Vortical flow topology in a curved duct with 90° bend

Tony W. H. Sheu¹, S. F. Tsai ²

¹ Dept. of Engineering Science and Ocean Engineering, National Taiwan University,
No. 1, Sec. 4, Roosevelt Rd., Taipei, Taiwan 106, R.O.C.
² Dept. of Marine Engineering, National Taiwan Ocean University,
No. 2, Pei-Ning Rd., Keelung, Taiwan 202, R.O.C.

E-mail: twhsheu@ntu.edu.tw

Abstract: We performed incompressible flow simulation in a square duct featured with 90° bend and curvature radius of 2.3 to extend our understanding of the vertical flow development in the bend. The solutions for the flow investigated at Reynolds number of \( Re = 790 \) are obtained in a tri-quadratic element system, where velocities stagger the pressure working variable, using the streamline-upwind finite element model and the BiCGSTAB iterative solver. The simulated results reveal that centrifugal force convects the quickly moving fluid particles towards the outer wall. The axial velocity, as a result, shows twin peaks and secondary flow in the curved channel. At about \( \theta = 66° \), the secondary flow shows three complex pairs of vortices. Also noteworthy is the formation of a downstream spiralling flow motion. To better elucidate the dominating three-dimensional flow nature, the topological study of limiting streamlines was undertaken. Insight into the longitudinal flow instability is gained by tracking the formation and diminishing of limiting cycles.

Key-Words: incompressible; square duct; 90° bend; topological study; limiting streamlines

1 Introduction

Fluid flows in curved ducts can be found in pumps, aircraft intakes, river bends, and cooling coils in heat exchangers. Their practical importance has motivated considerable research effort in the past. In curved ducts, centrifugal and viscous (Tollmien-Schlichting) instabilities may exist and interact strongly [5]. The resulting nonlinear interaction between these instabilities may cause the flow to evolve to exhibit turbulence at a higher Reynolds number. Advancing of knowledge about this three-dimensional curved flow is, thus, of fundamental importance.

One of the main flow characteristics in curved pipes or channels is the secondary flow formation, which was pointed out firstly by Eustice [11,12] in curved pipes and has an inegligible influence on the primary flow development in regions of prevailing centrifugal force. Later on, this experimentally observed phenomenon was analytically confirmed by Dean [9,10]. In the presence of secondary flow, the velocity may be skewed towards the outer wall. Therefore, the hydrodynamic performance, viscous power loss and heat transfer can be affected significantly. In addition, the large pressure drop, enhanced mixing, and non-uniform wall shear stress are major signatures of this type of flow. These make the curved channel flows differ significantly from that seen in the straight channels.

Since 1927, studies of curved flows in rectangular or circular channels have been numerous owing to its important role played in fluid mechanics. One way to obtain the secondary flow insights that are experimentally impossible is to exploit computational fluid dynamics technique. This problem is particularly amenable to numerical simulation due to the relative ease of mesh generation. Besides the excellent paper reviewed by Berger, Talbot and Yao [6], we shall only cite few of the representative articles such as those by Patankar, Pratap & Spalding [22], Humphrey [15], Soh & Berger [25], and Humphrey, Taylor & Whitelaw [16] because space does not permit a full list of them. Studies of flow development in curved ducts with square cross-sections have been addressed on the determination of critical Dean number, above which the formation-and-disintegration of secondary flow can lead to multiple-vortex-pair solutions [4,26]. The effect of the channel curvature on the secondary flow development has also been the subject of considerable interest, in addition to the study of bifurcation phenomenon due to centrifugal instability [10,31].

One way of exploring vortical flow in the currently investigated curved channel is to extract the physically meaningful insight from the simulated velocity vector field. In this study, we employ the topological theory to exhibit three-dimensional flow separation and reattachment. By virtue of the
simulated limiting streamlines [19] (or wall streamlines) or skin-friction lines [21], the kinematics of the three-dimensional flow motion can be revealed. We also plot the density of helicity or its normalized value [20] to reveal the curved duct flow.

The rest of this paper is organized as follows. In the next section, we solve the Navier-Stokes equations, subject to the divergence-free constraint condition and the well-posed boundary conditions, by using the in-house developed tri-quadratic Petrov-Galerkin finite element model [24]. The BiCGSTAB iterative solver [28] is used to overcome difficulties in association with the matrix indefiniteness and asymmetry. This followed by describing the problem in Section 3 and the numerical results in Section 4. The evolving counter-rotating vortices are addressed in the result section. Finally, we draw some conclusions in Section 5.

