1 Introduction

When a fluid moves through a curved duct, the centrifugal forces arising from the channel curvature set up a secondary motion which is superimposed to the primary flow. These phenomena give rise to an helical movement that has the effect of shifting high velocity regions toward the walls, with the consequent increase of the frictional losses.

Progress in internal fluid mechanics demands the study of this type of three-dimensional flow, because it constitutes an analysis tool allowing a better understanding of problems of practical interest. In view of the impossibility of finding analytical solutions for such phenomena, the research capability is intimately related to the availability of appropriate numerical tools.

The objective of this paper is to present a numerical scheme to predict some of the characteristics of the incompressible flow through ductings which are important in a design process. This is achieved by solving the three-dimensional time-dependent incompressible Navier-Stokes equations using a control volume approach.

The main difficulties associated with the solution of this type of problem are the treatment of the boundary conditions on the geometries that bound the domain, the choice of a proper storage location for the dependent variables and the lack of an explicit equation for the pressure.

The problem of the complex boundaries is treated by formulating and solving the conservation equations on a curvilinear coordinate system that matches the domain boundary. This is useful because the boundary conditions can be implemented accurately in the numerical solution of the governing equations. Different techniques can be used to numerically generate a curvilinear mesh, and a detailed review of the subject has been given in [1], and a specific 3-D generation procedure can be found in [2].

The computational discretization currently used for solving incompressible fluid flow problems is based on a staggered grid [3]. This technique requires a different location, together with a distinct computational cell for each velocity component and the pressure. In the present study it is proposed to compute the pressure and the velocity components at the same grid location. These parameters are located at the center of the same computational cell which is used for both the momentum and continuity balances. To avoid unrealistic fields, that normally would appear with such discretization, an opposed difference scheme for pressure and fluxes is used in the main flow direction.

Finally the pressure and velocity fields are coupled by means of a pressure equation derived on the basis of the SIMPLE method [4] modified for a curvilinear grid.

The present method has been applied to obtain the numerical solution of flows within ducts of different geometries. The results reveal the complex nature of the three-dimensional phenomena showing some aspects of the secondary flow.

2 Conservation Equations

The equations of motion written in the conservative form for a curvilinear system can be written as [5, 6]:

\[
\frac{\partial q}{\partial t} + \frac{\partial E}{\partial \xi} + \frac{\partial F}{\partial \eta} + \frac{\partial G}{\partial \zeta} = \frac{\partial R}{\partial \xi} + \frac{\partial S}{\partial \eta} + \frac{\partial T}{\partial \zeta}
\]

where \(\xi\) represents the "streamwise" direction, \(\eta\), the "normal" direction and \(\zeta\) the "binormal" direction as illustrated in Fig. 1
The flux and diffusion terms in equation (1) are:

\[
\begin{bmatrix}
q = J \\
u \\
w
\end{bmatrix}
\quad E = J \\
\begin{bmatrix}
U \\
u U + p \xi_x \\
w U + p \xi_z
\end{bmatrix}
\quad F = J \\
\begin{bmatrix}
V \\
u V + p \eta_x \\
w V + p \eta_z
\end{bmatrix}
\quad G = J \\
\begin{bmatrix}
W \\
u W + p \xi_y \\
w W + p \xi_z
\end{bmatrix}
\]

\[
R = \mu J \\
\begin{bmatrix}
g^{11} u_1 + g^{12} u_1 + g^{13} u_1 \\
g^{12} v_1 + g^{13} v_1 \\
g^{13} w_1 + g^{22} w_1 + g^{33} w_1
\end{bmatrix}
\]

\[
S = \mu J \\
\begin{bmatrix}
g^{21} u_2 + g^{22} u_2 + g^{23} u_2 \\
g^{22} v_2 + g^{23} v_2 \\
g^{23} w_2 + g^{32} w_2 + g^{33} w_2
\end{bmatrix}
\]

\[
T = \mu J \\
\begin{bmatrix}
g^{31} u_3 + g^{32} u_3 + g^{33} u_3 \\
g^{32} v_3 + g^{33} v_3 \\
g^{33} w_3 + g^{23} w_3 + g^{33} w_3
\end{bmatrix}
\]

where \(\mu\) represents the viscosity.

