Fluid-Structure Interaction Simulation of an Artificial Heart Valve and Unsteady Blood Flow

Manh Hieu Vu, Manh Hung Nguyen, Van Chien Vu,
Manh Duc Nguyen and Sang Truong Ha

December 12, 2022

Fluid-Structure Interaction simulation of an artificial heart valve and unsteady blood flow

Vu Manh Hieu, Nguyen Manh Hung, Vu Van Chien, Nguyen Manh Duc, Sang Truong Ha*

Department of Mechanical Engineering, Le Quy Don Technical University, 236 Hoang Quoc Viet, Hanoi, Viet Nam.

*Email: sanght.st@lqdtu.edu.vn

Abstract. The present work studies the interaction of incompressible fluid flow with an artificial heart valve using fluid-structure interaction (FSI) simulation. The finite element method (FEM) is employed both for fluid and solid domains. The strong coupling scheme is used for the coupling of fluid and structure to satisfy the kinematic and dynamic equilibrium conditions. The Navier-Stokes equations of an incompressible flow are solved using the integrated method based on the unstructured grid by Arbitrary Lagrangian-Eulerian (ALE) framework. The Lagrangian formulation is used for the non-linear behavior of the heart valve due to large deformation. The smoothing technique based on the harmonic extension is employed to improve the mesh quality on the fluid domain when the valve deformation is large. The present method investigates fluid flow through the heart valve in 3D geometries. The present method solves the two challenges of FSI simulation: The add-mass problem arising from the similarity of the density of fluid and solid and the large deformation of the solid wall. The fluid characteristic, such as the velocity and pressure in the valve, are evaluated and analyzed in detail. The simulation results can be used helpfully to predict and treat cardiovascular diseases.

Keywords: Incompressible fluid flow, artificial heart valve, fluid-structure interaction, FEM, ALE

1. Introduction

Heart valves are important parts of the circulatory system in the human body. The valves are open and close to control or regulate the blood flowing into the heart and then away from the heart. Three of the heart valves are composed of three leaflets or flaps that work together to open and close to allow blood to flow across the opening. Nowadays, heart valve diseases problem are among the most common cardiovascular defects. Thus, transcatheter heart valve implantation (TAVI) has been carried out as an alternative for patients with severe artery stenosis who are at high risk for surgical therapy. The biomechanical environment of TAVI is closely related to the interaction of the motion of the aorta and leaflets with blood flow. Therefore, fluid-structure interaction (FSI) simulation and an accurate blood prediction are essential for predicting and treating cardiovascular diseases. Recently, there have been many publications in FSI simulation for blood flow interacting with a heart/aortic valve in literature.

Amindari et al. [1] used the commercial software ANSYS to investigate the hemodynamics for FSI modeling of the aortic valve. The authors simulated the problem using fluid flow solver FLUENT and structural solver MECHANICAL APDL under ANSYS and coupled the solutions using the System Coupling Module to enable FSI. Based on that, the influence of leaflet calcification on hemodynamic stresses and flow patterns was investigated. B. Su et al. [2] study the intraventricular flow in a patient-specific left ventricle in two-dimensional space (2D) integrated with mitral and aortic valves. Based on MRI data, ventricular wall deformation was predefined, while leaflet dynamics were predicted numerically by FSI. A study of the aortic valve motion is also reported in [3], which used an immersed boundary method to avoid the mesh deformed in the fluid domain. The blood flow through the aortic and heart valve was researched using the FSI algorithm in 2D and 3D space [4,5].

Although many publications on the aortic blood flow through the aortic valve, the problem has still been challenged and needs more study. Because the fluid density is close to that of a solid in the aortic artery/valve, the add-mass effect is strong and leads to a complicated issue in the numerical method. Further, due to the large deformation, the grid of fluid part is big distorted. These challenging problems are now open issues for numerical research. This paper presents a numerical analysis of blood flow in an artificial heart using fluid-structure interaction simulation. The ALE framework with an unstructured grid in 3D space using the finite element method for both fluid and solid domains is employed for the present numerical method.

