Abstract

Cardiovascular diseases are one of the major causes of long-term morbidity and mortality in human beings. The nearly epidemic increase in prevalence of such diseases poses a serious threat to public health and calls for efficient methods of diagnosis and treatment. Non-invasive diagnostic procedures such as MRI are often used in this context, however these are limited in terms of spatial and temporal resolution and do not provide information on time-dependent pressures and wall shear stresses - key quantities considered to be partially responsible for the formation and development of related pathologies.

The present study is concerned with the numerical simulation of oscillatory flow through the abdominal aortic bifurcation. Computational fluid dynamics simulation of oscillatory flow in a branched geometry at high Reynolds numbers poses considerable challenges. The present study reports a detailed comparison of simulations performed with a finite volume and a finite element method, two approaches with significant differences in their discretization strategy, treatment of boundary conditions and other numerical aspects. Both solvers were parallelized, using loop parallelization of the BiCGStab linear solver for the finite volume and domain decomposition based on the Schur complement method for the finite element technique. The experience gained with these two approaches for the solution of flow in a bifurcation forms the focus of this study. While similar results were obtained for both methods, the computation time required for convergence was found to be significantly smaller for the finite element approach.

Keywords: Aortic bifurcation, Oscillatory flow, Numerical simulation, Comparison of FEM and FVM.
Nomenclature

\( u \) Cartesian velocity vector
\( x, y, \) and \( z \) Cartesian coordinates
\( p \) pressure
\( \phi_u \) solution vector for velocity
\( \phi_p \) solution vectors for pressure
\( b_u \) body force
\( A, B, B^T \) global matrices
\( S \) Schur complement
\( N \) total number of nodes
\( b \) Right hand side
\( K \) standard preconditioners
\( P^{-1} \) optimal preconditioner
\( n \) index of the corresponding node
\( \Delta r \) distance between the cell centroid and face centroid
\( p' \) total pressure correction
\( p_m' \) calculated pressure correction
\( p_s' \) smoothing pressure correction

Greek symbols
\( \gamma \) relaxation parameter in the finite volume method
\( \eta \) dynamic viscosity of the fluid
\( \rho \) fluid density

Subscripts
\( n_1, n_2, n_3 \) representation of centroids of the cell faces
\( f_1, f_2, f_3 \) faces of a tetrahedral element
1. INTRODUCTION

Blood flow in arteries is dominated by unsteady flow phenomena due to its oscillatory nature. The cardio-vascular system is an internal flow loop with multiple branches in which a complex liquid circulates. The arteries are living organs that can adapt to varying fluid loading conditions on short as well as long time scales. Under certain circumstances, unusual biophysical conditions create an abnormal biological response. The relationship between flow in the arteries, particularly the pressure pulses, the wall shear stress distribution and the sites where diseases develop has motivated much research on arterial flow in the last decade. It is now accepted that the sites where shear stresses are extreme or change rapidly in time or space are the ones that are most vulnerable. Pressure fluctuations can also play an important role as pointed out by Szczerba et.al. [1]. When an arterial wall loses its structural integrity, the result is the development of a balloon-like expansion called an aneurysm.

In order to investigate aneurysms in the artery, many mathematical and experimental studies have been conducted in the past. Even though some experiments use complex anatomically realistic geometries, most are limited to using simplified ones, such as straight bifurcations using glass or acrylic pipes. It is also difficult to measure or capture detailed sub-millimeter characteristics of the flow in an experiment. On the other hand, advancements in computer technology made numerical investigations a promising alternative for analyzing arterial hemodynamics, allowing detailed investigation of parameters affecting disease progression, such as wall shear stress (WSS), oscillatory shear index (OSI) and other typically not directly observable parameters. Although most numerical studies are limited to simple geometries, with or without pulsatile flow conditions, the research direction is towards simulating blood flow in real arterial geometries under realistic flow conditions.

Technologies related to medical image data acquisition and analysis such as computed tomography (CT), angiography and magnetic resonance imaging (MRI) allow the construction of three-dimensional, patient-specific models of blood vessels. Flow measurement techniques such as Doppler ultrasound or Phase-Contrast MRI have improved to a point where reliable partial information about the flow field is available. However, concomitant accuracy in measuring boundary conditions for the flow model is not as easily obtained. Pulsatile flow rate and axial velocities may be measured with reasonable accuracy and reproducibility, but the secondary velocities (the in-plane, swirling flows) are often an order of magnitude smaller than the axial flow and are either inaccessible to the measurement tools or contain prohibitively large uncertainties.
Computational fluid dynamics (CFD) is presently being used as a means of enhancing our understanding of fluid flow in arteries and the associated wall forces (Section 2). Utilizing available techniques, many researchers have conducted two- and three-dimensional calculations. While early CFD simulations were based on idealized vascular geometries and boundary conditions, in recent years attention has turned to realistic arterial models with \textit{in vitro} boundary conditions. The resulting complexity in the computational domain arises from bends, bifurcations, and excessive length-to-diameter ratios found in such models. Realistic geometries tend to exhibit a degree of non-planarity or curvature and these geometric features are known to strongly influence the flow patterns, especially for convection-dominated flows. The first step in any fluid simulation is to develop a regular mesh to cover this complicated physical domain. These computational meshes are typically body-fitted, preferably unstructured, three-dimensional and need to be sufficiently fine to capture the relevant flow details. Secondly, the solution methods must be time-dependent owing to the periodic (pulsatile) nature of the flow and stable over time. The timescales associated with recirculation phenomena are much smaller than that of the overall heart cycle. Thus, adoption of small time steps is needed to ensure the stability and accuracy of the simulation. The time-step size is limited even further when a moving grid is employed to track the evolution of the compliant arterial walls.

There have been several recent numerical studies reported to understand steady and pulsatile flow in the context of vascular hemodynamics. A variety of numerical techniques have been used. However, the finite element method (FEM) and the finite volume method (FVM) have emerged as the most common ones. The present study reports results obtained using FEM and FVM codes developed by the authors for the simulation of pulsatile flow in realistic arterial bifurcations.

Biomedical flow problems are typically very complex, due to intricate geometries, moving walls and the high accuracy required to completely cover the occurring flow phenomena. In this context a significant amount of effort is being devoted to parallelization and in the near future all competitive solvers applied to physiological flow problems will have to be parallel. Considering this, we parallelized both our FEM and FVM solvers, using two appropriate state-of-the-art methods.

The present article offers several contributions. We describe novel implementations of the well-known and widely used FEM and FVM methods, but using relatively new or uncommon derivations such as the smoothing pressure correction [2] or Schur complement [3]. Adapted parallelization schemes are applied to both solvers. These are comprehensively validated against published experimental data. An anatomical bifurcation problem is then considered in
great detail and used to compare the two solvers extensively, with respect to accuracy and computational efficiency. The main objective is to gain experience in the use of CFD tools for biomedical flow simulations and evaluate available options without the bias of commercial software optimization.

