COMPARATIVE ANALYSIS OF SPH AND FVM NUMERICAL SIMULATIONS OF BLOOD FLOW THROUGH LEFT VENTRICLE

Aleksandar S. Bodić[1], Marko D. Topalović[2], Miljan S. Milošević[2,3] Miroslav M. Živković[1], and Miloš S. Pešić[2]

[1] Faculty of of Engineering, University of Kragujevac, Kragujevac 34000, Serbia
e-mail: abodic@uni.kg.ac.rs , zile@kg.ac.rs
[2] Institute for Information Technologies, University of Kragujevac, Kragujevac 34000, Serbia
e-mail: topalovic@kg.ac.rs , miljan.m@kg.ac.rs , milospesic1736@gmail.com
[3] Belgrade Metropolitan University, Belgrade, Serbia

Abstract

The purpose of this research was to compare blood flow modeling inside the heart's left ventricle using commercial smoothed particle hydrodynamics (SPH) and finite volume method (FVM) solvers. These two methods are both based on continuum mechanics, and while FVM uses Eulerian material framework, SPH uses a Lagrangian formulation. In this study, the focus was only on CFD analysis of blood flow through the left ventricle using the two mentioned methods. Therefore, in the numerical analysis using FVM, walls were modeled as boundary conditions where fluid velocity was set to zero. The ventricle wall in the SPH was modeled using larger, fixed fluid particles, so at this point, there is no need for a specific contact definition. *LS-DYNA* software was used for modeling the left ventricle using the SPH method. In order to generate realistic fluid flow injection particles at the mitral valve and deactivation planes at the aortic semilunar valve were used where particle velocity was defined by time functions. *Ansys Fluent* software was used for modeling the left ventricle using FVM, within which a finite volume mesh was generated. Velocities at the inlet and outlet of the model are defined by functions using User Defined Function (UDF) so that the fluid flow corresponds to the realistic blood flow through the left ventricle. The results obtained by FVM were used as a verification of the results obtained using the SPH method. In the results section of the paper, the velocity field obtained by SPH and FVM methods is shown and compared. SPH offers greater possibilities to study FSI phenomena like the effects of wall deformations, or tracking the movement of solid particle inclusion, all within the single numerical domain. On the other hand, it requires elaborate contact definition, and prolonged analysis time in comparison to the finite volume CFD analysis.

Keywords: SPH, FVM, Lagrangian formulation, Eulerian formulation, left ventricle model, Bioengineering, Fluid flow

1. Introduction

Despite significant advances in medicine, cardiovascular diseases are still one of the leading causes of death in the world. For this reason, studies related to the evaluation of heart function are increasing significantly. Understanding cardiac blood flow patterns has various applications in studying haemodynamics as well as for the clinical evaluation of heart function [1]. Various experimental techniques are used to investigate blood flow patterns. Among the experimental techniques, imaging techniques such as Doppler ultrasound echocardiography [2, 3] and magnetic resonance imaging (MRI) [4, 5] are most common. Although these techniques have advanced significantly and provide a lot of information about the blood flow pattern through the left ventricle, there is still plenty of room for improvement, especially in terms of increasing the spatial and/or temporal resolution [6]. In addition to experimental techniques, computer simulations such as computational fluid dynamics (CFD) are also applied for these purposes, which are most often performed using the finite volume method (FVM), finite element method (FEM) and smoothed particle hydrodynamics (SPH). Their advantage is that they provide a much more complete picture of blood flow through the left ventricle with more details compared to classical methods.

SPH is a mesh-free numerical method [7], based on continuum mechanics, where the analyzed continuum is divided into sub-domains called pseudo-particles [8]. Originally designed for astrophysical problems [9,10], it was later extended to CFD [11] and solid mechanics [12]. SPH uses the Lagrangian material framework to observe the motion of a particular domain section [7], no matter the state or the composition, while the FVM which is also based on continuum mechanics, uses the Eulerian spatial formulation to observe a fixed volume through which the fluid flows [13]. While Eulerian formulation is accurate in predicting pressure and fluid velocity, a Lagrangian formulation is needed to observe particle movements and interactions within the fluid flow. Fig. 1 shows the difference between Lagrangian and Eulerian formulation using the finite element method (FEM) as an example [14], because this method can use both, although it was not used in this paper.

