11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

VALIDATION OF AN OPEN SOURCE FRAMEWORK FOR THE SIMULATION OF BLOOD FLOW

A. Quaini[1*], T. Passerini[2], U. Villa[3], A. Veneziani[4] and S. Canic[2]

[1] Department of Mathematics, University of Houston, Houston TX (USA), quaini@math.uh.edu and canic@math.uh.edu
[2] Siemens Corporation, Corporate Technology, Imaging and Computer Vision, Princeton NJ (USA), tiziano.passerini@siemens.com
[3] Lawrence Livermore National Laboratory, Livermore CA (USA), villa13@llnl.gov
[4] Department of Mathematics and Computer Science, Emory University, Atlanta GA (USA), ale@mathcs.emory.edu

Key words: *Computational Fluid Dynamics, Hemodynamics, Experimental validation, Fluid-Structure Interaction.*

We discuss the validation of an open source framework for the solution of problems arising in hemodynamics. The proposed framework is assessed through validation against experimental data for fluid flow in an idealized medical device with rigid boundaries and a numerical benchmark for flow in compliant vessels. The core of the framework is an open source parallel finite element library called LifeV (www.lifev.org) that features several algorithms to solve both fluid and fluid-structure interaction problems.

We first consider a benchmark proposed by the U.S. Food and Drug Administration [1], which consists in simulating the flow of an incompressible Newtonian fluid in an idealized medical device shaped like a nozzle (conical convergent, narrow throat, and sudden expansion; see Fig. 1, left). The system is studied in a variety of conditions: laminar, transitional, and turbulent flow regimes. The numerical results are in good quantitative agreement with the measured axial components of the velocity and pressures for different flow rates corresponding to the three regimes. Using validation metrics proposed in [2], for several axial locations we evaluate the relative mismatch between computed and measured values. We obtain very good estimates of the flow rate (errors within 1%) and good estimates of the velocity at all axial locations, under all exercise conditions. We emphasize the crucial role played by the accuracy in performing numerical integration, mesh, and time step to obtain a good match between numerical results and experimental measurements.

The second benchmark we consider was proposed in [3]. It is a numerical fluid-structure interaction benchmark that deals with the propagation of a pressure wave in a fluid-filled

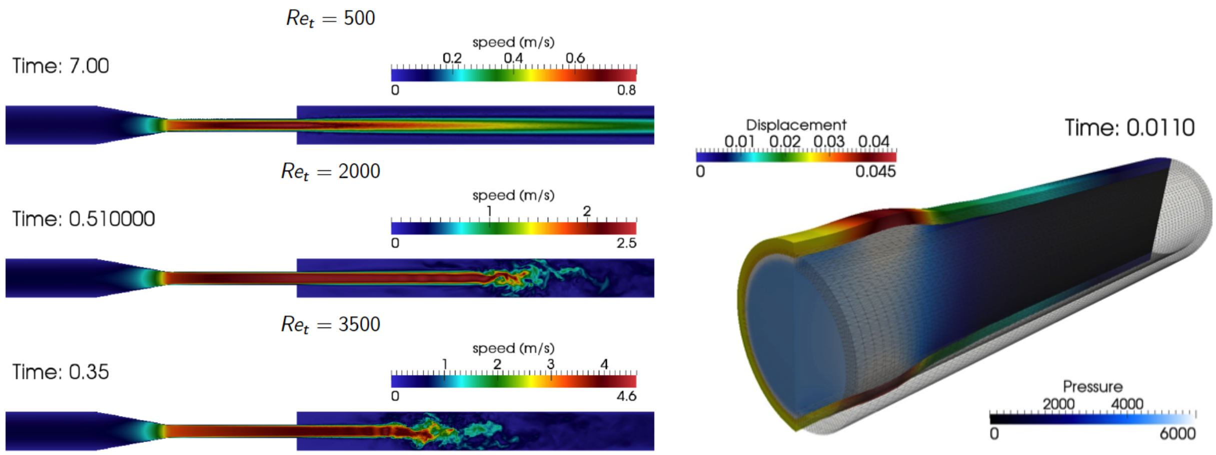

Figure 1: Left: Computed solution of the FDA benchmark in three flow regimes: laminar (top), transitional (middle), turbulent (bottom). Right: Computed solution of the fluid-structure interaction benchmark. The vessel wall is colored by the magnitude of the radial displacement (in cm), the fluid domain by the pressure (in dyn/cm2).

elastic tube (see Fig. 1, right). The computed pressure wave speed and frequency of oscillations, and the axial velocity of the fluid on the tube axis are close to the values predicted by the analytical solution associated with the benchmark (relative errors within 1%).

An important outcome of this work is the production of a suite of scripts and codes that are based on a completely open-source set of tools, and therefore will be readily shared with the community through the web portal www.lifev.org.

REFERENCES

[1] Computational Fluid Dynamics: An FDA's Critical Path Initiative. https://fdacfd.nci.nih.gov/.

[2] S. Stewart, E. Paterson, G. Burgreen, P. Hariharan, M. Giarra, V. Reddy, S. Day, K. Manning, S. Deutsch, M. Berman, M. Myers, and R. Malinauskas. Assessment of CFD performance in simulations of an idealized medical device: Results of FDA's first computational inter laboratory study. *Cardiovascular Engineering and Technology*, Vol. **3**(2):139–160, 2012.

[3] C. Greenshields and H. Weller. A unified formulation for continuum mechanics applied to fluid-structure interaction in flexible tubes. *Int. J. Numer. Methods Eng.*, Vol. **64**:1575–1593, 2005.