DOI: https://doi.org/10.14311/TPFM.2025.004

DESIGN AND NUMERICAL SIMULATION OF AN AXIAL BLOOD PUMP

T. Bodnár [1,2], I. Linkeová [1], L. Pirkl [3]

[1] **Faculty of Mechanical Engineering, Czech Technical University in Prague, Karlovo Náměstí 13, 121 35 Prague 2, Czech Republic**

[2] **Institute of Mathematics, Czech Academy of Sciences, Žitná 25, 115 67 Prague 1, Czech Republic**

[3] **CFD support, s.r.o., Sokolovská 270/201, 19000 Prague 9, Czech Republic**

Abstract

This paper shows a first version of a new geometrical and computational model of a Ventricular Assist Device (VAD). The main aim and purpose of this model is to serve as a computational test case and benchmark for mutual cross-comparison of numerical simulations employing different grids, solvers, and additional mathematical model extensions. The simplified VAD was designed as a fully parametric adjustable geometry, allowing for easy change of the essential model shape parameters to assess their effect on the VAD's performance. The description of the newly developed VAD geometry is presented, briefly describing its structure and main shape parameters. The first numerical simulations of the flow in the resulting VAD model are presented, based on the OpenFOAM finite-volume solver implemented in the TCAE Engineering Simulation Environment. The presented first numerical results show the applicability of the newly developed VAD's model geometry for computational fluid dynamics (CFD) evaluation and benchmarking.

Keywords: ventricular assist device, blood pump, parametric 3D model, incompressible Navier-Stokes, OpenFOAM

1 Introduction

The blood flow in human (and not just human) body is driven by the action of the hearth, periodically pumping the blood through the circulatory system. In the case of hearth weakness or malfunction, the blood flow generated is insufficient, leading to severe medical life-threatening problems. One of the possible solutions in such situations is the use of the so called Ventricular Assist Device (VAD). It is a mechanical blood pump that is surgically implemented in the body attached in parallel to the failing hearth, supporting the blood circulation. The blood flow in ventricular assist devices is very complicated, namely due to highly complex shape of both the rotor and the stator of the blood pump. Due to small physical dimensions of the blood pump, the device has to operate at quite high speeds (revolutions per minute, RPM's), leading to complicated flow field structure, characterized by important secondary flows and turbulence. Such flow conditions differ significantly from those in the standard circulatory system, leading to higher stresses acting on blood cells [8], that can be damaged or destroyed causing blood hemolysis [11], which can result in further severe medical complications. This is why one of the major challenges in the design of VADs is their optimization to minimize the blood damage (hemolysis) while maintaining the necessary performance parameters of the blood pump. To design a VADs is a very demanding and expensive challenge [6],[10], leading to number of competitive designs being actually produced and used in medical practice. Despite number of research teams working on VADs design and optimization, most of the used geometries are not disclosed in detail, preventing mutual comparison and assessment of different mathematical models and numerical tools on identical VADs. This led us to the idea to develop and share for public use a simplified model geometry of an axial blood pump, allowing any interested researchers to work on the same well described and fully disclosed case.

2 Geometrical Model

The newly developed parametric model of a simple axial blood pump consists of only few essential parts with simple, easy to describe shapes: straightener, impeller, diffuser, two clutches and shroud. The schematic view of the essential model dimensional shape parameters is shown in Fig. 1.

