CALCULATION OF THREE-DIMENSIONAL LAMINAR FLOWS IN T-SHAPED JUNCTIONS

A. SAMAGAIO* and N.S. VLACHOS
Department of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign, Urbana, IL 61801, U.S.A.

A numerical study of three-dimensional laminar flow in rectangular bifurcating ducts with the branching flow at 90° to the main flow was performed. The full set of Navier–Stokes equations governing the flow was solved with an existing numerical iterative procedure appropriate for recirculating flows. The method uses primitive variables such as velocity and pressure. The results show that two recirculation zones are established, one near the bottom of the main duct opposite the bifurcation and the other in the branching duct close to the upstream side. One of the cases studied reproduces the conditions of existing two-dimensional measurements with good agreement. Pairs of streamwise vortices are generated downstream of the junction zone with their centers moving towards the symmetry plane. It is shown that strong three-dimensional flow is produced in the recirculation regions of the T-shaped junction, especially at low aspect ratios.

Nomenclature

\[ H, L, W \text{ duct dimensions (Fig. 1)}, \quad w_{in} \text{ average inlet velocity}, \]
\[ p \text{ pressure}, \quad w_{in,\text{max}} \text{ maximum inlet velocity}, \]
\[ Q \text{ volume flow rate, (3)}, \quad x, y, z \text{ coordinates}, \]
\[ \text{Re } \rho w_{in,\text{max}}H/\mu = \text{Reynolds number}, \quad x_i \text{ coordinates in tensor notation}, \]
\[ u, v, w \text{ velocity component in } x-, y-, z- \text{ direction}, \quad \beta \text{ mass rate in branch/total mass flow rate ratio}, \]
\[ u_i \text{ velocity component in } x_i- \text{ direction}, \quad \mu \text{ dynamic viscosity}, \]
\[ \rho \text{ density}. \]

1. Introduction

The flow of fluids in bifurcating ducts is encountered in many engineering applications, such as fluid machinery, heat exchangers, heating and ventilating systems, chemical processing plants, and the human circulatory system. While this geometry is apparently simple, the flows generated by it are often complex. Depending on the Reynolds number, Re, and the dividing

* Present address: Universidade de Aveiro, Departamento de Ambiente, Campo Universitário de Santiago, P-3800 Aveiro, Portugal.

mass flow rate ratio, $\beta$, a number of recirculation zones (flow reversal regions) and secondary flows (fluid motion on planes perpendicular to the streamwise direction) may be found.

A literature survey showed that a set of problems related to the one under consideration has in recent years deserved the attention of the scientific community. This interest has certainly been stimulated by the advent of modern tools for the numerical simulation of complex flows (e.g., fast digital computers with large amounts of central memory) and new measurement techniques (e.g., hot-wire and laser Doppler anemometry) upon which most of the following investigations rely. Although laminar flows are not often encountered in industrial applications, they allow an assessment of the predictability of numerical procedures to be carried out, since the constitutive laws are known. For example, Liepsch et al. [1] proved that good agreement between laser Doppler anemometry measurements and numerical predictions could be obtained using a finite-difference method to compute laminar flows in a two-dimensional T-shaped junction.

Kawaguti and Hamano [2] performed a numerical study of two-dimensional steady laminar flow in a channel with a variable angle branch. Kawaguti and Hamano [3] extended their predictions to the case of pulsatile flow in a $90^\circ$ bifurcation. The numerical study of two-dimensional pulsatile laminar flow in an orthogonal branch was reported by O'Brien and Ehrlich [4]. Kawashima et al. [5] obtained experimental data for two-dimensional steady flow in a right-angle T-shaped confluence. Kawashima et al. [6] extended their study to the heat transfer characteristics of the same flow pattern. A numerical prediction of two-dimensional steady turbulent flow in a $90^\circ$ confluence was also reported by Kawashima et al. [7]. An experimental study of steady and pulsatile flow in a double branching arterial model was performed by Lutz et al. [8]. Fernandez et al. [9] numerically predicted the two-dimensional pulsatile laminar flow in a Y-shaped bifurcation. Experimental data on that geometry can be found in Bharadvaj et al. [10, 11]. Experimental results on two-phase flow in a T-junction was reported by Hong [12] and Azzopardi and Whalley [13].

Pollard and Spalding [14] predicted the three-dimensional turbulent flowfield in a pipe flow splitting T-junction. Three-dimensional computations of laminar flow in a T-junction were also reported by Pollard and Spalding [15]. Pollard [16] obtained numerical and experimental results for turbulent flow with wall heat/mass transfer in a similar geometry.