2 Streamline upwind tri-quadratic finite element Model

In this paper, the Navier-Stokes equations for incompressible fluid flows in the curved three dimensional channels are solved under the laminar assumption. In dimensionless form, the conservation equations considered at steady state are expressed in terms of the velocity vector \( \mathbf{u} \) and the pressure \( p \):

\[
\frac{\partial \mathbf{u}}{\partial t} = 0
\]

\[
\frac{\partial}{\partial x_m}(u_m u_n) = -\frac{\partial p}{\partial x_m} + \frac{1}{Re} \frac{\partial^2 u_n}{\partial x_n^2} \quad (1)
\]

In the above, \( Re \) is denoted as the Reynolds number.

\[
\frac{\partial^2 u_n}{\partial x_n^2} \left( u_n u_n \right) = \frac{\partial p}{\partial x_n} + \frac{1}{Re} \frac{\partial^2 u_n}{\partial x_n^2} \quad (2)
\]

Given that \( \omega \in H^1(\Omega) \times H^1(\Omega) \) and \( q \in L^2(\Omega) \) in a simply-connected domain \( \Omega \), the weak solutions for equations (1-2) are solved subject to \( u = \mathbf{g} \) on from the following equations

\[
\int_\Omega (u \cdot V) u - w \, d\Omega + \int_\Omega V_u : \nabla w \, d\Omega - \int_\Gamma p \, \nabla \cdot w \, d\Gamma = 0
\]

\[
\int_\Omega (V \cdot u) q \, d\Omega = 0
\]

In equation (3), \( \Gamma_i/\Gamma_n \) represents the complement of \( \Gamma_n \) on \( \Gamma = \partial \Omega \). If \( \phi \in \Gamma/\Gamma_i \) (i.e., \( n = \text{r} \)), it by definition, satisfies \( \phi \in \Gamma \) but \( \phi \notin \Gamma_i \). The unit vectors \( \mathbf{n} \) and \( \mathbf{s} \) are normal and tangent to \( \Gamma \), respectively. The above weak formulation is closed by prescribing the following natural-type boundary conditions

\[
-p + \frac{1}{Re} \nabla \cdot u = r \quad \text{on} \ \Gamma/\Gamma_n \quad (5)
\]

\[
\frac{1}{Re} \nabla \cdot u \times \mathbf{n} = 0 \quad \text{on} \ \Gamma/\Gamma_n \quad (6)
\]

The appropriate finite element spaces for the closed primitive variables are the key to success in obtaining the convergent solutions from the mixed analysis of incompressible viscous fluid flows. For stability reasons, the shape functions chosen for \( \mathbf{u} \) and \( p \) should satisfy the LBB (or inf-sup) condition [3,7]. For this reason, we employ the tri-quadratic polynomials \( N_i \) (i = 1 ~ 27) for \( \mathbf{u} \) and use the tri-linear polynomials \( M_i \) (i = 1 ~ 9) for \( p \). Another key in simulating the high Reynolds number flow is to properly select the finite element test spaces so as to enhance convective stability. The test function is obtained by adding a biased polynomial to the shape function. While the convective stability is enhanced, the resulting Petrov-Galerkin model may suffer from false diffusion errors in the multi-dimensional flow simulation. To avoid accuracy deterioration without sacrificing nonlinear stability, it is essential to employ weighting functions which can yield an artificial diffusivity along the streamline direction only [14]. Following this thought, \( \tau V = N_i \bar{\mathbf{U}} \cdot \nabla N_i \), where the upwind coefficient \( \tau \) is expressed as

\[
\tau = \frac{\alpha_i V_i h_i + \alpha_j V_j h_j + \alpha_k V_k h_k}{2V_i V_j V_k} \quad (7)
\]

In the above, \( V_i = \bar{\mathbf{u}} \cdot \nabla N_i \) is added to the shape function \( N_i \) to render the weighting function \( W_i = N_i + \tau \mathbf{u} \cdot \nabla N_i \), where the weighting function \( \omega = W_i \) in the one-dimensional quadratic element [24], where \( \delta \) is expressed in terms of \( \gamma = V_i h_i / 2 \nu \):

\[
\delta(\gamma) = \begin{cases} 
\frac{1}{2} & \text{at center-nodes} \\
\frac{1}{2} \left( \frac{\cosh(\gamma) - 1}{\gamma} \right) & \text{at end-nodes}
\end{cases}
\]

