Computer Simulation of the Coronary Hemodynamics: Image-Based Generation of the Mesh and Simulation Methodology

B. Quatember [a] **and H. Mühlthaler** [b]

[a] *Department of Computer Science, University of Innsbruck, Austria (bernhard.quatember@uibk.ac.at)*

[b] *Department of Vascular Surgery, University Clinic of Surgery, University of Innsbruck, Austria (h.muehlthaler@archiatros.com)*

Abstract: In the entire field of coronary artery heart disease, cardiologists make extensive use of biplane angiograms for diagnosis and therapy planning. However, they frequently confine themselves to a visual inspection of the images which yields inadequate results, since the images are only two different central projections and not three-dimensional representations of the flow domain. Moreover, these images contain no hemodynamic data. In the companion paper, we describe an advanced method for the three-dimensional reconstruction of (stenosed) coronary arteries based on biplane angiograms. Even this three-dimensional geometric data is insufficient, however, because cardiologists need a fair knowledge of the patient-specific hemodynamics, especially of the hemodynamic relevance of stenoses. Hence, there is a need for patient-specific computer simulation studies of the coronary hemodynamics. This paper describes a system for the simulation of the three-dimensional coronary blood flow based on the finite element method. This system provides components for an image-based generation of the mesh required for these numerical simulation studies. Our aim is to generate an optimal mesh that will allow us to compute the solution with a specified accuracy at minimal cost in terms of computing time. To do this, we must adapt the size of the elements to the flow conditions. Since a consequent adaptive procedure with an *a posteriori* error analysis would be too time intensive, we decided to use *a priori* criteria for the adaptation. Although these criteria are in principle heuristic in nature, they nevertheless reflect a fair, quantitative, *a priori* knowledge relevant to the coronary artery under investigation. This quantitative knowledge is derived from *a posteriori* analyses of computed flow conditions in so-called reference flow domains. Using the above-described techniques, we generated a mesh within the three-dimensionally reconstructed flow domain of a patient's circumflex artery with an eccentric stenosis (relatively low degree of eccentricity). We numerically simulated the patient's hemodynamic conditions around this stenosis for the (nearly) steady-state flow at the end of the diastole with FIDAP 8.52, an advanced commercial CFD software package.

Keywords: Hemodynamics; Biosystems analysis; Mesh generation; Computational fluid dynamics

1. INTRODUCTION

In the entire field of coronary artery heart disease, cardiologists make extensive use of biplane angiograms for diagnosis and therapy planning, but they frequently confine themselves to a visual inspection of the images, which yields inadequate results. Instead, the diagnoses and the planning of therapy should be based on reliable data of the patient's hemodynamic state. The clinical importance of a fair knowledge of the coronary hemodynamics has already been pointed out in the companion paper [Mühlthaler and Quatember, 2001]. Since the range of clinically applicable measurements of the blood flow in the coronary arteries is rather limited, computer simulations of the coronary blood flow have a great significance for the field of coronary artery disease. It has also been shown in the companion paper that the simulation studies must be patient-specific. In carrying out such simulation studies, it is crucial to fully consider the three-dimensional geometry and the dimensions of the epicardial arteries of the individual patient. The acquisition of these geometric data for our simulation studies has been carried out by using biplane angiograms and applying a newly developed three-dimensional reconstruction method which is described in the companion paper [Mühlthaler and Quatember, 2001]. In the present paper, we will give an overview of our simulation methods for coronary hemodynamics and the focus on the simulation of the three-dimensional flow of blood in the epicardial arteries, especially in the flow domain around stenoses. These simulations have been carried out by using the finite element method. We thus need a mesh for our computations. The generation of the mesh is a difficult task, since the flow domain within the epicardial arteries has a highly complex geometry. We decided to use structured meshes and took advantage of the multi-block mesh generation approach [Gatzke, 1999; George, 1996]. The elements of the structured mesh are hexahedra. The generated mesh has anisotropic characteristics [Tam et al., 2000].