The three-dimensional flow configuration considered in the present investigation is shown schematically in Fig. 1. Liepsch et al. [1] studied this problem both experimentally and numerically for rectangular ducts with an aspect ratio ($W/H$, for the main duct, and $W/L$, for the branching duct) of $8:1$. The measurements presented were obtained using laser Doppler anemometry and the respective calculations assumed the flow to be two-dimensional. Good agreement between prediction and experiment was achieved but some discrepancies were observed in the recirculation zones. These discrepancies were attributed to numerical diffusion and three-dimensional effects.

The present computations seek to qualitatively study the aforementioned three-dimensional effects. Since the flow is separating in the region of the bifurcation, the governing equations are elliptic and the calculation method iterative in nature. This method is based on the numerical solution of a finite-difference representation of the steady three-dimensional constant-property Navier–Stokes equations in primitive variable form. The equations, boundary conditions, and numerical method are described in the following section.
2. Mathematical model

The geometry under consideration and the corresponding solution domain are shown in Figs. 1 and 2, respectively. One can intuitively expect to find recirculation zones within the solution domain and, therefore, the equations governing the steady three-dimensional incom-
pressible Newtonian recirculating flow of interest to the present study are given in tensor notation as follows:

**Mass conservation equation**

\[ \frac{\partial u_i}{\partial x_i} = 0 . \]  

(1)

**Momentum conservation equation**

\[ u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( u \frac{\partial u_i}{\partial x_j} \right), \]  

(2)

where Einstein’s summation convention holds.

Since the above equations are elliptic, one must specify boundary conditions on all sides of the solution domain. At the inlet plane, which is located far upstream of the zone of influence of the bifurcation, a fully developed profile of the axial (z-component) velocity was imposed by the function (see, e.g., [17]):

\[ w(x, y) = \left( \frac{Q}{8 \frac{ab^3}{3} - \frac{16}{b} \sum_{n=0}^{\infty} \lambda_n^{-5} \tanh \lambda_n a} \right) \times \left( b^2 - (y - b)^2 + \frac{4}{b} \sum_{n=0}^{\infty} \frac{(-1)^{n+1}}{\lambda_n^3} \frac{\cosh \lambda_n(x - a)}{\cosh \lambda_n a} \cos \lambda_n(y - b) \right), \]  

(3)

where

\[ \lambda_n = \frac{(2n + 1)\pi}{2b}, \quad a = \frac{W}{2} \quad \text{and} \quad b = \frac{H}{2}. \]

In the above equation, \( Q \) denotes the inlet volume flow rate.

The assumption of fully developed flow at the inlet implies that the \( x \)- and \( y \)-components of the inlet velocity is zero. At the outlets, all variables were given zero normal gradients (von Neumann boundary conditions), except the tangential components of the velocity which were set to zero. A no-slip condition was imposed at all solid surfaces. A symmetry condition was also presumed across the mid-plane of the model duct (see Fig. 2).

The numerical solution of the steady two-dimensional form of the Navier–Stokes equations using pressure and velocities as dependent variables was described by Gosman and Pun [18]. Practical applications of the procedure were reported by various authors, e.g., Durst and Rastogi [19] and Vlachos and Whitelaw [20]. An extension of the present numerical method to three-dimensional laminar flows was reported by Gosman et al. [21].

Figure 2 schematically shows the grid, the cell arrangement and the location of the fluid properties and dependent variables. Pressure and fluid properties, such as density, viscosity, etc., are stored on the grid nodes. The velocity components, \( u, v \) and \( w \), are stored midway between the grid nodes. The grid is evenly spaced in the \( x \)-direction and unevenly spaced in the \( y \)- and \( z \)-directions to accommodate steep gradients.
The finite-difference forms of the differential equations are derived employing a semi-integral approach, which assumes a finite control volume surrounding each grid node. For each variable, the differentials are calculated by taking differences over the surrounding grid intervals. The variation of properties is assumed to be linear between successive locations. However, a hybrid numerical scheme is utilized which combines the central difference and upwind difference schemes, according to Spalding [22], in order to account for high convection and to secure stability of the solution procedure. The solution method is based on the SIMPLE algorithm of Patankar and Spalding [23]. The approximate solution is considered acceptable when the continuity and momentum equations are satisfied, i.e., when the normalized residuals of each equation summed over the whole calculation domain are smaller than 0.005.

3. Results and discussion

The computations presented herein comprise two cases. The first case reproduces the conditions of the experimental investigation of Liepsch et al. [1] (Re = 496, \( \beta = 0.44 \), \( W/H = 8.0 \), \( W/L = 8.0 \)). The second case, for which no experimental data could be found in the open literature, is similar to the first case, the sole difference being the low aspect ratio of the ducts which form the T-junction (\( W/H = 1.0 \), \( W/L = 1.0 \)). The latter situation is more commonly found in practical applications than the former. These cases will hereafter be designated Case 1 and Case 2, respectively.