The consequence of the resulting indefinite and asymmetric finite element matrix equations is the poor eigenvalue distribution. One can employ a Gaussian elimination direct solver to solve for the matrix equation. The storage requirement is, however, prohibitive for very large-size problems. For this reason, the Lanczos-type Bi-conjugate gradient stabilized (BiCGSTAB) [28] iterative method will be employed in the present three-dimensional curved flow analysis. This approach is effective because it can locally minimize the residual through the
generalized minimized residuals GMRES [23] and can avoid matrix transpose calculations. Nevertheless, the BiCGSTAB iterative solver still suffers both from the pivoting and Lanczos breakdowns. In order to reduce the matrix bandwidth, the matrix equations were element-by-element compressed [29].

3 Problem description and code validation

In this study, we simulated the duct flow which was studied experimentally by [16]. The duct schematic in Fig. 1 has a 90° bend with the mean radius of \( R_c = 2.3 \). The non-zero dimensionless length \( \delta (= r_v/\overline{r}_v = 1.586) \) can affect the balance of inertia, viscous, and centrifugal forces and, therefore, has an essential role in the flow development. The elbow of 1×1 square cross-section has the downstream straight extension with a length fourteen times of the hydraulic diameter \( D (=1) \). It is, thus, rational to specify a fully-developed flow at the truncated outlet plane EFGH. Since the deflected inlet vorticity may generate a strong transverse pressure gradient and, in turn, can influence the streamwise pressure gradient, a straight channel of length 2 is attached upstream to the elbow. With \( U \) as the characteristic velocity (inflow velocity), \( D \) the characteristic length (duct hydraulic diameter) and \( \mu \) the fluid viscosity, the channel flow will be investigated at \( Re = 790 \). In this benchmark exercise, the experimental condition of [16] was simulated. The fluid kinematic viscosity was \( \nu = \frac{UD}{Re} \), with a bulk velocity magnitude of \( U=1 \). The corresponding Dean number \( D_e (Re^{4/5}/R_c^{3/5}) \), which is another key parameter in the development of secondary flow, was 368.

We performed here the steady-state flow analysis in the full channel to model the Coanda effect [30]. The present calculations were performed in a non-uniformly discretized domain, with 201 grid points distributed in the streamwise direction and 41×41 grid points at each (y,z) cross-section. The simulated solutions at 337,881 nodal points has been shown to be capable of resolving the secondary flow according to the grid-independence test.

To confirm the validity of the present finite element calculation, we have compared the simulated solutions with both the experimental [16] and numerical [27] data for the streamwise velocity \( u_x(r) \) obtained in the curved sections at two planes \( y = 0.25 \) and 0.5. In Fig. 2 good agreement is seen to achieve except at planes near \( \theta = 60^\circ \). The reason for the simulated discrepancy between the experimental and numerical results remains unclear. Comparison of the axial velocities in the radial direction was also carried out at different angles \( \theta \). As Fig. 3 shows, the agreement between the numerical solutions is better than that between the numerical and experimental data except at \( \theta = 60^\circ \).

4 Numerical results

Near the inner bend, the axial flow in Fig. 2 undergoes a rapid decrease in velocity and is seen to form a step-like profile. Farther downstream, the continuously eroded step-like axial velocity forms a new local maximum near the inner bend. A deep valley is, thus, seen to show its presence in between the two local peaks. This valley remains deep up to the plane near \( \theta = 90^\circ \). Afterwards, a continuous flatting is seen in the direction towards the exit plane. The axial velocity profile manifested by the local maxima and the valley present in between was experimentally confirmed by Humphrey, Taylor & Whitelaw [16] in curved channels. The step-like axial velocity profile near the inner bend was also experimentally reported by [1] and numerically simulated by Soh & Berger [25] for a circular pipe.

