1 Introduction

Although arterial bypass grafting is a commonly performed surgical procedure, there is a loss of patency in 50 percent of vessels over a ten-year period [1]. One of the major causes of loss of patency in bypass grafts is intimal hyperplasia (at later times there is superadded atherosclerosis) where encroachment on the lumen of the vessels occurs. The local flow field and in particular the wall shear stress markedly influences vascular biology [2,3]. The physiological mechanisms behind the development of this pathology are not completely understood. There is, however, strong evidence of a correlation between the local flow patterns and the regions where intimal thickening has been shown to occur preferentially [4–6]. For example, the work of Sottiusi et al. [5] suggests a relationship between the occurrence of intimal hyperplasia at the so-called heel, toe, and bed of an anastomosis (see Fig. 1) and regions of low time-averaged shear where one might expect a long particle residence time. Ojha et al. [4], Ojha [7], and Lei et al. [8] have also suggested a connection between the spatial gradient and sharp temporal variation of wall shear stress within these regions.

The association between local flow-determined parameters (for example degree of shear reversal and particle residence times) has not only been postulated for intimal hyperplasia but also for the development of atherosclerotic lesions [9,10]. Yamamoto et al. [11] showed, via in vivo measurements on the origin of the canine renal artery, that atherosclerotic lesions are also prone to develop in regions with low time-averaged shear, flow separation, and oscillation of the flow. Asakura and Karino [12], using fixed human coronary arteries, also found a strong correlation between regions of low wall shear and recirculating flow patterns and the location of atherosclerotic plaques.

Correlation between the local haemodynamics at bypass grafts and the onset of intimal hyperplasia is not a new hypothesis and has been widely researched, both numerically [13–15] and experimentally [5,16,4,7]. However, while these investigations generally support the overall hypothesis, little research has focused on the role of the three-dimensional character of the geometry; such geometry would occur in a physiologically correct anastomosis [17–19]. As a natural starting point, most of the numerical and in-vitro experiments have concentrated on geometries where the centerline of the bypass vessel and the centerline of the host vessel lie in a plane, as shown in Fig. 1(a), and may be termed planar. However, in this investigation we consider not only the flow in a planar model but also the effect of an out-of-plane distal anastomosis model where the centerline of the bypass vessel and the centerline of the host vessel no longer lie in a constant plane and may be termed nonplanar. There are many possible configurations for a nonplanar model depending on the local vascular geometry, for example the nonplanarity of the aortic bifurcation [19], the use of vein cuffs [20] or the introduction of a hood [21]. In the present study, we have restricted our attention to an anastomosis model where the bypass vessel is described by a torus that lies in a plane perpendicular to the original planar geometry, as shown in Fig. 1(b).

For our study, we have assumed that the model is noncompliant, that the flow is steady, and the fluid is Newtonian. The flow in arteries is clearly unsteady, the blood has non-Newtonian properties, and the walls are compliant. The local haemodynamics, however, as in any internal flow problem, are determined not only by these intrinsic properties of the flow but also by the boundary conditions at inflow, outflow, and on the wall, as induced by the local geometry. In modeling the flow in the large arteries (with diameters of millimeters or above), it appears that the overall behavior is primarily affected by the basic geometry, the inflow and outflow conditions, and by the unsteadiness of the flow. Although the wall elasticity and non-Newtonian character of the fluid may be of considerable significance in, for example, transport mechanisms, several studies suggest they are of somewhat lesser importance as far as the gross features of the flow are concerned. Moore et al. [23] compared MRI measurements of flow in the abdominal aorta in vivo and in a glass model (designed to reproduce the three-dimensional in vivo anatomical geometry), and concluded that in vitro modeling could represent the major...
features of blood flow (such as flow reversal and nonaxisymmetric profiles in the human aorta). Further discussion on the repercussions of assuming rigid walls and neglecting non-Newtonian effects to model in vivo flow is contained in the review of Friedman [24].

In sections 2.1 and 2.2, we review the experimental and numerical techniques employed and provide appropriate validation for each technique. In section 3.3 we show the comparison of the results of the MRI and CFD investigations within a planar geometry and discuss the salient features of this flow. Finally, in section 4.1, we discuss the significant flow characteristics in the planar and nonplanar models.

2 Materials and Methods

2.1 Experimental Technique

2.1.1 Model Construction. The planar anastomosis model was constructed using two Perspex pipes each of 8 mm internal diameter. The junction between the two pipes was milled to form a 45°± 1 deg junction and then the two pipes were adhered together.