2. Numerical method

2.1. Equations for incompressible blood flow

The fluid domain is denoted by Ω^f with the boundary Γ^f. The incompressible Navier-Stokes equations can be written in Ω^f by arbitrary Lagrangian-Eulerian formas follows [6,7]:

$$\rho\left(\frac{\partial u}{\partial t}+(u-u^*)\frac{\partial u}{\partial x}+(v-v^*)\frac{\partial u}{\partial y}+(w-w^*)\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \tag{1a}$$

$$\rho\left(\frac{\partial v}{\partial t}+(u-u^*)\frac{\partial v}{\partial x}+(v-v^*)\frac{\partial v}{\partial y}+(w-w^*)\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{1b}$$

$$\rho\left(\frac{\partial w}{\partial t}+(u-u^*)\frac{\partial w}{\partial x}+(v-v^*)\frac{\partial w}{\partial y}+(w-w^*)\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{1c}$$

And the continuity equation in Ω^f given by:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2}$$

where ρ is the fluid density, u, v, w represents the velocity in x, y, z direction, g_i represents the body force in i-direction, and μ denotes the dynamic viscosity of the fluid. u^*, v^*, w^* represents the components of mesh velocity, and p is fluid pressure.

The equations (1) and (2) is the governing equation for fluid dynamics. Solving this equation to obtain the velocity and pressure field by the FEM method is described in ref. [7].

2.2. Equation for elastic displacement of heart valve

The solid domain is denoted by Ω^s with the boundary Γ^s. The deformation of the solid in the Lagrangian form is written by:

$$\rho^s\frac{\partial^2 d_x}{\partial t^2}=\frac{\partial\,\boldsymbol{\sigma}_{xx}}{\partial x}+\frac{\partial\,\boldsymbol{\sigma}_{xy}}{\partial y}+\frac{\partial\,\boldsymbol{\sigma}_{xz}}{\partial z}+\rho^s g_x \tag{3a}$$

$$\rho^s\frac{\partial^2 d_y}{\partial t^2}=\frac{\partial\,\boldsymbol{\sigma}_{yx}}{\partial x}+\frac{\partial\,\boldsymbol{\sigma}_{yy}}{\partial y}+\frac{\partial\,\boldsymbol{\sigma}_{yz}}{\partial z}+\rho^s g_y \tag{3b}$$

$$\rho^s\frac{\partial^2 d_z}{\partial t^2}=\frac{\partial\,\boldsymbol{\sigma}_{zx}}{\partial x}+\frac{\partial\,\boldsymbol{\sigma}_{zy}}{\partial y}+\frac{\partial\,\boldsymbol{\sigma}_{zz}}{\partial z}+\rho^s g_z \tag{3c}$$

where ρ^s is density of solid, d_x, d_y, d_z represents the displacement in x, y, z direction of solid, and $\boldsymbol{\sigma}_{ij}$ denote the components of stress tensor:

$$\sigma = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix} = \begin{pmatrix} \sigma^x \\ \sigma^y \\ \sigma^z \end{pmatrix}; \tag{4a}$$

$$\sigma_{xy} = \sigma_{yx};\ \ \sigma_{xz} = \sigma_{zx};\ \ \sigma_{yz} = \sigma_{zy} \tag{4b}$$

The relation of stress and strain can be written by compact form as follows:

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{bmatrix} = [D] \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{xy} + \varepsilon_{yx} \\ \varepsilon_{yz} + \varepsilon_{zy} \\ \varepsilon_{zx} + \varepsilon_{xz} \end{bmatrix} \tag{5}$$

where ε_{ij} is components of strain tensor:

$$\varepsilon = \frac{1}{2}\left\{\left(\nabla \mathbf{d}\right)^{\mathrm{T}} + (\nabla \mathbf{d}) + \left(\nabla \mathbf{d}\right)^{\mathrm{T}}(\nabla \mathbf{d})\right\}; \tag{6}$$

and

$$\nabla \mathbf{d} = \begin{pmatrix} \frac{\partial d_x}{\partial x} & \frac{\partial d_x}{\partial y} & \frac{\partial d_x}{\partial z} \\ \frac{\partial d_y}{\partial x} & \frac{\partial d_y}{\partial y} & \frac{\partial d_y}{\partial z} \\ \frac{\partial d_z}{\partial x} & \frac{\partial d_z}{\partial y} & \frac{\partial d_z}{\partial z} \end{pmatrix} \tag{7}$$

The constitutive equations of the solid domain in large deformation are written as [8].

