Journal of Computational Simulation and Modeling
ISSN: 2231-3494 & E-ISSN: 2231-3508, Volume 2, Issue 1, 2012, pp.-27-29.
Available online at http://www.bioinfopublication.org/jouarchive.php?opt=&jouid=BPJ0000258

TECHNICAL BRIEF: COMPUTATIONAL FLUID DYNAMIC (CFD) ANALYSIS OF BLOOD FLOW THROUGH HUMAN ARTERIES

PANTA Y.M[1]* AND HARVEY M.[2]
[1]Department of Mechanical & Industrial Engineering College of STEM, Youngstown State University, 1 University Plaza, OH-44555, USA.
[2]Northern States Metals, 3207 Innovation Place, Youngstown, OH 44509, USA.
*Corresponding Author: Email- ympanta@ysu.edu

Received: September 20, 2012; Accepted: September 28, 2012

Abstract- Computational fluid flow modeling with commercially available computational fluid dynamics (CFD) software is used to visualize and predict physical phenomena related to human fluid flow. Over many years, CFD has been widely used in medical, pharmaceutical, and biomedical applications to analyze manufacturing processes, device performance, physiological flows, fluid-structure interactions, and the effectiveness of drug delivery systems. In this study, different human aortic models including straight, bend, T-shaped and the main arterial branching areas were analyzed for their fluid flow variables. Although analysis was performed with an assumption of laminar flow with steady state cardiac muscle walls, these results were validated with some available literatures. Turbulent flow with complaint muscle boundaries will provide more realistic results compared to the clinical studies. As future investigations, human blood vascular systems will be analyzed and reported for transient fluid flow modeling with complaint arterial walls.

Keywords- Computational Fluid Dynamics (CFD), Blood Flow, Blood Vessel, Wall Shear Stress (WSS), FLUENT, Mathematical modeling, Numerical Solution.

Citation: Panta Y.M. and Harvey M. (2012) Technical Brief: Computational Fluid Dynamic (CFD) Analysis of Blood Flow through Human Arteries. Journal of Computational Simulation and Modeling, ISSN: 2231-3494 & E-ISSN: 2231-3508, Volume 2, Issue 1, pp.-27-29.

Introduction

Recently, there have been several research studies of blood flow through the aorta using computer simulation to better investigate the blood flow variables.

The mathematical model consists of a set of governing equations used for a closed-form solution for fluid flow that are also embedded within ANSYS Fluent 12.0. Modeling and simulation helps to analyze and describe the physical phenomena in a given fluid domain. Multiple governing equations for fluid flow do have with their own given characteristics to solve for certain values that are based upon the user's interest. The equations of continuity, and momentum, are basically used for modeling.

Incompressible flow - for negligible change in density of fluid flow

Steady flow - as time dependent analysis is not considered.

Similarly, the mathematical model for pulsating blood flowing through the artery enclosed by the cardiac muscle is defined as a time varying function for the pulsatile blood pressure.

The flow of blood is governed by the Navier Stokes equations and continuity equation.

Continuity Equation:

$$\nabla \bullet \mathrm{u} = 0 \quad (1)$$

Navier-Stokes Equations:

$$\rho \mathrm{u} \bullet \nabla u = -\nabla \rho + \mu \nabla^2 u + F \quad (2)$$

Where ρ is the fluid density, p is pressure gradient, F is the body force and μ is viscosity of the fluid. Fluid velocity u in which u, v, and w represent, respectively, the velocity components in the x-, y- and z- directions.

Simultaneously solving these two equations with appropriate initial and boundary conditions, general characteristics of fluid flow can be identified.

Computational Methods

Several geometries were created that attempted to mimic a human blood vessel. Models were created using Solid Works as well as Gambit software. A geometrical model of the aortic network was created in Solid Works and ANSYS Design Modeler, and then meshed in GAMBIT and ANSYS, preprocessing software packages. Simulation and post-processing were performed in ANSYS Fluent. As a final simulation, multiple user defined function (UDF) was used to imitate the pressure inlet and six pressure outlets in the arterial network based on a combination of measurements from patients over a cycle for heart beat. Prior to the completion of the final Aortic blood vessel model, several other preliminary blood vessel models were created. These include a straight section, elbows that consisted of 30, 45, and 60 degree bends, a "T" shape, a three way intersection, as well as a cross(4 way intersection) A laminar model was chosen for all of the trials. The governing equations that apply to the specified conditions of the particular problem are then applied to each sub domain. After meshing and boundary types were specified, the meshed model was exported as a .msh file and imported into Fluent software for analysis. Once in Fluent, material properties were specified that include blood properties, properties of the artery material, and properties of the surrounding cardiac muscle. The properties used are as follows: Blood (fluid): Density = 1060 kg/m^3, dynamic viscosity = .005 Ns/m^2, Artery (solid): Density = 960 kg/m^3, Cardiac muscle (solid): Density = 1200kg/m^3. The walls of the models (the artery material) were assumed to be stationary and no-slip with the blood flowing in. Convergence criteria were set in Fluent and the solution was iterated until convergence.