2.1.2 Imaging. The MRI pulse sequence was of a two-dimensional phase contrast type using a 1.5 Tesla ‘‘custom-built’’ small-bore scanner with 5.8 Gauss/cm gradient strength and a 300 μs rise time. The magnet was manufactured by Magnex (UK) Ltd., the imaging electronics were built by GE Medical Systems (Milwaukee, WI) and the RF coils and customized pulse sequences supplied by GE Research and Development Center (Schenectady, NY). The field of view was 4 cm using a 256×256 image matrix, which produced a 0.156 mm/pixel resolution. A 30 deg flip angle was applied and a slice thickness of 2 mm was selected. The flow was encoded using a velocity encoding (VENC) parameter of 22 cm/s for all velocity components. The VENC value was well above the highest measured velocities. Repetition time (TR) for the scans was 33 ms and the echo time (TE) or time between the center of the RF excitation pulses and the center of the echo was 17 ms. Given that data were extracted from a thin slice, 40 excitations (NEX) were employed to compensate for the inherently low signal to noise ratio (SNR) of the measurements. The acquisition took 22 minutes per slice location for all dimensions of flow encoding.

A computer-controlled flow simulator (Quest, Inc.) was used to provide the flow for the experiments. The circulating fluid used was a 60-40 by volume distilled water–glycerol solution. Its kinematic viscosity was measured with a capillary viscometer and was 3.416×10⁻⁶ m²/s at 20°C. For the planar model, a constant flow rate of 5.512 ml/s was applied corresponding to a Reynolds number of Re=250 based on the internal diameter of the pipe and the kinematic viscosity of the fluid used.

2.2 Computational Technique

2.2.1 Spectral/hp Algorithm. The computations were performed using a spectral/hp element algorithm [22]. In this technique the solution domain is decomposed into tetrahedral subdomains or elements as is typical of standard finite element or finite volume discretizations. However, unlike these standard techniques, each tetrahedral region is represented by a polynomial expansion. Convergence of the numerical solution may then be achieved by reducing the characteristic size of an element or increasing the order of the polynomial within each element. For smooth solutions, the advantage of a p-type approach is that high accuracy for a given amount of computational work can be obtained efficiently [22]. A further advantage of this method is that a refined simulation does not necessitate a redesign of the computational mesh, since a higher order polynomial expansion may be used within each tetrahedral element of the existing mesh. Finally, the complex curvature of the surface may also be accurately represented by the high-order polynomial expansion.

The spectral/hp element method has been combined with a high-order splitting scheme [25] to solve the time-dependent incompressible Navier–Stokes equations. The incompressible Navier–Stokes equations can be written as

\[
\frac{\partial \mathbf{v}}{\partial t} = -\nabla p + \nu \nabla^2 \mathbf{v} + \mathbf{N}(\mathbf{v})
\]

where

\[
\mathbf{L}(\mathbf{v}) = \nabla^2 \mathbf{v}
\]

and \( \mathbf{v} \) and \( p \) denote the velocity vector and the pressure, respectively. The numerical splitting scheme can be written in three steps as

\[
\begin{align*}
\mathbf{v}^{n+1} - \mathbf{v}^n &= \frac{j_{n-1}}{\Delta t} \beta_q \mathbf{N}(\mathbf{v}^{n-q}) \\
\mathbf{v}^n &= \mathbf{v}^{n+1} - \nabla \mathbf{p}^{n+1} \\
\mathbf{v}^{n+1} - \mathbf{v}^n &= \frac{j_{n-1}}{\Delta t} \nu \sum_{q=0}^{j_{n-1}} \gamma_q \mathbf{L}(\mathbf{v}^{n+1-q})
\end{align*}
\]

In the first step, (1a), the nonlinear advection terms are advanced using either a convective or rotational form and are integrated in time using a multilevel Adams–Bashforth scheme denoted by the coefficients \( \beta_q \). In the second step the time-averaged pressure \( \mathbf{p} \) is obtained by taking the divergence of (1b), and assuming \( \nabla \cdot \mathbf{v}^n = 0 \) to obtain a Poisson equation, which is supplemented with boundary conditions of the form

\[
\frac{\partial \mathbf{p}^{n+1}}{\partial n} = n \left[ \sum_{q=0}^{j_{n-1}} \beta_q \mathbf{N}(\mathbf{v}^{n-q}) - \nu \sum_{q=0}^{j_{n-1}} \beta_q \nabla \times (\nabla \times \mathbf{v})^{n-q} \right].
\]

These boundary conditions ensure that the splitting error associated with the scheme is consistent with the overall temporal discretization. Finally step (1c) is re-arranged into a Helmholtz equation for each velocity component, which implies an implicit
treatment of the viscous component using either an Euler backwards difference or Crank–Nicolson scheme, which are denoted by the coefficients $\gamma_t$.