The matrix D in equation (5) is given as follow for linear model of material behavior:

$$[D] = \begin{bmatrix} \lambda + 2\eta & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda + 2\eta & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda + 2\eta & 0 & 0 & 0 \\ 0 & 0 & 0 & \eta & 0 & 0 \\ 0 & 0 & 0 & 0 & \eta & 0 \\ 0 & 0 & 0 & 0 & 0 & \eta \end{bmatrix} \tag{8}$$

The Lamé constants λ and η can be computed from Young's modulus E and Poison ratio ν by:

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)};\ \ \eta = \frac{E}{2(1+\nu)} \tag{9}$$

2.3. FSI formulations

Let denote the interface between the fluid domain and solid domain (FS interface) by Γ^{fs}, $\Gamma^{fs} = \Gamma^f \cap \Gamma^s$. In the implicit coupling method, the velocity and traction need to satisfy the balance conditions. When a no-slip condition is applied, the velocity of the fluid is similar to that of solid on Γ^{fs}, and the balance condition can be written as follows

$$\mathbf{v} = \frac{\partial \mathbf{d}}{\partial t} \quad \text{on } \Gamma^{fs} \tag{10}$$

Due to the force equilibrium condition, the traction should also be continued along the FS interface:

$$\boldsymbol{\sigma}^f \cdot \mathbf{n}^f + \boldsymbol{\sigma}^s \cdot \mathbf{n}^s = 0 \text{ on } \Gamma^{fs} \tag{11}$$

The direction of the unit normal vector of solid $\mathbf{n}^s$ is opposite to that of fluid $\mathbf{n}^f$ when the grids of fluid and solid domain along the FS interface are conformed.

For the simulation of large deformation, the strong coupling [7,9] by using a monolithic approach is used to improve the convergence of the FSI problem. When the deformation is large, a smoothing technique based on the Laplace equation is employed to avoid the bad quality of the fluid mesh. The inner nodes are moved based on Laplace distribution. The interface nodes move with solid deformation, and nodes on other boundaries are fixed. The Lagrangian method is applied for nonlinear problems arising from the large deformation of the solid. The detailed expression of the FSI method is available in [9]. The obtained linear system can be solved by a multigrid method to reduce the simulation time [10]. It should be noted that the present method using strong coupling and smoothing techniques has successfully treated the problems of the add-mass issue arising from the similarity of the density of fluid and solid as well as the large deformation problem of the solid wall.

3. Results and discussions

3.1. Validation

Firstly, the code was validated by simulation of a simple case. A pressure wave propagation of incompressible fluid in a 3D flexible tube as reported in the previous works [7,9] was adopted in this section. The schematic and boundary conditions are shown in Fig. 1, the tube has a length of L = 5.0 cm, a diameter of D = 1.0 cm, and a wall thickness of δ = 0.1 cm. Density and dynamic viscosity of fuid are ρ^f = 1.0 g/cm^3 and μ = 0.03 Poise. The Young's modulus of E = 3×10^5 Pa and a Poisson ratio of ν = 0.3 were employed for the tube wall. The density of solid was set by ρ^s = 1.0 g/cm^3. The tube was fixed in all directions at the two ends. Both fluid and solid were initially at rest. At the inlet, a pressure of p_{in} = 10 mmHg was applied in the first short period of 3×10^{-3} s and then set back to zero later. A *P2/P1* tetrahedral mesh is used for both fluid and solid domains as shown in Fig. 2 in which the numbers of the nodes for fluid and solid domains are 55,045 and 32,320, respectively. The detail descriptions of the problem was listed in Ref. [2].

The time-step dt was set by 1×10^{-4} s, and 100 time steps were conducted to get 10 ms simulation. The pressure field at different instant times were illustrated in Fig. 3. Fig. 4 plots the evolution of axial movement at the centre point of the inner tube wall (point M in Fig. 2) for a quantitative comparison. It can be seen that the present results agree well with those obtained by Eken and Sahin [11].

