Journal of Biomechanics 57 (2017) 46–53

Contents lists available at ScienceDirect

Journal of Biomechanics

journal homepage: www.elsevier.com/locate/jbiomech
www.JBiomech.com

Cerebral aneurysm blood flow simulations are sensitive to basic solver settings

Kendall D. Dennis [a], David F. Kallmes [b,c], Dan Dragomir-Daescu [c,d,*]

[a] Division of Engineering, Mayo Clinic, Rochester, MN, USA
[b] Department of Radiology, Mayo Clinic, Rochester, MN, USA
[c] Mayo Clinic College of Medicine, Mayo Clinic, Rochester, MN, USA
[d] Department of Physiology and Biomedical Engineering, Mayo Clinic, Rochester, MN, USA

ARTICLE INFO

Article history:
Accepted 24 March 2017

Keywords:
CFD
Aneurysm
Sensitivity
Hemodynamics

ABSTRACT

Computational modeling of peri-aneurysmal hemodynamics is typically carried out with commercial software without knowledge of the sensitivity of the model to variation in input values. For three aneurysm models, we carried out a formal sensitivity analysis and optimization strategy focused on variation in timestep duration and model residual error values and their impact on hemodynamic outputs. We examined the solution sensitivity to timestep sizes of 10^{-3} s, 10^{-4} s, and 10^{-5} s while using model residual error values of 10^{-4}, 10^{-5}, and 10^{-6} using ANSYS Fluent to observe compounding errors and to optimize solver settings for computational efficiency while preserving solution accuracy. Simulations were compared qualitatively and quantitatively against the most rigorous combination of timestep and residual parameters, 10^{-5} s and 10^{-6}, respectively. A case using 10^{-4} s timesteps, with 10^{-5} residual errors proved to be a converged solution for all three models with mean velocity and WSS difference RMS errors less than <1% compared with baseline, and was computationally efficient with a simulation time of 62 h per cardiac cycle compared to 392 h for baseline for the model with the most complex flow simulation. The worst case of our analysis, using 10^{-3} s timesteps and 10^{-4} residual errors, was still able to predict the dominant vortex in the aneurysm, but its velocity and WSS RMS errors reached 20%. Even with an appealing simulation time of 11 h per cycle for the model with the most complex flow, the worst case analysis solution exhibited compounding errors from large timesteps and residual errors. To resolve time-dependent flow characteristics, CFD simulations of cerebral aneurysms require sufficiently small timestep size and residual error. Simulations with both insufficient timestep and residual resolution are vulnerable to compounding errors.

1. Introduction

Computational fluid dynamics is becoming a common tool for the study of hemodynamics in cerebral aneurysms. Using patient-specific geometries, it is possible to gain insight into the fluid structures present in aneurysms with great detail. Several groups have already applied CFD to study characteristics of aneurysm rupture and growth (Cebral et al., 2005b, 2011a, 2011b; Hodis et al., 2014; Kulcsar et al., 2011; Xiang et al., 2011, 2014b). Complex flow patterns, high Wall Shear Stress (WSS), low WSS and small impingement jets have all been suggested as risk factors.

Because these simulations are carried out making various assumptions and with only a small portion of the cerebral vasculature, sensitivity studies have also been performed to understand which of these parameters have significant effects on the results. Determining how to represent the lumen geometry is particularly important. The sensitivity of removing neighboring vessels (Zeng et al., 2010) and truncation location (Hodis et al., 2015) have been shown to alter intra-aneurysmal flow. Even subtle changes to geometry such as thresholding values used to isolate the lumen have nontrivial effects on the solution (Muller et al., 2012). Analyzing inlet boundary conditions is also critical as most studies will use idealized wave forms rather than patient specific data (Cebral et al., 2005a; Karmonik et al., 2010; Marzo et al., 2011; Venugopal et al., 2007; Xiang et al., 2014a).