2. LITERATURE REVIEW

Previous work that uses CFD methods for modeling blood flow in arteries is first reviewed. Past literature includes non-Newtonian blood flow in a two-dimensional aortic bifurcation (Lou and Yang [4]), unsteady blood flow in stenosed arteries (Tu and Deville [5], Lee [6], Gay and Zhang [7]), blood flow in three-dimensional modeled geometries (Tsui and Lu [8], Hunt [9]), blood flow in three-dimensional complex geometries (Hofer et al. [10]; Santamarina et al. [11]; Weydahl and Moore [12]; Lee et al. [13]; Lu et al. [14]) including multi-branch anatomy (Shipkowitz et al. [15]) and validation of flow simulations in a realistic artery against experimental data (Friedman and Ding [16]; Perktold et al.[17]). Issues such as non-Newtonian fluid behavior, wall movement and deformation, and baseline flow patterns have been investigated. A topic not covered in the literature is the long term effect of time-dependent fluid loading – shear and pressure - on wall weakening and permanent deformation. Such a study would require a carefully computed flow pattern with fully validated tools. However, solvers that compute three dimensional flow patterns in complex geometries and oscillatory flow conditions have not been rigorously examined from this view point.

Numerically, Taylor et al. [18-19] employed FEM for modeling three-dimensional pulsatile flows in the abdominal aorta. Steady flow in a model abdominal aorta system including seven branches (celiac, superior mesenteric, two renal arteries, inferior mesenteric and two iliac arteries) was studied numerically with a pressure-based FVM by Lee and Chen [20]. The averaged inlet Reynolds numbers were taken as 702 and 1424, based on the diameter. The baseline flow fields were obtained and the flow behavior was discussed from the viewpoint of fluid dynamics.

Perktold et al. [21] also used FEM to compute pulsatile flow of a Newtonian fluid in a model of a carotid artery bifurcation. Using a rigid wall approximation, the calculations gave access to a detailed representation of the velocity fields, pressure and wall shear stress. They also examined the effect of the bifurcation angle on hemodynamic conditions. In an investigation related to the effect of wall compliance on pulsatile flow in the carotid artery bifurcation, Perktold and Rappitsch [22] described the application of FEM to a weakly coupled fluid–structure interaction for solving the equations of fluid flow and vessel mechanics. The
fluid–structure interaction problem was investigated from a mathematical point of view by Formaggia et al. [23].

Milner et al. [24] also used FEM to perform a CFD simulation of the blood flow pattern in the carotid bifurcation, however, they used MRI data of two normal human subjects to provide the boundary conditions (geometry and flow rates). This study showed that conventional, time-averaged carotid bifurcation models adequately represent interesting hemodynamic features observed in realistic models derived from non-invasive imaging of normal human subjects. Moreover, inter-subject variations in the \textit{in vivo} wall shear stress patterns support the hypothesis that conclusive evidence regarding the role of flow parameters in the appearance of a vascular disease can be derived from individual studies.

Using the same imaging and simulation framework, Steinman et al. [25] reported image based simulation of flow of a Newtonian fluid in a giant, anatomically realistic human intracranial aneurysm with rigid walls. CFD analysis revealed high-speed flow entering the aneurysm at the proximal and distal ends of the neck, promoting the formation of both persistent and transient vortices within the aneurysm sac. Such vortices produced dynamic patterns of elevated and oscillatory wall shear stresses distal to the neck and along the sidewalls of the aneurysm.

Bathe and Kamm [26] conducted a finite element analysis to examine fluid-structure interaction of pulsatile flow through a compliant stenotic artery. The authors used ADINA, a commercial software package to develop an axisymmetric model of the flow field and the vessel geometry. With increasing degree of stenosis, an increase in the pressure drop and wall shear stress associated was seen.

In a similar investigation, Tang et al. [27] considered fluid-structure interactions of steady flow through an axisymmetric stenotic vessel using ADINA. The authors observed complex flow patterns and high shear stresses at the throat of the stenosis as well as compressive stresses inside the tube. Qiu and Tarbell [28] used FIDAP to study pulsatile flow in a compliant curved tube model of a coronary artery. In addition to the wall shear stress, the phase difference between circumferential strain in the artery wall and the wall shear stress was found to be important in locating possible coronary atherosclerosis.

Even though CFD is now established as a useful research tool for the understanding of cardiovascular diseases, it has yet to find its way to daily patient care. The main reasons for this lack of appropriate clinical tools are the often very lengthy computational times - which can even reach several months for fluid structure interaction problems in complex geometries - and the need for a dedicated specialist to set up and perform the simulations. To overcome these problems McGregor et al. [29-30] have proposed a fast alternative to CFD using proper
orthogonal decomposition. This requires a large amount of pre-computed simulations which could, for example, be generated using the methods proposed in this article.

A review of the literature shows two clear trends. Firstly, authors reporting original research on arterial flows have adopted simple geometries and well-defined boundary conditions. Secondly, for complex geometries and truly oscillatory flow, commercial packages such as Fluent, CFX, FLOW3D, COMSOL and ADINA have been utilized. In the absence of analytical solutions, the correctness of the flow solver itself has not been subject to scrutiny. To address this concern, two efficient computer codes have been developed in the present study for the simulation of three dimensional oscillatory flows in an arterial bifurcation. Codes based on the finite element and finite volume methods have been written to demonstrate their applicability to vascular hemodynamics. The formulations needed to arrive at computationally efficient packages are described. Both solvers are validated against published results including flow in a driven three dimensional cavity (straight and skewed). The two solvers are extensively compared in terms of the flow patterns in a real arterial bifurcation. Steady as well as oscillatory flow calculations are performed using the FVM and FEM solvers. Among the objectives, the ability of the solvers to capture short duration recirculation patterns in oscillatory flow with equal resolution is of great interest.

3. MATHEMATICAL FORMULATION

Numerical simulation of arterial hemodynamics is challenging because of the irregular three-dimensional nature of its geometry, flow sub-division, unsteadiness in flow and the necessity of accommodating wall motion if wall compliance is taken into account. Therefore, a general approach to solve the full Navier-Stokes equations governing fluid motion in arbitrary geometries is necessary. In the present work, numerical simulation of the flow equations using two techniques, namely the finite volume method (FVM) adopting the new smoothing pressure correction [2] and the finite element method (FEM) using the Schur complement method [3] are reported. Details of numerical implementation of these methods are presented below.

3.1 Governing equations

Flow of blood through an artery can be considered as time-dependent, three-dimensional, incompressible, laminar but vortical. Since the aorta has a large diameter and fast flow, the corresponding peak Reynolds numbers are large, and viscous effects are, on an average, less significant than inertial ones. For this reason, modifications in blood viscosity owing to non-
Newtonian effects are neglected in the present work. The formulation presented is, however, quite general.