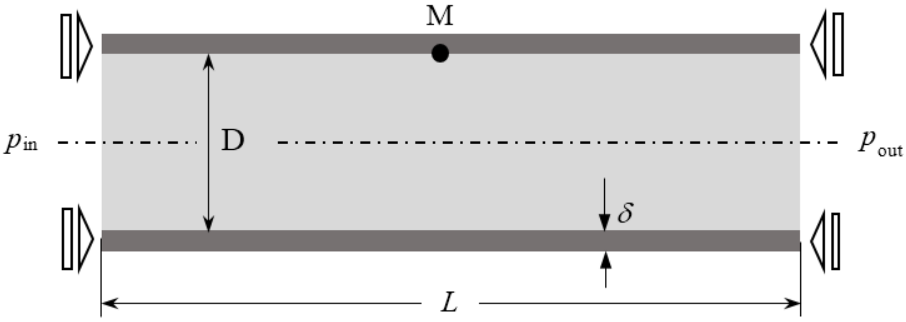
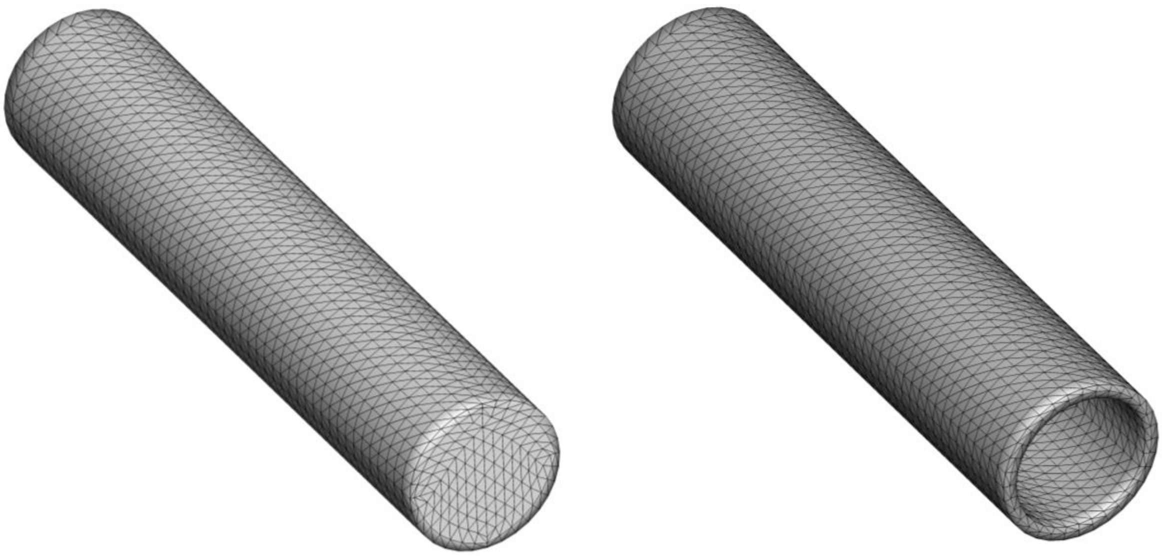
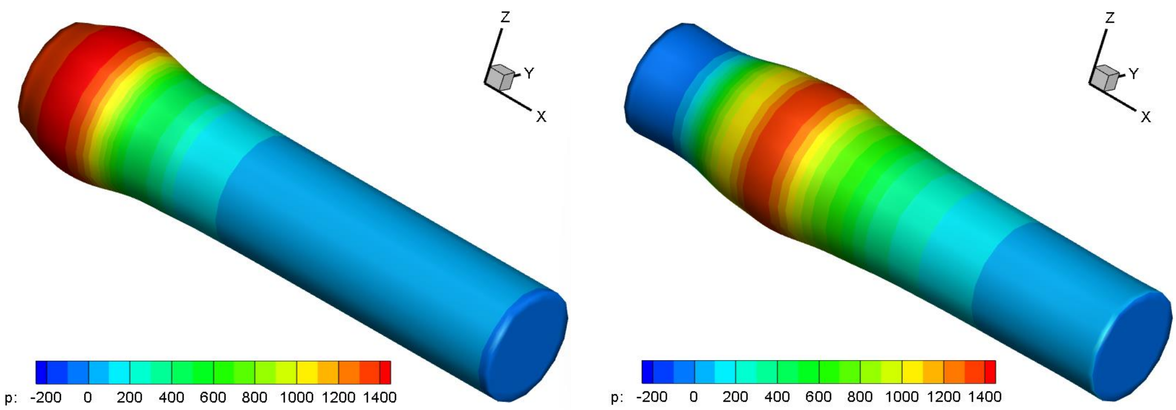


