Validation study of computational fluid dynamics models of hemodynamics in the human aorta

Jana Fuchsberger1,4,*, Elias Karabelas1,2, Philipp Aigner3, Heinrich Schima1, Gundolf Haase1, and Gernot Plank2

1 Institute of Mathematics and Scientific Computing, NAWI Graz, University of Graz, Heinrichstraße 36, A-8010 Graz
2 Gottfried Schatz Research Center, Biophysics, Medical University Graz, Neue Stiftungtalstraße 6IV, A-8010 Graz
3 Center for Medical Physics and Biomedical Eng., Medical University Vienna, Währengürtel 18-20, A-1090 Vienna
4 BioTechMed-Graz, Mozgasse 12/II, A-8010 Graz

Received: 31 May 2019 Accepted: 2 July 2019
DOI: 10.1002/pamm.201900472

We report on conducting a rigorous validation study where we attempt to compare simulated hemodynamics in a mock model of the human aorta with experimental bi-plane particle image velocimetry (PIV) data. Within the cardiac modeling framework CARP (Cardiac Arrhythmia Research Package) [12] we implemented a finite element CFD solver based on the residual-based variational multiscale (RBVMS) discretization of the incompressible Navier-Stokes equations. CFD simulation results compare favorably with experimental PIV data suggesting that our CFD analysis tools are representative of blood flow in the human aorta.

© 2019 The Authors Proceedings in Applied Mathematics & Mechanics published by Wiley-VCH Verlag GmbH & Co. KGaA Weinheim

1 Introduction

Computational fluid dynamics (CFD) models of blood flow in the left ventricle and aorta show high promise as a tool for analyzing the mechanistic links between myocardial deformation and flow patterns. Such models are able to provide additional computed biomarkers at a higher spatio-temporal resolution than currently feasible with clinical imaging modalities. However, before CFD tools can be applied within routine clinical evaluations, their limits of validity and accuracy must be characterized.

2 Experiment

A transparent human aortic block model (root diameter 19.5 mm) was created by a lost-core technique [9] and flow patterns were analyzed with a Particle Image Velocimetry (PIV) system (Dantec Dynamics, Skovlunde, Denmark). A defined inflow section into the aortic model was used to create a parabolic inflow into the model. Three fields of view were recorded, a view of the entire aorta and a close-up of the aortic root with two perpendicular imaging planes. The mock loop was filled with blood mimicking fluid and both stationary and pulsatile flows (Reynolds number from 750 to 8000) were created by a continuous or a volume driven piston pump, respectively. For comparison with CFD in the stationary settings the average of 4 s was taken and in the pulsatile setting the phasic average of 12 beats (100 frames per beat) was calculated.

3 Basic Equations and Discretization Concepts

The underlying equations are the incompressible Navier-Stokes equations. These are considered a sufficiently accurate description of hemodynamics in the aorta as blood in large vessels such as the left ventricle and the aorta complies with the assumptions of an incompressible, isothermal, Newtonian and single-phase liquid [7]. Specifically, we use

\[ \begin{align*}
\partial_t (\rho u_i) + \partial_j (\rho u_i u_j - \sigma_{ij}) &= 0 \quad \text{in } \mathbb{R}^+ \times \Omega, \\
\partial_t u_i &= 0, \quad \text{in } \mathbb{R}^+ \times \Omega, \\
\sigma_{ij} n_j &= h_i, \quad \text{on } \Gamma_n \\
u_i(0, \cdot) &= u_i^0 \quad \text{in } \Omega
\end{align*} \]

where \( \rho \) denotes the fluid density, \( p \) is the pressure, \( u_i \) and \( n_i \) are the components of the velocity field and the unit outward normal vector, respectively. To deal with turbulence occurring at higher Reynolds numbers the residual based variational multiscale (RBVMS) turbulence model, see [3,4,6,8], was employed. The RBVMS formulation has the additional property of stabilizing our method, which allowed the use of lowest equal order finite elements. The main idea behind the RBVMS model is to decompose the finite element functional spaces into coarse and fine scale subspaces in order to model the unresolvable fine scale quantities with element-wise residuals, see [3]. For time discretization the Implicit Euler and Crank-Nicholson schemes were used for the stationary and transient cases, respectively. The remaining non-linear system was treated using the Newton-Raphson method which results in a non-symmetric block system that has to be solved every time step. Solvers for the block system were taken from the PETSc library [1,2]. We used a right preconditioned flexible GMRES method with PETSc fieldsplit preconditioning [5,10] which in turn uses BoomerAMG [11] to approximate sub-block inverses.