We will also present a specific simulation study of the flow of blood around an eccentric stenosis and present patient-specific simulation results.

2. OVERVIEW OF MATHEMATICAL MODELING AND COMPUTER SIMULATION OF THE CORONARY HEMODYNAMICS

The coronary vessels have a highly complex tree-like structure with a huge number of bifurcations; they thus comprise a multitude of segments between adjacent bifurcation points. This complex structure makes it virtually impossible to carry out genuine three-dimensional hemodynamic simulation studies throughout the entire network of the coronary arteries. Hence, we must confine ourselves to simulations that are based on a lumped parameter modelling approach which is of course only a relatively rough approximation. In our investigations of the coronary hemodynamics, three-dimensional simulation studies are restricted to sections of the epicardial arteries with markedly disturbed flow, such as stenosed sections.

In our modelling and simulation approach, blood is treated as an incompressible non-Newtonian (shear thinning) fluid. It can easily be shown that the flow of blood of the entire coronary circulation is a laminar flow.

2.1 Lumped Parameter Modelling Approach

We developed a lumped parameter model of the entire cardiovascular system. In our model, only the subsystem of the coronary circulation has a large number of lumped components, whereas all other parts of the cardiovascular system are modelled in much less detail. In our lumped parameter modelling approach, the network of the coronary vessels comprises:

- a submodel of the epicardial vessels and
- a submodel of the intramyocardial vessels.

Unlike other lumped parameter modelling approaches which provide only linear descriptions of the coronary hemodynamics, our approach takes into account the most important non-linear characteristics of the coronary circulation. The non-linear character of the structural mechanical properties of the coronary vessel walls is expressed in terms of the relationship between the pressure and the cross sectional lumenal area by using distensibility diagrams that can be found in the literature. We also take into consideration the non-linear effect that changes of the cross-sectional lumenal area during a cardiac cycle have on the resistance to flow. For a detailed description of our lumped parameter modelling approach, see [Quatember, 2000].

However, we have to bear in mind that lumped parameter models only allow simulation studies of the global characteristics of the coronary hemodynamics, not of local characteristics such as the irregular flow patterns in regions of disturbed flow around stenoses.

2.2 Distributed Parameter Modelling Approach

A distributed parameter model of the coronary hemodynamics can be written in terms of partial differential equations (continuity equation and Navier-Stokes equation in their three-dimensional form [Truckenbrodt, 1996]). Simulation studies based on distributed parameter models involve solving a system of partial differential equations [Ang and Mazumdar, 1997; Fung, 1997; Lei et al., 1997]. They are thus more difficult and more expensive in terms of computing time than simulation studies based on lumped parameter models. As mentioned earlier, the expense of genuine three-dimensional simulation studies based on distributed parameter models can only be justified for specific sections of the coronary arteries with disturbed flow, since it is only in arterial sections of this kind that the knowledge of the three-dimensional flow pattern is of particular interest to physiologists and clinicians.

3. IMPORTANCE OF THE KNOWLEDGE OF THE THREE-DIMENSIONAL FLOW PATTERN AROUND A STENOSIS

In the field of coronary artery disease, the knowledge of the patient-specific flow patterns in the coronary arteries is important for an assessment of the adverse effects of patient's pathophysiological changes. This is particularly true for the flow conditions around stenoses in the epicardial arteries.

3. 1 Effects of Coronary Hemodynamics on the Processes of Atherosclerosis

Specific hemodynamic conditions such as the disturbed flow at bifurcations and around stenoses may contribute to the formation and the progression of atherosclerotic changes. The localisation of these pathological changes to certain arterial sites is strongly dependent on the three-dimensional flow pattern at these sites. In particular, these adverse processes of atherosclerosis are influenced by the spatial variation of shear stress in the fluid and along the inner surface of the arterial wall. In regions near the arterial wall with low shear stress and long residence times of the monocytes, for instance in recirculation zones, favourable conditions for the adherence of monocytes at the arterial wall exist. These low shear regions are predisposed for the development of atherosclerotic plaques. In such regions of blood flow, genuine three-dimensional simulation studies of the hemodynamics are a prerequisite for the investigations of these pathophysiological processes.