2.2.2 Computational Domains and Boundary Conditions. The computational domains for the planar and nonplanar geometries are shown in Fig. 1. For both meshes, the intersection of the centerline of the host and bypass vessels is located at the origin and the intersecting angle is taken to be 45 deg. In the planar geometry, shown in Fig. 1(a), the outflow was located 9 diameters ($D$) downstream from the origin and the occlusion and inflow boundaries were located at $3D$ and $5D$, respectively. The mesh contains 1742 elements with a viscous mesh region in the host artery of $0.1D$ thickness. In the nonplanar geometry, shown in Fig. 1(b), the outflow and occlusion were located at $10D$ and $3D$ from the origin, respectively. The nonplanar bypass vessel was decomposed into three sections: a region of straight pipe of length 1.5 connected to a 90 deg torus of radius $2D$ attached to another straight section pipe of length $1D$. The torus was oriented so that its centerline lay in a plane perpendicular to the plane described by the intersection of the pipes’ centerlines. For this geometry 1946 elements were used with a viscous region of $0.1D$ thickness in the host artery.

In both geometries, a Hagen–Poiseuille parabolic profile was imposed at inflow, which was suitably scaled to produce a unit mean velocity. All walls were treated as non-slip surfaces and the outflow was treated as a fully developed flow so that $\nabla \cdot \mathbf{v} = 0$, $\nabla \cdot \mathbf{n} = 0$, $\nabla \cdot \mathbf{v} = 0$. The pressure at outflow was assumed to be constant.

All computations were performed at three polynomial resolutions of order $p = 2, 4$, and 6, which correspond to 17,420, 60,970, and 146,328 local degrees of freedom per variable in the planar case and 19,460, 68,110, and 163,464 in the nonplanar case. As mentioned previously, polynomial refinement is hierarchical and so all of the degrees of freedom used in the lower resolutions were contained within the higher resolution simulation. The solution was time marched until a steady-state solution was reached. It was assumed that the steady-state solution was achieved when the fluctuation in the force acting on the pipe as compared with the mean flow velocity in five history points was smaller than $10^{-5}$ and a fluctuation of velocity was $10^{-5}$ when compared with the mean velocity. The history points were located at approximately $-5D$, $0D$, $1.5D$, $2D$, $3D$ along the centerline of the host vessel. Computational experiments on varying the length of the outflow boundary did not show any significant effect on the results within the region of interest.

2.2.3 Mesh Generation. The geometry of the model arteries is represented by means of CAD spline curves and surfaces to obtain a boundary representation (B-Rep) of the computational domain. The B-Rep is then used as the analytical definition of the boundary to produce a discretization of the domain into a tetrahedral mesh.

The mesh generation employs a modified advancing layers method for the near-wall regions [26] and a method based on the advancing front technique [27] for the rest of the domain. The resulting mesh of linear tetrahedra is finally transformed into a boundary conforming mesh of high-order elements.

2.2.4 Wall Shear Stress Calculation. A notable advantage of using a computational approach to simulate fluid flow is the ability to extract data on quantities that are extremely difficult, if not impossible, to determine experimentally. A flow quantity of particular interest in this investigation is the wall shear stress. The viscous stress $\tau = [\tau_1, \tau_2, \tau_3]^T$ acting on an area of fluid with a normal $\mathbf{n}$ is defined as

$$\tau_i = \mu \frac{\partial u_i}{\partial x_j} n_j$$

where $\mathbf{n} = [n_1, n_2, n_3]^T$, $\mathbf{v} = [u_1, u_2, u_3]^T$, and $\mu$ is the coefficient of dynamic viscosity. Therefore the wall shear stress acting on an element of fluid at the wall $\tau_w$ can be determined using the wall normal $\mathbf{n}_w$. Although the wall shear stress here is expressed as a three-dimensional vector, it can be shown that this vector is orthogonal to the wall normal and therefore acts in a plane tangent to the wall. We may represent the wall shear stress in terms of a magnitude $\tau_{wss} = |\tau_w|$ and unit vector $\hat{\mathbf{n}} = \tau_w / |\tau_w|$ as $\tau_w = \tau_{wss} \hat{\mathbf{n}}$.