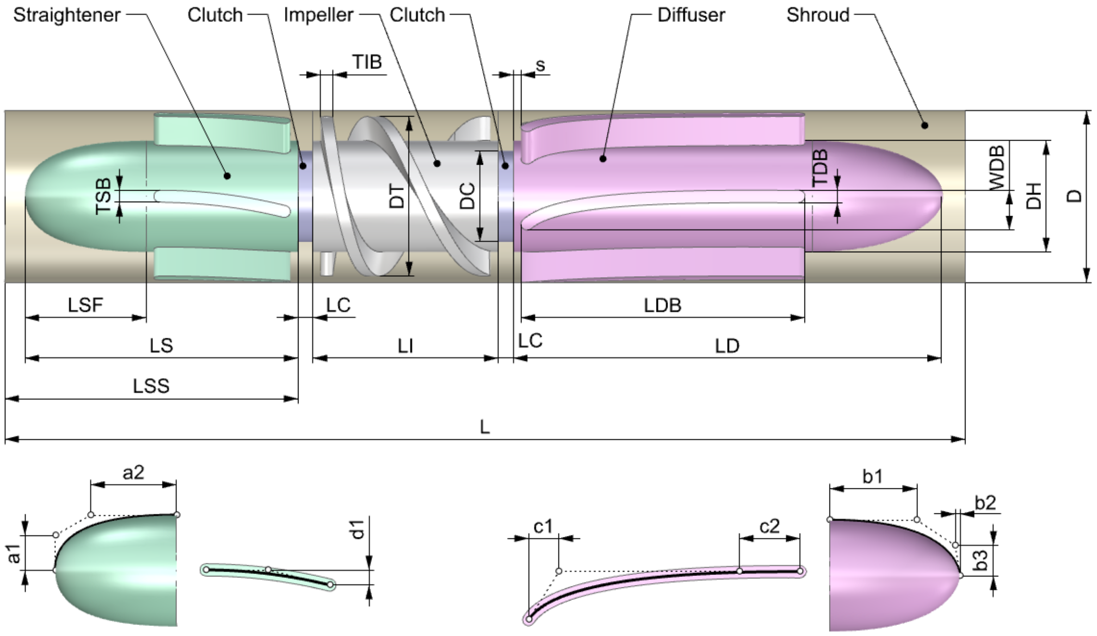


Figure 1: Schematics of the geometrical model parameters.

The description of shape parameters is given in Tab. 1 together with their values used in computational geometry depicted in Fig. 2.

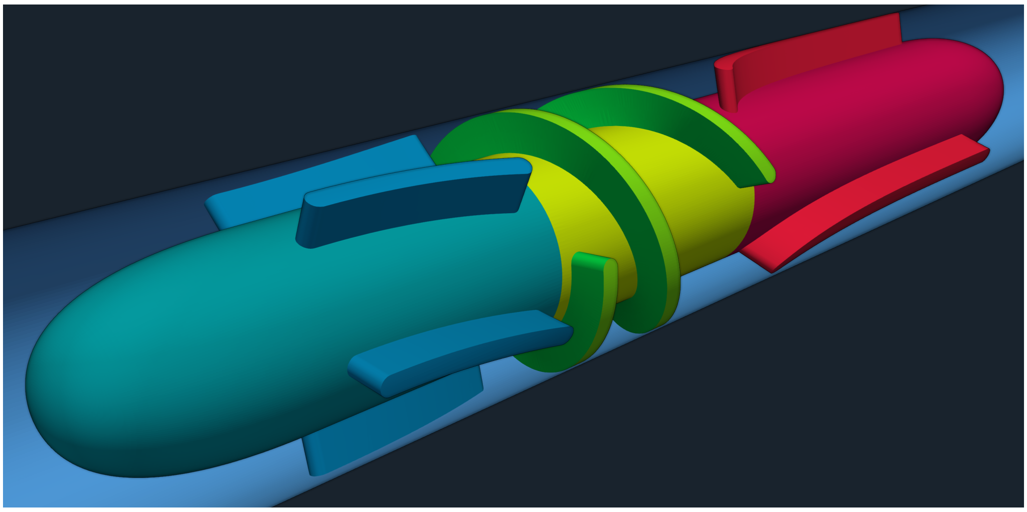


Figure 2: Computational geometry of VAD

The shroud is a cylinder of revolution of diameter D and length L. Cylindrical parts of the straightener, impeller and diffuser hubs have the same diameter DH. The meridians of the rounded parts of the straightener and diffuser hubs are defined by 3rd-degree Bézier curves. The shape of the meridians can be modified by appropriate setting of parameters a_1 and b_3 in vertical direction and a_2, b_1 and b_2 in horizontal direction. The centre line of straightener blade profile is 2nd-degree Bézier curve. Parameter d_1 affects the curvature of the blade as follows: $d_1 < 0$ the blade bends clockwise, $d_1 = 0$ the blade is straight, $d_1 > 0$ the blade bends counter-clockwise. The centre line of diffuser blade profile is 3rd-degree Bézier curve whose shape can be modified by parameters c_1 and c_2. The tip diameter of both the straightener and diffuser blades is equal to the shroud diameter D.

