International Journal of Computer Science and Engineering (IJCSE)
ISSN 2278-9960
Vol. 2, Issue 4, Sep 2013, 171-184

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

COMPUTER SIMULATION OF BLOOD FLOW IN LARGE ARTERIES BY A FINITE ELEMENT METHOD

HASANAIN A. A. SADEK[1], MOHAMMED J. KHAMI[2] & TALEB A. S. OBAID[3]

[1]Marine Science Center, University of Basra, Basra, Iraq

[2]Basra Technical Institute, University of Basra, Basra, Iraq

[3]Department of Computer Science, University of Basra, Basra, Iraq

ABSTRACT

Two-dimensional numerical simulations of blood flow are performed using the finite element method. This technique is carry out to investigate the influence of abnormal effects of artery geometry on the behavior of steady and laminar flow of an incompressible and Newtonian blood flow within a stenosed.

Blood **F**low **S**imulation (**BFS**) program using finite element method is written in Matlab. The BFS program have simple graphical user interface, it is easy to use and it has fully control for manipulating, visualization, and saving the results.

The stenoses severity intensifies the stretches the recirculation zones in the arterial flow. Shear stresses were estimated from velocity gradient for stenotic flow in arteries with 50% and 75% stenosis degrees. In the case of aneurysm model the Reynolds number changed in steps (216, 400, 800, and 1000). The velocity, pressure, and wall shear stress fields are visualized for a better interpreting and understanding of flow features.

The computation results, for the aneurismal and stenotic arteries are compared with other previous experimental results and gets a good agreement.

KEYWORDS: Computer, Numerical, Blood Flow Simulation, Element Method, Program

INTRODUCTION

Most of the researchers in the field of fluids and structures use numerical simulations, because simulations offer much cheaper and many times faster compared to experiments. In general, computations can be of considerable help in diagnosis, prediction of risk and prediction of effectiveness of model (devices). In the fluid field, the basic purpose of any simulation is to be able to predict the behavior of a test fluid in complex flows, when rheological data for the test fluid is available. The computational fluid dynamic model has been used to describe the flow patterns in different anatomical geometries, see [1-4].

Blood flow under normal physiologic conditions is an important field for study as is under disease conditions. The majority of deaths in developed countries result from cardiovascular diseases, most of which are associated with some form of abnormal blood flow in arteries. In certain circumstances, unusual hemodynamic conditions induce an abnormal biological response.

In this paper, simulate blood flow is performed in places called stenosis where arteries are abnormally pinched or narrow, and in dilatation places called aneurysms as well as normal artery. Arterial stenosis and aneurysm are due to vascular disease in human that, if untreated, leads to death. The stenosis causes the development of complex flow which reduces flow and flow choking, or it can form blood clots or cause plaque ruptures that result in strokes. Detection and quantification of stenosis contribute as the basis for surgical intervention. With regard to aneurysm, there is rupture of arterial aneurysms related to hemodynamic since they appear at specific locations of the vascular system. Wall shear stresses within the aneurismal bulge are considered as the predominant factor for arterial wall reconstructing, and static pressure is believed to be responsible for wall weakening, finally leading to rupture, [5].

Computer simulations of blood flow in arteries may help doctors make better individualized treatment decisions, or it can give researchers information on how blood is distributed through the vessels. This can help researchers in the field with designing specific treatments that will improve upon patient care.

The simulation is based on solving the two dimensional Navier-Stokes and continuity equations, which govern blood flow, by means of finite element method. Various hemodynamic parameters (e.g.,velocity, pressure, and shear stress) are visualized for a better understanding of flow characteristics such as distributions of the flow pattern, stagnation flow and recirculation zones.

Assuming that the blood is an ideal Newtonian fluid with constant viscosity, and also assumed that blood behaves as an incompressible fluid and the flow is steady, laminar, the vessel is straight and the walls are assumed to be rigid (inelastic), under these conditions a parabolic velocity profile could be assumed in the inlet of vessel.

Newtonian and Non-Newtonian Fluid

Shear rate and viscosity are directly related to the properties of the fluid. Non-Newtonian fluids are governed by a non-linear relationship between shear stress and shear rate. The fluid viscosity μ, depends on pressure, temperature and shear rate. In contrast, the viscosity of Newtonian fluids is independent from shear rate, as indicated in Figure (1), by the linear relation between shear stress and shear rate (velocity gradient).

For Newtonian fluids flowing upon a planar surface, shear stress $\boldsymbol{\tau}$, is determined according to Newton's law by the equation:

$$\tau = \mu \frac{du}{dy} \tag{1}$$

where **μ** is the viscosity, ***u*** the fluid velocity, **y** the distance from the surface, and $\frac{du}{dy}$ the velocity gradient (shear rate $\dot{\Upsilon}$), for details see [6].

