Analysis of artery blood flow before and after angioplasty

Michał Tomaszewski[1], **Paweł Baranowski**[1], **Jerzy Małachowski**[1] **and Krzysztof Damaziak**[1]*

[1] *Department of Mechanics and Applied Computer Science, Military University of Technology*
Gen. Sylwestra Kaliskiego 2, 00-908 Warsaw, Poland
e-mail: michal.tomaszewski@wat.edu.pl
e-mail: pawel.baranowski@wat.edu.pl
e-mail: jerzy.malachowski@wat.edu.pl
e-mail: krzysztof.damaziak@wat.edu.pl

Abstract

The study presents a comparison of analysis results of blood flow in two different arteries. One of them was narrowed to simulate an arteriosclerosis obstructing the blood flow in the vessel, whereas the other simulates the vessel after angioplasty treatment. During the treatment, a biodegradable stent is inserted into the artery, which prevents the vessel walls from collapsing. The treatment was simulated through the use of numerical analysis using a finite element method. An element mesh obtained from the analysis was exported to the dedicated software in order to create geometry in which a flow domain inside the artery with the stent could be created. The flow analysis was conducted in ANSYS Fluent software, with non-deformable vessel walls.

Keywords: coronary stents, percutaneous coronary intervention, CFD modelling techniques, artery flow, angioplasty

1. Introduction

The problem of an acute coronary syndrome affects a significant part of the society. Majority of these syndromes can be classified as insufficient transport of both oxygen and energy to the hearth, compared to the needs of the organ.

Fast pace of living combined with high stress levels and non-optimal diet can lead to syndromes such as arteriosclerosis. Arteriosclerosis plates can grow in blood vessels over time. These plates, when growing in arteries, can obstruct the blood flow. In the worst scenario, the artery can close completely through clotting of the blood and can lead to the patient's death [2].

Advancement in the field of numerical methods in engineering allows studying more and more difficult phenomena and processes. Numerical modelling of blood flow is a difficult task, requiring an advanced, combined knowledge of fluid mechanics and blood flow hemodynamic [1,5-7].

The study presents a comparison of the blood flow through a vessel before and after angioplasty through the use of a stent. There is presented an innovative approach, which allowed for two flow domains which are based on the geometry of the narrowed vessel. The other domain is based on the mesh created from the last step of non-linear analysis, where a balloon is expanded in order to put a stent in the vessel. The flow analyses were performed with use of the finite volumes method based on three conservation laws – conservation of mass, momentum and energy.

In the areas affected by the arteriosclerosis, flow disruption occurs. In these areas, the substances composing the arteriosclerosis plates can be formed. The analyses were aimed at finding the regions where flow turbulences occur and checking the distribution of the flow velocity along the vessel.

2. Geometric model

During the blood flow analysis, a flow domains needs to created. In the presented analyses, it consists of the internal volume of the vessel – the part filled with blood. Depending on the case, the approach to creating the domains was differed.

2.1. Narrowed artery

The geometric model (Fig.1) of the narrowed artery was created through recreation of the vessels outline (characteristic boundaries) based on results of the vessels cross sections medical scans and statistical studies of pathological changes in the them [3]. The total length of the coronary artery was assumed to be 16 mm. The cross-sections created based on the scanned pictures were used to create a surface, and then a solid which corresponded to the walls of the vessel.

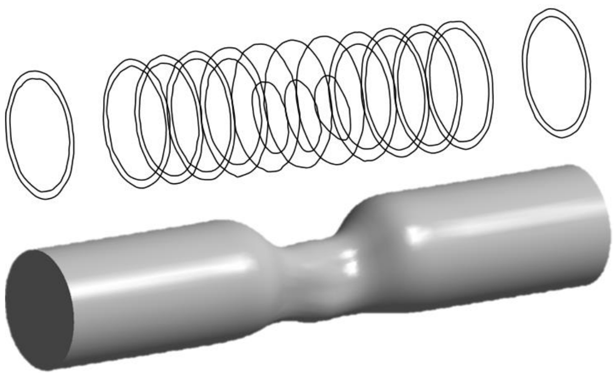

Figure 1: Geometrical model of stenosis

2.2. The artery with a stent

The first step during modelling the geometry of the artery vessel with the implanted stent after the angioplasty (Fig.2) was to export the mesh used in implicit analysis [2] to the Altair HyperMesh software. To create the blood flow domain, it was important to fill the empty space with an entity simulating the volume of the blood inside. Since it is impossible to create such a complicated geometry in a single operation, it was necessary to combine the outer surfaces into a single, continuous entity.

*The study was supported by the NCBiR within project "Apollo" (STRATEGMED) and the Interdisciplinary Centre for Mathematical and Computational Modelling (ICM) University of Warsaw under grant no GB65-19. This support is gratefully acknowledged.