3. 2 Hemodynamic Significance of Severe Stenoses

In regions with disturbed flow around a stenosis, both the values of shear stress within the flow domain and the values of shear stress along the inner vessel wall differ considerably from those of arterial sections under physiological conditions. These deviations from the physiological flow conditions may result in atherosclerotic changes in the arterial wall. In a region with significantly lower levels of shear stress, such as in the recirculation zone downstream from a severe stenosis, specific processes of atherosclerosis may occur, and it has been shown that the local rates of atherosclerosis progression correlate inversely with the magnitude of shear stress along the arterial wall. Hence, the disturbed flow around a stenosis, especially around the apex of a stenosis, not only reduces the blood supply to the myocardium but also accelerates the progression of the atherosclerotic changes. It may also cause the formation and the development of thrombi. Knowledge of the three-dimensional flow pattern is thus a prerequisite for the assessment of the adverse effects of stenoses.

4. COMPUTER SIMULATION OF THE THREE-DIMENSIONAL BLOOD FLOW AROUND A STENOSIS

We solved the governing equations of the coronary hemodynamics in the problem domain within the coronary arteries by using the finite element method.
We employed the commercial CFD (computational fluid dynamics) system FIDAP [FIDAP 8.1, 1998a and b; GAMBIT I, 1998a and b] that comprises:

- two pre-processors (FI-GEN, GAMBIT) for the generation of the mesh,
- specific flow solvers based on the finite element method, and
- a post-processor (FI-POST) for the presentation (visualisation) of the simulation results.

A prerequisite for carrying out this numerical method is the construction of a mesh in the domain where the problem is to be solved, especially in sections of the coronary arteries with stenoses.

4.1 Image-Based Mesh Generation

The generation of a mesh in the flow domain of the arterial section under investigation is an important and challenging task, because the quality of the generated mesh affects:

- the accuracy of the solution,
- the CPU time, and
- the memory requirements.

We use a wire-frame model of the patient-specific flow domain that has been constructed from the patient's biplane angiograms. The construction principles have been described in the companion paper [Mühlthaler and Quatember, 2001]. We employed the aforementioned commercial mesh generator (pre-processor of the commercial CFD system FIDAP) GAMBIT I [GAMBIT I, 1998a and b] for the performance of all meshing operations.

In our image-based meshing approach, we used a structured mesh with hexahedra as elements. We decided to generate a mesh of this kind with anisotropic characteristics [Tam et al., 2000], since such a mesh compensates for the significant directional variation of the solution in our problem domains with disturbed flow.

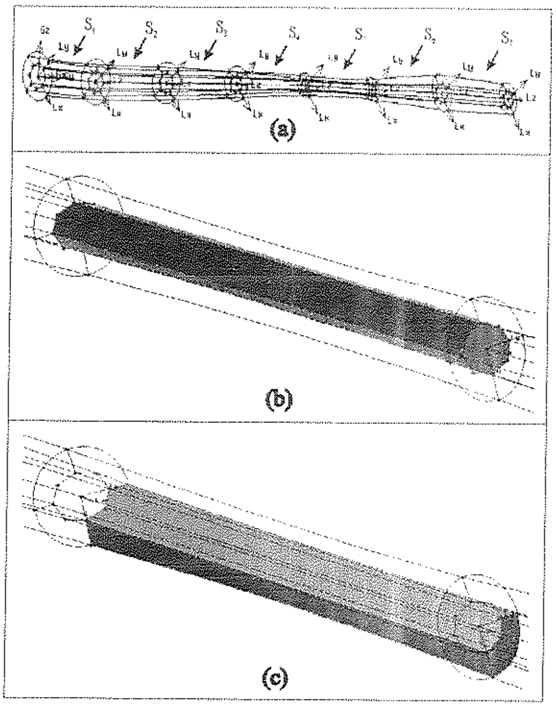

Figure 1. (a) Wire frame model after the subdivision into slices and volumes. (b) Core volume V_{c4}. (c) Peripheral volume V_{p34}.

