NUMERICAL AND EXPERIMENTAL RESULTS FOR FLOW THROUGH A FORWARD FACING STEP CHANNEL

Juan G. Barbosa-Saldaña 1, Oscar A. Morales-Contreras 2, José A. Jiménez-Bernal 3, Claudia del C. Gutiérrez-Torres 4 & Luis A. Moreno Pacheco 5

1,3,4,5 Instituto Politécnico Nacional
ESIME-Zacatenco-LABINTHAP Edif. 5 3er P. Col. Lindavista C.P. 07783 México D.F. México
Phone: 52-55-5729-6000 ext.: 54882
2 Universidad Autónoma de Baja California Centro de Ingeniería y Tecnología (CITEC)
Blvd. Universitario # 1000 Unidad Valle de las palmas
Tijuana, Baja California CP 2500

ABSTRACT

A numerical code based on the finite volume discretization technique is developed to simulate the flow through a three-dimensional forward facing step. A rectangular channel encloses the forward facing step with a contraction ratio of two and an aspect ratio of four. The channel total length in the stream-wise direction is thirty times the step height. The step occupies one third of the total axial length and is located at the channel exit. The numerical results were validated using PIV (Particle Image Velocimetry) measurements and visualization techniques realized for laminar airflow through an acrylic channel. The percent difference between numerical and experimental results is around 6%, giving an excellent approximation and therefore validating the numerical procedure. Numerical results presented in this work clearly show the tri-dimensionality of the flow structures at the vicinity of the step for the aspect and contraction ratio considered.

Keywords: Flow separation, Forward facing step channel, Numerical simulation, PIV.

1. INTRODUCTION

Flow separation and reattachment are phenomena inherently presented in several industrial devices such as diffusers, valves, turbine blades, pieces of electronic cooling equipment, cooling of nuclear reactors, flow in combustion chambers, flow over plates with ribs, flow in wide angle diffusers, etc. In some other cases these are induced for improving the heat transfer rate in compact heat exchangers or to enhance mixed conditions in combustion chambers [1]. Occasionally, flow separation appears in rectangular channels where the geometry suddenly or gradually is changed, for example in piping systems of cooling and air conditioning, in flow devices used in the nutritional sector, in transporting chemical agents, in polymer elaboration processes and in pipes equipped with baffles [2]. The flow separation in rectangular channels with sudden contraction generates a pressure drop and a recirculation zone or separation bubble. In addition, depending on the flow regime vibrations and noise can be found.

The study of the flow separation and reattachment is a complicated issue; however a valid simplification to study these phenomena is the analysis of the flow through a stepped rectangular channel. Although its simple geometry the flow separation and reattachment are highly evident and most of the flow features remain present. In this sense the experimental and numerical studies in these devices become the most affordable strategy for solving the flow problem first because the experimental construction do not require a high investment and from the numerical point of view the computational effort even for a three dimensional analysis is not high. In the last decade several numerical studies have been conducted to achieve a better knowledge and understanding of the hydrodynamic of the separated and reattached flow. In this aspect the backward-facing step has been the central objective for several researches, and even more this problem is considered as a benchmark problem for validating numerical codes and procedures [3 & 4]. Many less studies deal with the flow adjacent to a forward facing step (FFS) as is pointed out by Abu-Mulaweh [5].

The generalization of the behavior of the flow through a FFS can be observed in Fig. 1. Adjacent to the step and attached to the bottom wall a recirculation zone is presented and can be characterized by a length (r) in the axial direction and a height (a) in the vertical direction. Sometimes a zone of recirculation is allocated on the step and for higher Reynolds occasionally a separation zone is attached to the top channel wall.

Numerical efforts to analyze the fluid flow through a forward facing step were conducted initially for a two dimensional analysis. Among those first works are those presented by Dennis and Smith [6] and Mei and Plotkin [7], who concluded that the mean features of the flow are the large recirculation zone previous to the step and a small one presented on the stepped wall. Ratnish-Kumar and Naidu [8] developed a stream-function-vorticity formulation for solving the two dimensional Navier-Stokes equations for laminar flow. In their publication they did
not include the geometrical factors for the computational domain making their results difficult for being reproduced. In similar way, Houde et al., [9] used a stream-function vorticity formulation for analyzing the steady two dimensional laminar flow problem following a forward-facing step. They implemented a second order difference scheme to numerically solve the problem. In their study, they found a re-circulation zone at the step corner and also found that the flow separates from the bottom wall upstream the forward facing step. Even though their results presented an excellent agreement with previous results reported in literature, their approximation is for a two-dimension problem. Some studies have been carried out with the objective of characterized the recirculation zone at the step, for example, Wilhelm and Kleiser [10] developed a numerical code using spectral methods for discretizing the computational domain and analyzing the laminar flow in a channel with and contraction ratio equal to four. They reported results for Reynolds not larger than 1200 and concluded that the length \( r \) and height \( a \) of the separation bubble depends on the Reynolds number such that \( r=Re^{0.6} \) and \( a=Re^{0.2} \).

