BLOOD FLOW SIMULATION OF INTRACRANIAL ANEURYSMS

Wolfgang Fenz, Johannes Dirnberger[(a)]

[(a)] RISC Software GmbH, Research Unit Medical-Informatics, Softwarepark 35, 4232 Hagenberg, Austria

[(a)]{wolfgang.fenz | johannes.dirnberger}@risc.uni-linz.ac.at

ABSTRACT

In the course of the research project MEDVIS 3D (www.medvis3d.at), a clinical software application has been developed, capable of 3D reconstruction and visualization of intracranial aneurysms based on 2D medical image data. This system is now being extended with the functionality of a physically correct simulation of blood flow through aneurysms. It shall provide means to estimate rupture risks by calculating the distribution of pressure and shear stresses in the aneurysm, in order to support diagnosis and intervention planning. Due to the time-critical nature of the application, we are forced to use the most efficient state-of-the-art numerical methods and technologies. The elasticity equations for vessel walls and Navier-Stokes equations for blood flow are discretized via the Finite Element Method (FEM), and the resulting linear equation systems are handled by an Algebraic Multigrid (AMG) solver. The boundary conditions of both fluid and wall domains are coupled via Fluid-Structure Interaction (FSI) algorithms. In order to minimize computation time, the simulation will also be parallelized and distributed in the Austrian Grid network. First results using commercial Computational Fluid Dynamics (CFD) packages already show good medical relevance for diagnostic decision support. Our goal is to replace commercial modules step by step by our own implementations in order to end up with a license-free simulation system that is available for every hospital.

Keywords: computational fluid dynamics, flow simulation, fluid structure interaction, 3D visualization

1. INTRODUCTION

The various forms of vascular diseases are complex in their origins and their manifestations. In order to diagnose, prevent and treat such diseases a detailed knowledge of blood flow is essential. Vessels respond to stimuli and adapt to changes in blood flow and blood pressure. Hemodynamic (blood flow mechanic) factors are strongly correlated with the localization of atherioslcerotic plaque.

Atherosclerosis is primarily responsible for strokes, heart attacks and aneurysms, which are among the most frequent causes of death in western countries. In many cases the time of reaction is an important factor in the treatment of these diseases and is crucial for the convalescence of the patient. Due to atherosclerosis it is often necessary to dilate vessels to improve the blood flow. Stents are used there to support the vessel wall after dilation. We see a major application area in the simulation of blood flow in detected aneurysms and stent supported vessels.

The decision of the physician, which stent to use and where to place it, can be supported by the resulting flow patterns. As a future outlook also the stent design and stent optimization can be improved by simulating the hemodynamics. A simulation of the hemodynamics in a 3D geometry based on medical imaging should focus on the following:

- Correct modeling of the vascular system in 3D and an easy to handle tool for the physicians to segment the regions of interest and to insert and design stents and coils in the 3D environment.
- Fast and robust calculation of the hemodynamics in the relevant area.
- Visualization of the resulting flow patterns in a way that it can support the physician's decision without being an expert in mathematics and/or physics.

A decision support tool for the physician can therefore play a major role for recovery. To assure the relation to practice, a close participation of clinical partners during the whole project is intended.

2. METHODS

After having generated a new 3D reconstruction, segmentation and visualization library named REVOLTE (Baumgartner 2006; Thumfart 2007), the main focus of our project is the implementation of a realistic and physically correct simulation model of the blood flow through the visualized vessel geometry.

The geometry information gained by iso-surface extraction will be put into a computer simulation together with blood and tissue parameters, building up a virtual model of the patient's vascular system. Starting with measured blood velocity, a simulation procedure will initiate and calculate pressure and vessel deformation of the whole system at discrete time steps. The results have to be visualized in 3D, in order to quickly provide precious diagnostic information for the assessment of the aneurysm's rupture risk.

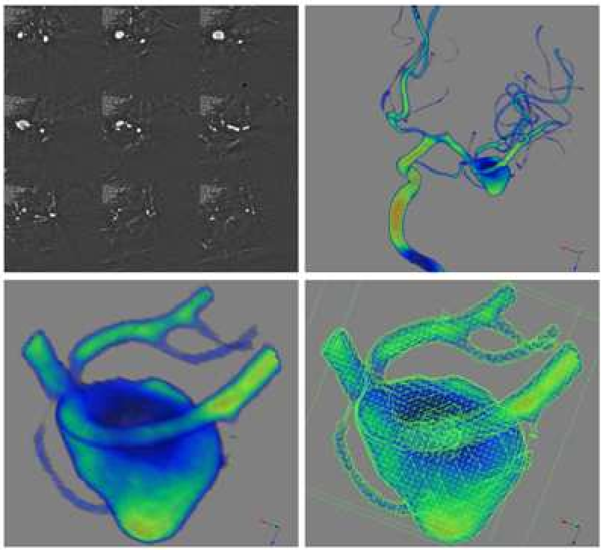


Figure 1: Biplane MRI data, 3D visualization, segmented aneurysm, surface triangulation generated by the skeleton climbing algorithm.