3 Validation Results

3.1 Estimate of Experimental Uncertainty. Our uncertainty analysis of the experimental measurements identified that there was a precision error, due to random noise in the MRI measurements, of 10 percent at a 95 percent confidence level (determined as the rms error as compared with the mean velocity). Initially a large systematic error was observed. This error was subsequently minimized by a recalibration of the MRI facility and a final systematic error of 2.5 percent with respect to the mean velocity was detectable. Therefore the uncertainty of the experimental process at a 95 percent confidence level, given by the root sum square of the bias and precision error, is 10.3 percent.

These estimates were obtained by scanning a straight rigid 8 mm internal diameter tube model (phantom) oriented along the axis of the magnet bore and under constant steady flow conditions. The MRI scanner and scan parameters were set to be the same as used in the model graft. Measurements were taken at different locations along the straight pipe and the experiment was repeated for a range of flow rates, which yielded Reynolds numbers between 250 and 600. Fully developed flow was ensured by only considering locations well beyond the entrance length ($L$) which was estimated as $L = 0.03D$ Re, where Reynolds number is based on the pipe diameter $D$ and mean flow velocity. The random and systematic errors were obtained by comparing the MRI profiles with the theoretical Hagen–Poiseuille flow profile. In order to avoid overestimation of the error levels due to the disproportionately high level of error in the near wall region, two pixels from each edge of the profile were discarded (which is less than 6 percent of the total number of pixels in our experimental profiles). Having determined the experimental profile, a parabolic fit was put through the data using a least-squares approximation.

The difference between the experimental data and this least-squares fit was then taken to be the random measurement noise, while the difference between the experimental and theoretical parabolic profiles was used to determine the systematic error. An illustrative sample of the experimental axial flow profile across the pipe is shown in Fig. 2 at Re = 390. Also shown is the least-squares parabolic fit, which is coincident with the theoretical profile.

3.2 Numerical Accuracy. To determine the accuracy of the numerical computations we have used two indicators. The first measure uses the integral of the Cartesian pressure and viscous forces acting on all the walls of the models, which gives a single quantitative value. The second measure is the distribution of normalized wall shear stress in the vicinity of the bed of the anastomosis, which provides a field of information in a more qualitative measure.

3.2.1 Force Integrals. We define the force coefficients, $C_x$, $C_y$, $C_z$, as

$$C_i = \frac{|F_i|}{\frac{1}{2} \rho u^2 A}$$

where $F_i$ is the integral of the pressure and shear forces over all walls in the ith direction, i.e.,

$$F_i = \int_{\text{walls}} \left( p n_i + \tau_i n_i \right) ds.$$
\( \rho \) is the density, \( \bar{u} \) is the mean velocity and \( A \) is the surface area of the walls (\( A = 51.3 \) for the planar geometry). For a mean Reynolds number \( \text{Re} = \bar{u}D/\nu = 250 \) the value of the force coefficients at steady state for different polynomial orders are shown in Table 1. From this table we see that the change in \( C_x \) and \( C_y \) between the simulations at \( p = 2 \) and \( p = 4 \) is 10.8 and 5.24 percent of the value at \( p = 6 \); however, the difference between the simulations at \( p = 4 \) and \( p = 6 \) is only 0.4 and 1.5 percent, respectively. This variation demonstrates the level of “mesh convergence” in the solution and at \( p = 6 \) is of the order of 1 percent in this flow variable. Finally we see from \( C_z \) that the symmetry of the problem is recovered as we increase the polynomial order since \( C_z \) is driven toward zero.

### 3.2.2 Normalized Wall Shear Stress

As discussed in section 2.2.4, the magnitude of wall shear stress \( \tau_{wms} \) is clearly dependent upon the first derivative of the velocity field \( v \). In the spectral/hp element approach, as well as the finite element and finite volume methods, the velocity field is represented by a piecewise continuous polynomial approximation with \( C^0 \) continuity across elements. That is to say the numerical representation of \( v \) is continuous over the elemental boundaries but the derivative of \( v \) is not. Therefore, if the wall shear stress is approximated using the same numerical representation as the velocity field, then it will not be continuous over elemental boundaries unless the solution is completely resolved so the numerical solution closely approximates the exact smooth solution.