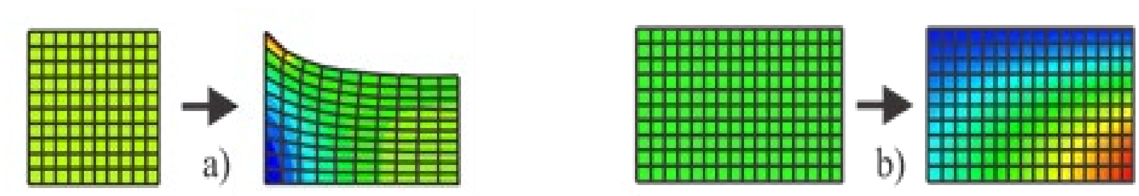

Fig. 1. Numerical formulations: a) Lagrangian b) Eulerian

In this paper, numerical simulations of blood flow through the left ventricle using commercial SPH and FVM solvers are conducted and results obtained from both methods are compared. This paper aims to facilitate the utilization of the SPH method in bioengineering modeling [15] of the cardiovascular system by addressing its challenges, primarily the need for comprehensive boundary conditions [16] and high computational demands that can be seen as a limitation or drawback, as SPH requires significant computing power and time to produce accurate results.

The paper also gives the procedure for modeling blood flow through the left ventricle using FVM. The results obtained by FVM within *Ansys Fluent* software were used as a verification of the results from *LS-DYNA* software obtained using the SPH method.

In chapter 2 of this paper, a brief retrospective of SPH kernel and particle approximation is given, as well as the procedure for modeling the left ventricle using the SPH method within the *LS-DYNA* software. In chapter 3, governing equations of fluid dynamics are given, as well as the procedure for numerical analysis of the left ventricle using FVM in the *Ansys Fluent* software. Also, the procedure for creating a model, defining regions on the model and setting boundary conditions on them is given. A comparison of the results obtained using both methods is given in chapter 4. The conclusion summarizes the outcomes shown in this paper and points out our future research.

2. Numerical analysis of blood flow through left ventricle using SPH method

2.1 SPH method

SPH models continuous matter with the kernel and particle approximations [7] and express the conservation laws of continuum mechanics as partial differential equations. These equations are transformed into integral equations with interpolation functions that estimate field variables at specific points. The exact value of the function $f(\mathbf{x})$ in integral form is given by equation (1):

$$f(\mathbf{x}) = \int_{\Omega} f(\mathbf{x}')\delta(\mathbf{x}-\mathbf{x}')d\mathbf{x}' \tag{1}$$

where $f(\mathbf{x})$ is a function of the position vector $\mathbf{x}$ of the observed pseudo-particle and $\mathbf{x}'$ is the position vector of the material point that belongs to the domain of influence Ω.

The Dirac delta measure [7] $\delta(\mathbf{x}-\mathbf{x}') = \begin{cases} 1 & \mathbf{x}=\mathbf{x}' \\ 0 & \mathbf{x}\neq\mathbf{x}' \end{cases}$ is not practical for computational applications, therefore it is replaced with bell-shaped kernel function $W(|\mathbf{x}-\mathbf{x}'|,h)$ where h is the smoothing length (bell base radius), which gives us a kernel approximation [7] of function $f(\mathbf{x})$:

$$\langle f(\mathbf{x})\rangle = \int_{\Omega} f(\mathbf{x}')W(|\mathbf{x}-\mathbf{x}'|,h)d\mathbf{x}' \tag{2}$$

Likewise, the integral form of equation (2) is not appropriate for numerical implementation because the continuous material being analyzed is divided into a finite number of particles, each with its own mass and occupying individual space. As a result, equation (2) is converted into a discrete form, which is expressed as a sum over all particles within the support domain [7]. The infinitesimal volume $d\mathbf{x}'$ is replaced by finite volume of the particle $\Delta V_\beta = m_\beta / \rho_\beta$ where m_β and ρ_β are particle mass and particle density [7]. With the summation of all particles within the support domain implemented in Eq. (2) we get particle approximation of a function $f(\mathbf{x})$ for particle α:

$$\langle f(\mathbf{x}_\alpha)\rangle \cong \sum_{\beta=1}^{NNP} f(\mathbf{x}_\beta)W(|\mathbf{x}_\alpha-\mathbf{x}_\beta|,h)dV_\beta = \sum_{\beta=1}^{NNP} \frac{m_\beta}{\rho_\beta} f(\mathbf{x}_\beta)W(|\mathbf{x}_\alpha-\mathbf{x}_\beta|,h) \tag{3}$$

where NNP is the number of nearest neighbouring particles [7].