Impeller blade is modelled as helicoidal surface with rectangular profile perpendicular to the

Symbol	Parameter	Dimension
D	Shroud diameter	16 mm
DT	Tip diameter	15.76 mm
DH	Hub diameter	11 mm
DC	Clutch diameter	11 mm
L	Shroud length	380 mm
LSS	Stator shroud length	127 mm
LS	Straightener length	27 mm
LSF	Straightener front length	12 mm
LI	Impeller length	18.3 mm
LD	Diffuser length	40 mm
LC	Clutch length	0.7 mm
n_S	Straightener blades number	5
n_I	Impeller blades number	2
n_D	Diffuser blades number	3
TSB	Straightener blades thickness	1.2 mm
TIB	Impeller blades thickness	1.2 mm
TDB	Diffuser blades thickness	1.25 mm
LDB	Diffuser blades length	28 mm
WDB	Diffuser blades width	4 mm
s	Blades offset	0.75 mm
w	Helix wrap angle	360°
p	Helix shape parameter	1.75
a_1	Straightener meridian definition	3.5 mm
a_2	Straightener meridian definition	8.5 mm
b_1	Diffuser meridian definition	7.5 mm
b_2	Diffuser meridian definition	0 mm
b_3	Diffuser meridian definition	3 mm
c_1	Diffuser blade centre line definition	3 mm
c_2	Diffuser blade centre line definition	6 mm
d_1	Straightener blade centre line definition	-1.5 mm

Table 1: Set of the geometrical model parameters.

directing helix. Helix shape parameter p controls the disproportionality between the translation along the axis of screw motion and angle of rotation about the axis of screw motion. Consequently, the cross-section area of the gap between the blades changes in the following way: $p < 1$ the gap decreases in the flow direction, $p = 1$ the gap is constant and $p > 1$ the gap increases. The tip diameter DI of impeller blades is less than the shroud diameter D.

All blades are rounded at both ends and offset from the edge of the cylindrical hubs by parameter s. The clutches separate the impeller from the straightener and diffuser.

The fully parametric geometry of the axial blood pump was developed in the graphical algorithm editor Grasshopper integrated in 3D computer graphics and CAD (Computer Aided Design) software Rhinoceros. The input is represented by the list of shape parameters given in Tab. 1 stored in a text file. The geometry is automatically generated based on the current parameter settings. Rhinoceros is not only focused on modelling spatial objects based on a mathematically precise NURBS (Non-Uniform B-Spline) representation, but also excels in its robust support for polygon meshes. Therefore, the generated geometry can be exported in NURBS representation (e.g. step, iges, sat format) or as a triangular node-to-node mesh in stl format.

Although the shape is rather simple, it is non-trivial, having most of the important features found in real VADs. The physical dimensions of the proposed model geometry correspond to real VADs, which allows to use realistic operating conditions (such as rotation speed, flow rates, pressure drops) in numerical simulations. This is of critical importance in the case of detailed modeling of turbulence, non-Newtonian rheology [7],[9] or blood hemolysis [3],[4],[5].

3 Numerical Simulation

The numerical simulations presented here show a preliminary results aiming to demonstrate the completeness and computational applicability of the newly developed VAD geometry. The simulation setup is very basic, with number of simplifications and generic choices. An incompressible Newtonian fluid (water) with viscosity $\mu = 1.0 \cdot 10^{-3}\,\mathrm{Pa} \cdot \mathrm{s}$ and density $\rho = 998\,\mathrm{kg} \cdot \mathrm{m}^{-3}$ was assumed. The Reynolds-Averaged Navier-Stokes equations were complemented by the SST $k - \omega$ turbulence model with wall function treatment at the no-slip boundaries and 2% turbulence intensity at the inlet. The grid had about 3×10^6 cells.

3.1 Computational domain and grid

The complete geometry of VAD was exported as a set of individual surfaces, defining the shape of the straightener, impeller and diffuser, but also including artificial surfaces representing the outer casing, computational domain inlet and outlet boundaries and interfaces between moving and static parts. The structure of these individual surfaces is shown in Fig. 3.

