirculation zone is formed near the sidewalls against the step, less intense than that found in the plane "x-z '. This situation is more evident in Figure 7B, where a darker colored area shown in the corners of the step near the walls. The scale shown in this color indicates null values speed, but in reality it is negative velocities, as indicated by the direction of current drawn lines; These images indicate that the flow near the walls presents a much more complex than had been reported so far behavior.

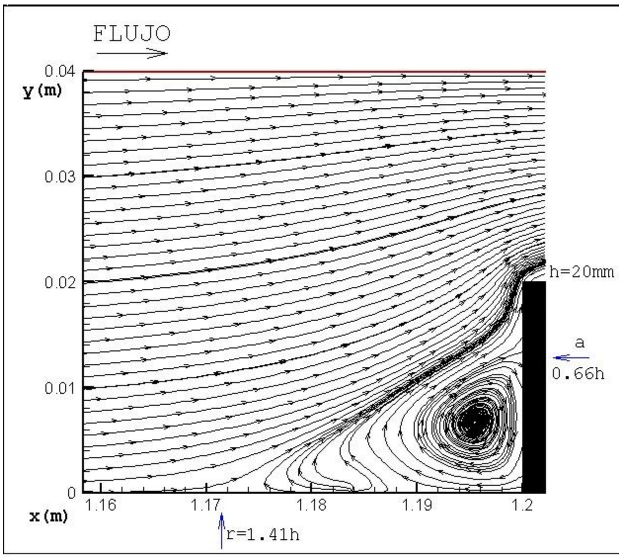


Fig. 6A: Vórtice en plano I con Re_h=400.

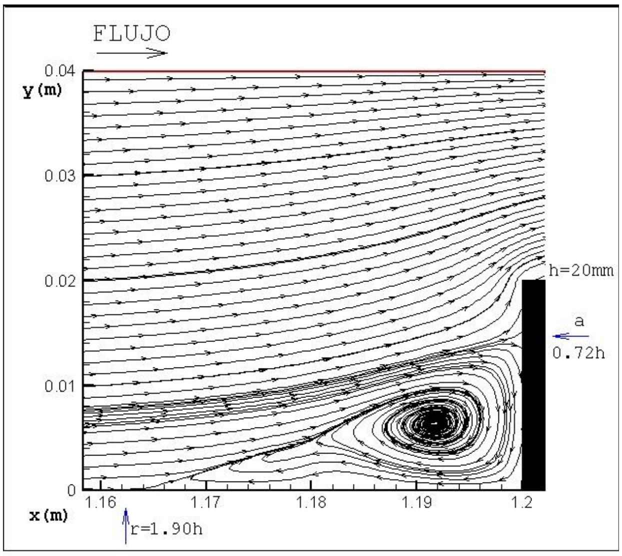


Fig. 6B: Vórtice en plano IV con Re_h=400.

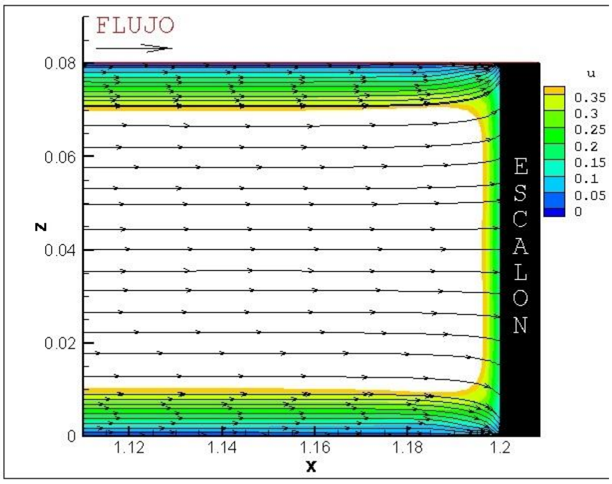


Fig. 7A: Contornos de velocidad "u" en plano A con Re_h=400.