Using the equation (3), which incorporates kernel and particle approximations, the conservation laws of continuum mechanics are programmed into the SPH solvers, which use an explicit integration scheme to calculate acceleration, velocity, displacement, energy and other values of interest, such as total stress tensor σ_{ij}, which in the viscous fluid [8] consists of the hydrostatic pressure p and viscous stress ${}^{visc}\tau_{ij}$:

$$\sigma_{ij} = -p\delta_{ij} + {}^{visc}\tau_{ij} \tag{4}$$

Viscous stress [7] can be calculated as:

$${}^{visc}\tau_{ij} = \mu\left(\partial_i v_j + \partial_j v_i - \frac{2}{3}\partial_k v_k \delta_{ij}\right) = \mu\varepsilon_{ij} \tag{5}$$

where μ is the coefficient of dynamic viscosity and ε_{ij} is the strain rate tensor [7]:

$$\varepsilon_{ij}^{\alpha} = \sum_{\beta=1}^{NNP} \frac{m_\beta}{\rho_\beta} v_j^{\beta\alpha} \frac{\partial W^{\alpha\beta}}{\partial x_i^{\alpha}} + \sum_{\beta=1}^{NNP} \frac{m_\beta}{\rho_\beta} v_i^{\beta\alpha} \frac{\partial W^{\alpha\beta}}{\partial x_j^{\alpha}} - \left(\frac{2}{3} \sum_{\beta=1}^{NNP} \frac{m_\beta}{\rho_\beta} \mathbf{v}^{\beta\alpha} \cdot \nabla_\alpha W^{\alpha\beta} \right) \delta_{ij} \tag{6}$$

Hydrostatic pressure p is calculated using the Murnaghan equation of state [7]:

$$p = k_0 \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right] \tag{7}$$

where ρ_0 is the density of the fluid at rest, while k_0 and γ are the material parameters.

2.2 SPH analysis of the left ventricle using LS-DYNA

The primary research and development focus of the commercial *LS-DYNA* solver [17] is not on fluid flow in SPH, especially not its application in the field of bioengineering. There are numerous published papers and examples utilizing academic or open-source SPH solvers, but they all have some solution for generation and deletion of SPH particles. Nevertheless, the LS-DYNA manual [17] contains a description of keywords such as BOX, BOX_LOCAL, BOUNDARY_SPH_FLOW, SPH_INJECTION, PRESCRIBED_MOTION_SET_BOX, CONTROL_SPH (BOXID) and SET_NODE_LIST_GENERATE which can activate and deactivate SPH particles. These can be used to create the variable fluid flow [18] shown in Figure 2. The inlet velocity function at mitral valve corresponds to realistic cardiac cycle (Fig. 2a). In the first approximately 0.5 seconds mitral valve is opened and the blood flows into the left ventricle from the atrium part. Subsequently, due to the ventricle contraction, blood is forced out through the aortic valve at a velocity defined by the function shown in Fig. 2b.

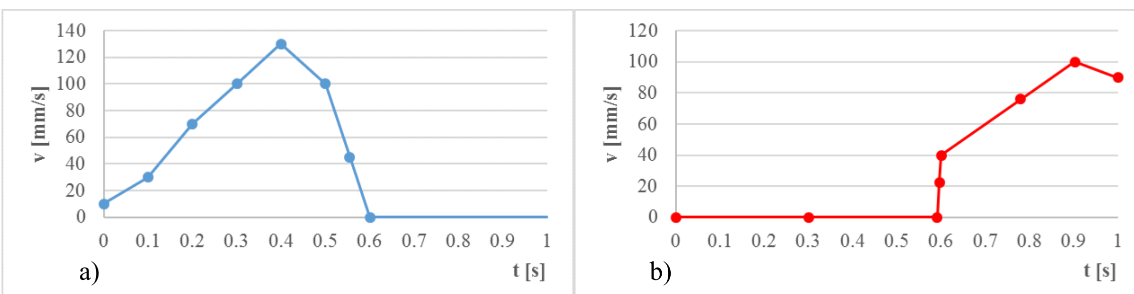

Fig. 2. Boundary flow: a) inlet velocity at mitral valve, b) outlet velocity at the aortic semilunar valve

Before *LS-DYNA* version R12.0.0 from 2020 which added option to define a variable speed of injection in the SPH_INJECTION keyword [17], this option could only produce a constant particle flow and could not be used for hemodynamic analysis. The only alternative was to use a BOUNDARY_SPH_FLOW keyword [17] with a long streak of SPH particles shown in Fig. 3.