Full modeling of flow in an artery requires the conservation principles of mass and momentum to be simultaneously satisfied. Expressed in compact tensor notation, the basic forms of the continuity and momentum equations in a fixed grid system are:

\[
\rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) - \eta \nabla^2 u + \nabla p = 0
\]

\[
\nabla \cdot u = 0
\]

Here \( u = u(x, y, z, t) \) denotes the velocity vector and \( p = p(x, y, z, t) \) the pressure; symbols used are \( \eta \) for the dynamic viscosity and \( \rho \) for density, each taken to be a constant parameter, depending on the choice of the fluid. Forcing terms may be added to the right side of the momentum equation to account for gravity as well as rigid body acceleration of the object as a whole.

Boundary conditions correspond to no-slip on all solid walls and a prescribed velocity variation at the inflow plane. The gradient condition for velocity is applied on the outflow plane even when the flow is oscillatory. In the finite volume method, pressure is set to zero on the outflow plane. In the finite element method, pressure is set to zero on the outflow plane in the momentum equation while pressure is implicitly determined from the constraint of mass balance. The validation problem considered is simpler in the sense that no-slip conditions are applicable on all bounding surfaces. Initial conditions are taken as a quiescent state; their selection is not important for the present work since the focus is on long term unsteadiness or simply the steady state.

The computational algorithm described below was implemented for unstructured three dimensional tetrahedral meshes. Grid generation was accomplished using the ICEM-CFD package. Grid quality was ascertained using indicators available in the software. Elements employed had an aspect ratio (i.e. the ratio of the maximum to the minimum dimensions) close to unity in all simulations.

3.2 Finite Element Method (FEM)

3.2.1 Discretization

In the finite element method for the solution of the governing equations a standard Galerkin finite element procedure [31] is used. A notable difference compared to the FVM method described in the next section is that the mass and momentum equations are simultaneously
solved by constructing a single global stiffness matrix. In view of the large Reynolds numbers involved, the calculation needs to be stabilized using techniques similar to upwinding, i.e., artificially strengthening the diagonal dominance. In the present work the streamline upwind method (SUPG) discussed in [32] and anisotropic diffusion were considered for stabilization. The latter was found to be a much more effective option and was implemented by solving the momentum equation in the form

\[ \rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) - \eta \nabla^2 u - \nabla \cdot [D] \nabla u + \nabla p = 0 \]

The time derivative is discretized using the backward Euler method, producing a nonlinear system of algebraic equations to be solved at each time-step. Picard iteration is applied, leading to a linear algebraic system to be solved within each time step.

Isotropic diffusion forces Peclet or Reynolds number to stay below the stability limit but can smooth out secondary flows. Anisotropic diffusion, while still perturbing the solution, does it only in the streamline direction, thus reducing smearing of the solution in the cross-stream direction. The anisotropic diffusion model was implemented in the present study by representing the artificial diffusion operator (in two dimensions) as

\[ \nabla \cdot [D] \nabla = \left[ \begin{array}{ccc} \frac{\partial}{\partial x} D_{11} & \frac{\partial}{\partial y} D_{12} & \frac{\partial}{\partial x} D_{12} \\ \frac{\partial}{\partial x} D_{21} & \frac{\partial}{\partial y} D_{22} & \frac{\partial}{\partial x} D_{22} \\ \frac{\partial}{\partial x} D_{21} & \frac{\partial}{\partial y} D_{22} & \frac{\partial}{\partial x} D_{22} \end{array} \right] \]

with

\[ [D] = D_{ij} = \alpha h \frac{u_i u_j}{|u|} \]

Here \( \alpha \) is a tuning parameter, \( h \), the element size and the velocity components are indicated as \( u_i \). Diffusivity as defined above has only one nonzero eigenvalue \( \delta h |u| \) with the corresponding eigenvector aligned with the flow direction.

### 3.2.2 Solving the linear equation system

Solution of a system of linear equations is probably computationally the most intensive step in the entire calculation. An approach towards parallelization of the linear equation solver in the context of the finite element method is presented below. In principle, it is derived from the domain decomposition approach.

The linear system arising from the Picard iteration has the block form
where \( \phi_u \) and \( \phi_p \) are velocity and pressure solution vectors respectively, and \( b_u \) is the body force. The matrix \( A_u \) represents the discretization of the time-dependent advection-diffusion equation being non-symmetric in the presence of advection and \( B \) is the discrete divergence operator.

Applying iterative techniques, a system such as (1) is solved without explicitly forming the inverse \( K^{-1} \) or factoring \( K \). Hence, less storage and a smaller number of operations may be required than by direct methods. The essential operation is the calculation of the matrix-vector product \( y \leftarrow Kx \) which can easily be parallelized on a distributed memory computer.

The iterative procedures of choice are Krylov subspace methods (such as GMRES), for which the number of iterations required for convergence scales roughly with the square root of the condition number. Since the condition number of \( K \) scales as \( 1/h^2 \) where \( h \) is the mesh size, preconditioning is mandatory for high-resolution. Our implementation uses GMRES and Schur complement preconditioning method, which allows (1) to be solved efficiently and in parallel. The parallelization is achieved by using a mesh partitioning approach implemented using PETSc. The full details of this implementation can be found in [33] and will not be repeated here.

### 3.3 Finite Volume Method (FVM)

The method is based on the finite volume principle of applying conservation laws locally to control volumes. It lends itself to easy physical interpretation in terms of fluxes, source terms and the satisfaction of local conservation principles. In the finite volume method, the governing equation is first integrated over a cell-volume \( V_{\alpha} \) and an integral form of the governing equation is obtained. The integral is replaced by summation and the required algebraic equation for the cell quantity is derived. In the finite volume method derived on an unstructured grid, the mass and momentum equations are decoupled so that the velocity components and pressure are determined from explicit equations (Date [2]). This step is to be contrasted with FEM described in Section 3.2 where a single global stiffness matrix is jointly constructed for velocity and pressure. Pressure-velocity decoupling has been presented in [2] as an extension of the SIMPLE algorithm and, except for a few differences in implementation, has been followed in the present study. In this approach, pressure is determined via a pressure-correction equation that indirectly satisfies mass-conservation.

On an unstructured mesh, it is most convenient to employ collocated variables so that
scalar and vector variables are defined at the same location. For such a collocated arrangement, Date [2] has derived an equation for pressure-correction with the aim of eliminating spatial fluctuations in the predicted pressure field. This approach is outlined in the following section.

3.3.1 Discretization

Convective terms of the Navier-Stokes are discretized using a higher-order upwind scheme. Here, cell face fluxes are calculated using an accurate estimation of the left and right states at the cell face. Barth and Jespersen [34] proposed a multidimensional linear reconstruction approach; Frink et al. [35] used this reconstruction route for upwind schemes. Following [34] and [35], higher-order accuracy in upwinding is achieved by expanding the cell-centered solution of each cell face with a Taylor series expansion

\[
\phi(x, y, z) = \phi(x_c, y_c, z_c) + \nabla \phi_c \cdot \Delta r + O(\Delta r^2)
\]

where \( \phi = [u, v, w, p] \), namely any one of the field variables. This formulation requires that the property gradient is known at the cell centers.