The Navier-Stokes equations comprise three PDEs in the state variables (velocity $v_x=u$, $v_y=v$ and pressure p), as in following:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial x^2}\right) \tag{2, a}$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{2, b}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2, c}$$

But For every fluid, to reach turbulent flow, there is a threshold value dependent on the average fluid velocity, density, viscosity and the diameter of the vessel (for internal flows). This characteristic value, which depends on all these quantities, is called the Reynolds number $\boldsymbol{Re}$, which is a dimensionless parameter. $\boldsymbol{Re}$ represents the ratio of inertial forces to viscous forces, and it is defined as:

$$\boldsymbol{Re} = \frac{\rho V D}{\mu} \tag{3}$$

Where ρ= fluid density in kg/m^3,

V = average fluid velocity in m/s,

D = vessel diameter in m, and

μ = fluid dynamic viscosity in Ns/m^2(in S.I. Newton(N)= kg m/s^2).

Laminar flow occurs in flow environments where $\boldsymbol{Re}$< 2000. Turbulent flow is present in circumstances under which $\boldsymbol{Re}$ > 4000. The range of (2000 < $\boldsymbol{Re}$ < 4000) is known as the **transition range** [7]. Moreover, Reynolds number helps us to predict the transition between laminar and turbulent flows, and it is also useful for predicting entrance length in pipe flow.

Finite Elements Method (FEM)

FEMs are widely used for solving problems in fluid mechanics and heat transfer. The FEM is a popular computational approach to the problems that has applications in diverse areas of cardiovascular biophysics, such as blood flow in arteries.

FEMs require division of the problem domain into many small sub domains or elements. Therefore, the problem domain consists of many finite element patches, each element is an essentially simple unit, the behaviour of which can be readily analysed. The complexities of the overall systems are accommodated by using large numbers of elements, rather than by resorting to the complicated mathematics required by many analytical solutions. The general steps of the finite element approach is given in a Figure (2).

Creating a mesh is the first step in a FEM, and mesh generation consider the starting point for the problem solution. A mesh consists of nodes that are grouped together to form elements. These nodes and elements define the computational domain of the numerical model. There are many different element types that one can select in discretization of the problem domain (triangle element is used in this work).

A mesh may assign additional information to the nodes such as boundary conditions assigned to nodes. This additional information is used as input data for the numerical model.

There are a number of commercial and open source codes that may generate high quality unstructured meshes that will be compatible with the data structure used in the selecting programming language. The basic requirements are that these mesh generators must provide the following parameters:

- A contiguous and unique numbering (labelling) of the nodes (i = 1, 2,…, n), where **n** is the number of nodes in the domain discretization.
- A vector of the nodal **x** and **y** positions, i.e., x_i and y_i coordinates,
- A contiguous unique numbering of the triangles in the discretization.
- A listing of the node numbers (labels) of the vertices of the triangles element preferably in counter-clockwise order, and / or
- A listing of the nodes that lie on the boundary of the domain.

Boundary Condition

Two different types of boundary conditions are applied to this study.

- **Within the Artery:** a ‘‘no-slip’’ boundary condition is assumed, i.e., the speed of the blood along the walls of vessel was zero,in other words, axial velocity **u**=0, transverse velocity **v**=0.
- **At the Inlet:** **u** is specified and **v** is set to zero, i.e., **u** = specified, **v** = 0. Fully developed flow condition at the inlet of the artery of diameter $\boldsymbol{d}$ is specified as follow:

$$u = u_{max} * 4\left(\frac{y}{d} - \frac{y^2}{d^2}\right) \tag{4}$$

Where u_{max} can be calculated as twice as the average velocity$V_{average}$, [8] or

$$u_{max} = 2 * V_{average} \tag{5}$$

and

$$V_{average} = \frac{(Re*\mu)}{(d*\rho)} \tag{6}$$

- **AT The Exit:** fully developed condition is assumed and hence gradients are set to zero, i.e., ∂u/∂x= ∂v/∂y=0

Blood Flow Simulation (BFS) Program

Since the blood flow problem has too many subdivision problems related to the geometry of the selected flow domain and the type of artery diseases (Aneurysm, Atherosclerosis and Stenosis), and also the selection of the proper FEM solution method. A Matlab GUI has been a specially written and designed to manage the solution, Figure (3) depicts the main GUI's window which has two main panels. The small axes panel and big panel with many buttons to activate the GUI. It has windows for inputting data, mesh generation (pre-process), solver (process), visualize (post-process), collecting, exporting data, and saving the results (storage management), Some of these windows are shown in Figure (4).

With BFS program, user can do and obtained the following:

- Select the artery disease type (Healthy, Tapered, Aneurysm, and Atherosclerosis or Stenosis artery).
- Input the geometric parameters of the specified artery.
- Generate suitable mesh for the given geometry with the opportunity of refining the mesh at any specified location on artery surface as much as the required accuracy .
- Clip (prune or delete) some of the boundary elements to change and resize the shape of the disease in the artery.
- Save generated (or load previously made mesh) parameter on/from any storage media.
- Calculate the main variables (velocity, pressure, and shear stress components).
- Display the results as a whole in different form (text, curve of points or arrows, stream lines, and other form of color graphics and figures).
- Obtain and plot any individual variable values at different points on the geometry to get an idea of that variable profile and tracing its overall variation at those locations.