The cartesian velocity components \(u, v, w\) and the contravariant velocity components \(U, V, W\) are related by:

\[
\begin{align*}
U &= u \xi_x + v \xi_y + w \xi_z \\
V &= u \eta_x + v \eta_y + w \eta_z \\
W &= u \zeta_x + v \zeta_y + w \zeta_z
\end{align*}
\]  

(2)

The metric terms \(\xi_x, \xi_y, \xi_z\), etc., the jacobian \(J\) and the contravariant metric tensor components \(g_{ij}\), are obtained from:

\[
\begin{align*}
\xi_x &= (y_1 z_1 - y_2 z_2) / J \\
\xi_y &= (z_1 x_1 - z_2 x_2) / J \\
\xi_z &= (x_1 y_1 - x_2 y_2) / J \\
\eta_x &= (z_2 z_1 - y_2 z_2) / J \\
\eta_y &= (y_2 y_1 - x_2 y_2) / J \\
\eta_z &= (x_2 x_1 - x_2 y_2) / J \\
\zeta_x &= (y_2 z_1 - y_1 z_2) / J \\
\zeta_y &= (z_1 x_1 - z_2 x_2) / J \\
\zeta_z &= (x_1 y_1 - x_2 y_2) / J
\end{align*}
\]

\[
\begin{align*}
J &= x_1 y_1 z_1 + x_2 y_2 z_2 + x_3 y_3 z_3 \\
&- x_1 y_1 z_2 - x_2 y_2 z_1 - x_3 y_3 z_1 \\
&- x_1 y_2 z_3 - x_2 y_3 z_1 - x_3 y_3 z_1 \\
g_{xx} &= \frac{\partial \xi_x}{\partial x} \frac{\partial \xi_x}{\partial x} \\
&\quad + \frac{\partial \xi_y}{\partial x} \frac{\partial \xi_y}{\partial x} \\
&\quad + \frac{\partial \xi_z}{\partial x} \frac{\partial \xi_z}{\partial x}
\end{align*}
\]

with \(\xi^1 = \xi, \xi^2 = \eta, \xi^3 = \zeta\)

3 Discretization

In the proposed grid structure the pressure and cartesian velocity components are stored at the center of the computational cell. Each of the general cell which is made up of one unit in the \(x\) direction and two units in the \(y\) and \(z\) directions. Figure 3 shows a two-dimensional cell configuration on the computational \(\xi - \eta\) (\(k = \text{const}\)) plane for case of discretization. The base grid is used for both mass and momentum equations in a two-dimensional domain. Central differences are used to discretize the pressure gradients in the \(\eta\) and \(\zeta\) directions.
directions. This yields a system of equations which requires the values of velocity at the $j+1,k$ and $j-1,k$ and $j,k-1$ and $j,k+1$ faces. These are not interpolated but are calculated by overlapping elements in those directions. The pressure is obtained by the averaging of neighboring points.

To gain insight into this overlapping procedure, we turn to an example in 2-D for a $k=\text{const}$ plane. To simplify the notation the third subscript will be omitted in the following discussion.

When solving the system (1) both cartesian and curvilinear components are required. The first set of components is calculated and stored at the center $i+1/2,j+1/2$ of the element (Fig. 3). As a result of the overlapping procedure in the $j$ direction as illustrated on Fig. 4, these properties are also known at the $i+1/2,j+1$ location that corresponds to the center of the one cell half unit above. The same reasoning applies for the $j-1,k+1,$ and $k-1$ levels.

With the cartesian velocity components known at all $j$ and $k$ levels, the $V$ and $W$ components are computed from equation (2) using:

$$V_{i+1/2,j+1} = u_{i+1/2,j+1}(\eta x)_{i+1/2,j+1} + v_{i-1/2,j+1}(\eta y)_{i+1/2,j+1} + w_{i+1/2,j+1}(\eta z)_{i+1/2,j+1}$$

for $k = \text{const},$ and

$$W_{i+1/2,j+1} = u_{i+1/2,j+1}(\eta x)_{i+1/2,j+1} + v_{i-1/2,j+1}(\eta y)_{i+1/2,j+1} + w_{i+1/2,j+1}(\eta z)_{i+1/2,j+1}$$

for $j = \text{const}.$

In the "streamwise" direction $\xi,$ no averaging or overlapping is used. Mass gradients are obtained by upwind differencing, so the flux through the downstream $i+1,j,k$ face is controlled by the velocity located at the center of the cell $i+1/2,j,k.$ With this in mind the $U$ velocity component is obtained from equation (2) for the $k=\text{const}$ levels as:

$$U_{i,j} = u_{i-1/2,j}(\xi x)_{i,j} + v_{i-1/2,j}(\xi y)_{i,j} + w_{i-1/2,j}(\xi z)_{i,j}$$