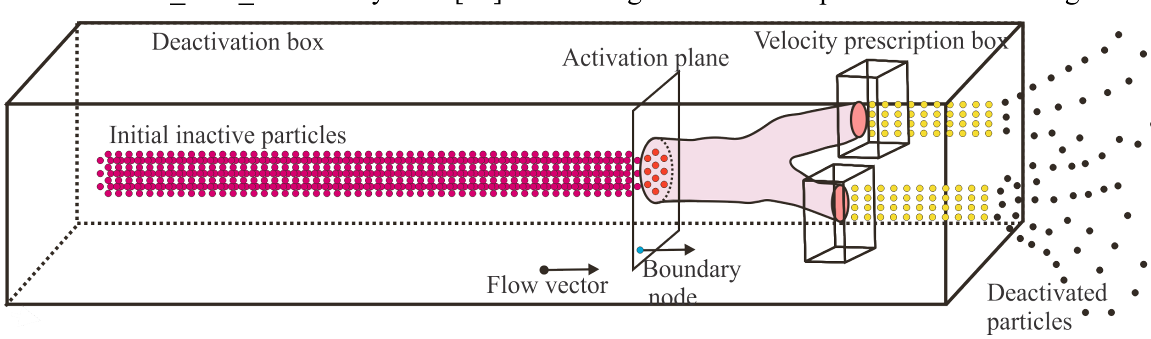

Fig. 3. Obsolete generation of fluid flow using BOUNDARY_SPH_FLOW keyword

The latest versions of LS-DYNA require a negative injection speed scale factor to be set for the SPH_INJECTION keyword [17] to prescribe a variable speed. This means that instead of a constant value, the scale factor now takes values from a DEFINE_CURVE keyword [17]. However, the new problem with this approach is that new SPH particles can collide with existing ones and cause an explosion or bounce away. To overcome this, a PRESCRIBED_MOTION_SET_BOX keyword is used, which creates a box in front of the injected particles that is at least two inter-particle distance wide. Within the PRESCRIBED_MOTION_SET_BOX keyword parameters [17], a DEFINE_CURVE is used to prescribe a continuous inflow of particles into the left ventricle of the heart according to Fig. 2 a). A similar approach is used at the aortic semilunar valve to define outlet velocity according to Figure 2 b). These keywords do not define motion for all of the SPH particles in the model, but only to those particles specified in the NODE_LIST keyword. To solve this issue, a PRESCRIBED_MOTION_SET_BOX keyword [17] is used twice at the aortic semilunar valve outlet: once with NODE_LIST for preexisting particles and the second time with NODE_LIST_GENERATE [17] for the new, injected particles. The functionality of the model is illustrated in the Fig. 4.

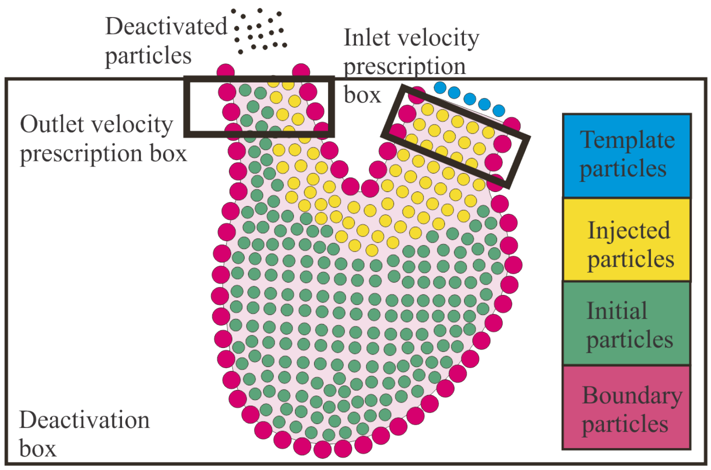

Fig. 4. Improved generation of fluid flow using SPH_INJECTION keyword

3. CFD analysis of left ventricle using Fluent

Blood flow through left ventricle is considered as laminar flow of incompressible fluid and can be defined by Navier–Stokes equations and continuity equation [15]:

$$-\mu\nabla^2 v_l + \rho\left(v_l \cdot \nabla\right) v_l + \nabla p_l = 0 \tag{8}$$

$$\nabla v_l = 0 \tag{9}$$

where v_l represents fluid velocity, p_l is the pressure, μ is the dynamic viscosity coefficient, and ρ is the fluid density.