Users of this program could be vascular surgeons, physician and biomedical engineers. In addition, the GUI was designed in a way that the users do not need to adequate knowledge for computational methods and modeling and can easily understand it and benefit from its results.

BFS Program Validation

Validation is defined as the process of checking whether the results that are produced by the program are in agreement with physical reality. In other words, are the assumptions and simplifications, that were made to build the simulation program, justified? This reality check means comparing the solutions obtained by the simulation to known theoretical or experimental results.

The validation is made by comparing the present results for three types of artery cases (Aneurysm, Stenosis and Atherosclerosis) with those results obtained by [9-11]. Aneurysm: In [9], the results obtained at five axial locations within aneurysm model with inlet **Re**=206 and kinematic viscosity 8.11 centiStokes (cSt). Figure (5) shows a reasonable agreement with the experimental data.

Stenosis: Also, computation result for the stenosis case is compared to the experiment data obtained by [10] at different axial locations distal to the smooth stenosis and inlet **Re**=500 and stenosis degree is 75%. Figure (6) shows this comparison, from this figure, the agreement is acceptable.

Atherosclerosis: Comparison between computed and published axial velocity profiles for the atherosclerosis case at three different locations is presented in Figure (7). It can be seen that upstream from the stenosis, the presented numerical results fit the experimental data and the numerical Fluent code results obtained by [11].

Upstream from the stenosis our simulated results overlap much better to experimental measured values than the Fluent results. At the throat and downstream from stenosis our simulated results agree with the computational data (Fluent).

Case Study 1: The Effect of Reynolds Number on Flow through Aneurysm

The common features of the aneurysm case are flow detachment at the entrance of the aneurysm and reattachment at its distal end, so that retrograde flow covers significant part of the bulge volume. The geometry of the selected portion of an artery with aneurysm is shown in Figure (8). The aneurysm model has parental diameter of 13 mm and the maximum diameter 32mm, the width of the aneurysm 45 mm, centre of aneurysm at point 50 on longitude axis. These values are according to [9], so later one can compare this simulation results with experimental results of this reference.

Figure (9) depicts the simulated velocity characteristics for the fusiform aneurysm when the Reynolds number is 206 and kinematic viscosity is ν=8.11 cSt. In Figure (9,c) it can be seen that towards the exit of the aneurysmal bulge, negative velocity maximizes close to the wall, the magnitude of the negative axial velocity increases with Reynolds as will be shown later.

Figure (10) shows the velocity distribution and contours in aneurysmal bulge at **Re** (400, and 1000) respectively. The wall shear stress takes a local maximum at the aneurysm exit as shown in Figure (11). Figure (12) illustrates how the artery dilatation is covered by a recirculation zone.

Figure (13) shows the velocity profiles for the used aneurysm model. The profiles are plotted at five positions along the total length of the model. These points are at distance of x= 30,40,50,60 and 70. The profiles are done for three different values of Reynolds number,(400 red line ,800 blue line and 1000 green line).

Case Study 2: The Effect of Stenosis Degree in Symmetric Stenotic Blood Flow

A flat artery with rigid walls and symmetric stenotic blood flow has been considered. The shape of the artery is in Figure (14). The assign measurements are as follow: total artery length = 50 mm, inlet diameter =6.2 mm, stenosis width 6 and its center at upper and lower sides are (10,6.2) and (10,0). The most important parameter that effects the stenotic blood flow is what is call the stenosis degree (Sd), which is defined as :

$$s_d = 1 - \left(\frac{d2}{d1}\right)^2 \quad (7)$$

Where $d1$ is the diameter of the artery and $d2$ is the diameter at the center of the blockage. Figure (14) also shows the radius (r_b) of the blockage, dependent on $d2$, which can be defined as, see [12]

$$r_b = \frac{d1-d2}{2} = \frac{d1}{2}\left(1 - \sqrt{1 - s_d}\right) \quad (8)$$

The generated mesh for the artery is shown in Figure (15,a). while the blood flow velocity, pressure, and shear stress distribution and contours are shown in . In Figure (15,b-d), flow separation is developed in the expansion region; between the central jet and the recirculation region where a strong shear layer develop. The flow separation at the stenosis edge results in a recirculation zone downstream of the stenosis. The length of this zone (distance between the peak and the reattachment of the separation streamline) increases as the stenosis degree, and thus Reynolds number, increases. In Figure (15) one can observe that near the stenosis the shear rate is high, corresponding to the high value of the velocity gradient and shear stress is low after the stenosed vessel region. A velocity profiles at points where x=1,10,16,18, and 30 are shown in Figure (15,e). Figure (15,e) depicts that flow velocity direction at the center line of the artery is always positive while the flow changes its direction near the expansion after leaving the stenosis. This is because the high shear rate at those regions with respect to that at the center line.