Others authors as Aseban et al., [11] had conducted their studies for the forward facing step geometry to analyze the mixed convective flow in vertical plates or for studying the mixed convective flow in two dimension channel for assisting and opposing flow as presented by Abu-Mulaweh et al., in several publications [12, 13 & 14]. In the last 20 years several numerical and experimental studies have been performed to analyze flow separation trough a FFS rectangular channel. Nevertheless, most of these consider the phenomenon of the separation for channels with aspect ratios larger than ten and then under this consideration they simplify the analysis to a two-dimensional problem. Therefore, the analysis of the three-dimensional behavior of the separation phenomenon is not complete.

\[ \text{Figure 1. Flow through a forward facing step (FFS).} \]

Nakamura et al., [15] showed results for a three-dimensional analysis by means of a DNS simulation. They compared their results against those of Shakouchi et al., [16] for a Reynolds number of 900 based on the step height. The comparison has an excellent agreement but the experimental results of Shakouchi shows three vortex on the stepped wall while the results of Nakamura only are capable of reproduce two of them, this discrepancy was associated to the grid size which is not as fine as necessary to completely represent the flow phenomenon.

One of the most important experimental work that treats the study of FFS step is the one published by Stüer et al., [17]. In their investigation they worked with a 0.01m step height, AR=4 and laminar flow. In this study the flow visualization with hydrogen bubbles technique was used. They concluded that the fluid within the separation bubble is parallel transported to the step in the span-wise direction and moves slowly. They reported that the height of the separation zone is 0.75h for Re=330.

The importance of studying the flow passing a forward facing step is described by Stuer et. al [17] and Ravindran [18]. They mentioned applications such that enhance heat transfer and flow mixing rates, flows over obstacles such
buildings and cooling of electronic equipments as well as in the control of fluid flow for designing fluid dynamical systems.

The objective of the present study is experimentally and numerically evaluating the three-dimensional behavior of the fluid structures, separation and reattachment of a flow through a forward facing step channel with a contraction ratio (CR) and aspect ratio (AR) equals to 2 and 4 respectively. The analysis for the tri-dimensionality and then the importance of the spanwise velocity component in the flow is the principal objective as it will be presented.

2. EXPERIMENTAL METHODOLOGY

An acrylic rectangular wind tunnel was built in the ESIME-Z-IPN in Mexico in order to carry out the experimental study. The tunnel was made of 9mm thick and their dimensions were parameterized respect to the step height (h=0.02m) such that the contraction ratio is equal to two (CR=2) and the aspect ratio is equal to four (AR=4). The step was built with Nylamid SL® and occupies one third of the total axial length and is located at the channel exit. The dimensions of the experimental setup are according to Fig. 1.

At the channel inlet the wind tunnel was equipped with a nozzle, which allows the fluid to be uniform in this section. At the channel exit, a transformation piece to connect the wind tunnel with the fan was installed. The nozzle was designed under ANSI norm specifications [19] while the transformation piece was built according to Barlow and collaborators [20]. A Digiquartz® meteorological station is used to give the weather conditions such as temperature [ºC], atmospheric pressure [Pa], and relative humidity [%]. By means of these data and an ideal gas equation the air density was computed and used to estimate the Reynolds number. The step height is used as characteristic length in the Reynolds number definition.

To determine the velocity field in the test zone the technique of particle image velocimetry (PIV) was employed. The system consists in a CCD high resolution camera (1600X1186 pixels), a twin laser Nd:YAG high energy (32mJ) and a CPU with the flow manager software as data acquisition system. The track particles used in the velocity field are obtained with a smoke generator TEKNOVA® that uses mineral oil (ρoil=854kg/m³). The particles are introduced to the test zone by means of an injector type I, which is 0.016m of diameter and has 40 holes of 0.0008m of diameter evenly distributed along the transversal channel’s direction. A general view of the experimental setup and the PIV measurement devices is presented in Fig. 2.

![Figure 2. Wind tunnel installed in the LABINTHAP.](image)
y planes in the span-wise direction including the central plane (Plane IV) and a plane that is 5mm away from the side wall as can be appreciated in Fig. 3.

