Abstract ID-247

NUMERICAL MODELLING OF THE BLOOD FLOW IN RIGHT CORONARY ARTERY USING EULER-EULER MULTIPHASE APPROACH

Maria Gracka, Bartlomiej Melka, Ziemowit Ostrowski

Biomedical Engineering Laboratory, Institute of Thermal Technology,
Silesian University of Technology, Poland
maria.gracka@polsl.pl, bartlomiej.melka@polsl.pl, ziemowit.ostrowski@polsl.pl

Keywords: hemodynamics, CFD simulations, cardiovascular blood flow, multiphase approach

Summary: In recent years Computational Fluid Dynamics (CFD) methods proved their applicability in bio-engineering. The cardiovascular diseases presently have become the leading causes of death in the world. Accurate model and understanding of the basic mechanisms and phenomena occurring in the cardiovascular system may be useful for early detection and diagnosis of developing lesions. The blood flow is related to the geometry of the vessels and to its rheological properties. To better understand cardiovascular diseases the hemodynamic date on the roles of physiologically critical blood particulates is needed. The numerical simulation could provide the blood flow patterns and the particulate buildup that could be useful in predicting the most vulnerable spaces to the accumulation of the atherosclerotic plaques.

The scope of this work was a numerical analysis of the blood flow within the human blood vessel. The multifluid transient CFD simulation for describing the hemodynamics in the realistic right coronary artery has been performed. The Eulerian multiphase approach was used in the model of the blood flow which assumed blood properties as a nonhomogenous mixture of its two main components: plasma and Red Blood Cells (RBC). The volume concentration of RBC was defined on the level of 45% and plasma constituted 55% of volume. The numerical model assumed rigid walls of the blood vessel. Using User Defined Functions (UDFs) the pulsatile boundary condition has been implemented to mimic periodic cardiac cycle. The velocity profile using typical cardiac waveform with a cardiac period of 0.735s has been set as an inlet boundary condition (BC) and the constant pressure condition as an outlet BC has been used. To develop numerical models of the blood flow within human coronary artery the commercial software ANSYS Fluent (ANSYS Inc., USA) has been used.

Acknowledgements: The research of was supported by Faculty of Energy and Environmental Engineering within Ministry of Science and Higher Education (Poland) statutory research funding scheme. This help is gratefully acknowledged herewith.