CONCLUSIONS

Numerical simulation for blood flow through aneurismal artery and symmetric stenotic is performed. This was obtained within the framework of Navier-Stokes equations. The finite element method is applied to solve the governing equations. Under steady conditions the effect of the dilatation, and also the effects of stenosis severity on the blood flow are presented and compared. The main conclusion drawn from this project can be summarized as follows:

With regard to aneurysm, the flow separates at the start of the aneurysm model and it reattaches at small distance upstream of its exit. This distance becomes smaller with increasing Reynolds number. Also the shear stress take a high values at the exit of the aneurysm model, see Figure (9) to Figure (13).

The steady flow is intensified and pressure falls more rapidly with the increasing stenosis severity. The recirculation zones are stretched and velocity profile varies suddenly in the region directly after the stenosis region. In general, the stenosis severity and geometry have important influence on recirculation length, see Figure (15) and Figure (16).

REFERENCES

1. Perktold K., Resch M and Florian H., **Pulsatile non-Newtoniasn flow characteristics in a three-dimensional human carotid bifurcation model**, ASME J Biomech Eng, 113:464–475, 1991.

2. Santamarina A., Weydahl E., John M., Siegel JR., and James E. Moore**, Computational analysis of flow in a curved tube model of the coronary arteries: effects of time-varying curvature" annals of Biomedical engineering**, Vol. 26, PP. 944-954, 1998.

3. Zeng D., Ding Z., Friedman MH and Ethier CR, **Effect of cardiac motion on right coronary artery hemodynamic**. Ann Biomed Eng, 31:420–429,2003.

4. Huo Y, Guo X., and Kassab GS**. The flow field along the entire length of mouse aorta and primary branches**. Ann Biomed Eng, 36(5):685–99, 2008.

5. Kleinstreuer C. and Zhonghua L., **Analysis and computer program for rupture-risk prediction of abdominal aortic aneurysms**, Biomed. Eng. Online 5 (19), 2006.

6. Sadeq H., **Computer Simulation of Blood Flow in Large Arteries by a Finite Element Method**, MSc thesis, college of science, Basra Univ., 2013

7. Stamatopoulos Ch., Papaharilaou Y., Mathioulakis D. S. and Katsamouris A., **Steady and unsteady flow within an axisymmetric tube dilatation**, Experimental Thermal and Fluid Science, 34, 915–927, 2010**.**

8. Ahmed S. A. and Giddens Don P**., Velocity measurements in steady flow through axisymmetric stenoses at moderate Reynolds numbers,** J. Biomechanics, 16(7): 505-507,1983.

9. Ai L., Zhang L., Dai W., Hu C., Shung K., and Hsiai T.K., **Real-time assessment of flow reversal in an eccentric arterial stenotic model**, J. Biomech., Vol. 43, No. 14, pp. 2678-2683, 2010.

10. Waite L. and Fine J., **Applied Biofluid Mechanics**, The McGraw-Hill Companies, Inc., 2007.

11. Frank M. W., Fluid Mechanics, 4th edition, McGraw-Hill Series in Mechanical Engineering, ISBN: 0072281928, 1998.

12. Taylor C. and Hood P., **A numerical solution of the Navier-Stokes equations using the finite element technique**, Comput. Fluids, 1:73–100, 1973.

APPENDICES

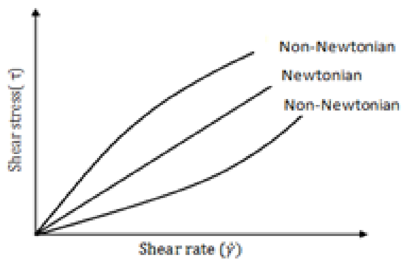


Figure 1: Shear Stress and Shear Rate Relationship for Newtonian and Non-Newtonian Fluid