![Figure 3. Planes placed in z-axis.](image)

The experimental characterization of the recirculation zone is carried out by measuring the height (a) as well as the location of the vortex core in the vertical ($\delta_y$) and axial ($\delta_z$) axes, as presented in Fig. 4. These measurements were made for each one of the above mentioned planes. In addition, some of the experimental results in the central pan-wise plane were used to validate the numerical results as will be discussed forthcoming.

![Figure 4. Measurements for characterizing the recirculation zone.](image)

3. NUMERICAL PROCEDURE
A FORTRAN code was developed to numerically study the stated problem for an incompressible fluid with constant properties (\( \mu = 1.81 \times 10^{-5} \text{ kg m}^{-1} \text{ s}^{-1} \) & \( \rho = 1.205 \text{ kg m}^{-3} \)). The governing equations expressing the mass conservation and momentum in Cartesian tensor notation are presented in eqs (1)-(2) [21].

\[
\frac{\partial}{\partial x_i} u_i = 0 \tag{1}
\]

\[
\rho \left[ \frac{\partial u_i}{\partial x_j} \right] = \rho g_i + \frac{\partial}{\partial x_j} \left[ -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \tag{2}
\]

A finite volume discretization technique was applied to solve the momentum equations inside the computational domain. The SIMPLE algorithm is used for linking the velocity and pressure distributions in the iterative procedure [22]. At the final step of every iteration the velocity field and pressure distribution are corrected and updated until reach convergence. The power law scheme was utilized to represent the convection-diffusion term at the control volume interfaces. Velocity nodes were located at staggered locations in each coordinate direction while pressure and other scalar properties were evaluated at the main grid nodes. At the channel exit the natural outflow boundary conditions assuming zero gradients of all flow variables were imposed [23]. To simulate the solid block inside the domain a very high diffusion coefficient for the momentum equations was chosen [24]. At the solid-fluid interface the diffusion coefficients where evaluated by a weighted harmonic mean of the properties in neighboring control volumes as described by Patankar [22].

The boundary conditions considered for the computational domain at the solid walls were those of no-slip condition while at the channel inlet the flow was considered as fully developed according to the correlation for a rectangular cross-section duct presented by Shah and London [25].

A combination of a line-by-line solver and the tri-diagonal matrix algorithm was used for each plane in x-, y-, and z-coordinate directions to compute the velocity components and pressure inside the computational domain. Under-relaxation for the velocity components (\( \alpha_u = \alpha_v = \alpha_w = 0.2 \)) and pressure (\( \alpha_p = 0.2 \)) were imposed to guarantee convergence. Convergence for the solution was declared when the normalized residuals for the velocity components and pressure were less than 1x10^{-8}.

A non-uniform grid size was considered for solving the numerical problem. In this sense, at the solid walls and at the edge of the step the grid was composed by small-size control volumes (fine grid) and the control volume size increased far away from the solid walls. The grid size was deployed by means of a geometrical expansion factor, such that each control volume is a certain percentage larger than its predecessor. A detailed description for the grid generation can be found in previous work presented by the authors [26].

### 3.1. Grid independence and numerical validation

In order to investigate if the predicted numerical results are grid independent, extensive studies using several grid densities for a Reynolds number (Re=365) were carried out. The point at the central plane where the shear stress (\( \tau_{xx} \)) is equal to zero was used as parameter to declare grid independence. A grid size of 150:40:80 does not represent important variation when compared with a 180:40:80 grid size. Hence, the former was proposed for the productive runs. Table 1 summarizes the grid independence study considering the 180:40:80 grid density as the base case.

<table>
<thead>
<tr>
<th>Grid size</th>
<th>x/s position at the central span-wise plane where ( \tau_{xx} = 0 )</th>
<th>Relative percentual difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>180:40:80</td>
<td>0.1820</td>
<td>-----</td>
</tr>
<tr>
<td>150:40:80</td>
<td>0.1816</td>
<td>0.2197</td>
</tr>
<tr>
<td>150:40:60</td>
<td>0.1812</td>
<td>0.4395</td>
</tr>
<tr>
<td>150:60:40</td>
<td>0.1810</td>
<td>0.5494</td>
</tr>
<tr>
<td>120:40:40</td>
<td>0.1917</td>
<td>5.3296</td>
</tr>
</tbody>
</table>
Once grid independence was established, the second step was to validate the numerical code. To satisfy this requirement the numerical results at the central span-wise plane were compared with the experimental results for the vortex core location and the height of the recirculation zone adjacent to the step. As presented in Table 2 the relative percentage errors are around 6.0% in the worst of the cases when the experimental and numerical results were compared. Hence the numerical code has a good approximation and is an excellent tool to be used for studying the flow structures in a forward facing step channel. In addition, the Fig. 5 shows the comparison of the numerical and experimental results obtained in the validation procedure for the parameters described in Table 2.