Material parameters of blood used in numerical simulations are given in Table 1.

Material parameter	Value
Density [kg/m^3]	1060
Viscosity [Pa·s]	0.001

Table 1. Material parameters of blood

Geometry of left ventricle [18] is created using user interface software CAD Field & Solid [19]. Based on this geometry, finite volume mesh is generated in software *Ansys Fluent*. The model is created using tetrahedral finite volume cells. Quality of the mesh is checked using Fluent Meshing, and maximum cell skewness was 0.84. The average size of cell is set as 2 mm, while the entire model consists of 62208 cells and 11972 nodes. The finite volume model of the left ventricle is shown in Fig. 5a.

Within the fluent meshing, regions are created on the model that represent the inlet, outlet, wall, and symmetry plane (Fig. 5b). Based on these regions, boundary conditions are set on the model. At inlet and outlet prescribed velocity functions are set as shown in Fig. 2.

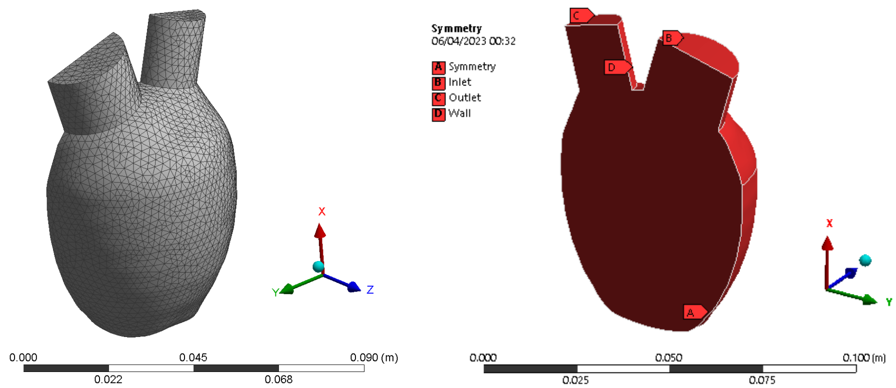

Fig. 5. a) Finite volume mesh of left ventricle and b) Defined regions on the left ventricle model

For the purpose of defining the prescribed velocities at inlet and outlet a User Defined Function (UDF) is created. As within the used version of *Fluent* there is no way to define fluid velocity at the outlet, the model consisted of "two inlet regions". At the inlet region from atrium part, a velocity is set using the UDF so that it corresponds to the function in Fig. 2a. On the other hand, at the "inlet region of the aortic valve" (model output), the velocity is set using UDF so that the function in Fig. 2b is reversed. In this way, the outflow of blood from the left ventricle through the aortic valve due to the contraction of the heart muscle walls was simulated. Transient analysis is performed in 500 time steps with time step size of 0.002 s.

4. Results and Discussion

As can be seen from Figure 2, the maximum value of inlet velocity occurs at 0.4 seconds [18], and we will compare velocity fields obtained using *LS-DYNA* and *Fluent* at that particular time, which will be shown in Figure 6. The maximum prescribed outlet velocity is defined at 0.9 seconds [18], but at that time fluid velocities within the volume of the ventricle are negligible, so more appropriate time of 0.8 seconds is shown in Fig. 7.