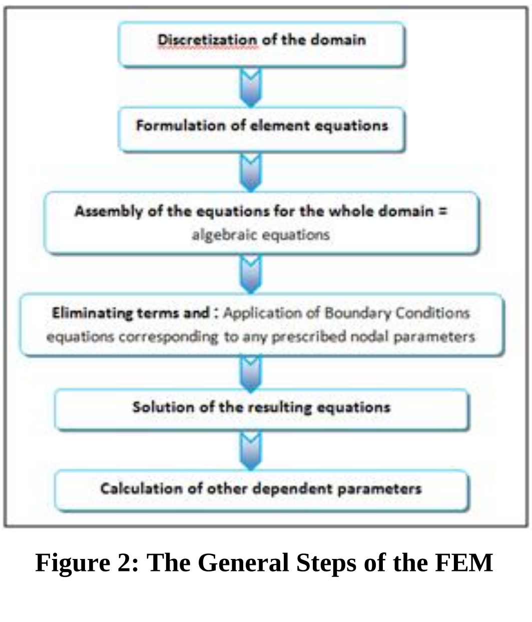


Figure 2: The General Steps of the FEM

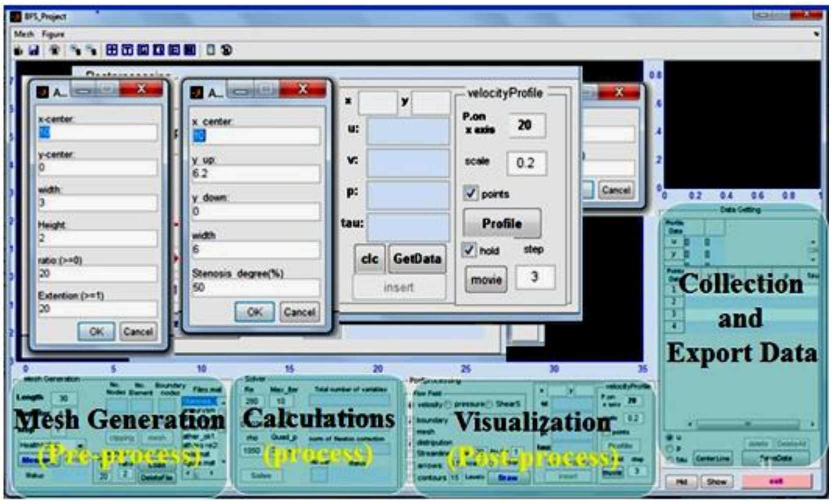
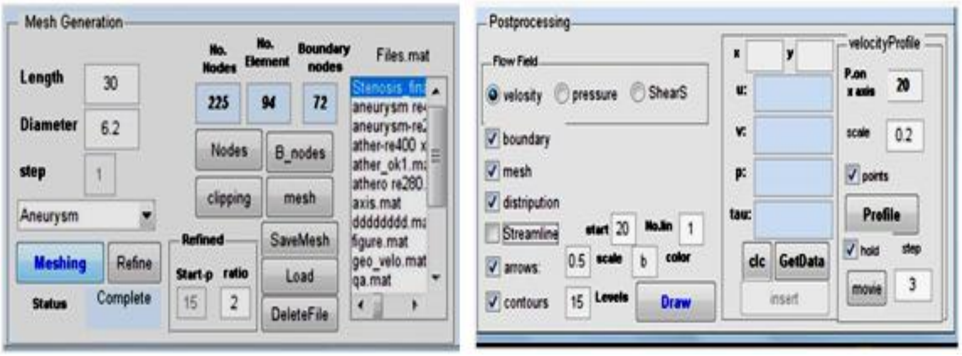


Figure 3: Graphical User Interface for BFS Program



Panel for Input Data　　　　Panel for Visualization

Figure 4: Mesh Generation and Post-Processing of the GUI

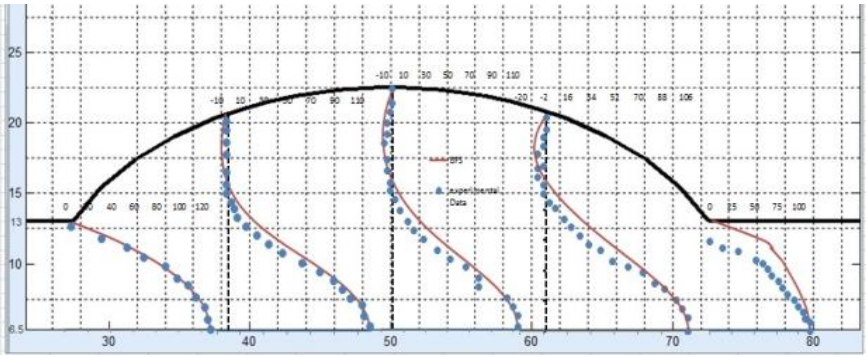


Figure 5: Velocity Profile of the Our Results (Sold) verses Experimental Data (Dotted) of [9]