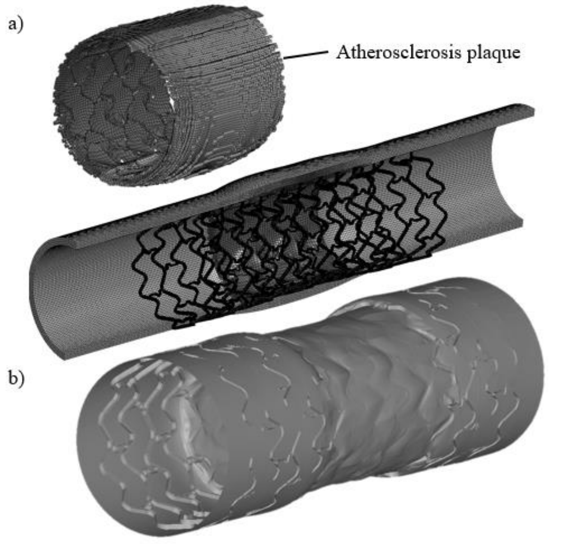

Figure 2: Models used to create the domains: a) FEM from the last step of the implicit analysis, b) Geometry of the domain after implanting the stent

3. Discrete model

The discrete model (Fig. 3) was created using tetragonal elements. The area where the flow disruption was estimated was modelled with a finer mesh to increase solution accuracy.

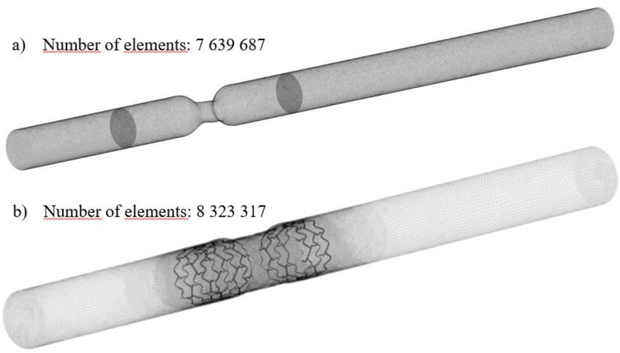

Figure 3: Discrete model: a) stenosis artery, b) artery after angioplasty procedure

4. Boundary conditions

For the considered case, it was necessary to define the following conditions: velocity at inlet, pressure at outlet, friction between the walls and the blood, and viscosity as a function of shear stress [4]. A simplification in the form of averaged pressure at exit (13 332 Pa) was used in the model. Velocity was defined as a periodic function, through UDF (user defined function), and was changing from 0.1 m/s during diastole to 0.5 m/s during contraction.

5. Results

As a result of the conducted analysis, distributions of velocity (Fig. 3), pressure and WWS shear stress were obtained for both domains. Flow analysis was conducted with the use of a coupling method with k-omega SST turbulence model available in Ansys Fluent software. The time step was set at 0.004 s, with 50 iterations per step. Visualisations of each value were shown through creation of additional surfaces, where the values were displayed.

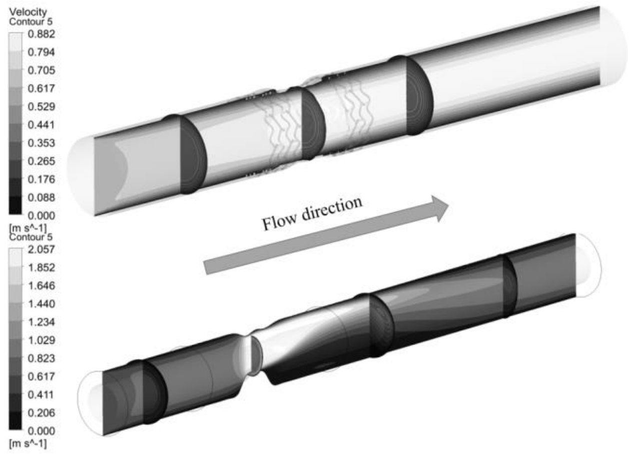

Figure 4: Velocity profile comparison for two analysed cases

6. Conclusions

The above presented results show that a stent significantly improves the blood flow in the vessel. The areas of lower flow velocities are smaller than in the narrowed vessel. Based on the velocity distribution, the highest velocities of the blood appear in the narrowed part of the vessel. It was also observed that the velocity of the flow dropped from 2 m/s to 0.9 m/s in the centre of the vessel after angioplasty procedure (Fig. 4).

References

[1] Auricchio, F., et al., Carotid artery stenting simulation: From patient-specific images to finite element analysis, *Medical Engineering & Physics*, 33, pp. 281-289, 2011.

[2] Bukała, J., Małachowski, J., Kwiatkowski, P., Numerical analysis of stent expansion process in coronary artery stenosis with the use of non-compliant balloon, *Biocybernetics and Biomedical Engineering*, 36(1), pp. 145-156, 2016.

[3] Jąkała, J., Ocena metodą ultrasonografii wewnątrznaczyniowej i wirtualnej histologii wyników przezskórnych interwencji wieńcowych u pacjentów z zawałem serca, *PhD thesis*, Jagiellonian University Medical College, Cracow, 2012.

[4] Jodko, D., et al., Numerical investigations of the unsteady blood flow in the end-to-side arteriovenous fistula for hemodialysis, *Acta of Bioengineering and Biomechanics*, 18(4), pp. 3-13, 2016.

[5] Gogas, B., et al., Novel drug-eluting stents for coronary revascularization, *Trend in Cardiovascular Medicine*, 24, pp. 305-313, 2014.

[6] Naghipoor, J., Ferreia, J., Oliveira, P., Rabczuk, T., Tuning polymeric and drug properties in a drug eluting stent: A numerical study, *Applied Methematical Modeling*, 40, pp. 8067-8086, 2016.

[7] Malve, M., et al., Numerical modeling of a human stented trachea under different stent designs, *International Communications in Heat and Mass Transfer*, 38, pp. 855-862, 2011.