4.1 Curved flow feature and topology

The entry flow plotted in Fig. 4 is characterized as a boundary layer type, with the peak axial velocity found at the plane of symmetry. Upstream of the curved duct, the flow is accelerated in regions near the inner-radius wall due to the favorable longitudinal pressure gradient. Conversely, the decelerated flow is observed in regions near the outer-radius wall because of the developed adverse pressure gradient downstream of \( \theta = 0^\circ \). The peak axial velocity is shifted towards the outer wall owing to the centrifugal effect. The degree of velocity skewness increases with the increasing turning angle. Such a skewed axial velocity can persist far downstream. The outer bend is referred to as the pressure side and the inner bend as the suction side. The evidence is given in Fig. 5, which plots the pressure coefficient \( C_p \) along the inner- and outer-radius walls in the vicinity of the symmetry plane. The difference in \( C_p \) at the inner and outer walls gradually diminishes at a location downstream of the 90° plane. The pressure obtained at the inner bend is seen to decrease gradually and monotonically with \( \theta \). The dramatic pressure rise along the outer bend results in the aforementioned adverse pressure gradient in the region close to the entry plane. Such an adverse pressure gradient developed along the outer bend may cause the streamwise separation to occur. The longitudinal recirculation, while being weak in vortex strength and small in eddy size, has been observed experimentally [16].
In Fig. 6, we plot $p(r)$ at $y = 0$, $y = 0.25$ and $y = 0.5$ planes to show the presence of inwardly directed radial pressure gradient in the bend. The force imbalance between this inwardly directed radial pressure gradient and the centrifugal force causes the secondary flow to form at the cross-section planes. The fluid in the core region is seen to move towards the outer wall and return to the inner wall along the channel wall, thereby resulting in a vortex. In the present full channel flow simulation, the simulated secondary flow has two counter-rotating vortices at the channel cross-section for $0^\circ \leq \theta \leq 40^\circ$. Two vortices are symmetric with respect to the plane of symmetry. The axial flow superimposed over the secondary flow makes the flow pattern a substantially three-dimensional type in the curved section, as schematically shown in Figs. 7-8. To assert that flow separation from the outer wall indeed exists, we plot the three-dimensional separation regions in Fig. 7. Immediately adjacent to the two end walls are the regions of streamwise flow separation. Along the direction towards the inner wall, the separated flow increasingly reduces its size and finally shows no separation near the plane of symmetry. The presence of outer-bend separated flows near the two end walls indicates the necessity of conducting the current three-dimensional flow analysis in the curved duct.

Figure 9 plots the streamlines at several chosen cross-sections. The number of vortex pairs is seen to increase from one to two and then three. Each vortex direction is opposed to its adjacent one. This physically rational bifurcation phenomenon was firstly predicted by Akiyama [2] and then experimentally confirmed by Joseph, Smith & Adler [18] in the curved channels as the Dean number is larger than $100$ [17]. According to Ghia & Sokhey [13], the increased vortex pairs at the cross-flow planes are attributable to the centrifugal instability. As Fig. 9 shows, weak vortex is present in the corner of the inner and end walls. The primary counter-rotating vortex is gradually distorted with the vanishing corner vortex near the end wall. This is accompanied by another vortex formed at the inner bend near the plane of symmetry. Farther downstream, this vortex is strengthened by the primary vortex that has been increasingly distorted and elongated. At about $\theta = 66^\circ$, the primary vortex becomes distorted and it can be divided into two large corotating vortices. In between the two vortices of different rotation signs, there shows a topological saddle point. The tendency of dividing the primary vortex into two vortices is deemed to be responsible for the step-like axial velocity profile [26]. The increased number of vortex pairs is accompanied by the decreased vorticity strength. This finding is logical since the flow will be fully-developed again in the approach to the exit plane.

Due to the formation of secondary flow and the presence of skewed axial velocity, the wall shear is varied circumferentially, with the maximum and minimum helicity magnitudes found at the outer and the inner walls of the bend, respectively. To confirm this, we plot in Fig. 10 the wall shear stresses at the inner and outer walls. It is found that the simulated shear stress contours at the outer bend are far more complicated than those found at the inner bend, with the observed oscillatory shear stress. An explanation for such stress pattern found near the outer bend is due to the fluid flowing over the concave wall. This tends to destabilize the curved flow. The degree of instability increases with the increasing Reynolds number. Under the circumstances, the Taylor-Görtler vortices may be developed in the flow.

Amongst the vector fields that can be chosen in the topological study of three-dimensional flow field, we employed the limiting streamlines, which are known to be the streamlines located immediately above the channel wall [19]. By plotting the topologically singular points, the simulated nodes, foci, and saddles on each curved channel wall in Fig. 9 can help us to reveal the kinematic nature of the flow based on the topological rule of Davey [8] and Lighthill [21]. In Fig. 11, it is seen that the simulated limiting streamlines are directed either towards or away from the topological singular point. The lines of separation are found to originate from the saddle point and terminate either at a spiral node in the flow interior or at the half-node at the intersection line of two adjacent walls. Unlike the lines of separation, the wall-streamlines adjacent to the line of reattachment repel from this singular line. We also plot in Fig. 11 the reattachment lines, which the emanated from the node and terminated at the saddle. The simulated reattachment lines have two nodes, in between of them there is a half-node located exactly at the intersection of two planes.