On the other hand, pressure gradients are calculated by downwind differencing. This can be interpreted as if the pressure at the center of the element acts on its upstream face $i,j,k.$

The following $u$ momentum equation summarizes the proposed discretization.

$$
J_{i+1/2,j,k} = \frac{(u_{i+1,j,k})_{i+1/2,j,k} - (u_{i,j,k})_{i+1/2,j,k}}{\Delta t} + \frac{(JuU)_{i+1,j,k} - (JuU)_{i,j,k}}{\Delta \xi} + \frac{(JuV)_{i+1,j,k+1} - (JuV)_{i+1,j,k}}{2\Delta \eta} + \frac{(JuW)_{i+1,j,k+1} - (JuW)_{i+1,j,k}}{2\Delta \eta} \\
+ \frac{p_{i+1/2,j,k+1} - p_{i+1/2,j,k}}{\Delta \xi} + \frac{(JuX)_{i+1/2,j,k+1} - (JuX)_{i+1/2,j,k}}{2\Delta \eta} + \frac{(JuY)_{i+1/2,j,k+1} - (JuY)_{i+1/2,j,k}}{2\Delta \eta} + \frac{(JuZ)_{i+1/2,j,k+1} - (JuZ)_{i+1/2,j,k}}{2\Delta \xi}$

$+$ VISC $\equiv 0$

where VISC represents the resulting viscous terms over the element.

A similar combination of backward and forward differences has been used by reference [7] for the solution of the compressible Euler equations. References [8, 9] have also used the opposed-differencing idea to solve the steady Navier-Stokes equations.

To evaluate the convected momentum fluxes and diffusion terms at the cell faces, the weighted upstream difference scheme of Raithby and Torrance [10] has been adopted. These authors propose the use of weights depending on the Peclet number to calculate the degree of upwinding.

4 Solution Procedure

The scheme is explicit and in a general form can be written as:

$$
\Delta q + \Delta t(R_{q} + F_{q} + G_{q}) = \Delta t(S_{q} + T_{q})
$$

where $\Delta$ denotes the forward time difference operator and the superscript $n$ the time level.

The sequence of calculations is as follows. The velocity and pressure fields are first guessed. Then the three cartesian momentum components characterized by equation (3), are solved to get the three velocity components over the whole domain. These intermediate values do not satisfy mass conservation.

The next step is to adjust the velocity field in order that the continuity equation be satisfied. This is achieved through a suitable variation of the pressure field. The velocity-pressure coupling is based on the SIMPLE method [4]. By using the momentum equations, corrections to the curvilinear velocity components can be related to corrections to the pressure as:

$$
\delta U = f^{U} (\delta \rho), \delta V = f^{V} (\delta \rho), \delta W = f^{W} (\delta \rho)
$$

These expressions together with the use of the continuity constraint lead to a Poisson-like equation for the pressure cor-
rection, which leads to the following relation:

\[ A_p \delta p + \Sigma A_{ao} \delta p_{ao} = -D_p / \Delta t \]  \hspace{1cm} (5)

Where the \( A_i \)'s represent coefficients which are functions of the metric terms, \( \delta p \) is the pressure correction, and \( D \) the velocity divergence. The subscripts \( p \) and \( nb \) denote the center of the neighboring nodes, respectively.

This can be significantly simplified by neglecting the contribution of the surrounding points, which then reduces equation (5) to:

\[ \delta p = -D_p / A_p \Delta t \]  \hspace{1cm} (6)

and the pressure correction can be computed directly in terms of the divergence of the pressure field.

Once \( \delta p \) is evaluated, the corresponding curvilinear velocity corrections \( \delta U, \delta V, \delta W \) are calculated. These are then combined with the inextant velocity and pressure fields in order to verify the mass constraint requirement; that is:

\[ U = U^* + \delta U \]
\[ V = V^* + \delta V \]
\[ W = W^* + \delta W \]
\[ p = p^* + \delta p \]

where \( U, V, W, p \) and \( U^*, V^*, W^*, p^* \) represent those values that do and do not respectively satisfy both mass and momentum equations. More details on the way that equations (5) to (7) are derived can be found in [6].

The sequence represented by equations (6) and (7) which involve the computing of the pressure change and the mass correction, respectively, is applied in such a way that no overlapping cells take part at this step. In particular, the correction procedure is applied to all the cells in the main flow direction, but only to every other cell in the secondary directions. In doing so, the continuity control volumes do not overlap (as the momentum cells do), and there is no particular difficulty associated with this discretization.