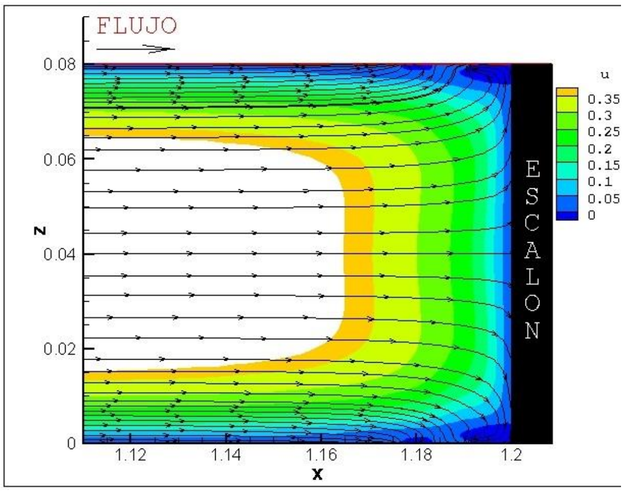


Fig. 7B: Contornos de velocidad "u" en plano B con Re_h=400.

To observe the influence of speed on the flow behavior in a conduit with step results for two different Reynolds numbers considering the same geometry shown, the results are shown in Table VI for Reynolds 300 and 500 respectively.

Tabla VI: Resultados numéricos en conducto con RA=4.

	Reynolds 300				Reynolds 500			
Plano	a/h	r/h	x'/h	y'/h	a/h	r/h	x'/h	y'/h
I (z=0.125b)	0.66	1.29	0.20	0.37	0.66	1.53	0.24	0.32
II (z=0.250b)	0.72	1.41	0.33	0.30	0.72	1.65	0.36	0.29
III (z=0.375b)	0.72	1.49	0.39	0.31	0.72	1.90	0.41	0.31
IV (z=0.500b)	0.72	1.65	0.41	0.31	0.72	2.03	0.43	0.32

From the numerical results it is concluded that if the Reynolds number increases the height of the recirculation zone does not vary its magnitude, this agrees with the results obtained by Largeau [8], and the height of the vortex occupies 72% of the step height in the planes II, III and IV.

Furthermore in case the length of the separation zone results indicate that as the Reynolds number also increases the length increases, this trend behavior occurs at all levels tested, this is due to the conservation of momentum in this area. Thus, when the Reynolds 300 500 increases the length will increase by 18% and 15% central plane in the plane I. As the length of the recirculation zone coordinates [x ', y'] its magnitude vary if the Reynolds number is increased, but in this case is increased up to 5% for all planes.

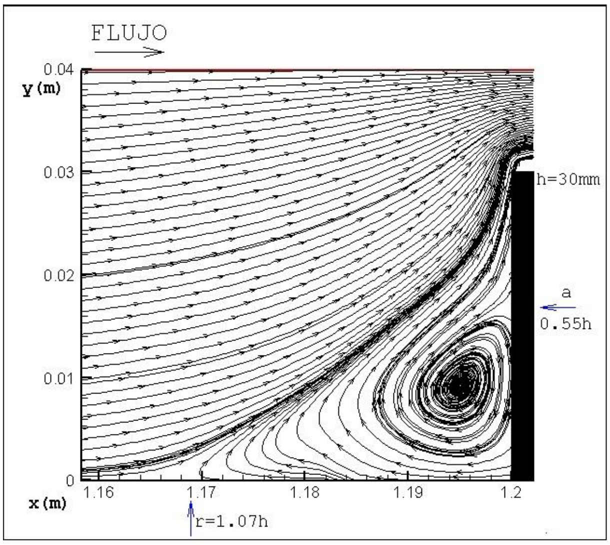


Fig. 8A: Vórtice en plano I para Re_h=400 y RA=2.7.