2.1. Mesh Generation

Commonly used methods in medical imaging are Magnetic Resonance Imaging (MRI), X-Ray Angiography and Computer Tomography (CT). All these methods provide series of cross-section pictures of the patient. The software *MEDVIS 3D* is able to reconstruct the 3D volume out of these cross sections and subsequently generate a finite element mesh of a semi-automatically selected region (volume of interest; see figure 1). For surface meshing the skeleton climbing algorithm is used, which is superior to commonly used marching cube schemes, whereas for the creation of tetrahedra we apply Delaunay triangulation.

The quality of both meshes will be further improved by applying area weighted Laplacian smoothers (2D) and iterative mesh improvement (3D).

2.2. Simulation

The simulation of a number of processes in engineering and nature requires the coupled solution of different physical problems. In the present research the fluid-structure interaction and its application to hemodynamics in the human vascular system is taken into account. The interaction between the fluid (blood) and the structure (vessel wall) takes place only at the interface. In large vessels, blood is usually modeled as a Newtonian fluid using Navier-Stokes equations. Due to the fluid pressure and shear stress, forces appear on the structural boundary. The vessel wall displacement due to these forces is small, therefore linear elasticity equations (Navier equations) are used for modeling the vessel wall. The vessel walls are homogeneous and we assume isotropic coefficients for simplicity of presentation. The solution of each subproblem is incorporated into the boundary conditions on the interface for the other subproblem. The fluid forces have to be taken into account within the structural dynamic subtask and the fluid domain has to be updated in a way that it fits to its new boundaries determined by the structural displacements. This interaction is non-linear and an analytical solution of Fluid-Structure-Interaction (FSI) problems is not possible at all in practical cases, therefore the FSI task has to be solved numerically.

Of course, the numerical solution of the coupled problem requires the numerical solution of the fluid and the structural subtasks. Although in the past decades a number of efficient solution methods have been developed for both subtasks, the development of efficient solvers for the FSI problem is still a challenge.

Our targeted method of choice is the stabilized loose coupling (Sieber 2001) which uses an iterative predictor-corrector scheme to guarantee the energy conservation and the equilibrium between the fluid and the structure. It acts as a compromise between fully implicit coupling algorithms that ensure exact energy conservation and mechanical equilibrium at the price of high computational demands and the so-called loose coupling or explicit methods that are faster but numerically unstable. All these coupling methods belong to the group of partitioned approaches, where the fluid and structure problems can be solved separately by specialized and highly efficient Computational Fluid Dynamics (CFD) and Computational Structure Dynamics (CSD) solvers, respectively, as opposed to the monolithic approach where the subproblems are reformulated into a coupled equation system and solved simultaneously.

2.2.1. Solving Subproblems

The vessel wall is described by a linear elasticity equation (Navier equation), which is given by

$$(\lambda + \mu)\nabla(\nabla u) + \mu\Delta u + f = \rho \frac{\partial^2 u}{\partial t^2} \tag{1}$$

where $\mathbf{u}(\mathbf{r},t)$ denotes the structural displacement at coordinate $\mathbf{r}$ and time t, ρ is the density of the wall tissue, λ and μ are the so-called Lamé parameters describing its elastic properties, and $\mathbf{f}$ is an external force. As a first step, the wall is modeled as a shell with displacements only in radial direction, ignoring the viscous shear stress acting on the wall. This can later be replaced by a finite thickness model, which, however, requires the medical images to be of high quality (for example, MRI images do not include information on the vessel walls).