4.2 Vortex stability

Although the investigated problem is simple in geometry, the flow is much complicated than our expectation. Especially noteworthy is the formation of spiraling flow. To get additional insight into the vortical flow development in the curved duct, we plot the vortical coreline, which is regarded as the global signature of the vortical flow. By definition, all three-dimensional foci span the vortical coreline. The velocity components orthogonal to the vertical coreline are zero at the spiral focal point. By virtue of
this definition, we can plot the vortical coreline in Fig. 12. Hereafter, we refer to \( u_\ast \) as the velocity component tangential to the vortical coreline. In light of the non-zero \( u_\ast \), we were led to realize that the fluid particles near the vortical coreline will continue their spiral journey downwards. To confirm this, the massless markers were seeded near each vertical end wall so as to enable us to trace their trajectories.

In this paper, we also plot in Fig. 13 the value of \( \lambda = (\hat{\partial}u_\ast /3s) \), where \( \hat{\cdot} \) denotes the unit tangent vector to the vortical coreline. Since \( u_\ast \) is zero along the vortical coreline passing through "o", schematic in Fig. 14, the equation of motion valid along this line can be simplified as

\[
\Delta = 1/\mu u_\ast |p/3s| - \mu (\hat{\partial}^2 u_\ast /\partial^2 + \hat{\partial}^2 u_\ast /\partial^2) + \hat{\partial}^2 u_\ast /\partial^2]
\]

where \( \Delta \) represents the divergence of the simulated velocity components. In the case of high Reynolds number (or in the case of negligibly small \( \mu \)), the sign of \( \Delta \) will be varied depending on the value of \( u_\ast \) and on the pressure gradient \( \hat{\partial}p/3s \) evaluated at the nodal point "o". Simple algebra shows that if \( u_\ast > 0 \) and \( \hat{\partial}p/3s > 0 \), then \( \Delta < 0 \) (or \( \lambda > 0 \)), thereby resulting in an accelerating flow (Fig. 13). Otherwise, if \( u_\ast > 0 \) and \( \hat{\partial}p/3s > 0 \), then we have \( \Delta > 0 \) (or \( \lambda < 0 \)). Under the circumstance, the flow is of the deceleration nature (Fig. 13). For completeness, we also plot in Fig. 13 the normalized helicity \( \lambda = 0 \) along the vortical coreline for showing the intensity of the simulated spiralling flow motion.

The secondary flow inside the rectangular curved channel is deemed to be responsible for the change of sign in \( \lambda \). At the upstream side, \( \lambda \) is seen to have the negative value. The streamlines at the transverse planes located upstream of \( \lambda = 0 \) repel from the spiral node and the streamline flow is of the decelerated type. Immediately downstream of \( \lambda = 0 \) the simulated cross-flow starts to show two families of spiralling flow to respond the presence of positive value of \( \lambda \). The flow at the outer part spirals towards the vortical coreline and the flow at the inner part repels spirally from "o". Both of them, however, proceed downstream in a clockwise direction. This inward-and-outward motion results in a ring (or limiting cycle) in the sense that the fluid particles inside of the ring can not spiral towards the vortical core. On the other hand, fluid particles outside the ring can not spiral outwards.

In view of the simulated limiting cycles on the cross-flow planes, it is instructive to reveal the subtle changes in vortex motion in the vicinity of \( \lambda = 0 \). In Fig. 15, the inward spiralling flow region is seen to decrease in size. The flow, as a result, becomes unstable as it proceeds downwards. This is followed by the decreasing value of \( \lambda \). The presence of unstable limiting cycle indicates the possible onset of Hopf bifurcation. Under these circumstances, flow unsteadiness may become increasingly pronounced in the subsequent flow development. The limiting cycle is seen again as \( \lambda \) becomes a negative value. As the limiting cycles appear, circles which are stable or unstable alternate their presence. It is impossible to find two consecutive cross-flow planes on which the limiting cycles are either stable or unstable [32]. Therefore, a vortex can be destabilized under the conditions of \( \hat{\partial}u_\ast /3s < 0 \) and \( u_\ast \geq 0 \) along the vortical coreline.