A grid-independence study was conducted for Case 1 in order to choose the size of the grid to be used in the final computations. Three different grids with \( 12 \times 17 \times 21 \), \( 12 \times 21 \times 21 \), and \( 17 \times 25 \times 25 \) nodes were used for that purpose. It should be pointed out that further refinement of the grid size was made impossible, due to the limited central memory of the CDC CYBER 175 computer available for the calculations. Figure 3 shows the \( w \)-velocity profiles obtained at a selected location within the recirculation zone for the various grids. The results are qualitatively consistent, although clearly inadequate to declare the computations independent of the grid size. Nevertheless, a comparison of the different profiles with the available experimental data (see Fig. 4) demonstrates that the agreement between the prediction and the measurements improves with the refinement of the mesh size. The hypothesis of an asymptotic behavior is, therefore, strongly supported. However, given the above mentioned constrains, in terms of computer storage, the grid with \( 17 \times 25 \times 25 \) (=10,625) nodes had to be accepted for the present study. It should also be pointed out that the symmetry condition yields an effective grid with \( 32 \times 25 \times 25 \) nodes for the whole T-junction. Finally, the time spent on the CDC CYBER 175 machine to compute Case 1 and Case 2, using a grid with \( 17 \times 25 \times 25 \) nodes, was equal to 1000 and 3300 s, respectively. The strong three-dimensional effects produced by the side walls were chiefly responsible for the time increase in Case 2.

The experimental values obtained by Liepsch et al. [1] of \( w \)-velocities in the main duct and \( v \)-velocities in the branching duct, respectively, are shown in Fig. 4. The corresponding numerical predictions are also plotted at the same locations. Some of the numerical results were linearly interpolated, since the position of the grid lines did not always coincide with that of the experimental stations. In general, the agreement between computed and measured
values is good, although some discrepancies can be noticed in the regions of recirculation. In these regions, the mathematical model performs poorly in the prediction of sharp gradients present in the actual flow. At the outlet of both ducts, the agreement is very good, thus proving that the discretization process is quite adequate in those regions where the flow is of parabolic nature.

Figure 5 shows the projection of the velocity vectors on selected planes for Case 1. Only the region of the computational domain surrounding the bifurcation zone is presented. Figure 5b confirms the findings of Liepsch et al. [1], i.e., two recirculation zones are established on the symmetry plane, one at the bottom of the main duct opposite the trailing edge of the bifurcation and the other in the branching duct, near the upstream side. It can also be observed that the velocities in the upper part of the main duct are larger than near the bottom wall, as a consequence of the mass transfer into the branching duct. Downstream of the reattachment point, the flow becomes parabolic and slowly regains characteristics of symmetry. The velocity vectors in the branch indicate the existence of an important zone of backflow in the region of the leading edge of the bifurcation. After leaving this region of negative velocities, the flow evolves into the typical shape of developed laminar flow in rectangular ducts, as in the main duct. Figure 5a, in conjunction with Figs. 5c–f, clearly shows the existence of important three-dimensional effects. The recirculation zones found on the symmetry plane are seen to extend across the bifurcation, in the x-direction, in spite of an increase in the complexity of the flow field close to the walls parallel to the symmetry plane. Pairs of steamwise vortices are generated in the main and branching ducts of the bifurcation. These vortices are diffused by the viscous action and their centers move towards the symmetry plane further downstream. The velocity vector field computed for Case 2 (not presented here
Fig. 4. Comparison of predicted and measured velocity profiles (Case 1): (a) \( w \)-velocity in main duct; (b) \( v \)-velocity in branching duct. —— exp., Liepsch et al. [1]; O num., 17 \( x \) 25 \( x \) 25 grid.
Fig. 5. Velocity vector field (Case 1): (a) on plane $x/H = 0.40$; (b) on plane $x/H = 4.00$; (c) on plane $y/H = 0.96$; (d) on plane $y/H = 1.95$; (e) on plane $z/H = 0.04$; (f) on plane $z/H = 1.88$. 
Fig. 6. $w$-velocity isotachs (Case 1): (a) $w(x/H, y/H, 0.0)/\bar{w}_1$; (b) $w(x/H, y/H, 1.0)/\bar{w}_1$; (c) $w(x/H, y/H, 3.3)/\bar{w}_1$. 
Fig. 7. $w$-velocity isolochs (Case 2): (a) $w(x/H, y/H, 0.0)/\tilde{w}_\text{in}$; (b) $w(x/H, y/H, 1.0)/\tilde{w}_\text{in}$; (c) $w(x/H, y/H, 3.3)/\tilde{w}_\text{in}$. 
Fig. 8. $v$-velocity isolachs (Case 1): (a) $v(x/H, 1.0, z/H)/\bar{v}_\text{in}$; (b) $v(x/H, 2.9, z/H)/\bar{v}_\text{in}$; (c) $v(x/H, 7.1, z/H)/\bar{v}_\text{in}$.
Fig. 9. $v$-velocity isolachs (Case 2): (a) $v(x/H, 1.0, z/H)/\bar{w}_i$; (b) $v(x/H, 2.9, z/H)/\bar{w}_i$; (c) $v(x/H, 7.1, z/H)/\bar{w}_i$. 