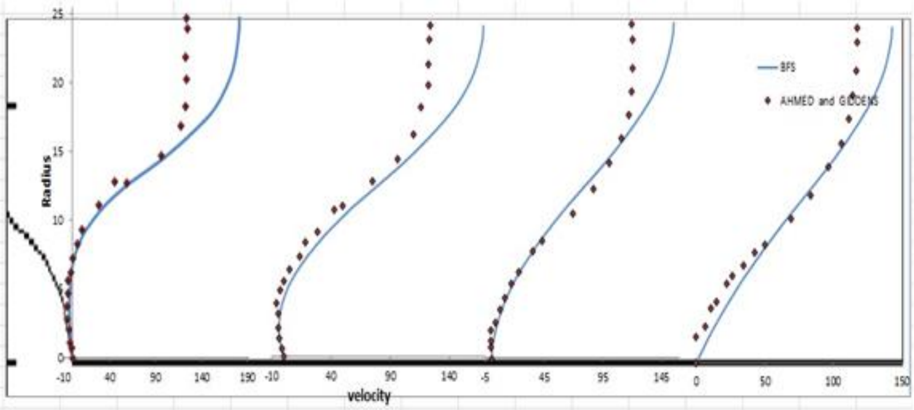
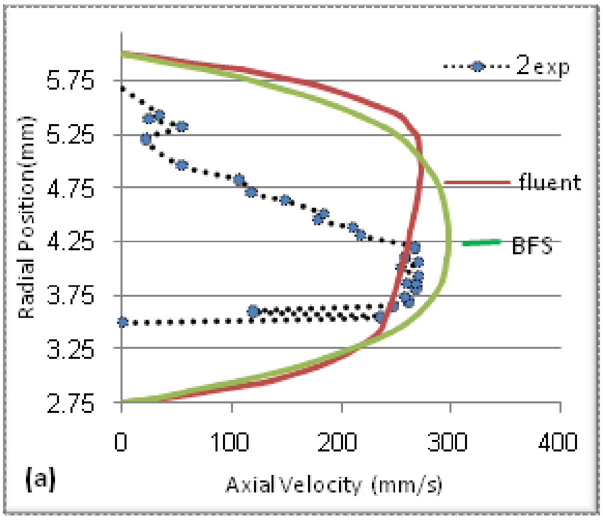
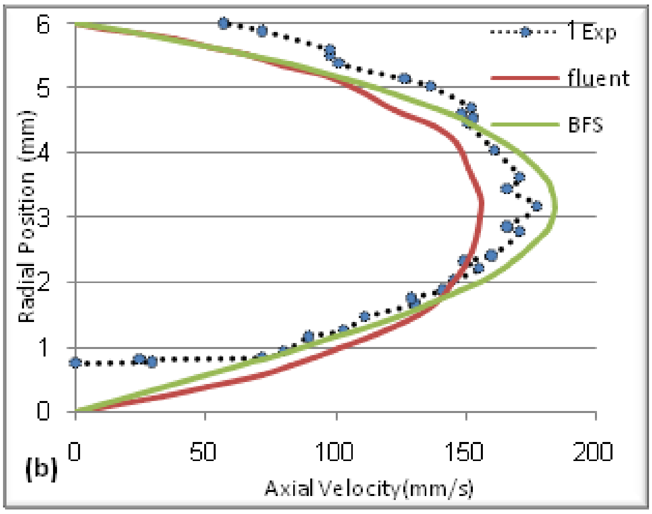
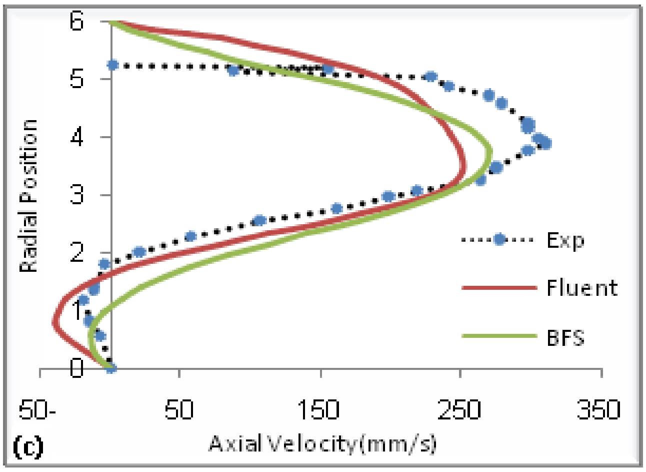


Figure 6: Computed Velocity (Sold) Profiles verses the Experimental (Dotted) Profiles of [10], with *Re*=500 and Stenosis Degree=75%





(a) Velocity Profile Upstream the Stenosis at x=7mm (b) Velocity Profile at the Throat x =20mm



(c) Velocity Profile Downstream the Stenosis at x=33mm

Figure 7: Our Computed Velocity Profiles at Three Axial Locations with the Experimental and Computational Profiles of [11], for Inlet Stenosis Degree=75%