Some authors have highlighted that many of these studies use meshes with insufficient spatial and temporal resolution. Hodis

* Corresponding author at: Department of Physiology and Biomedical Engineering, Mayo Clinic, 200 First Street SW, Rochester, MN 55905, USA.
E-mail address: dragomirdaescu.dan@mayo.edu (D. Dragomir-Daescu).

http://dx.doi.org/10.1016/j.jbiomech.2017.03.020
0021-9290/

et al. (2012) showed that mesh convergence can require millions of elements instead of hundreds of thousands as is frequently reported. High temporal and spatial resolution simulations using direct numerical solvers had also shown how important flow features can be overlooked in certain geometries (Valen-Sendstad et al., 2013). In particular, unsteady flow structures were found in bifurcation aneurysms that were given steady inflow conditions. Moreover, high resolution simulations have been shown to capture instabilities that were missed by lower resolution simulations (Valen-Sendstad and Steinman, 2014). The authors reported domain-averaged errors as high as 40% for the low resolution studies.

Maintaining sufficient resolution for commercial solvers is necessary to ensure the solution is converged, especially with the wide variability of solutions that can be obtained when different groups perform CFD studies on the same geometries (Berg et al., 2015; Steinman et al., 2013). Determining solver settings that adequately resolve the flow structures is key. To minimize computational resources, this study evaluated 2 solver settings that can be optimized to maintain solution accuracy and reduce computation time. The residual error for the mass and momentum equations and timestep settings drastically affect computation time and accuracy. We demonstrated not only the range of acceptable values for three aneurysm models, but also showed how the solution is vulnerable to compounding error.

2. Methods

Approval to use patient specific data was obtained from our Institutional Review Board. Simulations were carried out in ANSYS Fluent (ANSYS, Inc.; Canonsburg, PA) on patient-specific models of a left internal carotid artery aneurysm with a daughter sac, a middle cerebral artery aneurysm, and a basilar tip aneurysm. These geometries are shown in Fig. 1, which also identifies the locations of 3 planes where velocity values were analyzed across all simulations. The geometry shown in Fig. 1b was from a recent CFD challenge (Berg et al., 2015). WSS, time-averaged WSS (AWSS), and OSI were examined on the aneurysm wall surface. Because our prescribed inlet boundary condition requires a circular inlet, we extended the irregularly-shaped inlet vessel such that it extruded to a perfect circle. The diameter of the inlet circles are 4.74 mm and 2.2 mm for the geometries in Fig. 1a and b, respectively. The geometry in Fig. 1c had two inlets with diameters of 2.2 mm and 3.2 mm.

From there, we discretized the geometry with a tetrahedral mesh using ANSYS ICEM CFD (ANSYS Inc., Canonsburg, PA). The uniform meshes had an average edge length of 0.1 mm and 9 inflation layers along the boundary with a prescribed initial layer height of 0.01 mm and an inflation factor of 1.2. The resulting meshes contained 1.3–10.5 million elements. This meshing technique has been shown to produce converged flow solutions following the practice followed by Hodis et al. (2012).

Transient simulations were performed using Fluent 14.0 (ANSYS Inc., Canonsburg, PA) with second order implicit solver. A SIMPLE scheme was employed with QUICK discretization for momentum and second order scheme for pressure interpolation. We assumed an incompressible, Newtonian fluid to simulate blood with the following properties: density, $\rho = 1050\ \mathrm{kg/m^3}$ and dynamic viscosity, $\mu = 0.035$ Poise. A Womersly profile was selected for the inlet boundary condition with the Womersely number, $\alpha = 3.7$. The time-averaged flow rates for geometries A and B were 213 cc/min and 156 cc/min, respectively. For aneurysm C, 60 cc/min was prescribed for each inlet. The cardiac cycle time, T, equaled 1 s. We applied traction-free, zero relative pressure boundary conditions to the outlets and a no-slip condition along the vessel walls. We ran each simulation using 60 nodes on our computation cluster, which uses Dual Intel Xeon X5650 2.66 Ghz processors. Using the same number of nodes on the same machine across all simulations allowed for a comparison of the computation time given the different solver parameters. Four cardiac cycles were simulated for each case to wash away any effects of our initial conditions on the final solution. Post processing of the solution data was performed on Tecplot 360 (Tecplot, Inc. Bellevue, WA).