5 Concluding remark

In this study, the three-dimensional steady-state Navier-Stokes equations, subject to the incompressibility constraint condition, are solved by employing the streamline upwind finite element model so as to enhance convective stability and to minimize the false diffusion error. To resolve asymmetry and indefiniteness problems in the large-size finite element matrix equations, we have applied the element-by-element BiCGSTAB iterative solver for improving the convergent performance. At a streamwise plane that is upstream of the bend with a length of 2, the location with the peak axial velocity at the plane of symmetry shifts towards the outer wall owing to the centrifugal force. The skewed axial flow rapidly intensifies up to about \( \theta = 30^\circ \). This progressively developing flow is accompanied by an accelerated flow in regions near the inner-radial wall. In the bend between the 0° and 40° streamwise planes, the fluid flows near the outer wall are seen to be greatly decelerated due to the adverse longitudinal pressure gradient. Conversely, the favorable pressure gradient observed at the suction side of the bend can cause the flow to accelerate. This flow acceleration can be further strengthened by the high-speed fluid transferred by the secondary flow motion proceeding from the duct center towards the outer-radius wall. Such a pronounced profile is clearly observed at the downstream station. The simulated secondary flow is characterized by having the fluid particles moving towards the side wall along the outer-radius (pressure) wall and the symmetry plane along the inner-radius (suction) wall. In the curved channel, the streamlines at cross-sections are seen to spiral towards and away from the vortical coreline as \( \lambda (= \hat{\partial}u_\ast /3s) \) changes sign. The limiting cycle develops as a consequence of the inward-and-outward particle motion. The seevolving limiting cycles have different sizes and can be classified as stable or unstable, depending on the sign of \( \hat{\partial}u_\ast /3s \). The simulated
limiting cycles are stable when $\lambda$ is evolved from the positive value to the negative value. In contrast to the stable limiting cycle, the cycle is unstable if $\lambda$ changes its sign from the negative to positive. Once the limiting cycles appear, circles which are stable or unstable alternately take their presence. When $\partial u_s/\partial s < 0$ and $u_s > 0$, it is highly possible that the vertical flow will lose its stability.

References:


Fig. 1. Schematic of the investigated curved duct

Fig. 2 Comparison of the simulated velocity profiles

Fig. 3. Comparison of the simulated $u_r(r, y)$ against $r$ with other two solutions obtained at different angles $\theta$ defined in Fig. 1. (a) $\theta=30^\circ$; (b) $\theta=60^\circ$; (c) $\theta=90^\circ$.

Fig. 4. The simulated velocity vectors at different $y$-planes.

Fig. 5. Comparison of the pressure distributions
for \( p(s, \theta) \) along the inner and outer walls at the plane of symmetry. Here, \( \theta \) is defined in figure 1.

Fig. 6. The simulated pressure contours at three chosen \( y \)-planes. (a) \( y=0.5 \); (b) \( y=0.25 \); (c) \( y=0 \).

Fig. 7. The simulated three-dimensional velocity magnitude contours at the five chosen curved sections for showing the separation region.

Fig. 8. The simulated three-dimensional pressure contours at the five chosen curved sections.

Fig. 9. The simulated streamlines at different \( \theta \)-planes. The topological points are labelled for showing that the predicted solutions satisfy the topological rule \((N-S)+\frac{1}{2}(N'-S')=1\). Here, \( N, S, N' \) and \( S' \) denote the node, saddle, half-node and half-saddle, respectively.

Fig. 10. (a) The simulated wall shear stress contours; (b) The simulated wall shear stresses along the inner and outer walls of the channel at the plane of symmetry \( y=0.5 \).
Fig. 11. The simulated limiting streamlines at each no-slip wall.

Fig. 12. The simulated three-dimensional vortical corelines and the surrounding spiraling particle motions.

Fig. 13. The simulated velocity component $u_\lambda$, pressure $p$, normalized helicity and helicity gradient $\lambda(= \hat{\partial}_{\lambda}/\hat{s})$ along the vortical coreline having the unit vector $\hat{s}$.

Fig. 14. Illustration of the plane which is locally orthogonal to a vortical coreline. The longitudinal plane is also plotted.

Fig. 15. The simulated streamlines at the planes that are locally normal to the vortical coreline.