The general approach is to apply the mid-point trapezoidal rule to evaluate the surface integral by using the divergence theorem:

\[
\int_{\partial \Omega} \mathbf{n} \cdot \mathbf{F} \, ds = \\
\iiint_{\Omega} \nabla \cdot \mathbf{F} \, dV
\]

over the faces of each tetrahedral cell. Here \( V_{\Omega} \) denotes the volume enclosed by the cell \( \Omega \) and \( \hat{n} \) is the surface normal. Algebraic simplification is carried out by using the geometrical invariant features of triangles and tetrahedral elements. These features are illustrated for an arbitrary tetrahedral cell in Figure 1. The line extending from a cell-vertex through the cell-centroid will always intersect the centroid of the opposing face. Further, the distance from the cell-vertex to the cell-centroid is always three-fourths of that from the vertex to the opposing face. By using these invariants along with the fact that \( \Delta r \) is the distance between the cell centroid and face centroid, the second term in Equation 10 can be evaluated as:

\[
\nabla \phi_c \cdot \Delta r = \frac{\partial \phi_c}{\partial r} \Delta r \approx \left[ \frac{1}{3} (\phi_{n1} + \phi_{n2} + \phi_{n3}) - \phi_{n4} \right] / 4 \Delta r
\]

Thus Equation (10) can be approximated for tetrahedron cells by the simple formula:

\[
\phi_{f1,2,3} = \phi_c + \frac{1}{4} \left[ \frac{1}{3} (\phi_{n1} + \phi_{n2} + \phi_{n3}) - \phi_{n4} \right]
\]

As shown in Figure 1, the subscripts \( n_1, n_2, n_3 \) denote the nodes of face \( f_{1,2,3} \) of the cell \( c \) and \( n_4 \) corresponds to the opposite node.
For determining nodal quantities $\phi_n$, a weighted average of the surrounding cell-centered solution is used. It is assumed in the nodal averaging procedure that the cell centered values are known and their contribution to a node from the surrounding cells is inversely proportional to the distance from the cell centroid to the node. Hence

$$\phi_n = \left( \frac{\sum_{i=1}^{n} \frac{\phi_{i,j}}{r_i}}{\sum_{i=1}^{n} \frac{1}{r_i}} \right)$$

(14)

where

$$r_i = \left[ (x_{c,i} - x_n)^2 + (y_{c,i} - y_n)^2 + (z_{c,i} - z_n)^2 \right]^{1/2}$$

The subscripts $n$ and $c,i$ refer to the node and surrounding cell-centered values respectively. This reconstruction process utilizes information from all of the surrounding cells, thus producing a truly multidimensional higher-order expansion in Equation (13). For boundary nodes, the surrounding face-centered boundary conditions and respective distances are used in Equation (14). The diffusion terms are discretized using a 2nd-order central-difference scheme [2]. The discretized equations are solved using Stabilized Bi-Conjugate Gradient method (BiCGStab) with a diagonal preconditioner.

### 3.3.2 Pressure-correction scheme

The tendency to provoke checkerboard oscillations arising from the pressure–velocity decoupling is due to the use of the collocated storage arrangement, regardless of whether a structured or an unstructured grid method is used. This problem can be circumvented by a new derivation of the pressure correction equation that is appropriate for a non-staggered grid. Date [2] conducted this derivation and showed that the resulting pressure correction equation bears similarities with the staggered grid pressure correction equation. The pressure correction on a collocated grid additionally requires smoothing. Should one ignore this step, the new calculation scheme would be identical to the SIMPLE procedure of Patankar [36]. The new approach significantly simplifies computer coding as well.

The final form of the pressure correction equation can be shown to be as follows:

$$\frac{\partial}{\partial x} \left( \rho u' \frac{\partial p'_m}{\partial x} \right) + \frac{\partial}{\partial y} \left( \rho u' \frac{\partial p'_m}{\partial y} \right) + \frac{\partial}{\partial z} \left( \rho u' \frac{\partial p'_m}{\partial z} \right) =$$

$$\frac{\partial (\rho u')}{\partial x} + \frac{\partial (\rho v')}{\partial y} + \frac{\partial (\rho w')}{\partial z}$$

(15)
where the total pressure correction $p'$ is given by $p' = p'_m + p'_s$ and $p'_s = \gamma (p' - \bar{p}')$ with $\gamma = 0.5$, $p'_m$ being the calculated and $p'_s$ the smoothing pressure correction. Details of discretization of Equation 15 are available in [2] and will not be repeated here.

### 3.3.3 Code parallelization

Porting applications to high performance parallel computers is a challenging task. It is time consuming and costly. Parallel computer architectures traditionally can either be of shared-memory or distributed memory type. For distributed memory architectures, where the memory on remote processors is not directly addressable, inter-processor communication must be implemented through message-passing operations using, for example, the MPI library. For shared memory architectures, thread-level parallelization has often been advocated. One way of achieving thread-parallelization is through the use of the OpenMP programming model.

In the present study, the finite volume code was developed for shared memory architectures, thus simplifying programming by working in a globally accessible address space. The user can supply compiler directives to parallelize the code without explicit data partitioning. Computation is distributed inside a loop based on the index range regardless of data location and the scalability is achieved by taking advantage of the hardware cache coherence.

In the present study, a fork-join execution model is used to develop parallel code. A program written with OpenMP begins execution as a single process, called the master thread. The master thread progresses sequentially until the first parallel construct is encountered, at which point a team of threads is created. The enclosed statements are then executed in parallel by each thread in the team until a work sharing construct is encountered. This step then distributes the workload of a loop among the members of the current team. An implied synchronization occurs at the end of the loop unless otherwise specified. Data sharing of variables is implemented at the start of a parallel or work sharing construct. In addition, reduction operations (such as summation) can be used. Upon completion of the parallel construct, the threads in the team synchronize and only the master thread continues execution. The performance of FEM and FVM codes is discussed in Section 6.1.

### 4. CODE VALIDATION

The computer codes developed for FEM and FVM solvers were validated by first considering the well-known benchmark problem of flow in a lid driven square cavity. The cavity is square and the top lid is given unit velocity to the right. Both velocity vectors and a scalar variable such as temperature are considered. The effect of the anisotropic diffusion parameter $\alpha$ is also
studied. The Reynolds number considered for simulation is 400 while the Prandtl number is 7, corresponding to water as the working fluid. The presented results are obtained using 896K linear tetrahedral elements for the FVM solver and 112K quadratic tetrahedral elements for the FEM solver. The grid near the walls is considerably finer than at the core. For the FVM, the maximum momentum and mass residuals are reduced to less than $10^{-5}$ during the iteration process. The comparison is carried out at steady state. The convergence criterion on the residuals in FEM is also set to $10^{-5}$.

