Image based, geometry prescribed CFD of intra-cardiac flows on a dynamic heart phantom

R. Hvid[1], M. S. Traberg[1]

1. DTU Health Technology, Technical University of Denmark, Lyngby, Denmark

INTRODUCTION A pipeline for simulating intra-cardiac blood flows from computed tomography angiography (CTA) is presented. The pipeline computes a geometry prescribed CFD based on segmentation and registration of the CTA volumes. The simulation inputs are discrete which has previously been a challenge to solve robustly.

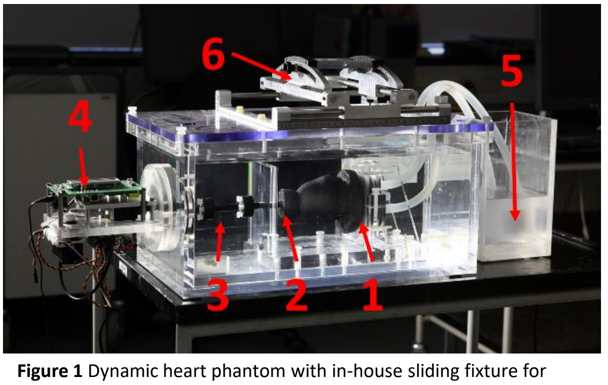

Figure 1 Dynamic heart phantom with in-house sliding fixture for ultrasound probes. **1:** Phantom base **2:** phantom apex **3:** Rod for compression **4:** Servo motors and programmable micro controller **5:** fluid reservoir **6:** Sliding fixture for ultrasound probe

EXPERIMENTAL METHODS Several experiments were made to provide model input and ground truth measurements for validation.

Phantom: The DHP mimics a human heart with two ventricles without heart valves. The phantom ventricles are compressed 10mm cyclically at 75 beats per minute in the direction of the long axis. Blood mimicking fluid is pumped through the inlet of the right phantom ventricle at 5L/min.

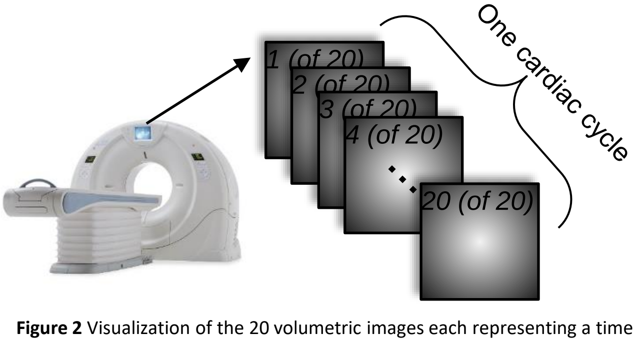

Figure 2 Visualization of the 20 volumetric images each representing a time instance of the cardiac cycle.

Ultrasound: The DHP is scanned using ultrasound vector flow imaging (VFI), see figure 4, providing planes with time-dependent fluid velocities (2D+t). 8 planes are obtained for multiple cardiac cycles each.

CTA: The DHP is scanned with CTA. The CTA data are processed and used as model input for the CFD simulation.

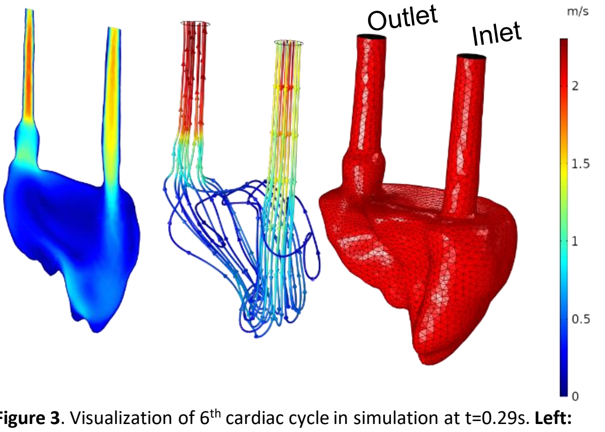

Figure 3. Visualization of 6th cardiac cycle in simulation at t=0.29s. **Left:** slice showing absolute velocity **Center:** Streamlines **Right:** Surface mesh with inlet and outlet annotation.

PRE PROCESSING The fluid domain was segmented in MATLAB as a surface mesh. The surface movement was estimated using volumetric image registration between the 20 CTA volumes. The resulting displacement field $\vec{d}(\vec{x},t)$ contains 3D vectors in a discrete 3D+t space.

$$\vec{d}(\vec{x},t) = \begin{bmatrix} d_x(x,y,z,t) \\ d_y(x,y,z,t) \\ d_z(x,y,z,t) \end{bmatrix}$$

COMPUTATIONAL METHODS A moving mesh was defined with a deforming domain and prescribed mesh displacement. The mesh movement is computed in a separate study. The CFD was modeled as a laminar, isothermal and non-compressible flow using Navier-Stokes equation. The flow was computed for 6 cardiac cycles.

Boundary conditions: Inlet: 5L/min, Outlet: 0 Pa, Wall: $\vec{u}_{wall} = \vec{u}_{\text{fluid}}$

RESULTS The CFD pipeline was successfully implemented. The flow patterns are visually assessed to be in correspondence with the measurement. See figure 4.

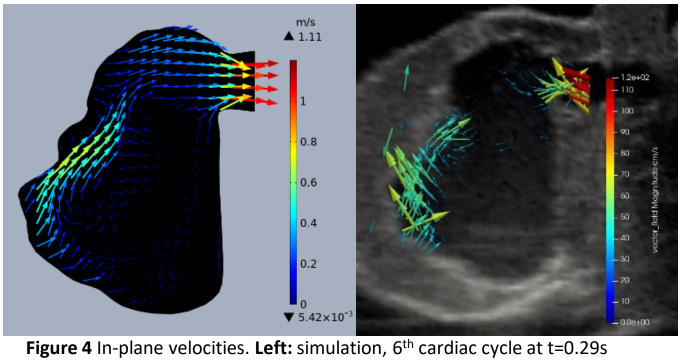

Figure 4 In-plane velocities. **Left:** simulation, 6th cardiac cycle at t=0.29s **Right:** VFI measurement of phantom 0.29s in cardiac cycle

Excerpt from the Proceedings of the 2020 COMSOL Conference