For the fluid domain, we consider the Navier-Stokes and the mass continuity equation,

$$\rho\left[\frac{\partial v}{\partial t}+(v\nabla)v\right]-\mu\Delta v+\nabla p=f, \tag{2}$$
$$\nabla v=0, \tag{3}$$

describing the velocity $\mathbf{v}(\mathbf{r}, t)$ and pressure field $\mathbf{p}(\mathbf{r}, t)$ of an incompressible Newtonian fluid with density ρ and viscosity μ. Both the fluid and structure subtasks are solved numerically by application of the Finite Element Method, which provides a discretization of the corresponding fields for arbitrary geometries, and replaces the partial differential equations with large systems of linear equations of the general form

$$Ax=b, \tag{4}$$

where A is a sparse matrix. Such equations are usually solved by iterative methods, most notably the so-called Krylov Subspace Methods such as Conjugate Gradient. The use of a preconditioner which transforms (4) into an equivalent system that is easier to solve can speed up the calculation considerably.

One quite recently developed method is Algebraic Multigrid described in Stüben (2001). It can be used either as a standalone solver or as a preconditioner, is fast and robust, but it requires the coefficient matrix A to be symmetric and positive definite.

For the Navier equation, this is automatically fulfilled. In the case of the Navier-Stokes equation, however, a splitting algorithm has to be applied first. Using the technique of Haschke and Heinrichs (2001), instead of one non-linear equation coupling velocity and pressure, four uncoupled equations of Helmholtz and Poisson type have to be solved, all of which yield a symmetric and positive definite system matrix after discretization.

As boundary conditions to the simulated system we assume a no-slip condition of the blood on the vessel walls and take the velocity variation of the incoming blood stream during one pulse cycle from measurements performed by our medical partners via Doppler ultrasound.

2.2.2. Current Implementation Status

For the 3D blood flow simulation, work on a FEM Navier-Stokes solver has started. It takes as input an arbitrary unstructured tetrahedral mesh in a simple file format, which can for example be exported by the open source mesh generator NETGEN (Schöberl, 1997), and reads further parameters such as fluid density and viscosity from a separate data file. Boundary conditions for each domain of the mesh surface (as defined in the mesh file) have to be provided too.

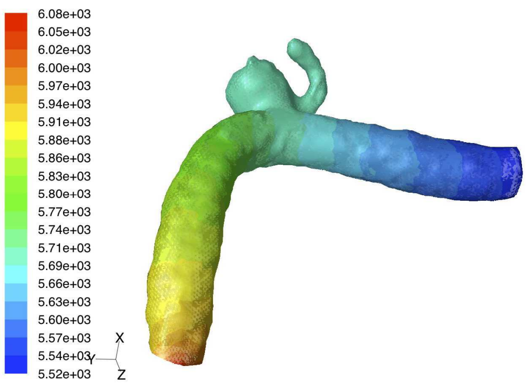
Our code uses Taylor-Hood elements with quadratic shape functions (10 nodes per tetrahedron) for the velocity and linear shape functions (4 nodes per tetrahedron) for the pressure. The unsteady incompressible Navier-Stokes (NS) equations are solved either in their coupled form using Oseen iterations or via the splitting algorithm after Frochte and Heinrichs (2009). In the first case, the coefficient matrix of the system of linear equations obtained via the FEM discretization is asymmetric, and thus one has to use a Biconjugate Gradient method to solve it. In particular, we are applying the so-called BiCGStab algorithm. In the other case, the original partial differential equations are replaced by three Helmholtz equations for the velocity components and a Poisson equation for the pressure. All of them yield symmetric and positive definite coefficient matrices, and therefore the Conjugate Gradient method combined with some preconditioner is applicable. Apart from a simple diagonal preconditioner we are using the Algebraic Multigrid (AMG) method. Our project partner Prof. Gundolf Haase of the Institute for Mathematics and Scientific Computing at the Karl-Franzens-University Graz provided the so-called Parallel Toolbox which includes an AMG routine. We have incorporated a parallel version of it into our solver, but have only used it with one CPU so far. In order to efficiently parallelize the AMG/CG iterations, some partitioning of the mesh has to be provided, which we have not produced yet.

Gradients of pressure and velocity occurring in the Helmholtz and Poisson equations are calculated via the Taylor based gradient recovery technique which was also introduced by Frochte and Heinrichs (2009).

Other tasks that have already been started but are not yet finished are the implementation of Streamline Diffusion in order to improve numerical stability for convection-dominated problems and the integration of mesh smoothing algorithms.