Velocity profiles in the cavity obtained by FEM and FVM were compared with the numerical results of Sheu and Tsai [37] in Figure 2. The centerline velocity profile on the mid-plane shown in Figure 2 demonstrates very good agreement between the three methods. Indeed the two solvers, FEM and FVM, converge to an identical answer, suggesting that the convergence limits set in the calculations are satisfactory. In the context of FVM, similarly good results were achieved with a skewed cavity.

The parameter $\alpha$ used in anisotropic diffusion stabilization plays an important role in FEM calculations. With reference to the equation appearing in Section 3.2, it is clear that the anisotropic diffusion model adds artificial viscosity in the streamline direction. The extent of this effect depends on the scale factor $\alpha$. For $\alpha=0$, solutions equivalent to central differences are obtained. For $\alpha > 0$, the contribution of the stabilization terms increases, the matrix is better conditioned and convergence is achieved in fewer iterations. In FEM, a speed increase of around 25% was seen for $\alpha$ increasing from 0 to 0.5. However, the improvement in speed is at the expense of accuracy. The resulting deviation for up to $\alpha=0.5$ is shown in Figure 3. $\alpha=0.25$ seems to offer a reasonable compromise between speed increase and the degradation in the solution.

Next the two solvers FEM and FVM were tested against each other in terms of the scalar field, namely temperature. The boundary conditions correspond to 353 K on the top moving wall and 300 K on all side walls. The velocity fields calculated by the respective solvers were used in the advection terms of the governing equation. Figure 4 shows the steady state temperature field on various planes of the cavity. Despite major differences in the solution methods, FEM and FVM produce nearly identical results, even when employing entirely different stabilization techniques, namely anisotropic streamline diffusion (FEM) versus upwinding (FVM). Both methods capture all the major features of the temperature field including thermal boundary-layers and an overall circulatory pattern in the cavity. Owing to the movement of the top lid, Figure 4(c) shows symmetry breaking in a horizontal plane with the eddy to the right having a larger size than the one on the left. This feature is well-predicted by both solvers.
5. SIMULATION PARAMETERS

The present study focuses on the application of CFD tools to biomedical flow problems, more specifically to oscillatory flow in bifurcating geometries. To this end, a real-life aortic bifurcation as shown in Figure 5 is used. This geometry was generated using MRI data, as described by McGregor et al. [38]. These were acquired with a Philips Achieva 1.5T device, using a T1FFE sequence (proprietary to Philips) which provided bright blood anatomy images of the lower abdomen of a healthy 35 year old male volunteer. The slices are taken perpendicularly to the infrarenal abdominal aorta. The images are 224 × 207 pixels in size for 20 slices, with a slice thickness of 5mm and an in-plane pixel spacing of 1.29 mm. The arterial lumen was segmented from the anatomy images and smoothed, so as to produce an initial surface mesh. This was then used as input to a novel meshing algorithm [39] resulting in a high quality tetrahedral mesh with refinement close to the walls.

The FEM solver requires quadratic tetrahedral elements, while for FVM the elements are linear. For comparison of the solutions obtained by the two methods, we require the total number of nodes in each calculation to be equal, at least as a first step. Based on this consideration, it was found that for the L mesh (see table 1), 1’199’728 linear tetrahedral elements in the FVM solver were equivalent to 149’966 quadratic elements for FEM, with the node distribution over the physical domain being preserved. To develop these meshes, 149’966 linear elements are first generated in ICEM-CFD software. Linear elements are subsequently converted to quadratic elements and sent to the FEM solver. The quadratic elements are now converted to linear by retaining vertex nodes and sent to the FVM solver. As discussed in Section 6.1, equality in the number of nodes is a necessary but not sufficient condition; the resulting matrix structures from the two methods are fundamentally different and result in additional differences in the numerical solution.

The time step in oscillatory flow calculations was taken to be 1% of cardiac cycle.

Figure 5 shows the three-dimensional surface rendering of an aortic bifurcation. It is clear from the figure that the anatomy has several complexities in its shape, such as a convex out-of-plane curvature towards the anterior abdominal wall in the aorta. This curvature can considerably alter flow patterns in the arteries. Figure 6 shows the mesh distribution on the inlet plane of the aorta, the overall flow direction being vertically downwards. As seen in Figure 6, the mesh is fine near the wall and the element size increases smoothly away from it. This arrangement is necessary, since flow recirculation is initiated at the solid wall. Numerical experiments show that an appropriate mesh generation is essential to resolve the details of the
flow field on the one hand and to deliver faster convergence on the other. The mesh shown in Figure 6 is the L mesh for FVM as mentioned above.

The inflow velocity profile (at the top boundary) is taken to be parabolic in shape in both steady and unsteady calculations. The outflow condition is of the gradient type. Rigid and no-slip boundary conditions are imposed at the vessel walls. Blood is modeled as a Newtonian fluid with a constant density ($\rho = 1020 \text{ kg/m}^3$) and viscosity ($\mu = 3 \times 10^{-3} \text{ kg/m-s}$).

The unsteady calculation is initiated by specifying the inlet velocity as a sinusoidal function mimicking the physiological heart cycle

$$U_f = 0.5 \sin(2\pi \omega t) + 0.3$$

where the frequency ($\omega$) in dimensional form is 1.333 Hz, corresponding to a heart rate of 80 beats per minute. Figure 7 shows the prescribed axial velocity variation at every point on the inflow plane within one cycle of the flow. The unsteady calculation is conducted for four cardiac cycles of 0.75 seconds each. The results corresponding to the fourth cycle are considered independent of the initial conditions and used for subsequent analysis.

For the parameters considered, the Reynolds number is in the range of a few thousand, reaching 6000 at the peak of oscillatory flow. The overall length-to-diameter ratio is 15 for the branches. Mesh independence was evaluated on small (S), medium (M), and large (L) meshes (see Table 1). FEM calculations are performed using cells on which velocity is interpolated as a quadratic function while pressure variation is taken to be linear.

6. RESULTS AND DISCUSSION

Numerical simulation was carried out using both the FEM and FVM solvers described above in an anatomical model of a human aortic bifurcation both for steady (Figures 8-17) and oscillatory flows (Figures 18-23). The results show interesting patterns of flow distribution including recirculation. A comparison of the solutions obtained using FEM and FVM is presented. In each figure, FEM and FVM are compared in terms of various components of velocity, velocity magnitude, pressure, and stream traces. For oscillatory flow, the comparison is carried out at selected phases within a cycle at dynamic steady state. Special attention has been paid on the recirculation patterns that form in the bifurcation.

6.1 Computational Time and Speed-up

The computations were all performed on a 64-bit 8 core (8×Intel® Xeon® 2.66GHz) machine with 32GB shared memory. On the largest mesh studied (the L mesh - see table 1), the FEM code converged in 17.4 hours (real time) to the fifth decimal place in velocity and pressure.
while FVM required 394.6 hours to achieve the same accuracy. Thus, with the chosen parallelization schemes and on the same machine, FEM was seen more than 20 times faster than FVM. This ratio was smaller on coarser meshes. The corresponding CPU times summed over all CPUs were 76.0 and 3’061.5 hours respectively, so the FEM code required 40.3 times less CPU time. The performance of the parallel FVM unstructured code in terms of speedup characteristics on a shared memory machine is about 5 on 8 CPUs for the full code, resulting in an efficiency of 62%. Speedup of BiCGStab algorithm alone is about 6 on 8 CPUs (efficiency of 75%). Comparable numbers for the FEM code parallelized using the Schur complement method are in the range of 6-7 (75-87%), depending on the mesh employed.