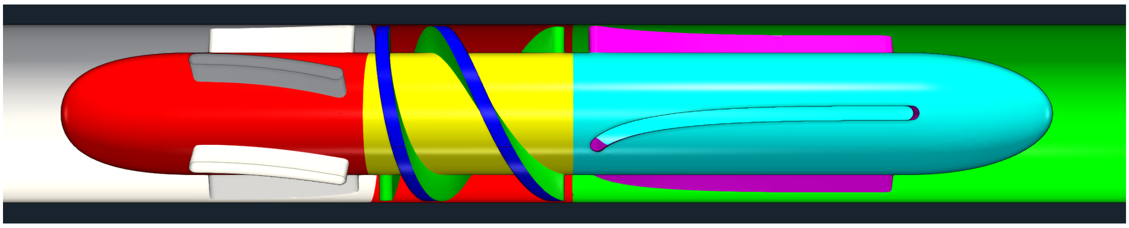


Figure 3: Structure of the surface representation of the VAD.

Based on these shapes and additional surfaces the initial unstructured volume grid is automatically generated, taking into account the physical as well as the artificial surfaces, where the boundary conditions need to be imposed.

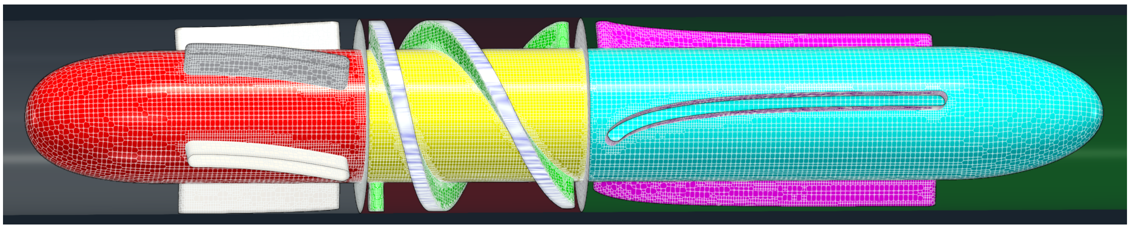


Figure 4: Surface grid for numerical simulations, including the stator/rotor interfaces.

The whole computational domain is represented by a long straight pipe, inside of which the actual VAD is placed for numerical testing. The artificial inlet and outlet boundaries are placed far enough to be able to consider and impose undisturbed and fully developed flow conditions at those boundaries.

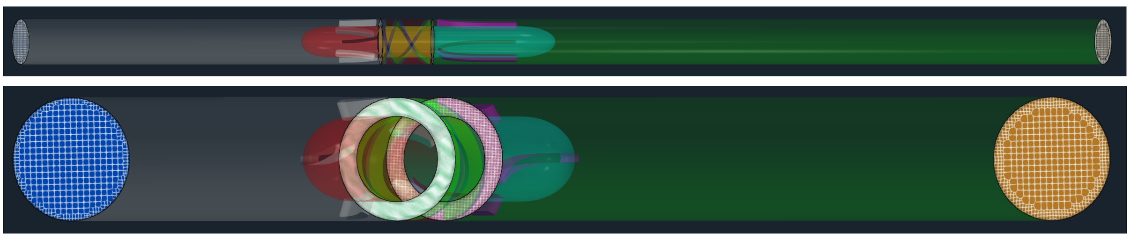


Figure 5: Computational domain with grid structure at inlet and outlet boundaries.

The automatically generated computational grid is refined where needed, depending on the local shape of the nearby surfaces, considering the corners and thin gaps between the impeller and outer casing.

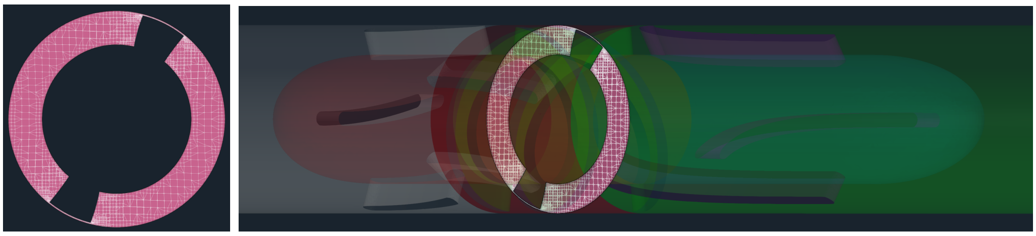


Figure 6: Grid structure in one of the impeler cross-sections.

