

Correct rheology simulation on compliant thoracic aorta model : comparison between CFD and MRI velocity measurements

R. MORENO*†, M. CHAU□, B. TAYLLAMIN‡, H. ROUSSEAU†, F. NICOU‡

† INSERM U 858 I2MR, CHU de Rangueil, Toulouse

□ ASA, Montpellier

‡ Institut de Mathématiques et de Modélisation de Montpellier CNRS - UMR 5149

*Corresponding author. Email: ramoroa@gmail.com

Keywords: CFD; MRI; Aorta

1 Introduction (times new roman 12 et gras)

Experimental modelling of blood flow through the pulsatile and compliant thoracic aorta is validated by combining computational fluid dynamics (CFD) and 7D magnetic resonance imaging (7D-MRI). This method provides a complete velocity field used here for flow boundary conditions setup (inlet and outlet) and for correlating numerical results with experimental data set.

We present a complete, optimized computational chain whose input comes from an entirely non-invasive 4D MRI protocol that provides time varying geometry and flow rates. Its output is a realistic functional imaging description of the arterial tree region of interest. Results were obtained through parallel computing with AVBP software (CERFACS, Toulouse, France) and classical Arbitrary Lagrangian Eulerian (ALE) formulation.

There is no doubt that advances in computational resources and infrastructure will benefit to the medical community. In return, more and more challenging applications will rise and stimulate research and development

2 Methods

All the images were obtained with a 1.5 T MR scanner (Siemens Medical, Germany).

The objective of the following processing steps was to translate the image data set into patient-specific

boundary conditions (velocity conditions) that are suitable for CFD calculations.

MR Angiography. Firstly, a routine contrast-enhanced MR angiography (CE-MRA) was performed using a three-dimensional (3D) slab covering the vessels of interest with an injection of 18 ml of gadolinium-diethylenetriamine penta acetic acid (Gd-DTPA) followed by 15 ml of saline, all delivered at 2 ml/sec. The sequence was performed on sagittal-oblique planes, parallel to the major aortic axis, in order to cover the whole aorta geometry (field of view, 450x450x126mm) with a spatial resolution of 0,88x0,88x0,88 mm³.

MR velocity quantification. Secondly, 7D velocity phase-contrast (PC) imaging was performed with the same orientation of contrast enhanced acquisition. Magnitude (anatomy) and phase-contrast (velocity) images were reconstructed over the cardiac cycle with spatial resolution of 1.5x1.5x4.5mm³. Velocities (V_x, V_y and V_z) were encoded in the three directions of space with an encoding level adapted to each patient to prevent aliasing.

Surface extraction and Moving wall. An initial computational grid was obtained by the discretization of segmented contrast-enhanced MR angiography (Amira 4.1, TGS, Mercury Computer Systems, USA). Wall movements were imposed to the initial grid according to dynamic acquisition (anatomy), by means of a “non linear transformation field”, an in-house algorithm (Matlab 7.0, the MathWorks, Inc).

Numerical simulation. The flow simulations were performed using the finite volume (FV) method, as implemented in the AVBP code (CERFACS, European Center for Research and Advanced Training in Scientific Computation, Toulouse, France). The FV method used in the code solves the full Navier-Stokes equations that governs the flow, by an efficient explicit Arbitrary Lagrangian Eulerian (ALE) formulation, which allows to impose the motion of tetraedral mesh within cardiac cycles. Hemodynamic boundary conditions (time-dependent functions) were synchronized with the wall motion and were imposed in the form of speed profiles at the entry (ascending aorta) and exit (descending aorta, supraortic vessels) of the numerical field (aorta district).

3 Results and Discussion

Numerical results and experimental measurements were exported to the visualisation toolkit format (VTK) for comparison.

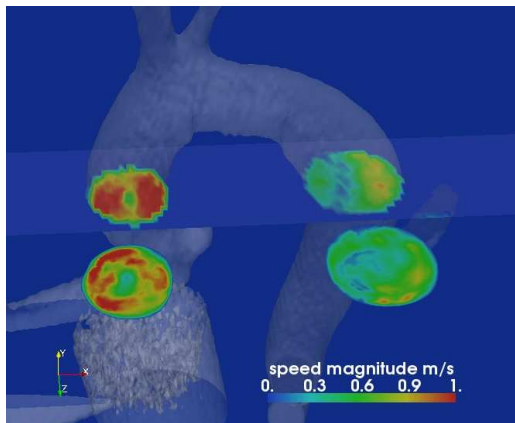


Figure 1: MRI (up) / CFD at 440ms, axial view.

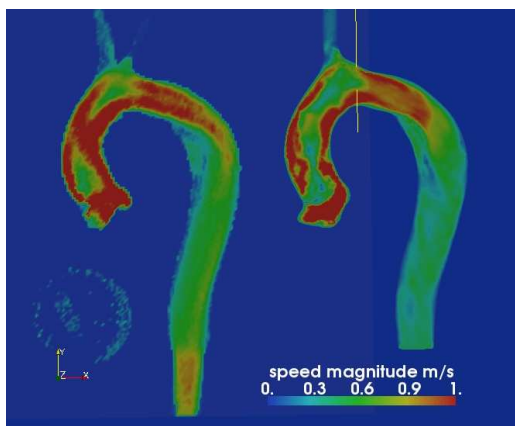


Figure 2: MRI (left) / CFD at 440ms, sagittal view.

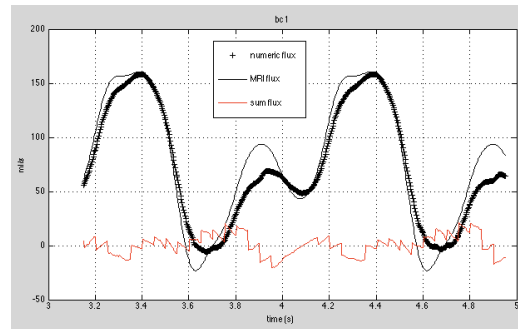


Figure 3: numerical (*) and experimental flow at inlet boundary condition (mL).

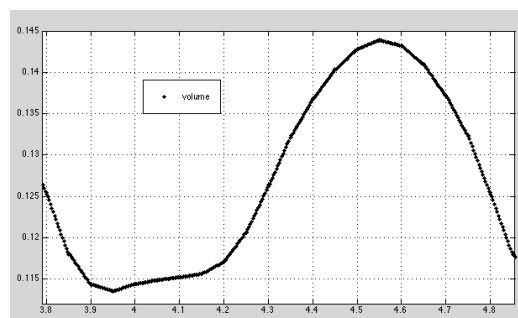


Figure 4: Grid volume évolution (L).

4 Conclusions

The proposed approach permits the computation of the blood flow under realistic in vivo, time evolving and flow controlled conditions. It is much simpler than the full coupled fluid-structure problem and has the potential to provide a better picture of the specific hemodynamic status. The method gives a direct way to impose realistic wall interaction to hemodynamic time boundary conditions, from medical examinations performed into a simple MR protocol. It seems obvious that a realistic simulation needs to be done according to the rheology of the wall.