Fig.1. Schematic of flow in a straight flexible tube [7,9]

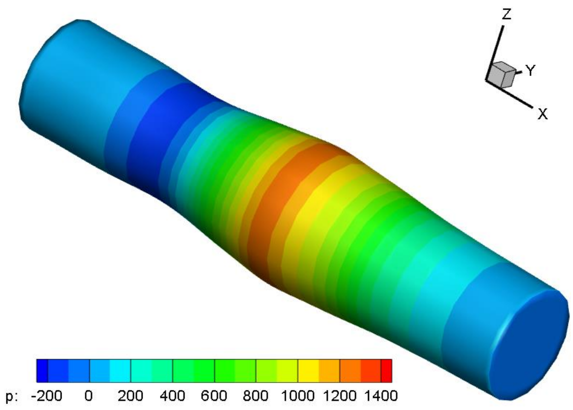


(a) *(b)*

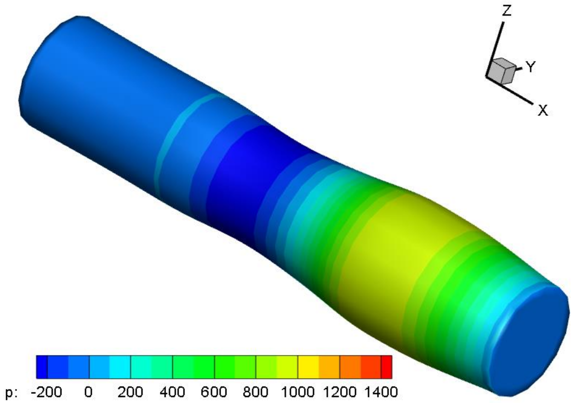
Fig. 2. Grid of fluid (a) and solid domains (b)



(a) Time = 2.5 ms (b) Time = 5.0 ms



(c) Time = 7.5 ms



(d) Time = 10 ms

Fig. 3: The pressure field at different instant times

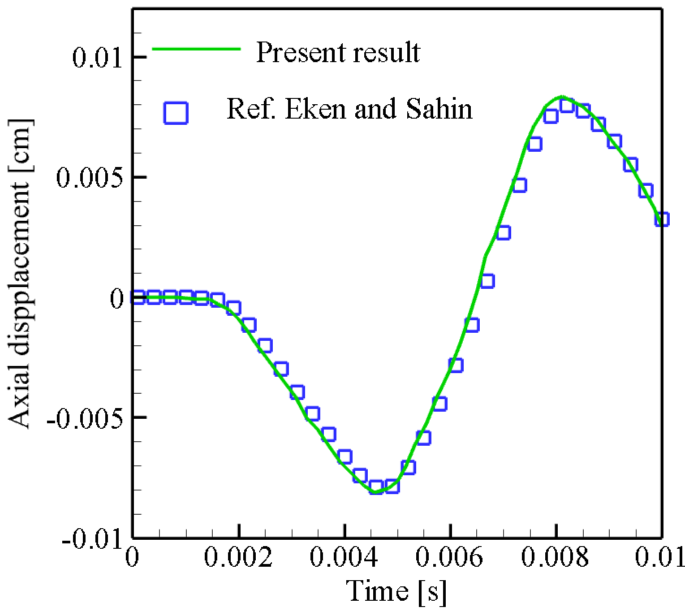


Fig. 4. Comparison of the movement of point M obtained from present work and results by Eken and Sahin [11]