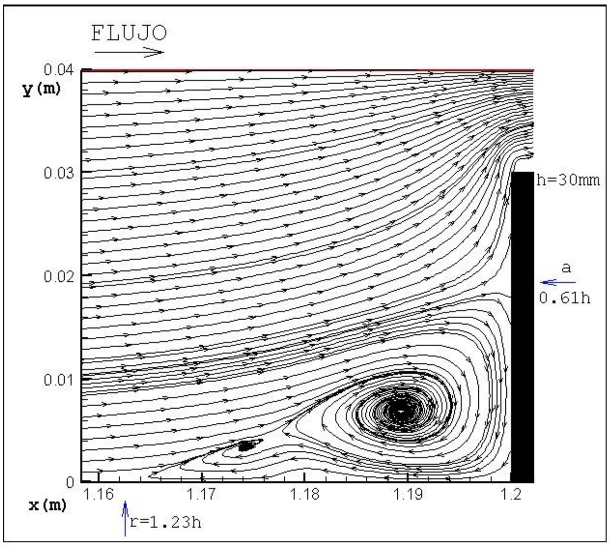


Fig. 8B: Vórtice en plano IV para Re_h=400 y RA=2.7.

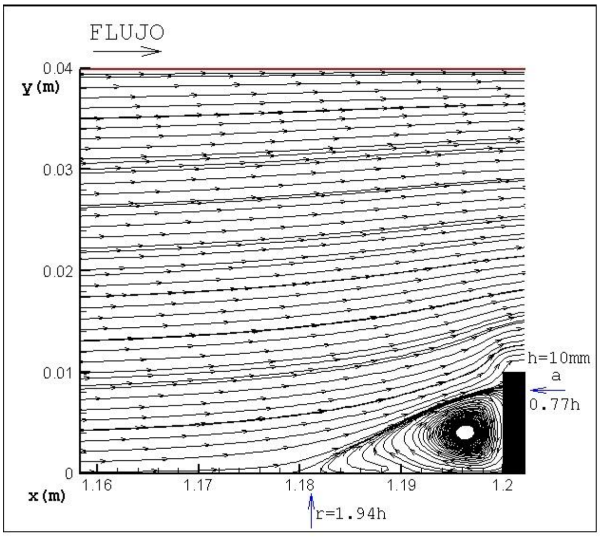


Fig. 9A: Vórtice en plano I para Re_h=400 y RA=8.

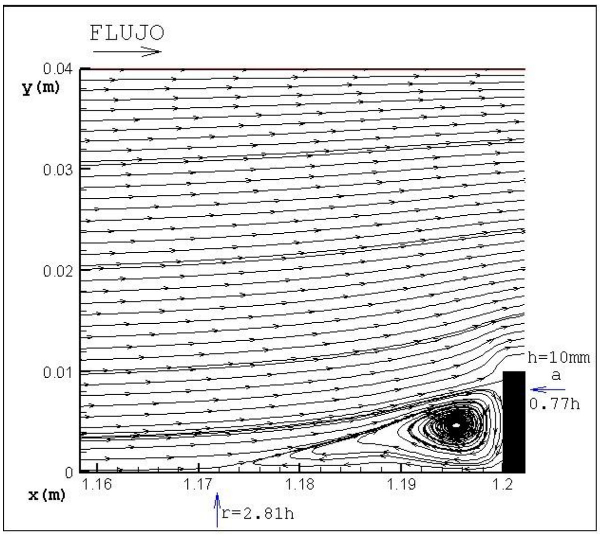


Fig. 9B: Vórtice para plano IV para Re_h=400 y RA=8.

To determine the influence of the aspect ratio in the flow behavior of two different step heights using a Reynolds 400 for both cases are analyzed, the results are shown in Table VII. The proposals for the step heights are 0.03myh = h = 0.01m resulting in RA = RA = 2.7 and 8 respectively.

In 8A and 8B vortices formed in the planes I and IV are respectively shown when having RA = 2.7, while in Figures 9A and 9B vortices formed in the planes I and IV shown respectively for when It has a RA = 8.

From the results in Table VII it shows that the height of the vortex will increase if the aspect ratio increases. When you have RA = 2.7 height of the vortex occupies 61% of the step height, but if you have a RA = 8, the height will be approximately 77% the height of the step. This happens because the increase (h) the area of the outlet section drastically, which causes an increase in fluid velocity, having a higher speed area, part of the low velocity fluid is not reduced within the recirculation zone it will focus on it, causing the height of the separation zone decreases. The same applies to the length of the recirculation zone, which shows that if the RA is increased, the length will also increase to a value of almost 3 times the height of the step at the central level.