Direct comparison of the two methods calls for certain explanations. For the computation times and speed-up data provided above, the L mesh as described in table 1 was used, so the number of nodes was 227’309 and was identical for both the FEM and the FVM simulations. The results are discussed in Sections 6.2 and 6.3. Differences in computational effort could arise because of the order of approximation of the field variables, being quadratic in velocity/linear in pressure for FEM and linear in velocity/constant in pressure for FVM. Accounting for the difference in the order of approximation, the number of cells in the two approaches turned out to be 1’199’728 (FVM) and 149’966 (FEM). The number of unknowns in FVM (three velocity components plus pressure) evaluated at the cell center was 4’798’912 while the number of unknowns evaluated at the vertices of the tetrahedral cells in FEM was 713’969, for more details see table 1. The total number of non-zeroes in the system of algebraic equations was practically identical in the two approaches, being around 24 million, including a reduction in the number of unknowns due to the boundary conditions. Thus, the matrix arising from FEM is smaller and denser as compared to the one from FVM which is larger but sparser. Based on the equality of the number of non-zeroes, one may conclude that the computational effort for the two approaches is comparable.

The second factor likely to be significant is the matrix structure arising in the two formulations. In FVM, two matrices are constructed: one for the three momentum equations and one for the pressure correction equation. Both systems are inverted using the preconditioned BiCGStab algorithm. In FEM, a single matrix is constructed jointly from the mass and the momentum equations, resulting in a large number of zeroes and greater bandwidth. The increase in complexity of the global matrix for FEM is related to the fact that pressure does not appear explicitly as a dependent variable in the mass balance equation. The linear system of FEM is solved using a preconditioned GMRES solver. The inversion effort for FEM is greater, however coupling of the mass and momentum equations make the pressure-velocity iterations, which are so dominant in FVM, obsolete, resulting in a significant overall
computing time advantage. This is an interesting result, especially considering that a large portion of the CFD community use FVM-based codes.

6.2 Steady flow in an anatomical bifurcation

Considerable attention has been given in the literature to determining the local fluid dynamic phenomena that occur in the branches and bifurcations of arterial models. The flow field in these regions is complicated, highly three-dimensional in nature and depends heavily on the geometry of the vessel. The focus of most investigations thus far has been on the secondary flows which are generated within a branch. Qualitatively, the flow in a bifurcation can be described as follows: the flow divides into two streams at the apex. In each stream a new boundary-layer is formed at the inner walls, with a high velocity stream just outside. Owing to the curvature of the streamline at the location of bifurcation the faster moving fluid is thrown inwards, indicating conservation of the angular momentum (the product of velocity and radial distance). At the same time, secondary motion towards the outer wall becomes apparent.

Favorite sites for atherosclerotic events are bifurcations and junctions. Most of these branchings in the cardiovascular system are asymmetric. The potential and actual variety of asymmetric arterial vessel branchings is much larger and the flow in them is considerably more complex, so it is not surprising that symmetric bifurcations have been more extensively studied as compared to asymmetric ones. Even for these, however, only partial results are available, being primarily experimental in origin and for steady flows. A more serious limitation is that most of the work is performed for two-dimensional bifurcations, which excludes certain features of the flow, such as secondary motions and circumferential variations of shear stress. These features may be critical to atherogenesis and plaque growth.

In this section, the steady state solutions for a physiological bifurcation are shown (Figures 10-17). In healthy cases, flow at the bifurcation is predicted to split into two relatively high velocity laminar streams at the central part of the vessel. Near the apex and the proximal side of the right and left branches of the artery, there are regions of slow recirculation. These features are clearly identified by FVM as well as the FEM solvers. Hence, under steady state conditions in the artery, it is clear that the two solvers compare quite well.

Secondary velocities are caused by an upstream curvature in the vessel. When this curvature is planar and the flow is steady, the well known Dean flow pattern develops [40]. Here, flow forms two symmetric counter-rotating vortices and an axial profile skewed to the outer wall of the bend. In generalized models of the aortic bifurcation [41] and particle deposition in the upper airways [42] influences of secondary flow have been found to be significant. Although general secondary flow characteristics have been observed or inferred from in vivo
measurements, the sensitivity of a model prediction to the complexity of the flow features and its implication on the weakening of the artery wall requires sophisticated measurements as well as detailed computations.

Figures 8-11 present contour plots of the three velocity components and the velocity magnitude over selected horizontal (cross-sectional) planes. A fully developed velocity profile is enforced at the inflow plane. On the two outflow planes, the gradient condition is applied. Figures 8-11 show that high velocities are skewed toward the outer side of the artery before the flow reaches apex. Thus, secondary flow develops near to the inner side of the artery. A difference in the mass flow rates between the two branches of the artery is observed. From Figure 11, it is clear that mass flow rate is higher in the left branch as compared to the right one. In both branches, flow shifts towards the left side of the artery, which sustains the secondary flow in the branch.

Figure 12 shows a surface distribution of vorticity over the inner surface of the artery. The magnitude alone is presented. WSS is of particular interest to the medical community as there is believed to be a high correlation between areas of low or time-varying WSS and ones where atherosclerotic plaque typically develops. WSS scales with wall vorticity. The apex of the bifurcation is a region of high shear. Since flow divides unequally between the two branches, shear stresses (and hence wall vorticity) on the left branch is, on average, greater than on the right one. Provided the pulsation frequency is small, the trends seen in steady flow may be realized in oscillatory flow under peak flow conditions. At higher frequencies (characterized by the Strouhal number $\omega d/U$), fluid inertia would play an important role. In the present study, Strouhal number is around 0.08 and we expect steady state results to have some utility in describing the flow field as well as wall vorticity.

Figure 13 clearly indicates the appearance of secondary flow in the main artery as well as near the apex and in the two branches. A Dean-type vortex structure is formed in the main artery ($z = 0.056$), typically seen in flows in curved tubes. The fluid moves from the side-A of the wall to side-B along the diameter, and then returns to the inner wall along the sides of the tube, forming two counter-rotating vortices. As flow approaches the apex, the pair of counter-rotating vortices vanishes and a strong single vortex forms near the inner wall ($z = 0.039$). The flow passes the apex, divides into two streams and the strength of the vortex is further reduced ($z = 0.023$).

Figures 14-17 present contour plots of the three velocity components and velocity magnitude over the vertical plane $y=0.1516$. The asymmetric flow division between the branches is once again quite apparent from these figures.
In summary, steady flow simulation in a physiological bifurcation shows good agreement between the FEM and FEM solvers. The major flow features in both geometries are nicely captured by both methods.