3.2. Artificial *heart valve*

After validation the code, a 3D problem of an incompressible fluid flow interacting with three leaflets of an artificial heart valve is simulated. The geometry is shown in Fig.5, and the dimension is set similarly to those in ref. [12] with a length of 20 cm and a diameter of 4 cm. A grid independent test is handled and a medium mesh with 227,448 nodes and 156,162 elements (145,971 elements for fluid and 13,191 elements for solid) as shown in Fig.6 is employed for this simulation. A time-dependent velocity is set at the inlet with a systolic phase in 0.4 s and a diastolic phase in 0.6 s while a zero pressure is set at the outlet for all simulation time [13]. This flow rate is from the left ventricle with the systolic phase (opened valve) and the diastole phase (closed valve) in a period of T = 1.0 s. Fig.7 shows the streamline and velocity field when the valve is at maximum opening. The 3D effect is appeared by many vortices after the valve since the geometry is not completely symmetrical. The deformation of the valve is large, and its movement together with Average Von-Mises (AVM) stress in several instants time is illustrated in Fig.8. It is shown that the Von-Mises stress obtain the highest value around the commissure of the valve. In addition, a larger deformation of the valve at the systole leads to a higher VMS distribution. This simulation results are help full for the prediction of the TAVI in the real case.

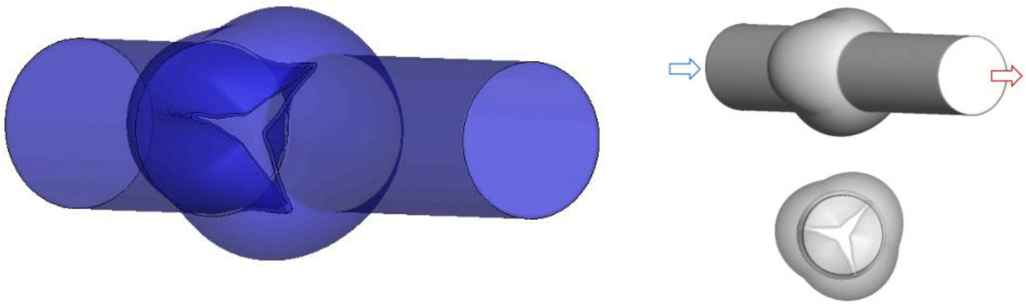


Fig. 5. Geometry of a 3D heart valve

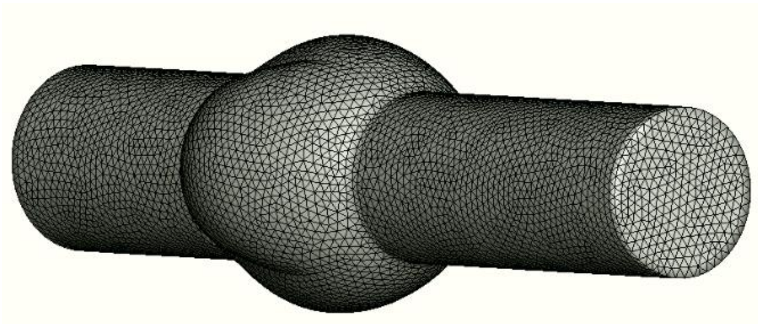
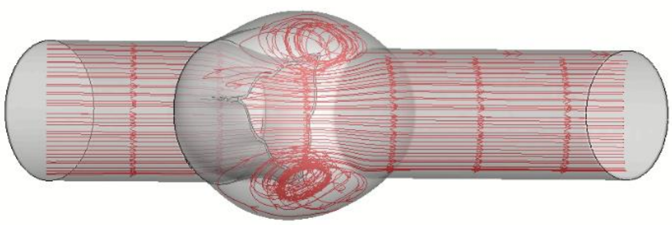
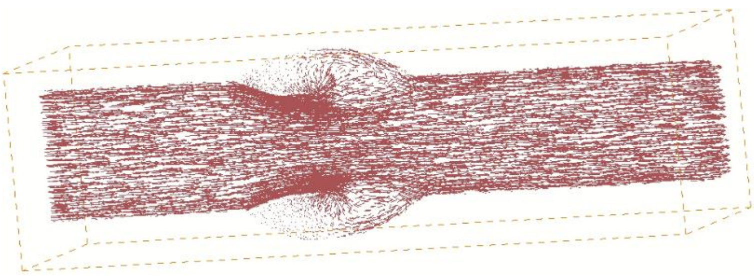


Fig. 6. Tetrahedral mesh for 3D simulation of artificial heart valve problem.



a) Streamline



b) Velocity field

Fig. 7. Blood flow through the valve at opening phase.