<table>
<thead>
<tr>
<th>Length</th>
<th>Experimental (mm)</th>
<th>Numerical (mm)</th>
<th>Percentage error E (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>δₓ</td>
<td>8.5</td>
<td>8.0</td>
<td>5.88</td>
</tr>
<tr>
<td>δᵧ</td>
<td>6.1</td>
<td>5.9</td>
<td>3.2</td>
</tr>
<tr>
<td>a</td>
<td>14.6</td>
<td>14.1</td>
<td>3.3</td>
</tr>
</tbody>
</table>

Table 2. Numerical validation central plane at the span-wise direction (Re=365)

Once the problem has been defined, the experimental and numerical solution’s techniques have been described, the grid independence and validation requirements have been satisfied the following is to present and discuss the results.

4. RESULTS AND DISCUSSIONS
In this section some aspects of the experimental and numerical results will be discussed. It is very important to mention here that the tendency of the flow behavior was similar for all the Reynolds studied, but the magnitude of the flow velocity field as well as the dimensions of the recirculation zone were higher as the flow regimen was increased.

Figure 6 shows the adjacent recirculation zone previous to the step for Re=475 at different planes in the z coordinate direction. It is clear that the vortex changes its shape and size in the plane presented along the z-coordinate direction. This behavior indicates that as was supposed the flow must be highly tridimensional and the velocity component in

Figure 5. Flow structure attached to the step for Re=365. a) PIV velocity vector field. b) Numerical results for the vortex.
the span-wise direction must be considered for analysis when the channel has an aspect ratio equal to four. Hence the reduction for a two-dimensional analysis is not valid for this aspect ratio.

Figure 6. Recirculation zone adjacent to the step $Re=475$ at different span-wise planes. a) Plane 0. b) Plane I. c) Plane II. d) Plane III. e) Plane IV.
Near the side wall (Fig 6a) the vortex is enlarged in the vertical direction. Here the vortex-core reports the highest position in the vertical direction when compared with the others planes. Also, in this plane the position in the axial direction of the vortex core is the nearest to the step. The vortex moves towards the bottom wall and opposite to the main flow direction as the central plane is achieved.

The characterization of the vortex is achieved by means of the location of the vortex core. Here it is observed that this point moves toward the bottom wall and upstream the main flow. The result is that the recirculation zone adjacent to the step moves along the vertical and stream-wise direction and then there is a helical movement of the flow in this zone, hence it can be said that the recirculation zone has a highly tridimensional behavior. It is also observed that there is a separation zone on the stepped wall; however for Re=269 and Re=365 it was not found.

Figure 7 shows a numerical approximation of the three-dimensional streamlines along the channel for Re=475. Here the shadowed zone represents the step, while the lines represent the flow stream lines. The main flow enters the channel and moves downstream along the channel and once the step zone is achieved it is observed that part of the flow moves towards the lateral walls and then jumps the step at this point, this is the reason why the vortex is enlarged to the vertical direction in figure 6a. It can be said that the flow has its higher momentum at the central part because the fully developed condition imposed and once the step is reached the flow must find a way to continue to the exit, and then part of the flow moves towards the side wall because the momentum here is much lower due to the no-slip condition at the side walls and at this zone the flow jumps the step and continue to the channel exit. Other part of the flow does not move toward the side walls, instead is trapped inside the recirculation zone forming the reattachment zone and vortices adjacent to the step. The flow that enters at the channel at the vicinity of the upper wall remains in this zone and when it arrives to the step zone is pushed in the vertical direction against the upper wall and depends on the intensity this provokes the recirculation zone adjacent to the top wall. Once the stepped zone is passed the flow in this region continues to the channel exit.

![Figure 7. Streamlines for a forward facing step channel Re=475.](image)

Contours of the w-velocity component at different y-planes are presented in Fig. 8. for Re=475. Here it is evident that the zone at the vicinity of the step is higher influenced from the w-velocity component and then has a tridimensional behavior. Also, it can be confirmed that the flow tends to move towards the lateral side wall when the plane is located below the step height (Figs. 8a and 8b). Figure 8c) shows that over the step the flow moves towards the side wall, but also a small portion is directed towards the central plane (blue region and red region) this particular behavior should be attributed to the side walls and to the recirculation zone formed over the step. Figure 8d) shows that at the top channel wall the flow is directed to the central stream-wise plane, also indicates that the importance of the w-velocity component is associated to the region of the step and at the channel inlet and exit the w-velocity does not impact the flow behavior.