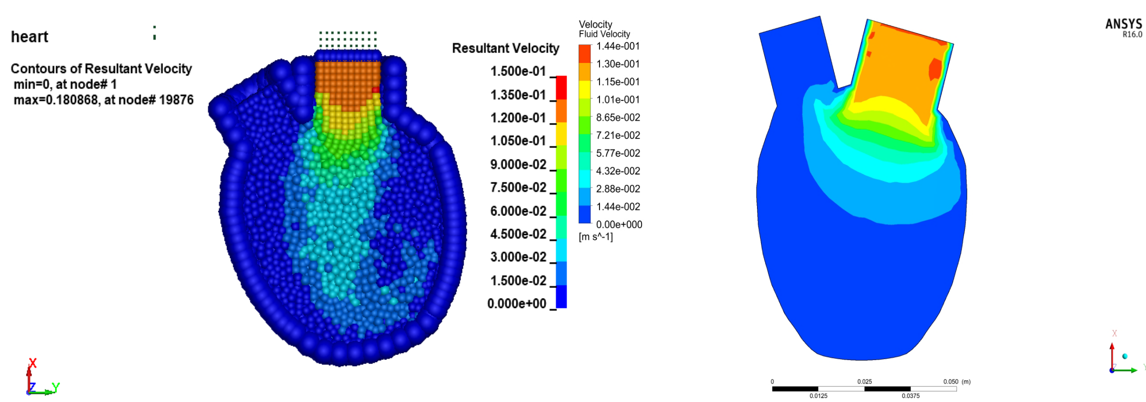

Fig. 6. Velocity field at 0.4 s: a) Ls-DYNA b) Fluent

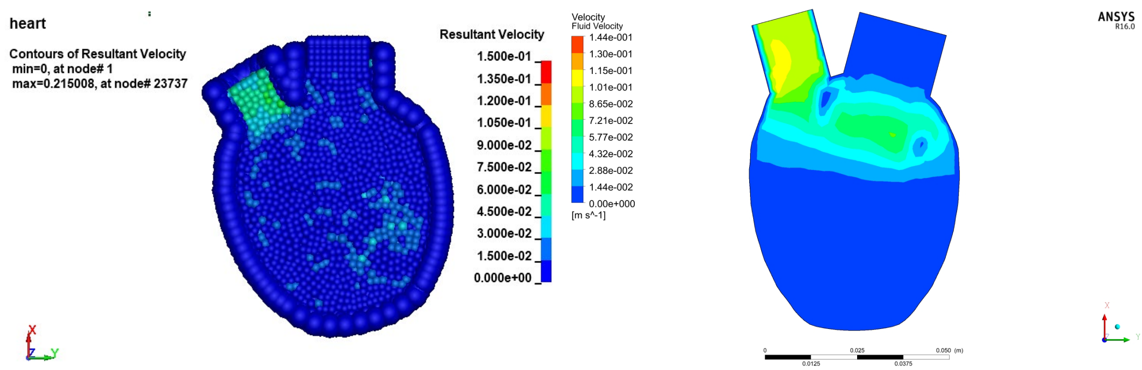

Fig. 7. Velocity field at 0.8 s: a) Ls-DYNA b) Fluent

As can be seen from previous figures, both SPH and FVM solvers produced similar velocity fields. Smaller deviations in the results can be observed after 0.6 s, when the blood starts to flow through the aortic valve. These deviations can be explained by the nature of the SPH method, due to which small fluctuations of particle velocity occur. It should also be noted that SPH analysis lasted 25 hours, while FVM took about 30 minutes on AMD Ryzen 7 5800X with 128 GB DDR4. For purely CFD simulations, FVM method is obviously the best choice, but if the multyphysics and multiscale calculations are required, SPH represents a flexible, scalable and unified base for further research.

5. Conclusions

SPH method is often used in maritime engineering to predict fluid-structure interaction between vessels and surrounding water, as well as in aeronautical engineering to predict bird strikes or crash-landing. However, these calculations are done with static fluid, and vessels or objects that impact these vessels travelling at high speeds. In this paper, we used SPH for bioengineering analysis that required modeling of continuous, variable fluid flow. Within the *LS-DYNA* software, blood flow through the left ventricle was modeled using the SPH method. Additionally, the same problem is modeled using FVM within the *Ansys Fluent* software. The results obtained using the FVM were used as a verification of the numerical analysis using the SPH method. By comparing the results obtained using both methods, it can be concluded that the results obtained within the *LS-DYNA* software are similar to the results obtained in the *Ansys Fluent* software.

Although the results in both software are similar, it should be noted that the prescription of boundaries is much more difficult in *LS-DYNA* in comparison to *Fluent*. In order to achieve realistic blood flow, we used a combination of several keywords in *LS-DYNA* input file and prescribed inlet and outlet velocities by defining appropriate boxes. Execution time is also significantly longer for SPH analysis, but this method can be used for more sophisticated analysis of fluid-structure interaction within the Lagrangian material framework, which will be the focus of our future research.