Typically, CFD computations of the velocity field are post-processed to recover a continuous approximation to the wall shear stress. However, this operation essentially smooths the wall shear stress. In the following computations we have calculated the wall shear stress using only the \( C^0 \) numerical polynomial representation of velocity and have not post-processed the results. Such an approach has been adopted since the jump in wall shear stress at elemental boundaries can be used as an indication of the resolution of the simulation. This point is illustrated in Fig. 3 where we see three plots of the numerical calculation of the magnitude of wall shear stress for the planar anastomosis model at \( \text{Re} = 250 \) around the bed region. The plot has been normalized with the Hagen–Poiseuille wall shear stress at this Reynolds number. In the first plot, 3(a), the calculation was performed using a polynomial order of \( p = 2 \) and the jumps in the wall shear stress magnitude directly correspond to the elemental boundaries of the computational mesh. In this calculation we have captured the general location of the maximum and minimum values, although the local shape of the features is not clear. In Fig. 3(b) a polynomial order of \( p = 4 \) was used. The general continuity of the contours improves and the shape of the extreme values becomes better defined. Finally in Fig. 3(c), where a polynomial order of \( p = 6 \) was applied, the majority of the contours are continuous and the shape of the maximum shear region is clear. The only noisy region is the region proximal to the heel. Here the flow is nearly stagnant, leading to a wall shear stress that is two orders of magnitude lower than the value at the peak.

### 3.3 Comparison of MRI and CFD Within the Planar Model

In this section we consider the steady flow in the planar anastomosis model using computational and experimental data, as previously studied by Steinman et al. [13], with a view to validating the computational model and reviewing the flow within this configuration. Figure 4 shows the comparison of the axial component of velocity on three slices of the host vessel: one at \( 0.25D \) distal to the toe of the anastomosis, as defined in Fig. 1, and at \( 2D \) and \( 5D \) distal to the toe. The computational data have been

![Fig. 2 Experimental measurement of the axial flow in a straight pipe at a Reynolds number of \( \text{Re}=390 \). Also shown is the least-squares fit to a parabolic profile, which is coincident with the theoretical parabolic solution at this Reynolds number.](image)

![Fig. 3 Numerical calculation of the normalized wall shear stress in the planar anastomoses model around the bed region using a polynomial order of: (a) \( p=2 \), (b) \( p=4 \), (c) \( p=6 \). The same contour levels have been used in each plot.](image)

<table>
<thead>
<tr>
<th>Polynomial order (( p ))</th>
<th>2</th>
<th>4</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_x(\times 10^{-2}) )</td>
<td>3.623</td>
<td>4.054</td>
<td>3.992</td>
</tr>
<tr>
<td>( C_y(\times 10^{-2}) )</td>
<td>7.549</td>
<td>7.965</td>
<td>7.934</td>
</tr>
<tr>
<td>( C_z(\times 10^{-3}) )</td>
<td>0.310</td>
<td>4.078</td>
<td>1.416</td>
</tr>
</tbody>
</table>
linearly averaged over five equally spaced slices of thickness 0.25D to take account of the slice thickness of the MRI data. All data have been normalized by the mean velocity, \( \bar{u} \). The mean Reynolds number, based on the mean velocity, was 250 for both the MRI and the computational data. Also shown in Fig. 4 is the comparison of the CFD and MRI data through a horizontal section along the centerline of the data with error bars indicating the 10 percent experimental error. The flow should be interpreted as having a positive velocity as it moves into the page, toward the outflow of the domain. Figure 5 shows a comparison of the crossflow components \( \bar{v} \) and \( \bar{w} \) captured using CFD and MRI at 0.3D distal to the toe of the graft. The agreement of the MRI and CFD data is generally good and within the experimental error. The plane of symmetry of the model about the centerline of the host and bypass vessel imposes a similar symmetry on the flow patterns as is illustrated by the cross sections in Figs. 4 and 5. We emphasize that this symmetry was not explicitly imposed in the computations.

The velocity profiles in Fig. 4(a), at 0.25D, demonstrates that the parabolic axial flow pattern of the Hagen–Poiseuille flow has been translated toward the outer wall. The crossflow velocities shown in Fig. 5 are consistent with this translation. We note that the \( v \)-component is of the same order of magnitude as the axial \( u \) component. This strong crossflow component results from the pressure gradient introduced to balance the centrifugal force required to turn the flow. Thus the lower pressure at the toe of the graft drives a secondary flow pattern, as shown by the \( v \)-component velocity in Fig. 5(a). Associated with this secondary flow is a \( w \)-component velocity, which is necessarily symmetric about the horizontal centerline as shown in Fig. 5(b) and the combined motion of the \( v \) and \( w \) velocities is shown in the crossflow streamline plot shown in Fig. 5(c).