3.2 Numerical results

The initial set of numerical simulations was performed to check the technical quality of the newly designed geometrical model. The primary main aim was to verify whether the individually generated model surfaces perfectly match and fit despite parameter changes and grid refinements. Secondary aim was to test the complete automated workflow, starting from the model geometry export, through its import and grid generation up to final CFD simulations and their evaluation. This is why only a very limited set of results is presented here, without going into much detail, documenting that the main aims and goals were accomplished.

The simulations were performed in the TCAE Engineering Simulation Environment [2], which uses an OpenFOAM based solver for CFD modeling. The VAD's evaluation setup follows closely the common methodology in performance and efficiency assessment of rotating pumps and turbines. In the presented simulations the fixed rotor speed (RPM=8000) is imposed, while the incoming flow rate varies within pre-specified range (ten points up to approx. $Q = 0.15$ l/s). This allows to evaluate the pump operation parameters under different flow regimes, leading to basic assessment of the pump design performance and optimal operating conditions [1].

Figure 7 shows the overall complex flow field structure. The kinematic pressure increase due to rotating impeller is shown in the first picture, detailing the gradual pressure increase along the spiral, with peaks on the leading edge and on the sharp edge close to outer casing. The thin gap between the rotor spiral blade and the outer casing is also partly responsible for the complex streamline structure shown in the lower part of the figure. The streamlines are colored by the velocity magnitude.

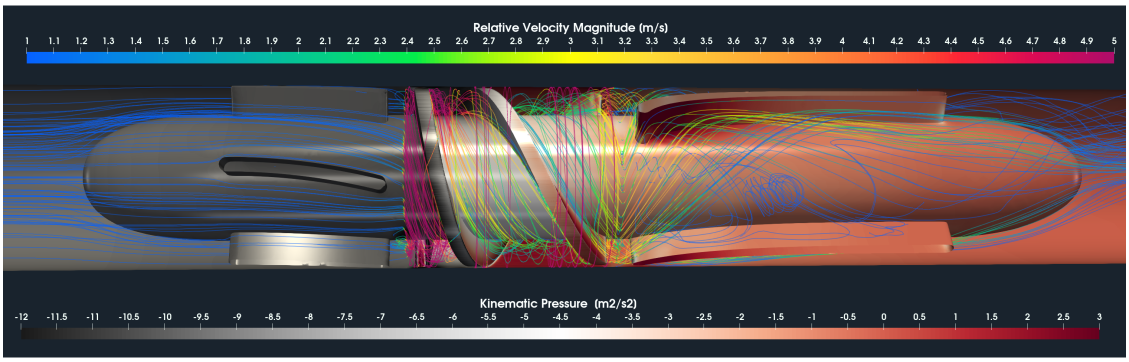
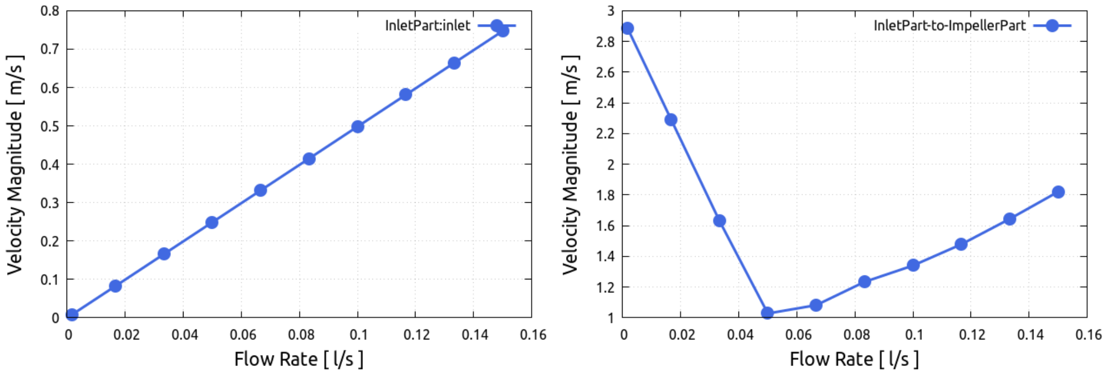


Figure 7: Kinematic pressure and streamlines for one flow regime.