For characterizing the recirculation zone the streamlines and contours of the u-velocity component at the central span-wise plane (Plane IV) for different Reynolds numbers are presented in Fig. 9. The non-colored zones represent negative values for the u-velocity component. In others words, is in these zones where the flow presents the vortex zone where the flow has a high tridimensional behavior. Table 3 presents the values of the length (r/h) and height.
(a/h) of the recirculation zone parameterized to the step height (h) for the Reynolds numbers analyzed. Results of Table 3 shows that the size of the recirculation zone is extended in the negative x-direction as the Reynolds is increasing, but the height of this zone has not considerable change even when the Reynolds is almost doubled.

Table 3. Characterization for the recirculation zone for different Reynolds

<table>
<thead>
<tr>
<th>Re</th>
<th>r/h</th>
<th>a/h</th>
</tr>
</thead>
<tbody>
<tr>
<td>269</td>
<td>0.95</td>
<td>0.68</td>
</tr>
<tr>
<td>365</td>
<td>1.09</td>
<td>0.70</td>
</tr>
<tr>
<td>475</td>
<td>1.25</td>
<td>0.70</td>
</tr>
</tbody>
</table>

Figure 8. Contours for the w-velocity component at different y=constant planes for Re=475. a) y=4.87x10^{-4} m b) y=1x10^{-2} m c) y=2.048x10^{-2} m d) y=3.45x10^{-2} m.

It is important to say that the computational domain geometry does not change for the study cases and in all the computations the step height remains equal, then this could be the reason the height of the recirculation remains invariable because the height of the step remains constant. It can be said that the height of the recirculation zone depends on the step height while its length depends on the Reynolds number.
Figure 9. Streamlines for the recirculation zone adjacent to the step at the central z-coordinate plane for different Reynolds (numerical).

Figure 9c) shows that for Re=475 there is a recirculation zone over the step wall, this zone is neither present for Re=365 nor for Re=269. The reason could be associated to the higher momentum of the flow in this case.

Figure 10 presents the experimental results for the recirculation zone adjacent to the step for the Reynolds number studied. The reasoning presented above for the numerical results is reinforced and figure 10 could be used to support the validation process.
5. CONCLUSIONS

The present work shows experimental and numerical results of air-flow behavior through a forward facing step channel. These results are used to point out the mean features of the recirculation and flow separation in this geometry. It was found that a recirculation zone is attached to the step and the flow in this region is highly three-dimensional flow. The flow in this region presents a helicoidal structure and the vortex is enlarged at the side wall while at the channel center presents a well defined vortex. The flow in the stepped zone is pushed to the side walls and jumps the step in this region. Then the flow continues outwards the channel exit.

An increase of Reynolds number is associated with an enlargement of the recirculation zone in the stream-wise direction (r). But the size in the normal direction (a) does not present a significant variation even if the Reynolds number is almost doubled.

The w-velocity component has an important influence on the flow structures. Finally the location of a recirculation zone attached to the stepped wall was found for Re=475. However its complete characterization was not the mean purpose of the present work and a future research about this topic is strongly recommended for the future.

Figure 10. Visualization of the recirculation zone adjacent to the step at the central z-coordinate plane for different Reynolds (experimental) a) Re=269, b) Re=365 and c) Re=475.
7. REFERENCES


8. NOMENCLATURE

\[ a = \text{Separation height (m)} \]
\[ b = \text{Channel width (m)} \]
\[ E = \text{Percentual error} \]
\[ h = \text{Step height (m)} \]
\[ H = \text{Channel height (m)} \]
\[ L = \text{Out length (m)} \]
\[ p = \text{Pressure (Pa)} \]
\[ \texttt{AR} = \text{Aspect ratio } [b / h] \]
\[ \texttt{CR} = \text{Contraction ratio } [H / h] \]
\[ r = \text{Reattachment length (m)} \]
\[ \texttt{Re} = \text{Reynolds number } [2 \bar{u} h \rho / \mu] \]
\[ S = \text{Length (m)} \]
\[ u, v, w = \text{Velocity components (m/s)} \]
\[ x, y, z = \text{Coordinates axes} \]
\[ \bar{u} = \text{Bulk velocity (m/s)} \]

Greek letters
\[ \delta_x = \text{Axial coordinate of vortex (m)} \]
\[ \delta_y = \text{Vertical coordinate of vortex (m)} \]
\[ \rho = \text{Density } (1.205 \text{ kg/m}^3) \]
\[ \mu = \text{Viscosity } (1.81 \times 10^{-5} \text{ kg/m-s}) \]
\[ \tau = \text{Shear stress tensor} \]