As a result of this methodology, only one corrected curvilinear velocity component is known on each face \( (U, V, W) \) on the \( \xi, \eta, \zeta \) faces, respectively, and also the corrected pressure is not available at all the stations needed for the computing of the momentum equations. This inconveniency is solved by obtaining the two corresponding contravariant components, as well as the unknown pressure, as the average of surrounding known values. The cartesian velocity components are decoded by using the inverse relations of equations (2).

To update the working variables over the entire domain, a similar practice to the MAC method [11] is used. The grid is swept point-by-point in successive planes in the inlet-to-outlet direction. Improved values are immediately used as the procedure advances; consequently the right-hand-side term of equation (6) is intrinsically modified as the iterations progress. This is repeated until a desired level of accuracy is reached.

Finally the time step is advanced and the cycle is repeated until steady state is reached. This is estimated by comparing the root mean square of a velocity component between two consecutive time steps.

5 Boundary Conditions

5(a) Velocity. In the present approach both cartesian and curvilinear velocities take part in the calculation procedure, so boundary conditions should be given for both of them.

At the inlet a velocity profile in terms of the cartesian and contravariant components is specified. At a no-slip surface only one curvilinear component has to be supplied, because the other two do not contribute to the flow balance over the adjacent elements through these surfaces. This means for example that for the triad \( U, V, W \) along the \( \xi, \eta, \zeta \) coordinates, only the \( W \) component is needed at a wall coincident with a \( \xi-\eta \) surface; and this value is zero.

In spite of the fact that the null mass flow is assured at the solid walls by the boundary condition on the curvilinear components, the cartesian velocity components are also required; they simply are \( u = v = w = 0 \).

At the outflow boundary, a zero gradient of the curvilinear components is specified from which the cartesian components are derived.

5(b) Pressure. The inspection of equation (6) reveals that the discrete form of the pressure correction equation will depend on the velocity components, including those next to the boundaries where the velocity is known (with the exception of the outlet). Consequently, the boundary conditions for the pressure correction equation are automatically incorporated through the right-hand-side term of equation (6) where the divergence of the velocity appears.

From this it follows that the pressure correction equation has no other boundary conditions than those applied on the velocity. However for the computation of the momentum equations, the pressure gradient has to be evaluated, and for the elements adjacent to the boundary a numerical boundary condition is needed.

At a solid wall a second order profile is fitted to the discrete points in order to obtain a pressure with a second order accuracy. This guarantees an approximation consistent over the entire computational domain. At the inflow boundary no condition is needed for the pressure because of the downwind scheme. At the outflow boundary with the assumption of a developed flow, a simple linear extrapolation is used, because the numerical error introduced by this calculation is not expected to propagate upstream. Once again, this estimation is carried out only for the momentum computation and not for the pressure correction equation.

6 Applications

6.1 Exponential Constriction. First, the behavior of the method was analyzed on a simple curvilinear geometry; the "hump test case" investigated by references [12, 13], who used a vector potential difference method. This geometry consists of a channel with an exponential constriction where the function \( y = 1 - 5e^{-2x} \) represents the lower surface for all depths; while \( y = 1 \) represents the flat upper surface. The mesh used was of \( 31 \times 11 \times 11 \) points as shown in Fig. 5. A developed profile specified as \( u = 36y(1-y)(1-z), v = 0, w = 0 \), set at the inlet completes the problem description.

Figure 6(a) shows the velocity field in the obstruction planes \( z = 0.1 \) and \( z = 0.5 \) for a flow Reynolds number of 80. This form of the recirculation zone, evidently due to the three-dimensional character of the flow agrees with the results obtained by reference [12]. This phenomenon can also be observed from the velocity vectors in \( \xi = \text{constant} \) sections, even though these sections are not strictly normal to the primary flow motion. This is presented in Fig. 6(b). When the flow reaches the obstruction, the bottom surface layer is forced towards the centerline. As it falls down the rear of the constriction, this layer is forced towards the centerline. Finally two symmetric vortices are developed.

[Fig. 5 3-D view of the duct mesh]
6.2 Circular Channel. A second calculation was carried out in a circular arc channel of square cross section with a Reynolds number of 80. For this case the Dean number defined as:

$$De = Re(H/R_m)^{0.5}$$  \hspace{1cm} (7)

is 50.9564, where $H = 1$ is the radial distance in the channel, and $R_m = 2.5$ is the channel mean radius of curvature. Upstream and downstream lengths of a straight channel of 1.254H and 3.1H, respectively, are attached to the curved duct. The turning angle of the elbow is 90 degrees. In the streamwise
direction 33 stations were used, while 17 × 13 points were used for the cross section. As in the previous case a parabolic velocity profile with no transverse component was set at the inlet.