Figure 4(b) shows the axial \( u \) component of velocity at 2D distal to the toe. The crescent form of this cross section, clearly captured in both the CFD and MRI results, is consistent with a Dean type flow, typically associated with flow within a torus [28]. Although it is not possible directly to identify the radius of curvature of the bypass pipe, it can be appreciated that the net effect of the bypass junction is to rotate the flow by 45 deg, which could have been achieved by a smooth transition in a curved pipe. Finally, Fig. 4(c) shows the axial \( u \) component of velocity at 5D distal to the toe. Although the crescent nature of the flow is still evident at this position, the viscous effects have started to play a greater role as the profile diffuses back to its steady-state parabolic profile, which would correspond to concentric circles in this figure. The rate of diffusion in the CFD and MRI results is similar. This should be expected as their respective Reynolds numbers are comparable.

Finally we note that the agreement between the MRI and CFD velocity fields is not as good in the \( w \) component as in the other crossflow components shown in Fig. 5(b). This is due to the fact that the magnitude of velocities in this direction is noticeably smaller and, consequently, the experimental noise is more significant.
4 Results and Discussion

4.1 Comparison of Planar and Nonplanar Geometries

4.1.1 Velocity Contours. Having considered the flow characteristics of the CFD and MRI results for the planar anastomosis model, which are generally in good agreement, we now compare the CFD results of the planar versus the nonplanar geometry. Figure 6 shows the axial $u$-component velocities normalized by $u$ at the toe, $2D$ and $5D$ distal to the toe for the planar and nonplanar model discussed in section 2.2.2. Both simulations were converged to a Reynolds number of Re=250 based on the mean velocity. The contours have been selected so that the planar and nonplanar cases are directly comparable at any given cross section. The nonplanar geometry exhibits a similar crescent formation typical of the Dean flow within a curved pipe. However, Fig. 6 shows that, in the nonplanar model, the crescent pattern is evident even at the toe in contrast to the planar model. Furthermore, the crescent has a bulk rotation as the flow moves distally and rotates in a clockwise sense as we look from the proximal toward the distal end of the host vessel. The motion is relatively pronounced having gone through a rotation of 180 deg in approximately five diameters. The sign of this rotation is consistent with the out-of-plane deformation of the bypass vessel where the bulk flow is pushed toward the outer radius of the torus section.

We recall that for a curve that lies in a plane, the curvature is a measure of the variation of the curve from a straight line. For a more general curve at any point we can define a local plane of curvature (using the local tangent and the derivative of the tangent...
along the curve. As we move along this general curve, the plane of curvature will change and the torsion is a measure of the variation of this plane. A circle is a line of constant curvature but zero torsion and a helix is a line of constant curvature and torsion. Therefore, in the same way that the planar geometry might be idealized as a planar bend with an equivalent curvature, which generates the crescent shape of the axial flow, we could also envisage the nonplanar case as being represented by an equivalent helical shape, thereby introducing torsion into the problem. For steady flow within a helical pipe with sufficiently large curvature and torsion, an axial velocity profile of crescentic shape also develops similar to the planar bend [29–31]. However, in a helical pipe the crescent is not symmetric, which is also evident in the results of Fig. 6. Further, the numerical and experimental results from Doorly et al. [32] have shown that when a helical pipe exits into a straight pipe, a bulk rotation in the axial velocity is established due to the rotational inertia of the flow that develops within the helical pipe. Once again the role of viscosity is evident as the flow moves down the pipe, forcing the axial flow to return to its steady-state parabolic profile.

Figures 7 and 8 show the comparison of the crossflow velocity components $v$ and $w$, normalized by $\bar{u}$, for the planar and nonplanar models at the toe, at $2D$ and at $5D$ distal to the toe. In the planar model there are two recirculation regions, clearly marked by the three cells in the $v$-component velocity and the four cells in the $w$-component velocity. The intersection of the zero contours of the $v$ and $w$ velocity indicate the two points at which the flow is moving purely in the axial direction. These points can be considered as the crossflow stagnation points or the equivalent location of a line vortex in the flow, which is oriented along the
x-direction. From the contours of \( v \)-component velocity in the planar geometry shown in Fig. 7 we see that these points move from their location near the outer wall at the toe toward the center of the vessel as the upper and lower cells grow in magnitude. Figure 8 shows a slight motion of the stagnation points from left to right as the flow moves distally from the toe. This slight motion is illustrated by the contour of zero \( w \)-velocity in the planar model at the toe, which is located slightly to the left of the center. However, by 2\( D \) distal to the toe, the contour has moved to the right of the centerline, and by 5\( D \) the contour has established a more central doubly symmetric location.