We simulated results for all three geometries at three different timesteps: 10^{-3} s, 10^{-4} s and 10^{-5} s. For each of these timesteps we used 3 different residual parameters for the mass and momentum equations: 10^{-4}, 10^{-5}, and 10^{-6}. Our most conservative simulation was our baseline case with a timestep of 10^{-5} s and residuals of 10^{-6}, which required 2.3 weeks to fully compute a single cardiac cycle for the largest of our models, geometry A. We compared hemodynamic variables with respect to this baseline by using a mean difference error, ε_{Xmd}, for given variables (Eq. (1)).

$$\varepsilon_{Xmd}^{(i)} = \frac{\left|X_0^{(i)} - \frac{X_0^{(i)}+X_n^{(i)}}{2}\right| + \left|X_n^{(i)} - \frac{X_0^{(i)}+X_n^{(i)}}{2}\right|}{\frac{X_0^{(i)}+X_n^{(i)}}{2}} \times 100\% \tag{1}$$

where $X_n^{(i)}$ is the hemodynamic variable of a solution at mesh point *(i)* in a current simulation, n, and $X_0^{(i)}$ is the same variable at mesh point *(i)* of the baseline simulation. Velocity results were analyzed on 3 internal planes shown in Fig. 1 while WSS was analyzed on the aneurysm surface. For a quantitative analysis, we tabulated the RMS of these errors from these planes and surfaces using Eq. (2).

$$\mathrm{RMS} = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(\varepsilon_{Xmd}^{(i)}\right)^2}, \quad n = 1, 2, \ldots \tag{2}$$

Because the term in the denominator for ε_{Xmd} could approach 0 when the respective hemodynamic variable is exceptionally low, the RMS terms do not include points where the variable is less than 10% of the variable maximum on the analysis plane or surface. This prevents points of insignificant value from disproportionately inflating the RMS calculations.

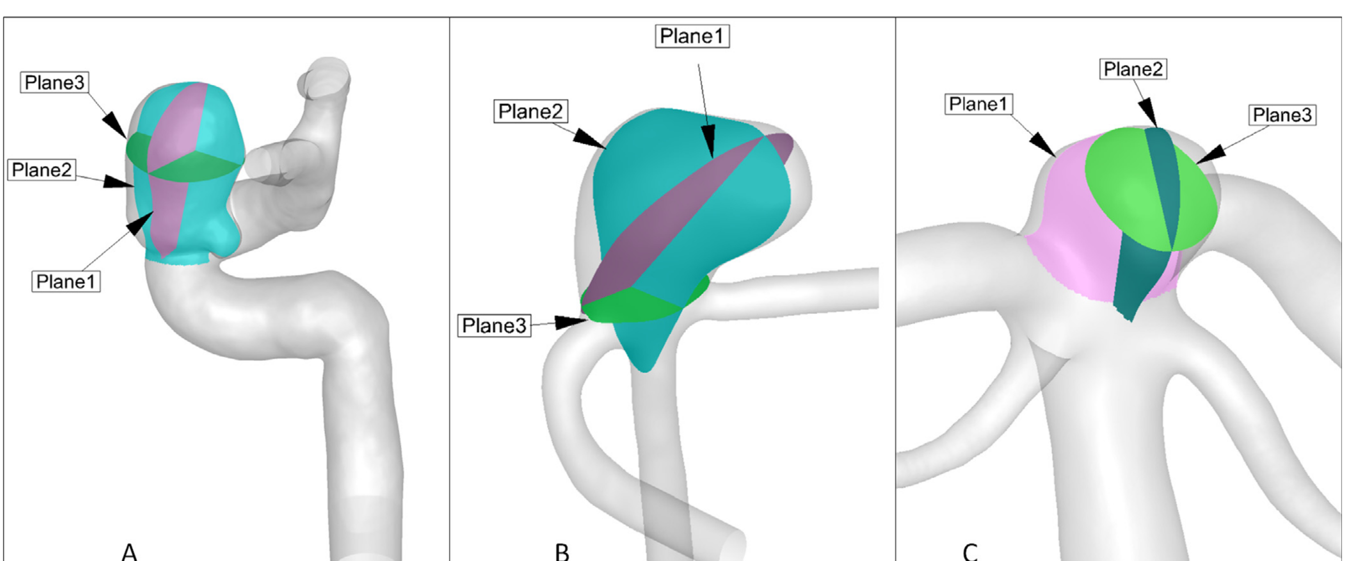

Fig. 1. Patient-specific aneurysm geometries with analysis planes. (A) is a sidewall, ICA aneurysm, (B) is a bifurcation aneurysm of the MCA, and (C) is a basilar tip aneurysm.