The distribution of the pressure on the bounding surfaces is shown in Fig. 7(a) by contours of constant values of the pressure. The importance of the viscous influence can be appreciated if one compares this result with the potential pressure solution obtained by Yang [13], Fig. 7(b).

The development of secondary flow is illustrated in Fig. 7(c). Low momentum fluid is drawn from the side wall and convected downstream towards the suction surface and high streamwise velocities near the centerline are displaced ac-
cordingly toward the pressure surface. This helical motion is observed from the beginning of the turning of the channel and increases as the flow progresses in the duct. At the exit the secondary flow is not as strong but still does not disappear entirely because the downstream extension is not sufficiently long to allow a redevelopment of the flow.

To assess the present solution, a comparison of the computed fully developed streamwise velocity profile with the available experimental and numerical data has been carried out. This is illustrated on Fig. 7(d) which shows a good qualitative and quantitative agreement between the present calculation and the numerical predictions obtained by references [14, 15]. The present numerical solution does not agree well with the experimental measurements obtained by Mori et al. [16], but this is also the case of other computer results using totally different formulations [14, 15].

6.3 Pipe Bend. In this test the flow in a pipe described in reference [17] was computed and studied. It consists of a 90 degrees bend with a mean radius of curvature of 3.2 times the diameter of the pipe, and where inlet and outlet extensions of 2 and 3.2 times the diameter respectively complete the geometry. The grid chosen for this case is 33 points in the $\xi$ direction, 17 in the $\eta$ direction and 13 in the $\zeta$ direction. The inlet flow was fully developed, and the flow Reynolds number is 1093.

In Fig. 8(a) the development of the flow in the streamwise direction is shown. Figure 8(b) depicts the secondary flow at the locations 30 deg, and 60 deg, and at one diameter length downstream from the end of the pipe turning. From these representations it can be seen that the center of the symmetric pair of vortices moves from the inside to the outside of the pipe as the turning angle progresses. Finally the vortex motion weakens as it leaves the bend. This behaviour is as expected and confirm the results obtained by reference [18].

All Figs. 8(c), 8(d), and 8(e), the calculated axial velocity contours at the 30 deg and 60 deg planes and at a distance equal the pipe diameter from the exit bend, are displayed and are compared to the measurements of reference [17]. From these figures we can conclude that the agreement of the computed results with the experimental data is generally satisfactory.

6.4 Twisted Elbow. In order to illustrate the general prediction capability of the present model, a final numerical application was carried out on a fully three-dimensional geometry.

The channel chosen is shown in Fig. 9(a) and its geometric characteristics were devised by Yang [14]. The cross section is a square, upstream and downstream tangents of 0.524H and 2.1H (H being the radial distance), respectively, are added to the elbow section. This section has a 60 degrees turning angle
together with a 60 degrees twist around its central line, so the three-dimensionality is fully present.

The discretization was carried out using \(31 \times 11 \times 11\) mesh points, a flow Reynolds number of 80, and as in the previous numerical experiments a parabolic profile with no transverse components was set at the inlet.

Figure 9(b) shows the pressure plotted in contours of constant values viewed from opposite directions.

Probably the most interesting phenomena in such a complex geometry is the development of the secondary flow, which is presented in Fig. 9(c). It consists of the generation of a vortex pair that remains normal to the plane of the duct turning. The twisting seems to have no effect on the location of the vortices; however it does increase the strength of one side of the vortex pair, while decreasing the other. This influence becomes more evident after \(\theta = 40\) deg (Fig. 9(c)). After the channel stops twisting and turning at \(\theta = 60\) deg, the flow begins to recover, but as the length of the downstream tangent is relatively short for the present Reynolds number, it still cannot reach the straight channel flow type.

7 Concluding Remarks

The preliminary goal of the present work was the development of a numerical procedure to solve 3-D incompressible laminar flows in general curved passages. The reported results are in general good agreement with the available data. The proposed method predicts the complex nature of the three-dimensional viscous flow phenomena inside ductings and in particular the helical shape and changing strength of the secondary flow. A typical computation for the tested geometries requires about 0.4 seconds of CPU time per grid point on an IBM 3441-II.

References