A. Samagaio, N.S. Vlachos, Three-dimensional laminar flows in T-shaped junctions
for reasons of space) show that, in general, the zones of recirculation identified in the study of Case 1 are also discernible. However, the two pairs of streamwise vortices are larger and the recirculation zones are substantially reduced in size.

Figure 6 shows the w-velocity isotachs at selected locations of the main duct for Case 1. The two-dimensionality of the flow field, i.e., the fulfillment of the requirement $\partial w/\partial x = 0$, in this case, is seen to be roughly limited to the central half of the duct cross-section. At the plane corresponding to the leading edge of the bifurcation (see Fig. 6a) the largest values of the velocity in the two-dimensional zone are found about the location $y/H = 0.6$, thus giving further evidence that the flow is being accelerated to the upside, as a consequence of its downstream splitting. A small region of negative velocities, located at the lower left corner of the duct, can also be detected. Further downstream, at the plane corresponding to the trailing edge of the bifurcation (see Fig. 6b) this region grows in size and strength. Negative velocities, as small as 20% of the average inlet velocity, may be found in this area. A smaller recirculation zone may also be noticed at the upper left corner of the duct. Figure 6c shows the flow field downstream of the junction. At plane $z/H = 3.3$ a large region of very small negative velocities is present in the lower left corner of the duct.

The w-velocity isotachs for Case 2, calculated at the locations chosen for Case 1, are shown in Fig. 7. The flow field is, at all planes considered, fully three-dimensional. Nevertheless, the recirculation zones are only observable below the trailing edge of the bifurcation (see Fig. 7b). At this higher position, and in contrast with what was found for Case 1, the larger negative velocities are located on the upper left corner of the duct. Close to the bottom wall, the cross-section of a large recirculation 'bubble' is also visible. In general, the w-velocity contours on the remaining plane help demonstrate that the recirculation zone in the main duct is smaller for Case 2 than for Case 1.

Figure 8 displays the v-velocity isotachs at the inlet and two downstream locations of the branching duct for Case 1. At the inlet, the flow field is seen to be strongly accelerated next to the wall opposite the symmetry plane and towards the trailing wall of the branch. At plane $y/H = 2.9$ (see Fig. 8b) the flow is further accelerated towards the trailing wall and decelerated next to the opposite wall, giving rise to a region of backflow. Downstream, at plane $y/H = 7.1$, the recirculation zone disappears, but the flow is still highly asymmetric.

Results obtained at the same locations for Case 2 are shown in Fig. 9. The principal features of the u-velocity distribution in the branch, found for Case 1, are also observable. The u-velocity is seen to reach its maximum values close to the trailing wall of the branch and the side wall, at the inlet (see Fig. 9a). The locus of largest velocities is much closer to the symmetry plane at plane $z/H = 2.9$ (see Fig. 9b) and approaches the duct center line, further downstream, at plane $z/H = 7.1$ (see Fig. 9c). The v-velocity contours at this latter plane are clearly more symmetric than those presented for Case 1, thus showing that the flow becomes more rapidly parabolic in Case 2.

4. Conclusions

From the discussion of the previous section, the following general conclusions can be drawn:

(i) The present numerical procedure is capable of predicting the major features of
three-dimensional laminar flow in rectangular bifurcating ducts of different aspect ratios. The predictions are in good agreement with experimental data obtained using laser Doppler anemometry for the limiting case of a large aspect ratio.

(ii) Two separation zones are established, one at the bottom wall of the main duct and the other at the upstream side of the branching duct. These zones are reduced in size with a reduction of aspect ratio.

(iii) Two pairs of counter-rotating streamwise vortices are formed downstream of the bifurcation in the main and in the branching ducts. The centers of these vortices move towards the symmetry plane and are diffused further downstream as the flow redevelops. These vortices may result in an increased pressure loss and heat transfer in the bifurcation, compared to the predictions based on a two-dimensional flow.

(iv) The flow is strongly three-dimensional in the regions of separating flow.

Finally, it should be pointed out that further refinement of the grid may result in more accurate predictions of the flow field in the bifurcation.

References


