Enhanced Fluid Dynamics Modeling Capability for Anatomically-Correct Endovascular Flows

Over the past several years, computational fluid dynamics (CFD) has been increasingly used as a tool for simulating hemodynamic flows within the human body. CFD simulations have the advantage of providing 3-D, time-dependent velocity and pressure fields that are not easily measured with *in vivo* diagnostics or *in vitro* experimental techniques. Combined with advances in medical imaging technology, such as magnetic resonance imaging (MRI) and computed tomography (CT), CFD can offer patient-specific data of hemodynamic flows within endovascular geometries acquired through these imaging methods. Doctors can then use this image-based data to make more informed medical diagnoses and to predict the outcomes of various treatment alternatives.

Project Goals

The purpose of this project is to demonstrate a modeling and simulation capability for the hemodynamic flow through an endovascular geometry obtained from CT scan data. To successfully achieve this goal, we identified the steps and software tools required to extract an arterial surface from raw CT scan data, to generate a volumetric mesh within this surface, and to simulate the hemodynamic flow throughout the mesh.

Relevance to LLNL Mission

The technical capability to extract surface geometry information from CT scan data, produce a 3-D

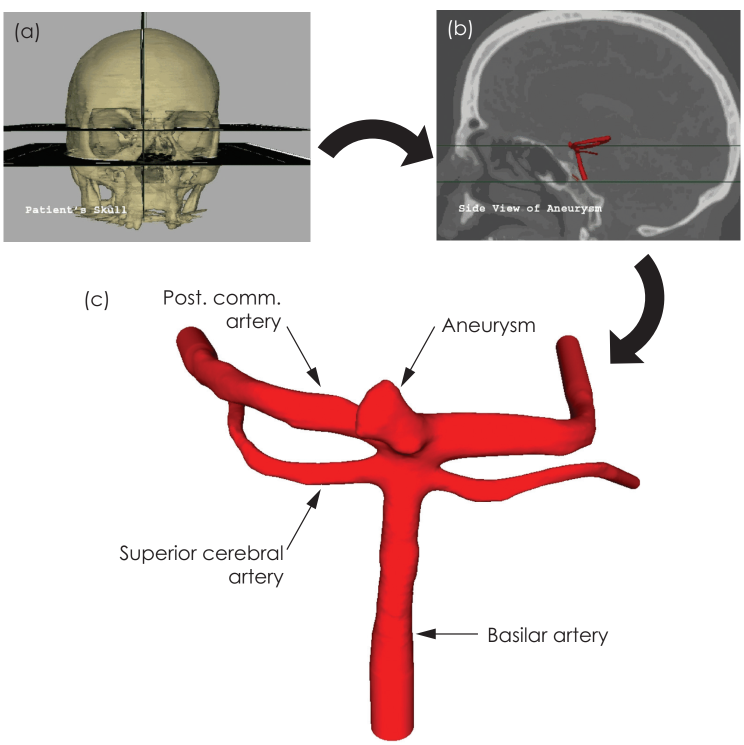

Figure 1. (a) CT scan data of the skull of a 51-year-old female patient with a basilar brain aneurysm. (b) Position of the endovascular geometry within the patient's brain. (c) Arteries and aneurysm included in the CFD simulation.

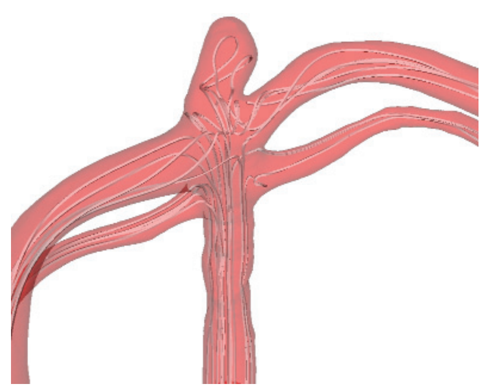

Figure 2. Streamlines of the steady-state velocity field through the bifurcation and aneurysm.