### 6.3 Oscillatory flow in an anatomical bifurcation

Under physiological conditions, the flow has a strong influence on the formation and development of stenosis, atherogenesis and plaque growth. In particular, regions of low velocity produce low WSS and high pressures. These features have an effect of weakening the wall [1]. Thus, a flow feature of significance is the appearance of recirculation patterns in steady as well as oscillatory flows.

As long as the flow moves forward without creating negative velocities, recirculation patterns are absent and no difficulties arise. Vortices formed in the branches move forward slowly, with the pulse wave. However, if negative velocities are created, indicating the presence of flow separation regions, the shear gradient between the forward and backward flow in these regions is high and creates high shear stresses close to low shear stress regions where particles or cells can adhere to the wall.

For the present study, the pulsatile waveform shown in Figure 7 was adopted. It consists of a brief systolic phase (acceleration and deceleration) and a diastolic phase with flow reversal. Traditional Strouhal number analysis of pulsatile flow uses only one time scale. In contrast, the heartbeat can be considered to have two important time scales: a systolic time and the pulse period. During systolic phase, vorticity is pumped into the flow and so the systolic time scale should characterize the size of the vortices in the artery. Figure 7 defines the six different phases of pulsatile behavior of flow through the physiological bifurcation. Phase-a represents the acceleration of systolic phase, phase-b the peak of systolic phase and phase-c is the deceleration of the systolic phase. Phase-d is at the end of the systolic phase and starting of diastolic phase, whereas phase-e corresponds to the peak of the diastolic phase, and phase-f to the end of diastolic and the starting point of the systolic phase. Velocity fields at these phases are reported in the discussion below. It is important to note that the abdominal aortic bifurcation is one of the rare locations in the vascular system where reverse flow occurs. Indeed, the inversed-direction flow seen here at phase-e is by no means a common feature. This is believed to be one of the reasons why this bifurcation is particularly prone to vascular diseases.

Velocities computed using FEM and FVM are presented in three different fashions. First, to give an overall picture of velocity variations during a cycle, six horizontal planes are chosen.
perpendicular to the blood stream direction. Velocity components and velocity magnitude contours at these planes are displayed at the six phases (a-f) within a cycle. Secondly, to show secondary flow patterns in detail, three horizontal planes are considered. Thirdly, the mid-plane \((y = 0.1516)\) is chosen for the vertical component of velocity to demonstrate the effect of the apex of the bifurcation on flow division.

In Figure 18, the two horizontal components of velocity are combined into stream traces and shown as secondary flow on three different planes. These two velocity components are much smaller than the vertical one. Hence, these traces serve to exaggerate the differences between the predictions of FVM and FEM. Considering this factor, the agreement between the two methods is satisfactory, being particularly good during the peak phases of (b) and (e) while showing largest deviations at the zero phases (d) and (f).

An examination of the data of the \(u\) and \(v\) components of velocity clearly indicates that a secondary flow pattern is created during the accelerating phases a-d. The magnitudes of both the components increase near the apex at phase-a and reach a maximum at phase-b, the peak of the systolic phase. Thereafter, both components diminish and reach a minimum during the diastolic phase. The development of the secondary flow during the heart cycle is clearly demonstrated in terms of the stream traces as well. On the first plane \((z = 0.056)\) the FVM solver shows that two counter-rotating vortices are formed during phase-a. Further, they transform into Dean-like vortices when the systolic peak is reached. At the end of systolic phase (phase-d) the pair of vortices shifts more towards the inner side of the main artery. Once again, the formations of Dean vortices is apparent for the peak of the diastolic phase. Near the apex, secondary flow with greater complexity develops, and is seen over the second plane \((z = 0.039)\). During the accelerating systolic phase (phase-a) there is a single vortex formed at the center of the artery which is convected until the peak is reached. During the deceleration of the systolic phase (phase-c), a pair of counter-rotating vortices is formed. At the peak of the diastolic phase, a single vortex is formed and it stays within the artery until the systolic peak is reached again. Beyond the apex (third plane, \(z = 0.023)\), skewness in the flow is visible from the secondary pattern. At phase-a, flow is shifted towards the right branch of the artery. During the diastolic phase, flow starts to shift towards the left branch of the artery.

Figures 19 and 20 present the contour plots of the vertical velocity component \(w\) and velocity magnitude respectively at six time instances within a cycle. Being the main component of velocity, \(w\) gives a better idea about mass flow rate in the artery. At phase-b, the skewness is apparent from Figure 19 and the flow is unequally divided between the two branches. During
the deceleration of the systolic phase (phase-d), there is a reduction in the mass flow rate. Reverse flow is clearly visible during phases e-f. A comparison of the peak of the systolic phase (phase-c) with the peak of the diastolic phase shows the following: for the systolic phase, flow is shifted towards the outer side while during the diastolic phase the shift in flow is towards the inner side. These shifts in flow create a secondary flow pattern within the artery during the zero phase.

Figure 21 shows surface plots of wall vorticity (or equivalently, WSS) along the arterial wall. The wall vorticity distribution at the peak phase exhibits considerable similarity to the steady state distribution shown in Figure 12. The predictions of both FEM and FVM are generally very similar. Patches of high vorticity are seen near the apex during the peak phase. Patches form during the diastolic phase as well (in particular, phase (d)) in the respective branches. Since such patches carry a pressure peak within, Figure 21 indicates periodic pressure pulses in the artery – once during the peak phase (b) at the apex and once over the sides at the zero phase (d). The fluid loading at (b) is considerably higher than at (d) but their implication for the physiological condition of the artery can be quite different [41-42].

This oscillatory nature of the pressure and shear stress loading is interesting to investigate as it plays an important role in atherosclerosis and the dynamics of wall weakening. Indeed, strong positive correlation has been established between areas of large spatial or temporal WSS gradients and positions of intimal thickening and plaque deposition [18]. The associated pressure peak has also been hypothesized to contribute significantly to local weakening of the endothelium [1] and the combination of these two factors can lead to a snowball effect, leading to local development of vascular disease [38].

Figures 22 and 23 respectively present the distribution of the vertical velocity component \( w \) and velocity magnitude on the mid-plane of the artery. Unequal flow division between the branches is clear, a feature not as easily visible in the cross-sectional view (Figures 19-20). At the peak systolic phase (phase-b) a sharp increase in velocity magnitude is seen near the apex of the bifurcation. During the diastolic phase strong flow reversal is clearly present at various locations in the artery. These trends can be seen in the results generated by both solvers.