Moreover, we chose a multi-block meshing method. As depicted in Figure 1(a), the problem domain is subdivided into individual slices $S_i, i=1...M$. Each slice S_i is then subdivided into five volumes, namely in one core volume V_{ci}, which can be seen in Figure 1(b), and in four peripheral volumes $V_{p1i},...,V_{p4i}$ (the peripheral volume V_{p34} is depicted in Figure 1(c)).

In each of these volumes, we generate an anisotropic structured mesh. Such a meshed volume of the problem domain is called a block. The mesh in the entire problem domain is obtained when all these blocks are joined seamlessly. However, this can only be carried out if

- each interface between two adjacent blocks is common to these blocks, and, moreover, if
- each of the M-1 groups of five block interfaces between two contiguous slices have the same topology.

An example of such a group of five block interfaces between two contiguous slices is depicted in Figure 2.

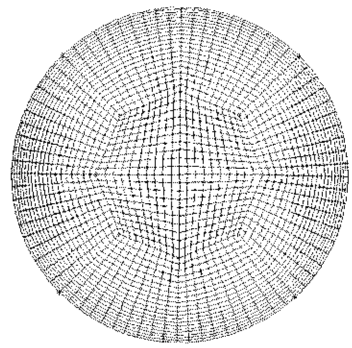

Figure 2. Interface between two contiguous slices (five block interfaces).

Our meshing concept offers enough flexibility to fill the problem domain (flow domain) with appropriately shaped elements, even in the region around the apex of the stenosis. At this stage of development, we will confine ourselves to concentric stenoses and stenosis having a relatively low degree of eccentricity. We aim at the generation of a high-quality mesh that comes close to an optimal mesh. An optimal mesh would allow the computation of a solution that achieves a specified accuracy with a minimal number of elements or degrees of freedom. To achieve this goal, we must adapt the size of the elements to the characteristics of the solution in the entire problem domain.

The construction of an optimal mesh would require a consistently adaptive mesh generation procedure that starts with a numerical calculation using an initial mesh, followed by an error analysis the specification of an improved mesh [Huerta et al., 1999]. This process is then continued iteratively until the optimal mesh is eventually attained. Such an adaptive process using *a posteriori* criteria would be extraordinarily expensive in terms of computing time, since several large-scale finite element computations must be carried out in a loop. The very long computing time required by such a classical adaptive mesh generation method would exceed acceptable time limits in clinical settings. Cardiologists need the simulation results as soon as possible. Therefore, we developed an alternative technique that is especially time efficient. Our adaptive strategy is in principle heuristic in nature, although it relies on the use of relevant quantitative information. It uses *a priori* criteria that are based on a specific quantitative *a* *priori* knowledge derived from analyses of already computed flow conditions in so-called reference flow domains. These domains are axially symmetric and quite similar to the geometry of the stenosed section of the coronary artery (*a posteriori* analyses). This knowledge is therefore relevant to the flow domain under investigation. As reference domains we have chosen concentric stenoses. We use the parameters

ε (degree of stenosis) and
δ (length of stenosis)

to characterise the shape of such a reference domain. We create numerous flow domains by systematically varying the aforementioned parameters. In each reference domain, we generate an extraordinarily fine mesh and compute the flow pattern using the finite element method. As a boundary condition, we assume a volume flow through the section of the artery which is 50% above the typical rate of flow under resting conditions, the highest rate of flow that we will consider. Because we use an extraordinarily fine mesh, we can justifiably assume that the computed solution is the exact solution for the particular reference domain.