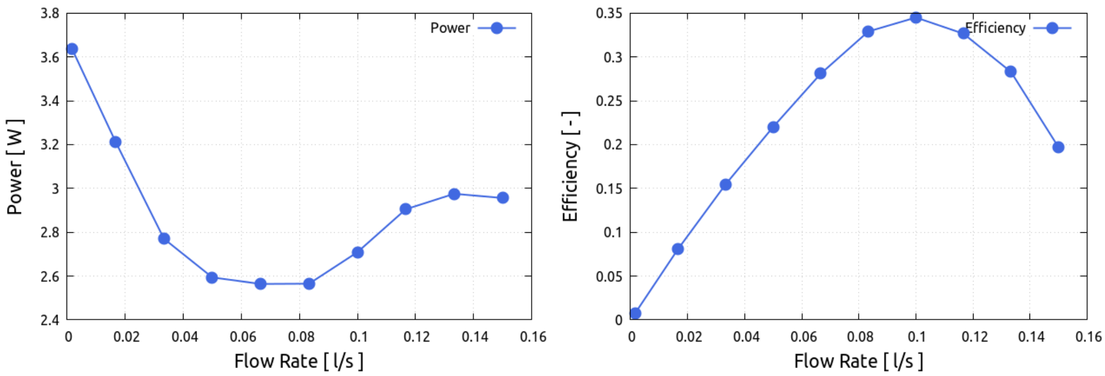
The flow in the VAD can be assessed at different flow regimes defined by the imposed flow rates. Figure 8 shows the linear increase of the velocity magnitude with flow rate at the inlet. The velocity there is parallel to the axis of the inlet cylindrical channel and the whole cross-section is available for the fluid flow, therefore the velocities grow monotonically (linearly) with the flow rate, achieving values up to approx 0.75 m/s for the highest flow rate. Later on, at the interface between the inlet and the impeller part, the velocities are much higher, due to flow blockage by the VAD core and partly due to increase in tangential and radial velocity components.



(a) Inlet (b) Impeller interface

Figure 8: Velocity magnitude at different flow rates.

In the presented simulations the rotation speed of the impeller is kept constant for all simulations (8000 RPM), but the imposed flow rate differs in ten steps between 0 and approximately 0.15 l/s. This leads to variation in the power required to drive the rotor (shown in Fig. 9a) which affects the pump efficiency (shown in Fig. 9b). This may indicate the optimal operating regime for the given specific design of the VAD.

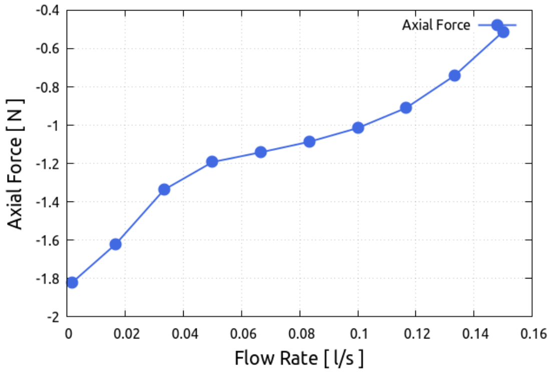


(a) Power (b) Efficiency

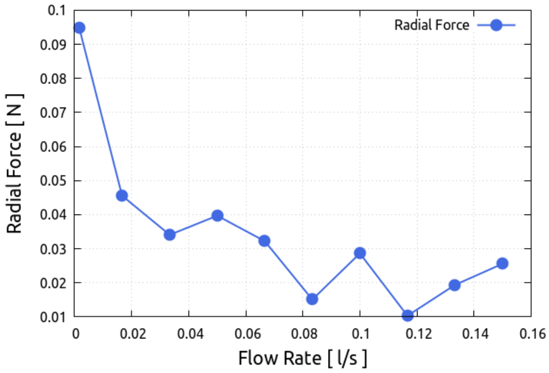
Figure 9: Power and efficiency at different flow rates.