2.3. Visualization

In contrast to the abstract mathematical results of vascular flow simulation the presentation of these results in a medical application should make the figures ready for interpretation and verification tasks. Therefore, the intuitive graphical visualization of flow patterns and turbulences within the reconstructed 3D structures of the patient's morphology plays an important role for the acceptance and deployment of computational simulation systems in medical environments. Together with our medical partners we will develop blood flow visualizations which allow physicians to easily explore a patient's vascular system and to locate parts of enhanced risk. Displaying results in real-time will require fast and flexible strategies in the field of computer graphics, and we have to extensively exploit the capabilities of modern hardware.



ISBN 978-84-692-5417-2

Figure 2: Total pressure of sample data calculated with FLUENT/ABACUS.

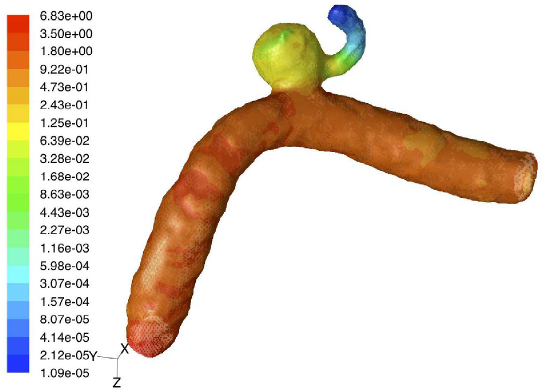


Figure 3: Wall shear stress of sample data calculated with FLUENT/ABACUS.

3. RESULTS AND DISCUSSION

So far, we have calculated the blood flow with our simulation module implemented in C++. Based on geometry data of a cerebral aneurysm measured on a human patient via magnetic resonance imaging (MRI), a 3D mesh was generated and velocity and pressure fields for one pulsatile cycle have been computed.

As a verification basis, on the same geometry velocity, pressure and wall shear stress have been obtained with *Fluent* (CFD) and *Abaqus* (CSD). Moreover, an explicit coupling algorithm providing the Fluid-Structure interaction was implemented by our project partner dTech Steyr GmbH, which will allow further verification of vessel wall displacements due to the blood flow in the future.

3.1.1. Implementation

As a starting point for our implementation, we have mainly studied simple aneurysm model geometries consisting of a cylindrical part representing the blood vessel and a spherical part for the aneurysm. In order to simulate pulsatile inflow, a parabolic velocity profile with periodically varying peak value is used for the boundary condition of the incoming fluid on one end of the cylinder. As outflow condition we set the pressure on the other end equal to zero, while at the vessel walls a no-slip condition is applied. Starting with a fluid at rest, the peak inflow velocity is gradually increased for the first one or two periods, until a stable oscillation is reached. Such a procedure was found to be more efficient than calculating an initial flow configuration with finite inflow velocity with the steady NS equation, especially for higher Reynolds numbers ($Re \geq 100$) where convergence proved to be difficult. We chose the period of the oscillating inflow velocity with 1s, and used time steps of $\Delta t = 0.01$s. Each time step took about one minute of CPU time on a single core of an Intel Core2 Duo Q6700.

We have also compared the results of our solver with calculations performed with *Fluent* (version 6.3) on the same mesh, using the same boundary conditions and parameters. For the results obtained with the coupled NS equations we observed a good agreement, whereas for the splitting algorithm there were some deviations that are probably due to problems with the implementation of the Neumann boundary conditions for the pressure equation. We are still working on that part of our code.

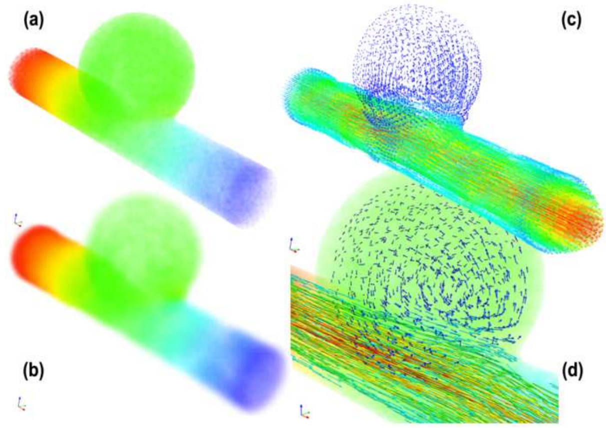


Figure 4: Visualization of pressure [(a), (b)] and velocity field [(c), (d)] for the simplified model geometry, with a peak inflow velocity of 1.5 m/s and Re=100 accomplished with our visualization module.