* Corresponding author: e-mail: jana.fuchsberger@unigraz.at
4 Simulation Setup

The parameters \((\rho = 1060 \text{ kg m}^{-3}, \mu = 4.0 \times 10^{-4} \text{ Pa s})\) and mesh geometry for the CFD simulation were chosen in agreement with the PIV experiment. For the stationary cases a parabolic inflow profile was chosen as inflow boundary condition. For the transient cases two dimensional inflow profiles were recovered by polar interpolation of the one dimensional velocity data sampled from the two orthogonal PIV imaging planes at the inflow region. We used pure tetrahedral meshes with \(\approx 1200 \times 10^3\) points, \(\approx 7000 \times 10^3\) elements corresponding to an average edge length of \(0.25 \text{ mm}\). This resulted in \(\approx 5000 \times 10^3\) degrees of freedom. For both stationary and transient simulations simulations were ran from \(t = 0\) to \(t = 2\) s with a time step size of \(\Delta t = 1\) ms. Simulation times for all experiments ranged at maximum up to 4 h. All simulations were executed on VSC3 using 512 MPI processes with an average computation time per time step of \(\approx 7\) s.

5 Validation and Results

In order to validate the simulated flow profiles with experimental data, patterns of characteristic vortices were compared, using the scaled Q-criterion \(Q_s := \frac{1}{2} \left( \frac{\| \Omega \|_F}{\| S \|_F} - 1 \right)\), where \(S\) and \(\Omega\) denote the symmetric and antisymmetric part of the velocity gradient tensor, respectively.

For the stationary cases we observe good agreement of vortex positions between PIV measurements and simulation results within the range of a maximum relative error of 22\% (see Tab. 1). Fig. 1 suggests a close agreement between measured and simulated vortex patterns.

![Fig. 1: Comparison of PIV measurements to simulated data for Re = 2000, where v1 and v2 indicate the positions of vortex 1 and vortex 2.](image)

| Re | Error Vortex 1 [%] | Error Vortex 2 [%] |
|----|-------------------|-------------------|
| 750| 2.5               | 10.1              |
| 1500| 10.6              | 21.8              |
| 2000| 17.2              | 2.0               |
| 3000| 4.3               | 6.0               |
| 4000| 14.9              | 12.4              |

Table 1: Relative errors from comparison of vortex positions. Errors have been calculated as \(\frac{2 |x_{\text{PIV}} - x_{\text{SIM}}|}{|x_{\text{PIV}}| + |x_{\text{SIM}}|}\), with \(x_{\text{PIV}}, x_{\text{SIM}}\) being the respective measured and simulated vortex centers.

Due to limited resolution and the reduced dimensionality of measured data the quantitative validation of the transient case remained inconclusive. While computed flow patterns appear plausible and showed qualitative agreement with measurements using an eyeball metric, a full quantitative analysis requires further research.

6 Conclusion and Future Work

In the stationary cases the CFD simulation results compare favorably with experimental PIV data suggesting that our CFD analysis tools are suitable to represent blood flow in the human aorta. The validation of transient flow would benefit from PIV with higher spatio-temporal resolution, two dimensional measurement of inflow profiles or 4D tomographic PIV data.

References

[1] S. Balay, S. Abhyankar, M. F. Adams, J. Brown, P. Brune, K. Buschelman, et al., PETSc Users Manual. Technical Report ANL-95/11 - Revision 3.7, Argonne National Laboratory (2016a).
[2] S. Balay, W. D. Gropp, L. C. McInnes and B. F. Smith, Modern Software Tools in Scientific Computing edited by E. Arge, A. M. Bruaset, and H. P. Langtangen (Boston, MA: Birkhäuser Press), 163–202.
[3] Y. Bazilevs, V. Calo, J. Cottrell, T. Hughes, A. Reali and G. Scovazzi, Comput. Methods Appl. Mech. Eng. 197, 173–201 (2007).
[4] Y. Bazilevs, K. Takizawa and T. E. Tezduyar, Computational Fluid-Structure Interaction: Methods and Applications (John Wiley & Sons, Hoboken, 2013)
[5] H. Elman, V. E., Howle,J. Shardid, R. Shuttleworth and R. Tuminaro, J. Comput. Phys. 227, 1790–1808 (2008).
[6] J. R. Hughes, Comput. Methods Appl. Mech. Eng. 127, 387 - 401 (1995).
[7] W. Nichols, M. O’Rouke, and C. Vlachopoulos, McDonald’s Blood Flow in Arteries (CRC Press, London, 2011)
[8] L. Pauli and M. Behr, Int. J. Numer. Methods Fluids 85, 189–209 (2017).
[9] M. Stoiber, T. Schloglohofer, P. Aigner, C. Grasl, H. Schima, Int J Artif Organs 2, 131-134 (2013).
[10] D. Silvester, H. Elman, D. Kay andA. Wathen, J. Comput. Appl. Math. 128, 261–279 (2001).
[11] H. Van Emden, and U. M. Yang, Appl. Numer. Math. 41, 155–177 (2002).
[12] E. J. Vigmond, M. Hughes, G. Plank, and L. J. Leon, J Electrocardiol. 36, 69-74 (2003).