# Comparison between Fluid-structure interaction simulations and computational fluid dynamics applied to thoracic aorta

Anja Horvat[*], Željko Tuković[*]

[*]*University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Zagreb, Croatia*

*E-mails: {anja.horvat, zeljko.tukovic}@fsb.hr*

*Keywords: Fluid – structure interaction, aorta, computational fluid dynamics, finite volume method*

## 1. Introduction

Aortic dissection is serious cardiovascular disease, where the inner layer of aorta is injured enabling blood flow inbetween layers. Complications can lead to death, however, preferred method of treating isn't yet defined. In order to assist clinical decision making, patient – specific computational modelling has lately been used. In this study, partitioned FSI solver based on finite volume discretisation is used to calculate interaction between blood flow trough idealised thoracic aorta and flexible vessel wall, as a first step towards the application of solver on more complex geometries. In order to determine whether the FSI solver is necessary, the comparison of blood flow through the same geometry with rigid vessel wall and flexible vessel wall has been made.

## 2. Methods

Partitioned FSI solver based on finite volume discretisation is used to calculate interaction between blood flow trough idealised thoracic aorta and flexible vessel wall. The flow of incompressible, Newtonian fluid is modelled as laminar, described with Navier Stokes equations in arbitrary Lagrangian – Eulerian formulation. Although the blood is non – Newtonian fluid, the assumption of Newtonian model is common, as the non – Newtonian effects of blood are not pronounced in large vessels [1].

The solid deformations is described by the incompressible, neo-Hookean hyper-elastic material model in total Lagrangian formulation. Solid properties are homogeneous and isotropic throughout its thickness. Fluid and solid properties are given in Table 1.

**Table 1. Fluid and solid properties**

| Property | value | Unit |
|---|---|---|
| Fluid properties | | |
| Fluid density | 1000 | $kg/m^3$ |
| Kinematic viscosity | $3^{-6}$ | $m^2/s$ |
| Solid properties | | |
| Solid density | 1200 | $kg/m^3$ |
| Young's modulus | $8^5$ | $N/m^2$ |
| Poisson | 0.5 | - |

Second – order accurate finite volume validamethod is used for the discretisation in space for both fluid and solid and first – order accurate implicit Euler method is used for temporal discretisation. The solid domain is discretised by the 4 – layered prismatic mesh. On the other hand, mostly polyhedral unstructured mesh with 4 prismatic layers at the interface is used to discretise the fluid domain. The total number of control volumes employed for solid and fluid domain is 74088 and 112002, respectively. Solid domain is divided into inlet, outlets, interface and outer wall. The radial slip boundary condition is imposed on outlets and inlets which enables radial displacements and prevents displacements in the axial direction. The outer wall is defined as traction free. The spatial discretisation of interface on the solid domain and the fluid domain is not identical. Consequently, GGI (General Grid Interface) interpolation is performed to transfer fields from one interface to another. The fluid domain is divided into inlet, outlets and interface. Time – varying flow is imposed on the inlet obtained by measuring flow through aorta and three – element Windkessel is imposed on outlets. The Windkessel parameters were set ensuring 10% outflow from smaller branches, and 70 % outflow from the main outlet. Coupling between fluid and solid is performed using Robin – Neumann partitioned procedure proposed in [2], where the fluid pressure at the interface is limited by solid inertia and thus, ensuring stability. Simulations were performed using open – source toolbox solids4Foam [3] developed for OpenFOAM, capable of solving fluid – structure interaction.

## 3. Results

CFD and FSI simulations of volumetric flow are compared on idealised geometry of the aorta in figure 1. FSI simulations were performed on stress-free geometry. CFD simulations were performed on mean volume geometry obtained from FSI simulation. CFD simulation predicted higher velocity magnitudes at the moment of maximal flow compared to FSI simulations. On the contrary, CFD predicted lower velocity magnitudes at the moment of minimal flow.

