Abstract ID 56

A STUDY ON THE COMPUTATIONAL FLUID DYNAMICS AND FLUID STRUCTURE INTERACTION MODELS FOR THE HEMODYNAMIC OF ASCENDING THORACIC AORTA ANEURYSM

Rodrigo Baptista Valente[1], André Filipe Geraldes Mourato[1], Moisés Gonçalo de Brito[1], José Manuel Cardoso Xavier[1], António Cruz Tomás[2], Stéphane Avril[3]

[1]Nova School of Science and Tecnology, DEMI, Lisbon, Portugal
[2]Hospital Santa Marta, Department of Cardiothoracic Surgery, Lisbon, Portugal
[3]University Hospital of Saint-Étienne, Department of Radiology, Saint-Étienne, France
rb.valente@campus.fct.unl.pt, af.mourato@campus.fct.unl.pt, moisesbrito@fct.unl.pt, jmc.xavier@fct.unl.pt, acruztomas@gmail.com, avril@emse.fr

Keywords: Ascending thoracic aorta aneurysm, Computational fluid dynamics, Fluid-structure interaction, Finite Element Method, hemodynamics

Summary: The ascending aorta is the first arterial segment that follows the left heart and is mechanically stressed by the blood flow that initiates the systemic circulation. Over time and due to changes in the microstructure of the vessel wall, the aorta may lose elasticity and dilate. If it increases by 50% over its original diameter in the ascending aorta, it is called an aneurysm. According to the European Society of Cardiology (ESC), this pathology should be diagnosed by measuring the diameter of the aorta. However, in practice, some patients have been observed to develop rupture or aortic dissection even before the criterion of minimum diameter has been reached. In this sense, it is generally accepted that the geometric diameter criterion must be assisted by other approaches that analyze multiple aspects to support the clinical medical diagnosis. Computational Fluid Dynamics (CFD) can provide a suitable tool, as it allows a real-time evaluation of the hemodynamic behaviour and is able to estimate future life-threatening situations, even after surgery, by geometry modification. To improve the reliability of the results, however, the CFD approach can be enhanced by considering the possibility to couple Fluid-Structure Interaction (FSI), thus incorporating the elastic properties of the structure and their effects on the hemodynamics to be evaluated in the numerical simulations. The present work aims to develop a methodology to perform a numerical study of CFD with FSI for the hemodynamic of ascending thoracic aorta aneurysm (ATAA). The geometric model and boundary conditions were extracted from Computed Tomography scan (CT) and 4D Magnetic Resonance Imaging (4D-MRI) imaging of the pilot patient. The open-source SimVascular software was used as a 3D segmentation tool, mesh generation, and numerical solver of Navier-Stokes equations and FSI. This methodology will improve the process of generating CFD numerical models to support the clinical outcome. In the future, this process will be automated to ensure consistent results.