In contrast, the crossflow within the nonplanar model demonstrates a noticeably different structure. As is to be expected, there is no longer a plane of symmetry, and although there are initially two recirculation regions, as indicated by the three cells in \( v \)-velocity and four cells in \( w \)-velocity, the bottom recirculation region is weak. As we move distally from the toe, the top and bottom cells in the \( v \)-component velocity of Fig. 7 grow under the action of diffusion and bring the crossflow stagnation point toward the center of the cross section. The bottom cell, however, is clearly weaker and at the 5\( D \) section is almost indiscernible. It can be noticed, from the \( w \)-component velocities in the nonplanar model of Fig. 8, that there is a transition from two to one recirculation cells. Initially at the toe, we see four cells indicating two recirculation regions. As the flow progresses distally, at the 2\( D \) section, the top left and bottom right cells have started to coalesce, causing the bottom left cell to become increasingly smaller. Finally at 5\( D \) the existence of two \( w \)-component cells indicates that only one recirculation region is present. As noted in Zabliski and Mestel [31], a single recirculation cell is a particular feature of non planar flows due to torsion. The occurrence of a single cell is typically found at low Dean numbers, which are associated with a large radius of curvature, as is clearly the case in the straight section of the host vessel after the anastomosis. On a more physical level, the dominance of one recirculation region is to be expected, since the nonplanarity causes a non symmetric injection of the flow into the host artery, with the centroid of the velocity being displaced below the horizontal plane of symmetry of the host artery.

4.1.2 Flux of Flow at Heel. After considering the flow development as we move distally from the anastomosis, we now analyze the proximal region to the anastomosis within the host vessel. In a fully occluded vessel there is a stagnant region proximal to the anastomosis. Such a region may be physiologically undesirable due to the potential development of intimal hyperplasia at the heel. We would therefore like to consider the role of the nonplanar geometry in the flow characteristics at this region. Figure 9 shows the axial \( u \)-component of velocity, normalized by \( \bar{u} \), in the cross-plane located at the heel for the planar and nonplanar models. Using the same convention as in previous plots, flow moving toward the outflow is represented by a negative contour. Positive contours indicates flow entering the occluded region proximal to the anastomosis. The bypass vessel is located on the right-hand side of the cross sections and therefore the negative contours indicate an entrainment of flow into the junction. This flux of flow out of the occluded region is replaced by a positive flux located near the bed, as shown on the left hand side of the section. Since there is no other source of flow entering this region, the net flux of flow across the plane is necessarily zero. However, the magnitude of the contours provide an indication of the amount of fluid recirculating within this region. In both the planar and nonplanar models, the magnitude of axial velocity is relatively small, having a value in the interval \(-0.065 \leq \frac{|u|}{\bar{u}} \leq 0.047\) in the planar model and a value in the interval \(-0.101 \leq \frac{|u|}{\bar{u}} \leq 0.097\) in the nonplanar model, as compared with a peak velocity of 2 at the inflow. However, the introduction of nonplanarity has almost doubled the peak of axial velocities crossing this section. The large velocities do not necessarily indicate an integrated effect of flow across the whole section and, perhaps, the average integrated absolute velocity over the cross section, i.e.,

\[
[\bar{u}] = \frac{\int |u|ds}{\pi r^2}
\]

where \( r \) is the pipe radius is a better measure. For the planar model we find that \[\frac{|u|}{\bar{u}} = 1.93 \times 10^{-2}\] and for the nonplanar model we find that \[\frac{|u|}{\bar{u}} = 3.51 \times 10^{-2}\] where \( \bar{u} \) is the mean flow. Although this represents an 80 percent increase the absolute flux only rises from 1.93 to 3.5 percent of the mean flow, which is still comparatively small.
4.1.3 Wall Forces. The final part of the investigation of the role of out-of-plane geometry considers the direct and tangential stresses on the host vessel in the region of the anastomosis. Figure 10 shows a map of the magnitude of wall shear stress \( \tau_{wss} \) for the two models. These values have been normalized with respect to the wall shear stress of the Hagen–Poiseuille flow within a straight pipe of the same diameter. The map should be interpreted so that the horizontal axis represents distance along the axis of the host vessel and the vertical axis represents the circumferential distance along the wall of the host vessel. The map has been chosen so that the bed region appears at the center. For the planar geometry shown in Fig. 1(a) the symmetry of the flow is once more evident, reaching a peak value of 3.72 in the region of the bed just distal to the toe. The circular region of low stress just adjacent to this peak can be attributed to the stagnation point, while the low flow rate at the heel seen in Fig. 9 is clearly associated with a region of low wall shear stress. As mentioned previously, the wall shear stress has been calculated directly from the velocity field and so the numerical wall shear stress might not be continuous across element boundaries. The overall continuous appearance of the wall shear stress is an indication of the resolved nature of the computation. However, there is some evidence of jumps in the wall shear stress in the low wall shear stress region proximal to the anastomosis. The wall shear stress in this region is about two orders of magnitude smaller than the peak shear stress at the bed, which is consistent with the level of numerical error at a polynomial order of \( p = 6 \) as discussed in section 2.2.2.

