Medical Engineering and Physics 000 (2016) 1–11

Contents lists available at ScienceDirect

Medical Engineering and Physics

journal homepage: www.elsevier.com/locate/medengphy

Investigation of hemodynamics during cardiopulmonary bypass: A multiscale multiphysics fluid–structure-interaction study

Michael Neidlin*, Simon J. Sonntag, Thomas Schmitz-Rode, Ulrich Steinseifer, Tim A S Kaufmann

Department of Cardiovascular Engineering, Institute of Applied Medical Engineering, Helmholtz Institute, RWTH Aachen University, Aachen, Germany

ARTICLE INFO

Article history:
Received 8 December 2014
Revised 4 January 2016
Accepted 31 January 2016
Available online xxx

Keywords:
Computational fluid dynamics
Fluid structure interaction
Cannulation
Cardiopulmonary bypass

ABSTRACT

Neurological complications often occur during cardiopulmonary bypass (CPB). Hypoperfusion of brain tissue due to diminished cerebral autoregulation (CA) and thromboembolism from atherosclerotic plaque reduce the cerebral oxygen supply and increase the risk of perioperative stroke. To improve the outcome of cardiac surgeries, patient-specific computational fluid dynamic (CFD) models can be used to investigate the blood flow during CPB.

In this study, we establish a computational model of CPB which includes cerebral autoregulation and movement of aortic walls on the basis of in vivo measurements. First, the Baroreflex mechanism, which plays a leading role in CA, is represented with a 0-D control circuit and coupled to the 3-D domain with differential equations as boundary conditions. Additionally a two-way coupled fluid–structure interaction (FSI) model with CA is set up. The wall shear stress (WSS) distribution is computed for the whole FSI domain and a comparison to rigid wall CFD is made. Constant flow and pulsatile flow CPB is considered.

Rigid wall CFD delivers higher wall shear stress values than FSI simulations, especially during pulsatile perfusion. The flow rates through the supraaortic vessels are almost not affected, if considered as percentages of total cannula output. The developed multiphysic multiscale framework allows deeper insights into the underlying mechanisms during CPB on a patient-specific basis.

1. Introduction

Perioperative stroke is a common complication during cardiopulmonary bypass (CPB), a standard technique used in cardiac surgeries. One of the main causes of stroke is the embolization of cerebral vessels and the reduced oxygen supply to the brain's tissue [1]. A proper perfusion can reduce the risks of stroke and improve the success rate of open heart surgeries, a challenge cardiac surgeons are constantly facing [2-4]. Mobilization of ruptured atherosclerotic plaque due to elevated wall shear stress (WSS) and wall pressure values is the major reason of the emerging vessel occlusion [5–7]. Therefore, assessment of these risk factors is a key point in biomedical research.

Investigations of the blood flow during extracorporeal circulation have been conducted by means of computational fluid dynamics (CFD) in the past [7–11]. Especially, the positions of the outflow cannula and the perfusion technique (pulsatile vs. non-pulsatile) have been analyzed with regard to the cerebral perfusion and areas of increased wall shear stresses. However, the movement of the vessel walls has been neglected in these studies. A consideration of compliant walls needs an implementation of structural deformations in the CFD domain and an establishment of a fluid–structure-interaction (FSI) framework. Analyses of blood flow in human blood vessels via FSI have been carried out in several studies with idealized and real geometries [12–14], nevertheless this has not been transferred to CPB conditions.

Since commonly only continuous flow CPB is analyzed, one argument against the use of FSI in simulations of CPB is that a constant flow does not create any vessel movement. However, this statement has not been investigated numerically or experimentally. Further on, pulsatile CPB is still available and used in the clinic as there is controversy concerning risks and advantages of each technique [7,15,16].

In this study, we develop a two-way fluid–structure-interaction model for physiological flow in the human aortic arch and apply this model to continuous flow and pulsatile flow CPB. Additionally, we implement a recently presented model of cerebral autoregulation in the FSI framework [17,18] to establish a multiscale and multiphysic description of CPB. The aim is to decide whether, and if so, how to apply FSI in computational studies of CPB.

* Corresponding author. Tel.: +49241/8080540.
E-mail address: neidlin@hia.rwth-aachen.de (M. Neidlin).

http://dx.doi.org/10.1016/j.medengphy.2016.01.003

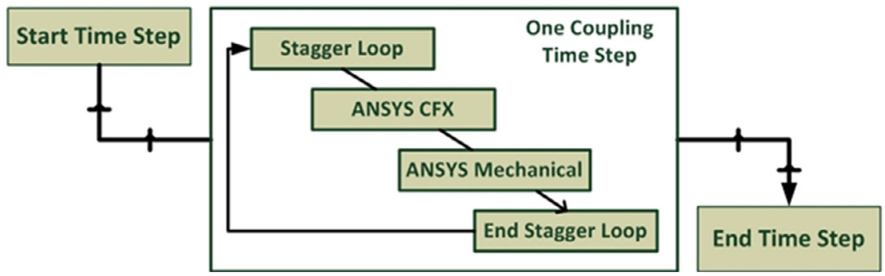

Fig. 1. Overview of one coupling time step.