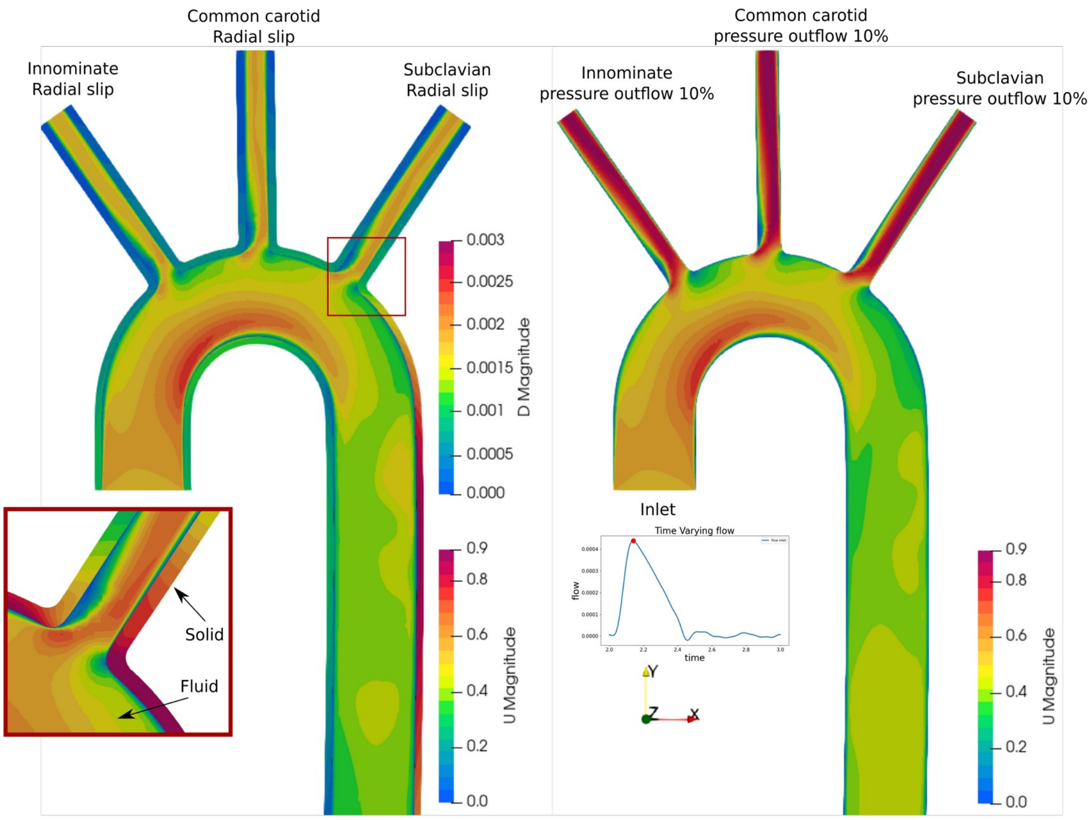


**Figure 1. Comparison between velocity fields obtained with FSI and CFD simulations (left *FSI*, right *CFD*)**

## 4. Conclusions

The comparison between FSI and CFD simulations and insight into differences in velocity fields between them has been presented. Due to the differences, the use of more computationally demanding FSI simulations is required in future work.

## 5. Acknowledgments

This work was supported by grants from the Croatian Science Foundation (project IP-2020-02-4016).

## References


[1] Sochi, Taha, *Non – Newtonian Rheology in Blood Circulation*. University College London, Department of Physics & Astronomy, London, 2013.

[2] Ž. Tuković, M. Bukač, P. Cardiff, H. Jasak, and A. Ivanković, *Added mass partitioned fluid–structure interaction solver based on a robin boundary condition for pressure*, in Selected Papers of the 11th Workshop, pp. 1–22., Springer International Publishing, 2019.

[3] P., Cardiff, A., Karač, P. D., Jaeger, H., Jasak, J., Nagy, A., Ivanković, and Ž., Tuković, *An open-source finite volume toolbox for solid mechanics and fluid-solid interaction simulations*, 2018.