In our attempts to create an optimal mesh in each reference flow domain, we aim at an equidistribution of the relative interpolation error and justifiably neglect all other sources of error. Because we already know the (virtually) exact solution in the reference domain, we are able to calculate the sizes of the elements of the optimal mesh, which will, however, vary throughout the entire reference domain. These reference flow domains serve as the basis for the determination of the *a priori* criteria for the stenosed arterial section under investigation. We now select from the totality of the reference flow domains we have created the one whose parameters come closest to the parameters of our stenosed arterial section. The spatial variations of the size of the elements of the reference flow domain selected are then used as *a priori* criteria for the generation of the mesh in the arterial section under investigation. For a detailed description of this calculation see [Quatember and Mühlthaler, 2001]. As mentioned above, we confine our-selves to concentric stenoses and eccentric stenoses with a relatively low degree of eccentricity. In cases of eccentric stenoses, we must generate the mesh in a genuine (not axially symmetric) three-dimensional flow domain. In our heuristic approach, we exploit the above results for the (axially symmetric) reference domains as *a priori* criteria for the mesh generation (the details of this heuristic approach are given in [Quatember and Mühlthaler, 2001b]). Although the above *a priori* criteria for the generation of the mesh are based on quantitative data that are highly relevant to the flow domain under investigation, they are nevertheless heuristic in nature and lack mathematical rigor. However, numerical experiments such as the simulation study described in the next section have proven the serviceability of our method for the generation of high-quality meshes that come close to the optimal mesh.

4.2 Simulation Methodology

As stated above, we employ the commercial CFD (computational fluid dynamics) software system FIDAP. We confine ourselves to the almost steady flow at the end of the diastole and make the simplifying assumption of rigid walls. As mentioned earlier, the flow within the entire network of the coronary vessels is laminar. Moreover, we assume:

- "no slip" conditions at the wall of the stenosed artery,
- "natural" boundary conditions at the outlet (The term "natural" boundary conditions at the outlet means that the normal stress is equal to zero. As the contribution of viscosity to the normal stress is relatively small, "natural" boundary conditions force the pressure to be close to zero at the outlet.),
- a paraboloid velocity profile at the inlet, and
- rigid walls of the stenosed coronary artery.

The finite element analyses of our fluid flow problems produce a wealth of numerical data. This is particularly true in the case of our three-dimensional model of the blood flow through a stenosed coronary artery. However, excessively long lists of numerical data would be very difficult to analyse. For this reason, we usually represent the simulation results as surface plots, contour plots, diagrams and other graphics with the post-processor FI-POST.

4.3 Simulation Results

Using the above-described image-based mesh generation method, we generated a mesh within the three-dimensionally reconstructed flow domain of a patient's circumflex artery with an eccentric stenosis (moderate degree of eccentricity). This three-dimensionally reconstructed arterial section and its mesh can be seen in Figure 3(a). As a complete three-dimensional view of the velocity vector field would no be appropriate in our case of a relatively complex arterial geometry, we created a longitudinal cutting plane (cf. Figure 3) and use this cutting plane for the visualisation of the variation of the absolute value of velocity (speed) as a contour plot, which is depicted in Figure 3(b).

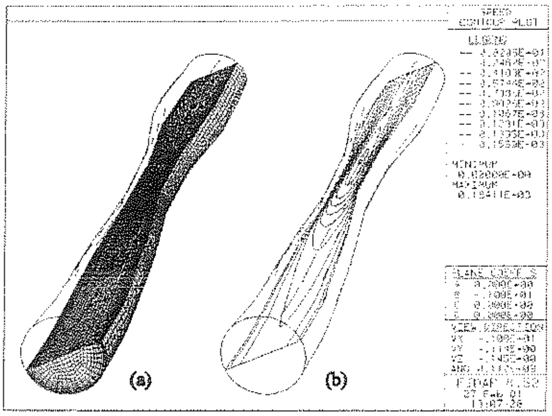

Figure 3. (a) Representation of the generated mesh. (b) Contour plot of the variation of the absolute value of the velocity(speed) on longitudinal cutting plane.