Results and Discussion

Model 1: Various 2D blood vessels

Several preliminary blood vessel models were created; each with a specified input pressure of 11,208 Pascal and outlet pressures of 11,148 Pascal.

Straight blood vessel: The first blood vessel that was created was one that consisted of simple straight pipe geometry. The model has an inner diameter of 10mm and a length of 30mm

45 Degree elbow blood vessel: This blood vessel model was created with a simple 45 degree bend. Each leg of the geometry is 20mm in length, and has an inside diameter of 10mm. It should be noted that 30 and 60 degree elbow geometries were created that were consistent with the results of the 45 degree elbow.

"T" Shape blood vessel: The last preliminary blood vessel that was created was that of a "T" shape. Again ach leg has a length of 20 mm, and an inner diameter of 10mm.

Model 2: 2D Vascular System

2D Aorta Model: For the final trial a blood vessel geometry was created that resembles an aorta. Dimensions of this blood vessel are shown in the 3 dimensions in [Fig-1], however the solution was obtained in 2 dimensions using COMSOL model library [Fluid-Structure Interaction in a Network of Blood Vessels Model ID: 660].

Using data provided by the local hospital, a final model was also created and simulated for some initial results of pulsatile blood flow model.

Above [Fig-2] and [Fig-3] demonstrate the velocity contours and wall shear stress distributions in various models including the arterial network. Consistent results were obtained for all of the attempted models. In the straight blood vessel it is shown that the maximum velocity occurs at the center of the blood vessel. With the no slip wall condition for the blood flow specified at the walls. The 45 degree bend, and the T shape blood vessels confirm this. Where a high velocity occurs, it corresponds to a low pressure at that same location as shown in [Fig-4]. In addition to this, the final aorta model agrees with the previous findings. The maximum velocities achieved in all models are similar as it should be expected given the same inlet and outlet pressure conditions.