3.1.2. Benchmarking

For obtaining a verification data set, we have computed the blood flow based on the constant density Navier-Stokes equation in commercial simulation toolkits (*Fluent* and *Abaqus*). The Reynolds number based on the vessel diameter for the intracranial aneurysm presented in figure 5 and figure 6 is about 200. Hence, the blood flow can be considered laminar (e.g. Finol et al. 2003). Nevertheless, at some larger vessel regions the Reynolds number may exceed the critical value of about 2300, which defines the transition between laminar and turbulent flow regimes. Turbulence models based on Reynolds averaging are valid only for fully turbulent flows and hence, not applicable. An alternative consists of using a Large Eddy approach, whereas a huge increase of computational effort has to be accepted. With regard to the project aims we will neglect turbulence considerations.

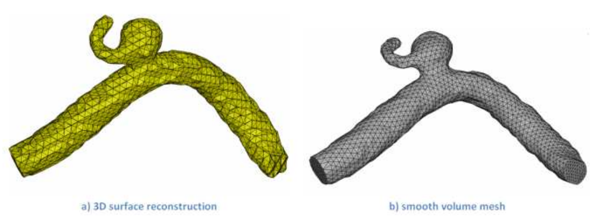


Figure 5: Comparison of 3D surface reconstruction and the resulting flow volume mesh of an intracranial aneurysm

It is common to consider blood as Newtonian fluid (e.g. Finol et al, 2003), Scotti and Finol, 2007)). However, blood is a suspension of red blood cells in plasma, whereas the viscosity of the blood depends on the volume fraction of the red blood cells in plasma. A possible approach to take into account the inhomogeneity of blood is to use the Herschel-Bulkley fluid model (e.g. Valencia et al. 2006). In this work we use the Newtonian material law because of a more stable behavior and the smaller set of parameters compared to the Herschel-Bulkley model.

To incorporate the coupling effects between the blood flow and the vessel wall, a coupling interface between the CFD software Fluent and the FE code Abaqus has been developed. Here, we assume a local equilibrium between the blood flow and the deformation of the vessel wall. From the CFD simulation we obtain the total pressure and shear forces acting on the vessel wall. By using the coupling interface, these forces can be utilized as boundary conditions for the structural analysis. The resulting deformations are passed back to the CFD simulation and applied to the CFD mesh. Furthermore, it may be necessary to remesh the blood flow volume due to the deformations of the vessel wall, which is achieved by a dynamic mesh algorithm.

To stabilize the fluid structure interaction simulation, we use a subcycling algorithm: i.e. a specific number of CFD iterations (Fluent) are executed between two succeeding structural static finite element iterations (Abaqus).

In figure 6 the displacements and the von Mises stresses caused by the total pressure and shear forces exerted by the blood flow onto the vessel wall for the intracranial aneurysm from Figure 5 are shown. It can clearly be seen that the pressure maximum at the aneurysm causes a peak in the stress distribution.

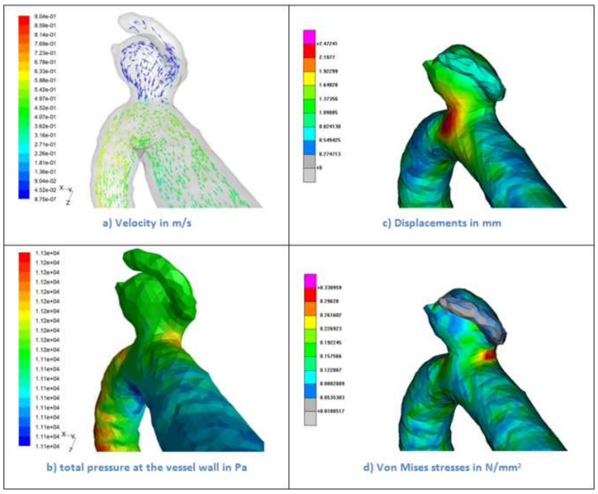


Figure 6: Results of the Fluid Structure Interaction Problem using the example of an intracranial aneurysm.

4. OUTLOOK

As a next step, we are planning to replace the commercial modules by open-source or self-developed libraries.

Moreover, the results obtained in the simulations have to be checked and verified. Forthcoming simulations will be performed on in-vivo measured aneurysm geometries using much larger meshes, and applying as a boundary condition the velocity variation of the incoming blood stream during one pulse cycle, measured on the patient by our medical partners via Doppler Ultrasound.