Tabla VII: Resultados numéricos en conducto con Re=400.

	Relación de aspecto de 2.7				Relación de aspecto de 8			
Plano	a/h	r/h	x'/h	y'/h	a/h	r/h	x'/h	y'/h
I (z=0.125b)	0.55	1.07	0.16	0.30	0.77	1.94	0.35	0.37
II (z=0.250b)	0.60	1.07	0.27	0.23	0.77	2.37	0.36	0.37
III (z=0.375b)	0.61	1.23	0.33	0.22	0.77	2.81	0.45	0.41
IV (z=0.500b)	0.61	1.23	0.35	0.22	0.77	2.81	0.42	0.43

Finally the three-dimensional flow behavior for the aforesaid cases is shown in Figure 10A flow in the flow duct having an aspect ratio of 8 is shown in the conduit with an aspect ratio of 2.7 and figure 10B .

It can be seen as the flow in the vicinity of the step moves along the "z" axis to the sidewalls. The reason for this particular behavior is that the flow time is greater in the central part than in the vicinity of the side walls, because in these has imposed no-slip condition, so that when the flow is the step, the latter acts as a barrier preventing the passage of the flow stream, the flow therefore seeks to meet the nature of conservation of momentum and moves the side walls helically as described by Wilhelm [4], Chiang and Sheu [10] and Stüer et al. [11].

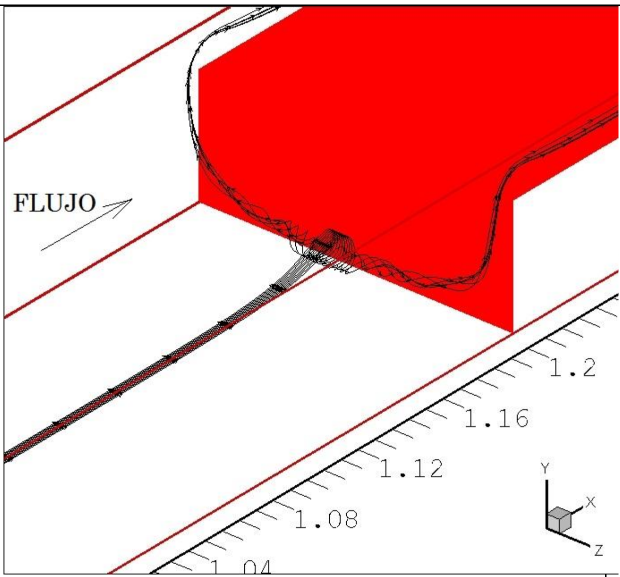


Fig. 10A: Zona de recirculación a lo largo de "z" con Re_h=400 y RA=2.7.

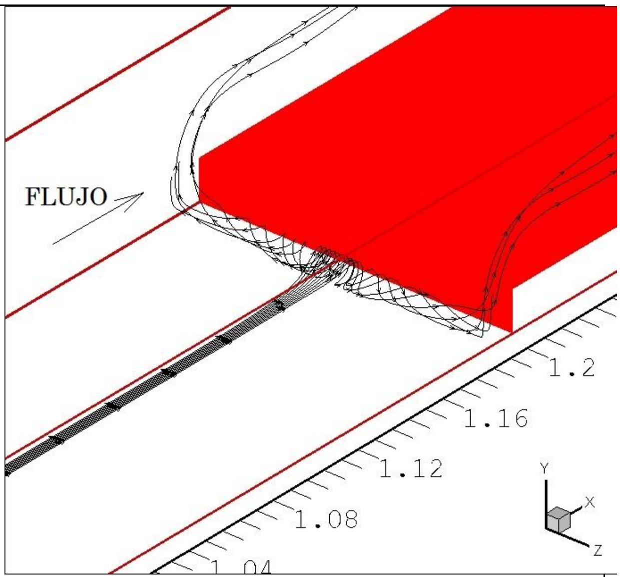


Fig. 10B: Zona de recirculación a lo largo de "z" con Re_h=400 y RA=8.