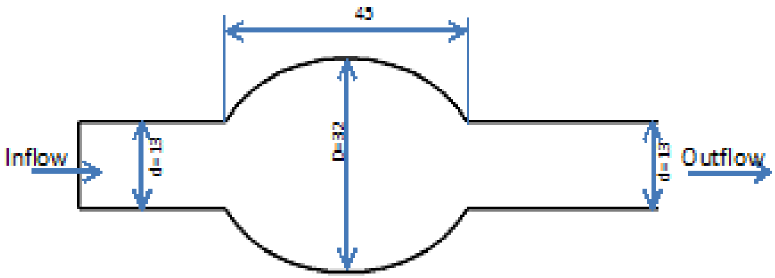
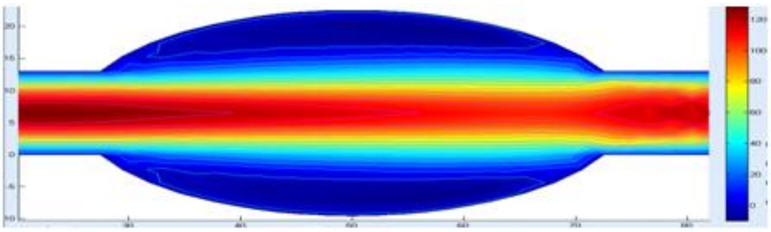
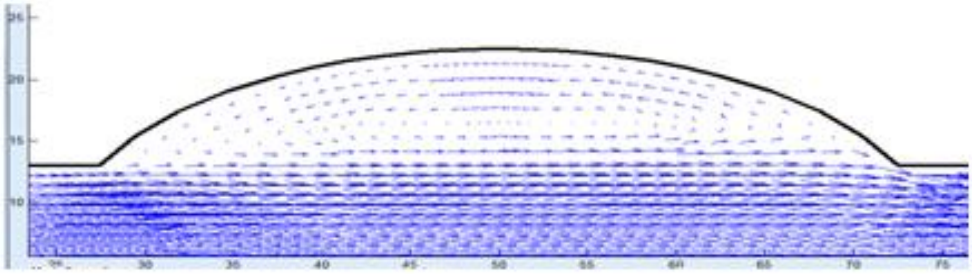


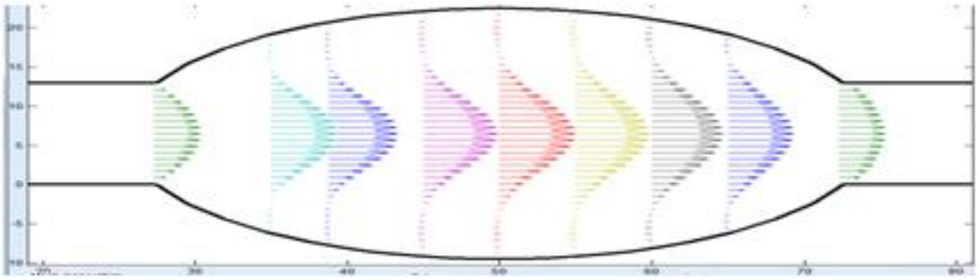
Figure 8: Artery Aneurysm Model, Dimensions in mm



(a) Velocity Distribution

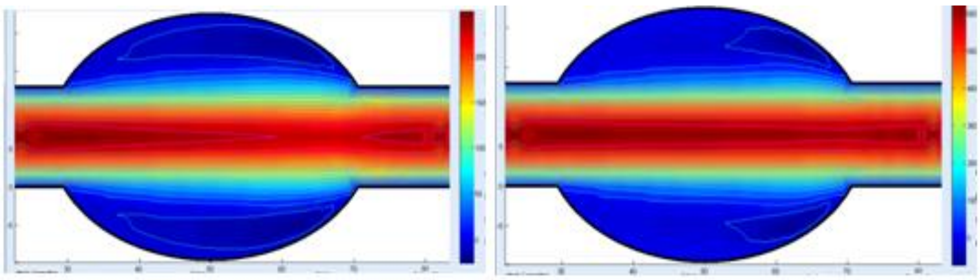


(b) Velocity Field



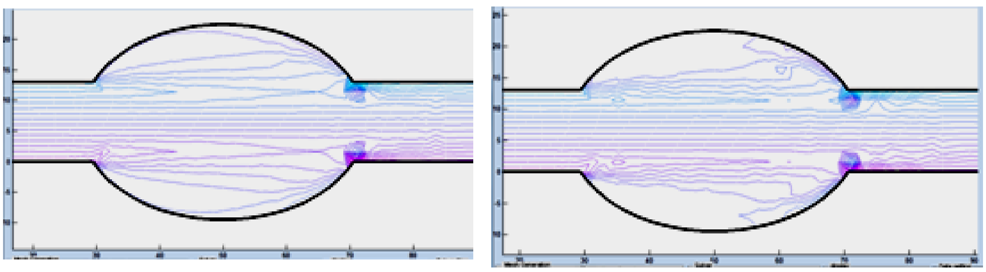
(c) Velocity Vectors

Figure 9: Simulation Results for the Fusiform Aneurysm



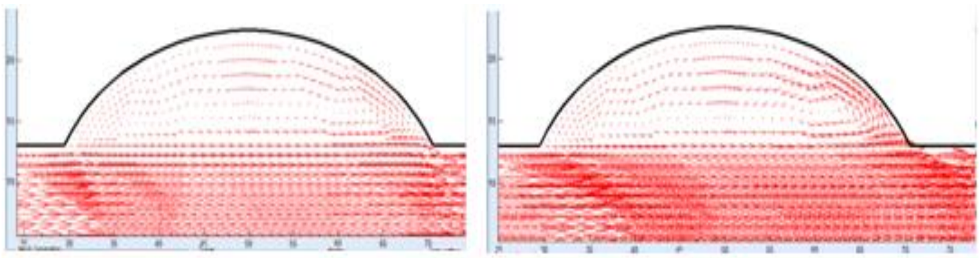
a. b.