Acknowledgements: This research is supported by the Ministry of Science, Technological Development and Innovations, Republic of Serbia, Agreement No. 451-03-47/2023-01/200378 and Grant TR32036.

References:

[1] Moosavi MH, Fatouraee N, Katoozian H, Pashaei A, Camara O, Frangi AF. Numerical simulation of blood flow in the left ventricle and aortic sinus using magnetic resonance

imaging and computational fluid dynamics, Comput Methods Biomech Biomed Engin., Vol. 17, Iss. 7, 740-749, 2012.

[2] Pedrizzetti, G., La Canna, G., Alfieri, O. et al., The vortex—an early predictor of cardiovascular outcome?, Nat Rev Cardiol, Vol 11, 545–553, 2014

[3] Mele, D., Smarrazzo, V., Pedrizzetti, G., Capasso, F., Pepe, M., Severino, S et al., Intracardiac Flow Analysis: Techniques and Potential Clinical Applications, Journal of the American Society of Echocardiography, Vol. 32, Iss. 3, 319-332, 2019.

[4] Elbaz, M. S. M., van der Geest, R. J., Calkoen, E. E., de Roos, A., Lelieveldt, B. P. F., Roest, A. A. W., Westenberg, J. J. M., Assessment of viscous energy loss and the association with three-dimensional vortex ring formation in left ventricular inflow: In vivo evaluation using four-dimensional flow MRI, Magnetic Resonance in Medicine, Vol. 77, Iss. 2, 794-805, 2016.

[5] Eriksson, J., Dyverfeldt, P., Engvall, J., Bolger, A. F., Ebbers, T., Carlhäll, C. J., Quantification of presystolic blood flow organization and energetics in the human left ventricle, American Journal of Physiology-Heart and Circulatory Physiology, Vol. 300, Iss. 6, H2135-H2141, 2011.

[6] Xu F., Kenjereš S., Numerical simulations of flow patterns in the human left ventricle model with a novel dynamic mesh morphing approach based on radial basis function, Computers in Biology and Medicine, Vol. 130, 2021.

[7] Liu G., Liu M., Smoothed Particle Hydrodynamics, World Scientific Publishing, 2003.

[8] Vignjević R., Campbell J., Brief review of development of the smooth particle hydrodynamics (SPH) method, IConSSM 2011 - The 3rd International Congress of Serbian Society of Mechanics, Vlasina lake, Serbia, 5-8 July 2011, 24-43.

[9] Gingold R. A., Monaghan J. J., Smoothed particle hydrodynamics: theory and application to non-spherical stars, Monthly Notices Royal Astronomical Society, Vol. 181, 375-389, 1997.

[10] Lucy L. B., A numerical approach to the testing of fusion process, Astronomical Journal, Vol. 88, 1013-1024, 1997.

[11] Monaghan, J., J., Pongracic H., Artificial viscosity for particle methods, Applied Numerical Mathematics Vol. 1, 187-194, 1985.

[12] Libersky L. D., Petschek A. G., Carney T. C., Hipp J. R., Allahdadi F. A., High Strain Lagrangian Hydrodynamics: A Three-Dimensional SPH Code for Dynamic Material Response, Journal of Computational Physics, Vol. 109. 67-75, 1993.

[13] ANSYS Fluent Theory Guide, Ansys Inc., 2013.

[14] Bathe K., Finite element Procedures, Prentice Hall: Pearson Education, Inc., 2006.

[15] Kojic M., Filipovic N., Stojanovic B., Kojic N., Computer Modeling in Bioengineering: Theoretical Background, Examples and Software, John Wiley & Sons, 2009.

[16] Vacondio R., Altomare C., De Leffe M., Hu X., Le Touzé D., Lind S., Marongiu J-C., Marrone S., Rogers B., D., Souto-Iglesias A., Grand challenges for Smoothed Particle Hydrodynamics numerical schemes. Computational Particle Mechanics, Vol. 8. 575–588, 2021.

[17] LS-DYNA Theory Manual, Livermore Software Technology Corporation, 2022.

[18] Tomasevic S., Milosevic M., Milicevic B., Vladimir Simic V., Prodanovic M., Mijailovich S., Filipovic N., Computational Modeling on Drugs Effects for Left Ventricle in Cardiomyopathy Disease, Pharmaceutics, Vol. 15, No. 3. 793, 2023.

[19] Milošević M., CAD Solid and Field, https://github.com/miljanmilos/CAD-Solid-Field