CONCLUSIONS

In this paper a numerical code called FLUSS was conducted to analyze the behavior of laminar flow through a rectangular duct with different RA and Reynolds, and the key findings are:

The difference between the numerical results with the program developed FLUSS and experimental study show a difference of 4%, when the results are compared in the central plane of the duct.

The recirculation zone adjacent to the step shifts the center of the conduit towards the side walls thereof as a helical vortex.

The length of the separation zone increases in value as the flow rate, ie, is directly proportional to Reynolds increases.

The height of the separation zone in the center of the conduit does not change its magnitude when the Reynolds number increases

The coordinates of the center of the vortex to change position as it approaches the wall, and while the coordinate x 'decreases, y coordinate' increases.

Bibliography

H. Abu-Mulaweh, B. Armaly and T. Chen, Laminar natural convection flow over a vertical forward-facing step, J. Thermophys. Heat Trans., vol. 10, pp. 517–523, (1996).

M. Sherry, D. Lo Jacono, J. Sheridan, R. Mathis and I. Marusic, Flow separation characterization of a forward facing step immersed in a turbulent boundary layer, Sixth International Symposium of Turbulence and Shear Flow Phenomena. pp. 1325-1330, (2009).

I. Ali, S. Becker, J. Utzmann and C. Munz, Aeroacoustic study of a forward facing step using linearized Euler equations, Physica D., pp. 1-6, (2008).

D Wilhelm, C. Härtel and L Kleiser, Computational analysis of the two-dimensional-three-dimensional transition in forward-facing step flow, J. of Scientific Computing., vol. 17, (2003).

T. Ando and T. Shakouchi, Flow characteristics over forward facing step and through abrupt contraction pipe and drag reduction, Res. Rep. Fac. Eng. Mie Univ. vol. 29, pp.1-8, (2004).

S. Becker, M. Escobar, C. Hahn, I. Ali, M. Kaltenbacher, B. Basel and M. Grunewald, Experimental and numerical investigation of the flow induced noise from a forward step, 11ª AIAA/CEAS. AIAA 2005-3006, (2005).

W. Moss and S. Baker, Recirculating flows associated with two-dimensionnal steps, Aero Q 32. pp. 693-704, (1979).

J. Largeau and V. Moriniere, Wall pressure fluctuations and topology in separated flows over a forward facing step, Exp. Fluids. vol. 42, pp. 21-40, (2007).

E. Fiorentini, R. Camussi, F. Pereira, M. Felli, G. Aloisio and A. Di Marco, Velocity and wall pressure correlations over a forward step flow, Advances in Turbulence XI. vol. 117, (2007).

T. Chiang and T. Sheu, Numerical studies of a three-dimensional flow in suddenly contracted channels, Physics of Fluid., vol. 14, pp. 1601-1616, (2002).

H. Stüer, A. Gyr and W. Kinzelbach, Laminar separation on a forward-facing step, Eur. J. Mech. B/Fluids. vol. 18, pp. 675-692, (1999).

S. V. Patankar, Numerical Heat Transfer and Fluid Flow, Taylor and Fracis, Philadelphia. (1980).

American Nacional Standard, Laboratory Methods of Testing Fang for Rating, ANSI/AMCA 210-85, AMCA Standard, ANSI/ASHRAE. (1985).

J. Barlow, W. Rae and A. Pope, Low-speed wind tunel testing, Third Edition, Wiley-Interscience Publication, pp. 80-83, (1999).

R. Figliola and D. Beasley, Análisis de incertidumbre, Mediciones mecánicas teoría y diseño, 3a edición, Ed. Alfaomega, México D.F., (2003).

P. Freymuth, F. Finaish and W. Bank, Three dimensional vortex patterns in a starting flow, J. Fluids Mech. vol. 161, pp. 239-248, (1986).