7. CONCLUSIONS

The implementation of a finite element adopting the Schur complement method [3] and a finite volume solver relying on a novel smoothing pressure correction scheme [2] and their application to investigate flow conditions in an anatomical arterial bifurcation are discussed. Steady as well as oscillatory flows are considered. An unstructured mesh was used in
both simulations. FEM utilized quadratic elements (with linear pressure) while FVM had linear elements for velocity (with constant pressure within a cell). The number of cells in the two approaches was adjusted to keep the total number of non-zero entries in the coefficient matrices approximately equal. Notable differences between the two methods include the following:

a) a simultaneous (coupled) solution of the mass and the momentum equations was implemented in FEM as opposed to the pressure correction (decoupled) strategy in FVM;

b) a variant of GMRES was utilized for FEM while BiCGStab was used for FVM;

c) in FVM, pressure was set to zero on the outflow plane, while in FEM, the composite term containing pressure and the velocity gradient was made equal to zero;

d) FEM employed the anisotropic diffusion method of stabilization while a hybrid upwind procedure was used in FVM.

The two codes were developed entirely by the authors and validated against published results in the literature. The results generated by the two methods matched well in terms of velocity for all flow configurations – steady as well as oscillatory. In steady flow, the flow division between the branches was unequal. Large shear stresses were seen at the apex. In oscillatory flow, the fields of velocity and vorticity resembled steady flow at the peak phases. Recirculation patterns were seen in the artery at the zero phases of the heart cycle. These patterns are revealed as vorticity patches with a pressure extremum contained within. Thus, pressure peaks are expected to act on the walls periodically at these time instances, which correspond to net zero flow.

While the FEM implementation was parallelized using domain decomposition via the Schur complement method, OpenMP on a shared memory machine was used for FVM. While both methods led to over 60% scaling efficiency on up to 8 processors, the coupled FEM approach was found to be faster than the decoupled FVM pressure correction approach by a factor of above 20 on the meshes studied.

This study shows that both the FEM and FVM methods implemented by the authors are capable of simulating intricate flows in complex anatomical geometries and are thus well-suited to biomedical flow problems. Given the selected implementation and parallelization options aiming at optimizing the resulting software package the FEM solver massively outperformed FVM. Whether or not this result can be generalized to other implementations of these schemes might be subject to debate, however, it is highly probable that the overbearing advantage of FEM over FVM observed here does not only depend on the actual implementation, but also on the fundamental properties of these methods, specifically the (de-)coupled treatment of the mass and momentum equations. Therefore we believe that our observations can serve as guidance for future implementation decisions.
The present schemes may serve to expand research development in allied areas of biofluid mechanics. These include biomagnetic micropolar flows [43], cranial blood flows [44], artificial valves [45] and peristaltic non-Newtonian biomagnetic flows [46]. These areas need rapid expansion and our methods can be successfully adapted to study oscillatory phenomena in these contexts.

REFERENCES


34. Barth, T. J. and Jespersen, D.C. The design and application of upwind schemes on unstructured meshes. AIAA paper 1989; 89:0366.


Table 1: Mesh and resulting matrix sizes for the physiological bifurcation model.

<table>
<thead>
<tr>
<th></th>
<th>FVM</th>
<th>FEM</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Name</td>
<td>S</td>
</tr>
<tr>
<td></td>
<td>Cells</td>
<td>88'24'0</td>
</tr>
<tr>
<td></td>
<td>Nodes</td>
<td>16'85'7</td>
</tr>
<tr>
<td></td>
<td>Unknowns</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Non-zeros</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 1: Geometrically invariant feature of a tetrahedral element
Figure 2: Center-line velocity profiles for a square cavity, Re=400, at the plane of symmetry \( z = 0.5 \) computed using FVM and FEM and compared with the literature.

Figure 3: Center-line velocity profiles for a square cavity, Re=400, at the plane of symmetry \( z = 0.5 \) computed using FEM; effect of \( \alpha \), the anisotropic diffusion parameter.
Figure 4: Contour plots of steady state temperature distribution on different planes of a square cavity. (a) Symmetric mid-plane (b) Vertical plane perpendicular to the symmetric mid-plane (c) Horizontal mid-plane parallel to the moving lid. Predictions of FVM (left column) are compared to FEM (right column).
Figure 5: Geometry of the physiological bifurcation model.

Figure 6: (a) Geometric model with boundary conditions (b) Grid description of the physiological bifurcation model at the inflow plane.
Figure 7: Time varying trace of axial velocity prescribed at the inflow plane of the bifurcation model. Points a-f indicate key instances of time at which the flow fields are plotted. Results obtained after the passage of several such cycles are presented.

Figure 8: Physiological bifurcation: Steady state distribution of the $x$ component of velocity ($u$) over selected horizontal planes.
Figure 9: Physiological bifurcation: Steady state distribution of the \( y \) component of velocity \((v)\) over selected horizontal planes.

Figure 10: Physiological bifurcation: Steady state distribution of the \( z \) (vertical) component of velocity \((w)\) over selected horizontal planes.
Figure 11: Physiological bifurcation: Steady state distribution of the magnitude of velocity over selected horizontal planes.

Figure 12: Physiological bifurcation: Steady state distribution of the magnitude of vorticity ($s^{-1}$) over the surface of the artery.
Figure 13: Physiological bifurcation: secondary flow patterns on selected horizontal planes.

Figure 14: Physiological bifurcation: Steady state distribution of the $x$ component of velocity ($u$) over the vertical mid-plane.
Figure 15: Physiological bifurcation: Steady state distribution of the $y$ component of velocity ($v$) over the vertical mid-plane.

Figure 16: Physiological bifurcation: Steady state distribution of the $z$ (vertical) component of velocity ($w$) over the vertical mid-plane.
Figure 17: Physiological bifurcation: Steady state distribution of the magnitude of velocity over the vertical mid-plane.
For caption see next page
For caption see next page
Figure 18: Physiological bifurcation: Secondary flow at three different horizontal planes at six time instances throughout the cardiac phase. The peak phase when flow is in the vertically downward direction is indicated by the time instance (b). The peak phase when flow is vertically upward is (e) while the phases when inflow is zero are (d) and (f).
Figure 19: Physiological bifurcation: Contour plot of the vertical velocity component $w$ over selected horizontal planes at selected phases of the cycle of oscillation. At each time instance, the left column shows FVM, the right FEM results.
Figure 20: Physiological bifurcation: Contour plot of the magnitude of velocity over selected horizontal planes at selected phases of the cycle of...
Figure 21: Physiological bifurcation: Contour plot of the magnitude of vorticity over the wall at selected phases of the cycle of oscillation. The peak phase when flow is in the vertically downward direction is indicated by the time instance (b). The peak phase when flow is vertically upward is (e) while the phases when inflow is zero are (d) and (f).
Figure 22: Physiological bifurcation: Contour plot of the vertical velocity component $w$ over the vertical midplane in oscillatory flow. The peak phase when flow is in the vertically downward direction is indicated by the time instance (b). The peak phase when flow is vertically upward is (e) while the phases when inflow is zero are (d) and (f).
Figure 23: Physiological bifurcation: Contour plot of the velocity magnitude over the vertical midplane at selected phases of the cycle of oscillation. The peak phase when flow is in the vertically downward direction is indicated by the time instance (b). The peak phase when flow is vertically upward is (e) while the phases when inflow is zero are (d) and (f).