2. Materials and methods

2.1. Simulation setup overview

There are mainly two approaches to perform a two-way coupled FSI simulation, the monolithic and the partitioned approach. The monolithic approach solves the fluid and solid domain simultaneously within the same solver and thus by using one set of equations, resulting in relatively stable simulations. However, separate structural simulations are typically performed using the Finite Element Method (FEM), while Computational Fluid Dynamics are typically performed using the Finite Volume Method (FVM). Consequently, the usage of commercial packages for the monolithic approach is limited [19].

The second approach for FSI simulations is the partitioned approach, in which the fluid and solid domain are solved separately with two different solvers and the information between both systems is transferred through a domain interface. In this study, the partitioned approach was applied using commercial software (Ansys Multiphysics, Ansys Germany Inc., Otterfing, Germany). The solid domain was solved within ANSYS Mechanical and the fluid domain was solved within ANSYS CFX. The coupling was performed using the Ansys Multifield solver (MFX). Up to 20 internal coupling iterations were executed within each domain per coupling time step until the results converged. The procedure of one coupling time step is shown in Fig. 1. The CFX solver calculates the solution of the fluid domain and thereby also the load on the fluid–solid interface, in this case the vessel wall. The structural solver receives the load and calculates the displacement of the vessel wall, which is transferred back to the fluid domain. This procedure is one iteration loop and is repeated until the loading and the displacement are converged. Before a partitioned two-way fluid–structure interaction simulation can be performed, the two solvers need to be set up separately.

2.2. Model creation and mesh generation

The model creation process is divided into two parts. Part one is model creation of the CFD model which represents the fluid part of the cardiovascular system. The second part is model creation for the structural analysis which represents the vessel wall. The inner surface of the vessel wall and the outer surface of the fluid domain are the interface between both models.

First, magnetic resonance imaging (MRI) scans of a 28 year old, healthy male volunteer were obtained. Based on this data, a three-dimensional model of the aortic arch including the subclavian arteries and the carotid and vertebral vessels was created with commercial software (Mimics, Materialise Inc., Leuven, Belgium). For the segmentation of the structures a threshold mask based on Hounsfield units was applied. Afterwards standard image processing tools as static and dynamic region growing and morphology operations were used. Manual corrections were necessary prior to creation of a 3-D surface geometry. This geometry was post-processed with 3matic 7.0 (3matic, Materialise Inc., Leuven, Belgium) to repair the model, smoothen the vessel walls and place an 18 Fr outflow cannula in the ascending aorta, a standard positioning technique in CPB.

2.3. Fluid domain mesh

An unstructured tetrahedral mesh was generated within the geometry for the model (Ansys ICEM CFD, Ansys Germany Inc., Otterfing, Germany). In addition to tetrahedral elements, three layers of prismatic elements were generated around the inside of the vessel wall to resolve the boundary layers of the blood flow. In a mesh independence study the volume flow rate through the outlets and the maximum pressure at the area where the cannula jet hits the aortic wall were taken as the control parameters. Independence was assumed, after the changes of these parameters were smaller than 3%. The number of elements of the chosen mesh was approximately 1 million.

2.4. Solid domain mesh

The FEM model creation was based on the same CAD model, but only the boundary representing the vessel wall was considered. At first, a triangle surface mesh was created on that surface. Based on this mesh, prismatic elements were extruded in the outward direction to represent the vessel wall. The thickness of the extruded layer was 1.0 mm. For a better stability of the structural simulation, these elements were converted to a higher order, resulting in type Solid-186 with 20 nodes per element and a total number of 49,000 elements.

2.5. Fluid domain simulation setup

The Computational Fluid Dynamics (CFD) simulations were set up based on [17,20]. The fluid was a non-Newtonian blood model taken from [21] with a density of 1056.4 kg/m^3 and a hematocrit of 44%. The Shear Stress Transport model was used for the description of turbulence flow areas with an upwind advection scheme and implicit first order backward Euler method as the time-integration technique. The near wall flow was resolved with automatic wall functions. The according maximum y^+ values for the maximum cannula flow were y^+ (cf-CPB) = 14 and y^+ (pf-CPB) = 4 8 at a thickness of the first prismatic layer of 0.3 mm. At the inlet, the turbulence intensity was set to 0.05 and the ratio between eddy viscosity and dynamic viscosity was set to 10. Up to 4 internal iterations were solved per time step until the average changes in the transport equations were equal or smaller than the specified convergence target, which was $1*10^{-4}$. The boundary representing the vessel wall was set as a fluid-solid-interface. The subclavian arteries and the descending aorta were modeled as an opening with an additional pressure loss. The blood entering or leaving the domain has to overcome this pressure difference Δp. Therefore unrealistic backflow as observed in [20] can be prevented. This term can

Please cite this article as: M. Neidlin et al., Investigation of hemodynamics during cardiopulmonary bypass: A multiscale multiphysics fluid–structure-interaction study, Medical Engineering and Physics (2016), http://dx.doi.org/10.1016/j.medengphy.2016.01.003