Not only the performance and efficiency parameters differ for various operating regimes, but also the structural forces acting on the pump exhibit non-trivial dependencies on the imposed flow rate. This is shown in Fig. 10 depicting the axial and radial forces versus the actual flow rate.

Many other parameters can be evaluated from the complete three-dimensional flow fields, allowing for detailed mutual cross-comparison of different mathematical models, turbulence models, numerical solvers and computational grids. The above provided outputs only serve as a basic examples documenting the usability of the newly developed geometrical model of simplified VAD.



(a) Axial force



(b) Radial force

Figure 10: Axial and radial force at different flow rates.

4 Discussion and Conclusions

The presented newly developed design of simplified axial blood pump has proved to be a viable alternative to numerous proprietary academic and industrial designs used in CFD modeling and development of VADs. The initial numerical simulations indicate that the developed geometry, its accuracy, structure and formatting can guarantee that the model is easy to import to mesh generators and CFD solvers for further processing. This high quality 3D model design is preserved even in case the adjustable dimensional and shape parameters are varied, without the need of any manual intervention and geometry fine tuning and post processing. The future steps in our research will focus on automated script-driven model and grid generation together with refined automatic CFD evaluation of different design configurations with outlook to their possible shape-optimization.

The presented initial study is a first step in creation of an open-access benchmark for numerical implementation and mutual cross-comparison of various mathematical models and numerical methods. The aim is to publish and fully disclose the complete geometry of the case and the OpenFOAM code setup which can be shared by numerous research groups working on mathematical modeling and numerical simulation of blood flow in VADs. The focus will be on non-Newtonian models for blood, models of turbulence and the various blood damage (hemolysis) models.

Acknowledgment

The financial support for the present work was provided by the Czech Academy of Sciences under the *Praemium Academiae of Š. Nečasová* and RVO:67985840.

References

[1] Flow in a ventricular assist device - pump performance & blood damage prediction: Application challenge AC7-03. https://www.kbwiki.ercoftac.org/w/index.php/AC7-03, 2022. Accessed: 2024-12-20.

[2] TCAE – engineering simulation environment. https://www.cfdsupport.com/tcae.html, 2024. Accessed: 2024-12-20.

[3] Arwatz, G. & Smits, A.J.: A viscoelastic model of shear-induced hemolysis in laminar flow. *Biorheology*, 50:45–55, 2013.

[4] Bodnár, T.: On the Eulerian formulation of a stress induced platelet activation function. *Mathematical Biosciences*, 257:91–95, 2014.

[5] Dirkes, N., Key, F. & Behr, M.: Eulerian formulation of the tensor-based morphology equations for strain-based blood damage modeling. *Computer Methods in Applied Mechanics and Engineering*, 426:116979, 2024.

[6] Fraser, K.H., Taskin, M.E, Griffith, B.P. & Wu, Z.J.: The use of computational fluid dynamics in the development of ventricular assist devices. *Medical Engineering & Physics*, 33:263–280, 2011.

[7] Galdi, G.P, Rannacher, R., Robertson, A.M., & Turek, S. editors: *Hemodynamical Flows - Modeling, Analysis and Simulation*, volume 37 of *Oberwolfach Seminars*. Birkäuser, 2008.

[8] Konnigk, L., Torner, B., Bruschewski, M., Grundmann, S. & Wurm, F.-H.: Equivalent scalar stress formulation taking into account non-resolved turbulent scales. *Cardiovascular Engineering and Technology*, 12(3):251–272, 2021.

[9] Sequeira, A. & Bodnár, T.: Blood coagulation simulations using a viscoelastic model. *Mathematical Modelling of Natural Phenomena*, 9(6):34—45, 2014.

[10] Torner, B., Konnigk, L., Abroug, N., & Wurm, H.: Turbulence and turbulent flow structures in a ventricular assist device – A numerical study using the large-eddy simulation. *International Journal of Numerical Methods in Biomedical Engineering*, 37:e3431, 2021.

[11] Yu, H., Engel, S., Janiga, G. & Thevenin, D.: A review of hemolysis prediction models for computational fluid dynamics. *Artificial Organs*, 41(7):603–621, 2017.