In Fig. 10(b) we see the nonplanar geometry distribution using the same contours as shown in Fig. 10(a). We note that for the nonplanar model the region of peak wall shear stress at the bed of the junction has been reduced to a peak value of 3.37, which is approximately 10 percent lower than that of the planar model. Although there is only a 10 percent reduction in the peak value, the contours of the maximum around this peak have been noticeably altered as indicated by the absence of contour levels at 3.4 and 3.6. The region of maximum wall shear stress at the bed has also extended proximally, as compared with the planar model, reducing the extent of the low shear region around the stagnation point. Further, the contours of low wall shear stress to the right of the heel decrease at a slower rate consistent with the increase of flow recirculating in the nonplanar model, as compared with the planar case.

For completeness, Fig. 11 shows the pressure coefficient along the wall of the host vessel in the region of the bed of the anastomosis. The pressure coefficient is defined as

\[
C_p = \frac{p - \bar{p}}{\frac{1}{2} \rho \bar{u}^2}
\]

where \( \bar{p} \) denotes the average pressure at the outflow location of the planar model; \( \rho \) is the density and \( \bar{u} \) is the mean velocity. From this figure we can conclude that the reduction in peak wall shear stress in the nonplanar simulation is associated with an increase in the peak pressure coefficient relative to the pressure coefficient at 9D distal to the junction. As discussed previously, there is clearly a negative pressure gradient in the circumferential direction from the bed toward the toe.

5 Summary

The comparison between the MRI measurements and the CFD simulations within a 45 deg planar model anastomosis at Re = 250 has been used to validate the computational technique and gain understanding of the steady Newtonian flow characteristics. The inherent symmetry of this flow model has been removed by considering an out-of-plane deformation of the bypass vessel. The role of introducing this three dimensionality can be viewed in terms of a helical-type motion of the flow as it moves from the bypass to the host vessel as compared to an idealization of a curvature induced transition in the planar model. Considering the planar model in terms of a curvature induced motion explains the occurrence of a crescent shape in the axial flow component associated with crossflow velocity established as the flow is turned through 45 deg. However, when considering the helical motion of a nonplanar anastomosis, model torsion as well as curvature effects are present. The introduction of torsion breaks the symmetry of the planar model and we see evidence of a single recirculation region developing in the crossflow velocity components as the flow moves distally, compared to the existence of two recirculation regions in the planar model. Further, the nonplanar model reduces the peak in the wall shear stress magnitude by 10 percent at the bed of the anastomosis as compared with the planar model and introduces an 80 percent increase in absolute flux of velocity into the occluded region proximal to the anastomosis. Nevertheless the absolute flux of velocity into the occluded region is still comparatively small being only 3.5 percent of the mean flow.

A simple modification of the geometry of an end-to-side anastomosis, whereby the graft is rendered approximately helical, has been shown to profoundly affect the flow field and can also be expected to influence the particle residence times. In view of the reported association between the flow field and vascular biology, the development of intimal hyperplasia, and atherosclerosis, the results may have pathophysiological implications in relation to geometry. Current investigations are now focused on the role of pulsatility within the context of planar and nonplanar geometries of end-to-side anastomosis.
Acknowledgments

The authors would like to acknowledge the Department of Health and the generous support provided by the General Electric Company (USA) for funding of the MRI experiments. We would also like to acknowledge the Garfield Weston Foundation for partial support. O. Shah is supported by the EPSRC and Harefield Advanced Technology under a CASE award. The computations were performed on the Imperial College Parallel Computing facility Fujitsu AP3000 and the biomedical visualisation Origin 2000 system.

References