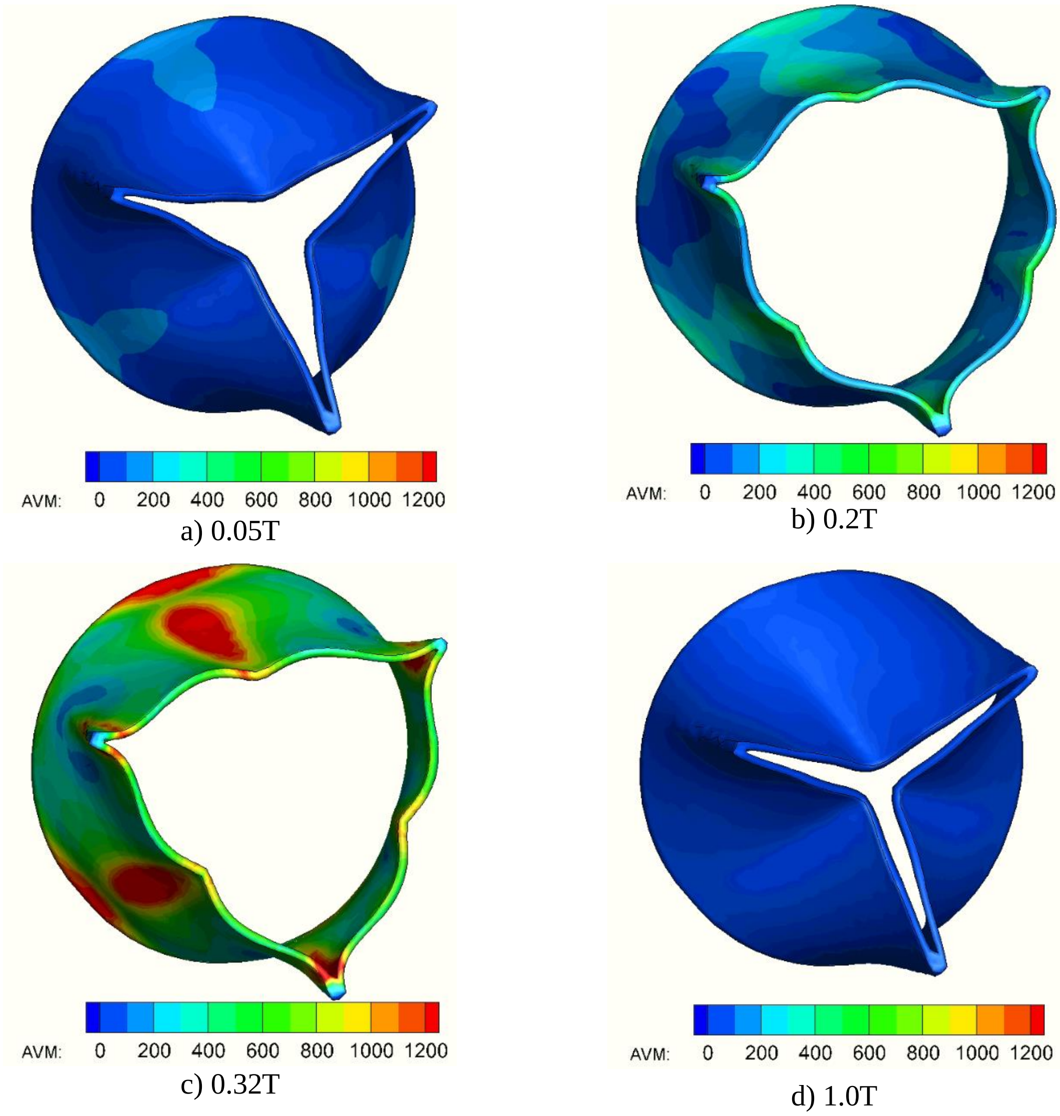


Fig. 8. Deformation of valve and average Von-Mises Stress at different phases.