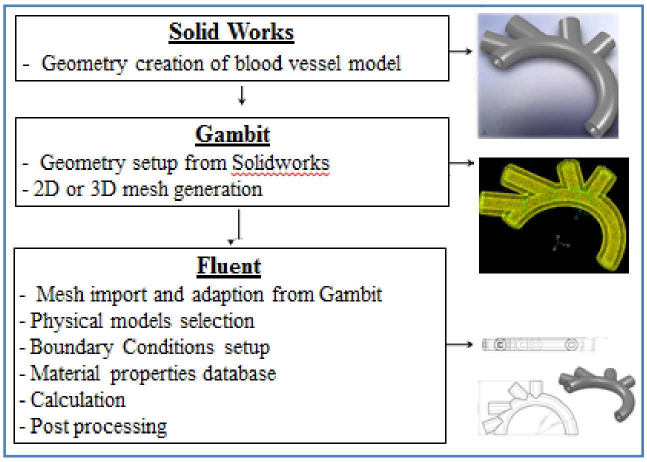

Fig. 1- Computational Modeling Procedure

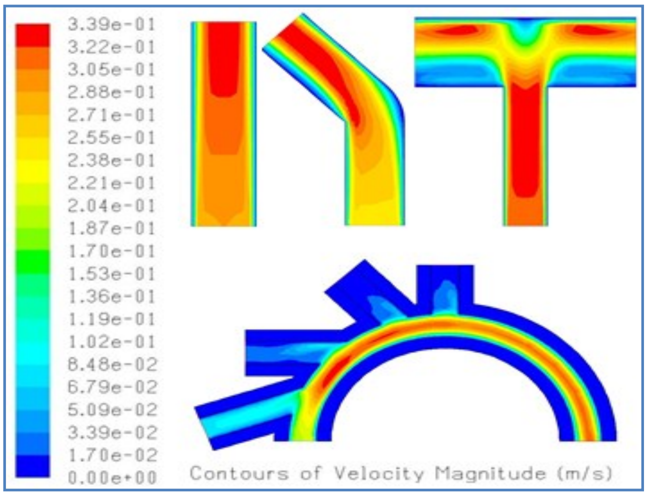

Fig. 2- Velocity Contours

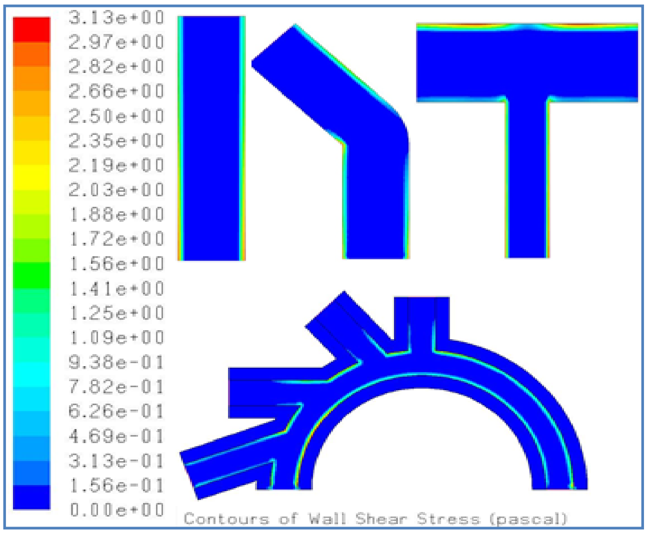

Fig. 3- Wall Shear Stress (WSS) Contours

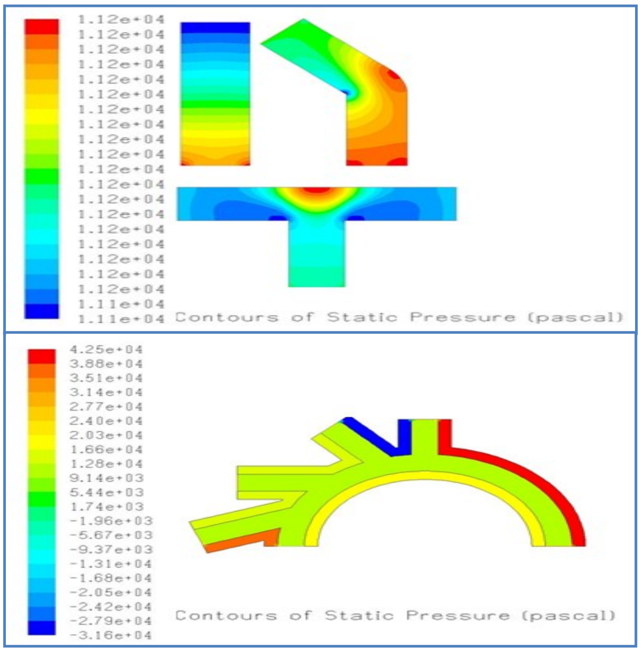

Fig. 4- Pressure Contours

This work is still under investigation and will be interpreted and presented later in the forthcoming publications.

Conclusions and Future Work

The study points out the significant change in velocity of the blood based on the shape of the arteries such as straight, bend, branches and the main arterial network models. Before implementing and trusting the results in practice, it is important to obtain consistent results with more accuracy and support the simulation results with experimental results which will be our future work. This preliminary work will further be extended with experimental validation, theoretical verification and more accurate computational modeling and simulation. The analysis of blood flow developed in blood vessels will be taken a step further in the future by performing a complete 3 dimensional analysis. In addition to this, modifications to assumptions that were used in these trials will be used. Changes to these assumptions will be taking into account the effect of body forces, a moving boundary, pulsatile blood pressure, a time dependent analysis and more realistic fluid structure interaction.

Acknowledgements

One of the authors (YP) acknowledges St. Elizabeth Hospital, Youngstown, Ohio (Resident Doctor: Jeremy Heffner) for providing some clinical data, School of Graduate Studies and Research at YSU for providing research release hours through RP Award. Also, M. Harvey was awarded undergraduate student intern through the Glen Stokes Minority Scholarship-OSEA to work on this project.

References

[1] Yearwood T.L. and Chandran K.B. (1984) *J. Biomech.,* 15(9), 683-704.

[2] Farthing S., Peronneau P. (1979) *Cardiovasc. Res.,* 13, 607-620.

[3] Ottesen J.T., et al (2004) *Applied Mathematical Models in Human Physiology,* 128.

[4] Shahcheraghi N., et al (2002) *J. Biomech. Engr.,* 124, 378-387.

[5] Jafari A., Mousavi S.M. and Kolari P. (2006) *Communications in Nonlinerar Science and Numerical Simulation,* 13, 1615-1626.

[6] Maura G., Daniele C., Morbiducci U. and Gaudio C.D. (2005) *Journal of Biomechanics,* 38, 1375-1386.

[7] Lorenzini G., Erminio C. (2008) *Journal of Biomechanics,* 41 1862-1870.

[8] Emilie M., Willemet M. and Lacroix V. (2009) *Medical Engineering & Physics,* 31, 131-144.

[9] COMSOL Inc. (2010) *Product Manual & Model Documentation: Introduction to COMSOL Multiphysics*, Version 4.0a, USA.

[10] ANSYS Inc. (2009) *User's Guide: ANSYS Fluent,* Version 12/12.1 Documentation, USA.

[11] Tushar G., Jeremic A. (2006) *COMSOL Users Conference,* Boston.

[12] Panta Y., Marie H. and Harvey M. (2009) 62*nd American Physical Society- Division of Fluid Dynamics (APS-DFD) Conference,* Minneapolis, MN.

[13] Harvey M., Panta Y. (2009) *OSEA Conference,* Columbus, OH.

[14] Robert F.W., Pritchard P.J. (2010) *Introduction to Fluid Mechanics,* 8th ed., Wiley.