5. CONCLUSIONS

In this paper, we dealt with patient-specific simulation studies of the coronary hemodynamics, especially in the flow domain of stenosed arterial sections. Patient-specific studies of this kind can only be performed with a fair knowledge of the three-dimensional geometry of the coronary arteries.

Our simulations are based on the finite element method and on a three-dimensional reconstruction of the coronary arteries from biplane angiograms using a newly developed method which is described in the companion paper [Mühlthaler and Quatember, 2001]. Our method for the generation of the mesh is so time efficient that we are able to accomplish both the generation of the mesh and the simulation run in a period of time that would be acceptable in clinical settings.

In the future, our simulation methods will be refined. We plan to perform transient studies and will also consider fluid/structure (blood/vessel wall) interactions.

6. REFERENCES

Ang, K.C., and J.N. Mazumdar, Mathematical Modeling of Three-Dimensional Flow through an Asymmetric Arterial Stenosis, *Mathematical and Computer Modeling*, 25(1), 19–29, 1997.

Fung, Y.C., *Biomechanics: Circulation*, Springer Verlag, New York, 1997.

FIDAP 8.1: User's Manual, FLUENT Incorporated, Lebanon NH, 1998a.

FIDAP 8.1: Theory Manual, FLUENT Incorporated, Lebanon NH, 1998b.

GAMBIT I: User's Manual, FLUENT Incorporated, Lebanon NH, 1998a.

GAMBIT I: Modeling Guide, FLUENT Incorporated, Lebanon NH, 1998b.

Gatzke, T., Block-Structured Applications, in J.F. Thompson, B.K. Soni, N.P. Weatherill (Eds.), *Handbook of Grid Generation*, CRC Press, Boca Raton, 13-1–13-24, 1999.

George, P.L., Automatic Mesh Generation and Finite Element Computation, *Handbook of Numerical Analysis*, Vol. IV, 69–190, 1996.

Huerta, A., A. Rodríguez-Ferran, P. Díez, and J. Sarrate, Adaptive finite element strategies based on error assessment, *International Journal of Numerical Methods in Engineering.*, 46, 1803–1818, 1999.

Lei, M., C. Kleinstreuer, and J.P. Archie, Hemodynamic Simulations and Computer-Aided Designs of Graft-Artery Junctions, *ASME Journal of Biomechanical Engineering*, 119, 343–348 1997.

Mühlthaler, H., and B. Quatember, Three-Dimensional Reconstruction of Stenosed Coronary Arteries from Biplane Angiograms and its Importance to Computer Simulation Studies of Blood Flow, in Proceedings of the MODSIM 2001 Congress, Canberra, 2001.

Quatember, B., Simulation of the Cardiovascular System: Lumped Parameter Simulation Model of Coronary Hemodynamics, in Proceedings of the International Conference on Health Sciences Simulation, SCS, San Diego, 139–148, 2000.

Quatember, B., and H. Mühlthaler, Generation of CFD Meshes from Biplane Angiograms: An Example of Image-Based Mesh Generation and Simulation, submitted to *Applied Numerical Mathematics*, 2001.

Tam, A., D. Ait-Ali-Yahia, M.P. Robichaud, M. Moore, V. Kozel, W.G. and Habashi, Anisotropic mesh adaptation for 3D flows on structured and unstructured grids, *Computer Methods in Applied Mechanics and Engineering*, 189, 1205–1230, 2000.

Truckenbrodt, E., *Fluidmechanik, Band 1: Grundlagen und elementare Strömungsvorgänge dichtebeständiger Fluide*, Springer Verlag, 1996.