Besides the comparison to other published results it is planned to compare the calculated flow velocities to the velocities measured by phase contrast MRI or Doppler Ultrasound. Furthermore, we plan to follow the Lagrangian interpolation method introduced by Cheng et al. (2002) to get data about the wall shear stress, the oscillatory wall shear stress and the cyclic strain to compare calculated values with in-vivo measurements. As an outlook, one can also think about simulating stenting, coiling and clipping of aneurysms by a change in the geometry (additional mesh from the stent with appropriate boundary condition to ensure the smoothness and regularity) and changes in the material parameters.

ACKNOWLEDGMENTS

This research project is funded by the Austrian Research Promotion Agency (www.ffg.at) in the frame of the "MODSIM Computational Mathematics" program.

The project board would like to thank Prof. Gundolf Haase (University Graz) and Prof. Heinrich Langer (University Linz) for their valuable input and remarks on our research activities and results.

REFERENCES

Baumgartner S, 2006. Volumenvisualisierung von MR-Daten zur Behandlungsunterstützung bei Schlaganfällen. *Master Thesis*, Institut für Graphische und Parallele Datenverarbeitung, Johannes Kepler University Linz

Cheng CP, Parker D, CAT, 2002. Quantification of wall shear stress in large blood vessels using Lagrangian interpolation functions with cine phase-contrast magnetic resonance imaging. *Ann Biomed Eng*. 30, 1020

Finol EA, Keyhani K and Amon CH, 2003. The Effect of Asymmetry in Abdominal Aortic Aneurysms Under Physiologically Realistic Pulsatile Flow Conditions. *Journal of Biomechanical Engineering*. 125:207-217

Frochte J, Heinrichs W, 2009. A splitting technique of higher order for the Navier-Stokes equations. Journal of Computational and Applied Mathematics. 228:373-390

Frochte J, 2005. Ein Splitting-Algorithmus höherer Ordnung für die Navier-Stokes-Gleichung auf der Basis der Finite-Element-Methode, *Dissertation*, Universität Duisburg-Essen

Haschke H, Heinrichs W, 2001. Splitting techniques with staggered grids for the Navier-Stokes equations in the 2D case. *Journal of Comp. Phys.* 168, 131
Haschke H, 2001. Splitting-Techniken zur spektralen Approximation der Navier-Stokes und Boussinesq-Gleichungen. *PhD thesis*, Universität-GH Essen
Schöberl J, 1997. NETGEN - An advancing front 2D/3D-mesh generator based on abstract rules. *Computing and Visualization in Science.* 1(1), pages 41-52
Scotti CM and Finol EA, 2007. Compliant biomechanics of abdominal aortic aneurysms: A fluid-structure interaction study. *Computers & Structures*. 85:1097-1113
Scotti CM, Shkolnik AD, Muluk SC and Finol EA. 2005. Fluid-structure interaction in abdominal aortic aneurysms: effects of asymmetry and wall thickness. *BioMedical Engineering OnLine*. 4:64
Sieber G, 2001. Numerical Simulation of Fluid-Structure Interaction Using Loose Coupling Methods. *PhD thesis*, TU Darmstadt
Stüben K, 2001. A review of algebraic multigrid. *J. Comput. Appl. Math.* 128, 281
Thumfart S, 2007. Visualisierung und Volumenmessung von Aneurysmen auf Basis von Angiographiedaten. *Master Thesis*, Institut für Graphische und Parallele Datenverarbeitung, Johannes Kepler University Linz
Valencia AA, Guzman AM, Finol EA and Amon CH, 2006. Blood Flow Dynamics in Saccular Aneurysm Models of the Basilar Artery. *Journal of Biomechanical Engineering*. 128:516-526

AUTHORS BIOGRAPHY

Wolfgang Fenz studied Technical Physics at the J. Kepler University of Linz and finished his PhD in 2004. For the following four years he was research assistant at the Institute for Theoretical Physics there, working on different projects dealing with molecular simulation of fluids, liquid state theory and the theory of phase transitions. At the beginning of 2009 he joined the Research Unit Medical-Informatics at the RISC Software GmbH and is now developing the software for blood flow simulation in the project MEDVIS 3D.

Johannes Dirnberger studied Software Engineering at the Polytechnic University of Applied Sciences Hagenberg. After graduation in 2001, he started his scientific career in the research department of the Polytechnic University in Hagenberg. In 2003, he started a research project for assessment of human burn injuries at the Upper Austrian Research GmbH. Since 2008, he works as senior researcher at the RISC Software GmbH and is responsible for project administration and computer visualization in the Research Unit Medical-Informatics.