Engineering Modeling and Simulation

For more information contact
Jason Ortega
(925) 423-3824
ortega17@llnl.gov

computational mesh based upon this geometry, and conduct CFD simulations using this mesh has numerous applications to LLNL's mission. For example, this capability could be used to examine the flow field through an autonomous pathogen detection system (APDS) returning from the field, and to quantitatively assess the impact of bio-fouling or particulate contamination on the flow through the collector system.

FY2005 Accomplishments and Results

Numerical simulations were performed with a commercial code, STAR-CD, on an endovascular geometry extracted from CT scan data of a 51-year-old female patient with a brain aneurysm (Fig. 1). The complexity of the flow field within this geometry is highlighted by the 3-D streamlines shown in Fig. 2.

A streamline plot at one cross-section of the computational domain (Fig. 3) shows that the flow at the bifurcation is comprised of a jet from the basilar artery that impinges upon the aneurysm neck at multiple locations to form a swirling, vortical structure. Figure 4 shows a plot of the shear stress over the entire arterial geometry. It can be seen that there is significant variation of the wall shear stress at the aneurysm neck due to the underlying flow patterns. Such behavior in the wall shear stress and wall shear stress gradient has often been identified as one of the prime mechanisms for arterial wall degradation.

Related References

1. Berger, S. A., and L.-D. Jou, "Flows in Stenotic Vessels," *Ann. Rev. Fluid Mech.*, **32**, pp. 347-382, 2000.
2. Cebral, J. R., M. A. Castro, O. Soto, R. Lohner, and N. Alperin, "Blood-Flow Models of the Circle of Willis from Magnetic Resonance Data," *J. Eng. Math.*, **47**, pp. 369-386, 2003.
3. Chatziprodromou, I., V. D. Butty, D. Poulikakos, and Y. Ventikos, "Pulsatile Blood Flow in Anatomically Accurate Vessels With Multiple Aneurysms: A Medical Intervention Planning Application of Computational Hemodynamics," *Flow, Turbulence, and Combustion*, **71**, pp. 333-346, 2003.
4. Stuhne, G. R., and D. A. Steinman, "Mesh Resolution Requirements for the Numerical Analysis of Flow Through Stented Aneurysms," *Summer Bioengineering Conference*, Sonesta Beach Resort, Key Biscayne, Florida, 2003.

FY2006 Proposed Work

To enhance the capability for simulating endovascular flows, we will implement and demonstrate non-Newtonian and temperature-dependent blood viscosity models with STAR-CD. The models will be validated using experimental data from the literature and demonstration simulations will be performed with an endovascular geometry obtained from CT/MRI scan data.

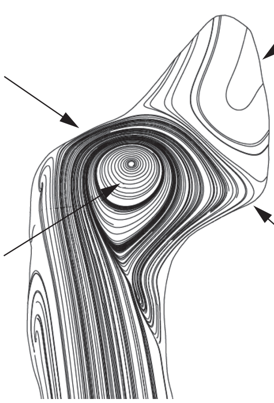

Figure 3. Velocity streamlines at a cross-section of the bifurcation and aneurysm.

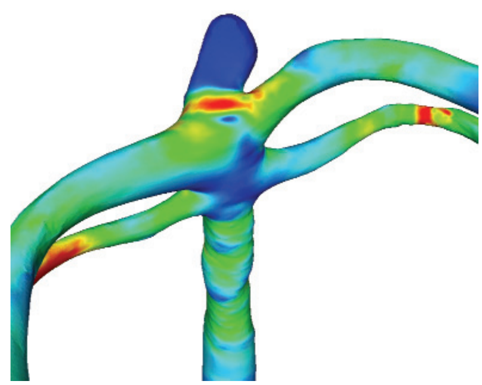

Figure 4. Shear stress distribution over the arterial walls.