15th International Symposium on Computer Methods in Biomechanics and Biomedical Engineering
and
3rd Conference on Imaging and Visualization
CMBBE 2018
P. R. Fernandes and J. M. Tavares (Editors)

Abstract ID-219

SIMULATION OF AN INTRACORPOREAL MEMBRANE CATHETER FOR CO2 REDUCTION IN BLOOD

***Michael Harasek*[(1)], *Margit Gföhler*[(1)], *Benjamin Lukitsch*[(1)], *Christoph Janeczek*[(1)], *Alen Karabegovic*[(1)], *Florentine Huber-Dangl*[(1)], *Claus Krenn*[(2)], *Roman Ullrich*[(2)]**

[(1)]Vienna University of Technology, Austria
michael.harasek@tuwien.ac.at, margit.gfoehler@tuwien.ac.at, benjamin.lukitsch@tuwien.ac.at, christoph.janeczek@tuwien.ac.at, alen.karabegovic@tuwien.ac.at, florentine.huber-dangl@tuwien.ac.at

[(2)]CCORE Technology GmbH, Austria
claus.krenn@ccore.at, roman.ullrich@ccore.at

Keywords: mass transfer modeling, intracorporeal membrsne catheter, membrane device, computational fluid dynamics, artificial lung, acute respiratory distress syndrome

Summary: An approach to prevent the acute respiratory distress syndrome (ARDS) is to reduce the blood CO_2 concentration before the blood reaches the lungs. Intracorporeal membrane catheters have recently been proposed as a novel approach. The catheter is implanted into a large body vein where CO_2 can be transferred from blood side to an take-up fluid which is pumped to an extracorporeal regeneration device. Here, CO_2 is removed from the secondary fluid and the fluid is ready to be pumped back into the body in a closed loop.
The main challenge is to design the membrane module in a way that it is small enough to be implanted as catheter into the patient body still providing the required CO_2 removal capacity.
The influence of key parameters on the performance of membrane catheter designs is critical. Using experimental methods is usually very time consuming and expensive. They provide point-wise information which might not be sufficient for perfect optimization since local dead zones or other flow irregularities might be missed. Simulation tools can provide a rather fast and flexible platform for investigation of different designs and operational parameters. Computational fluid dynamics (CFD) is a method using the solution of fluid flow equations (Navier-Stokes and continuity equations) based on numerical approaches with time and space resolution. In this study an open-source CFD platform (OpenFOAM®) was used for development of a new solver for 3D membrane modeling. Driving forces for mass transfer - partial pressure differences - are calculated based on local information. The code was validated against measurements and other 1D simulation tools for both carbon dioxide and oxygen transfer.
The flow structure in the shell side of a newly designed membrane module for CO_2 removal was simulated and the flow pattern and residence time of the blood in the shell was studied. The simulation gives insight to flow and pressure fields and residence time distribution which can be used for improvements and re-design of device such as membrane fiber arrangement, turbulence promotion, reduction of concentration polarization effects and prevention of dead zones.