4. Conclusions and future work

The study used the finite element method to separate the liquid and solid domains for the fluid-structure interaction (FSI) problem of complex geometric heart valves in 3D. A strong coupling method is used to ensure convergence, and smoothing techniques are applied to improve the mesh quality of the fluid region. Large deformation model used for artificial heart valves. The program (code) is validated by simulating the pressure wave propagation of an incompressible fluid in a flexible tube. Then it is applied to simulate the blood flow interacting with the artificial heart valve in the fields. 3D composite. An oscillating flow at the inlet moves the valve in the systolic and diastolic phases. The results show that the code can simulate the FSI problem with efficient performance. The simulation results can be used to predict valve movement in the arteries and thus can help to solve biomechanical problems, especially for transcatheter heart valve implantation, such as predicting blood-clotting problems in valves.

References

[1] A. Amindari, L. Saltik, K. Kirkkopru, M. Yacoub, H. C. Yalcin. Assessment of calcified aortic valve leaflet deformations and blood flow dynamics using fluid-structure interaction modeling. *Informatics in Medicine Unlocked*, **9**, (2017), pp. 191 – 199.

[2] B. Su, L. Zhong, X. K. Wang, J. M. Zhang, R. S. Tan, J. C. Allen, S. K. Tan, S. Kim, H. L. Leo. Numerical simulation of patient-specific left ventricular model with both mitral and aortic valves by FSI approach. *Computer methods and programs in biomedicine*, **113.2**, (2014), pp. 474 – 482.

[3] A. M. Bavo, G. Rocatello, F. Iannaccone, J. Degroote, J. Vierendeels, and P. Segers. Fluid-structure interaction simulation of prosthetic aortic valves: comparison between immersed boundary and arbitrary Lagrangian-Eulerian techniques for the mesh representation, *PloS one*, **11.4**, (2016), pp. e0154517.

[4] G. Marom, R. Haj-Ali, E. Raanani, H. J. Schäfers, and M. Rosenfeld. A fluid–structure interaction model of the aortic valve with coaptation and compliant aortic root, *Medical & biological engineering & computing*, **50.2**, (2012), pp. 173 – 182.

[5] K. Dumont, J. M. A. Stijnen, J. Vierendeels, F. N. van de Vosse, and P. R. Verdonck. Validation of a Fluid–Structure Interaction Model of a Heart Valve using the Dynamic Mesh Method in Fluent, *Computer Methods in Biomechanics and Biomedical Engineering*, **7**(3), (2004), pp. 139 – 146.

[6] S.T. Ha, H.G. Choi. Simulation of the motion of a carotid artery interacting with blood flow by using a partitioned semi-implicit algorithm, *Korean Soc. Comput. Fluids Eng* (2019).

[7] T. S. Ha, V. C. Vu, M. H. Nguyen, and M. D. Nguyen. Numerical simulation for fluid-structure interaction of a blood flow with the aortic valve using the fem monolithic formulation, *Journal of Science and Technique*, **16**(03), (2021), pp. 49-60.

[8] G. A. Holzapfel. Nonlinear solid mechanics: A continuum approach for engineering Science, *Meccanica*, **37**(4/5), (2002), pp. 489 – 490.

[9] S.T. Ha, H.G. Choi. Investigation on the effect of density ratio on the convergence behavior of partitioned method for fluid–structure interaction simulation, *Journal of Fluids and Structures*, **96**, (2020), 103050.

[10] S.T. Ha, H.G. Choi. A meshless geometric multigrid method based on a node-coarsening algorithm for the linear finite element discretization, *Computers & Mathematics with Applications*, **96**, (2021), 31-43.

[11] A. Eken and M. Sahin. A parallel monolithic algorithm for the numerical simulation of large-scale fluid structure interaction problems. *Int. J. Numer. Methods Fluids.* (2016) 687–714.

[12] S.T. Ha, T.D. Nguyen, V.C. Vu, M.H. Nguyen, M.D. Nguyen, A Study of Fluid-Structure Interaction of Unsteady Flow in the Blood Vessel Using Finite Element Method, *In Modern Mechanics and Applications*, Springer, Singapore, (2022), pp. 1089-1101.

[13] H.S. Jeannette, J. Johan, J. Niclas, H. Johan. D Fluid-Structure Interaction Simulation of Aortic Valves Using a Unified Continuum ALE FEM Model, *Frontiers in Physiology*, **9**, (2018): 363.