Figure 10: Velocity Distribution and Velocity Contour in Aneurysm with: a. Re= 400, and b. Re=1000



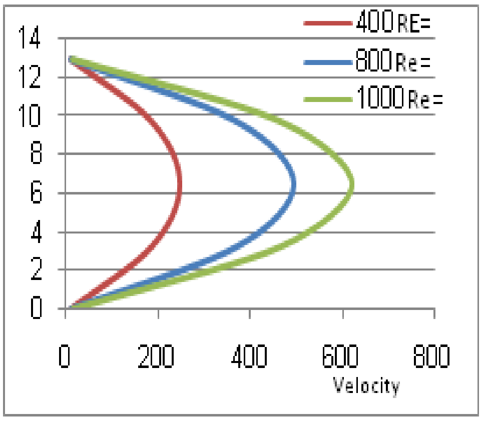
a. b.

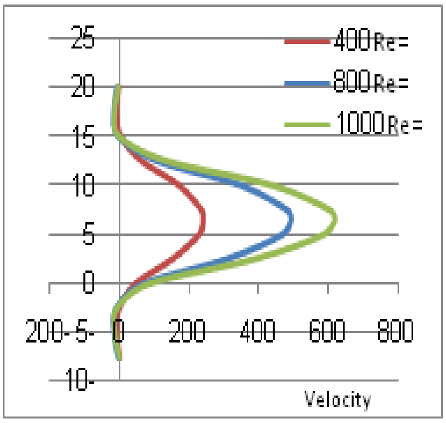
Figure 11: Shear Stress Contours, a. Re=400, b. Re=1000



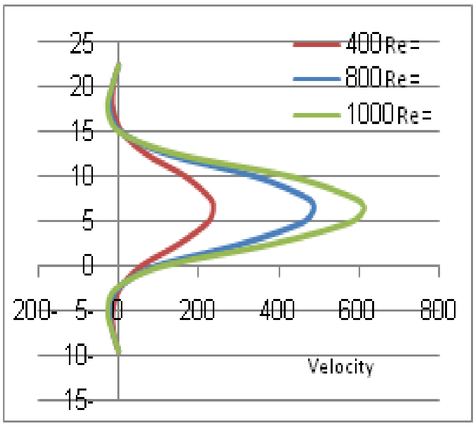
a. b.

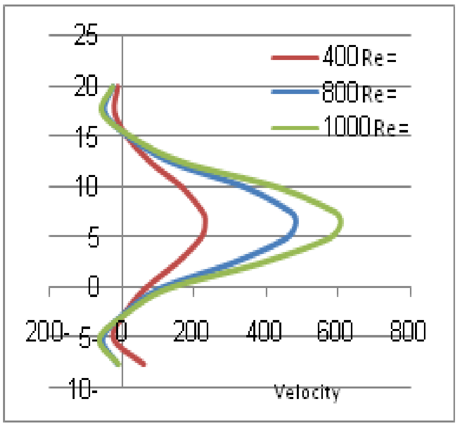
Figure 12: Velocity Direction, a. Re=400, b. Re=1000



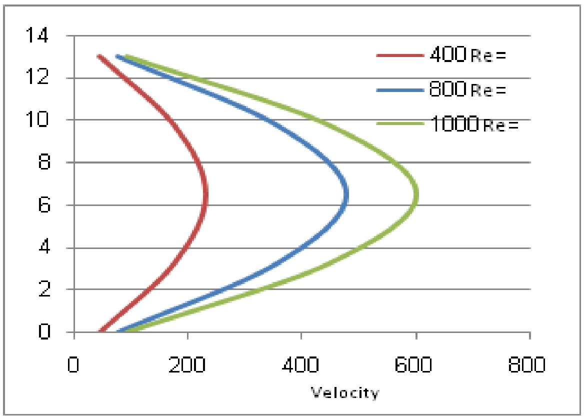


a. Profiles At x=30 **b. Profiles At x=40**





c. Profiles At x=50 **d. Profiles At x=60**



e. Profiles at x=70

Figure 13: Velocity Profiles for Aneuresm with Re=400,800, and 1000

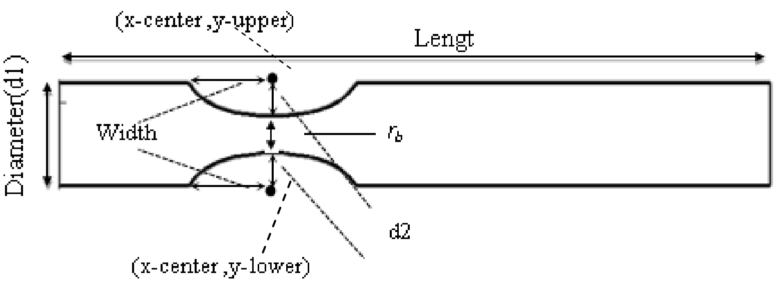
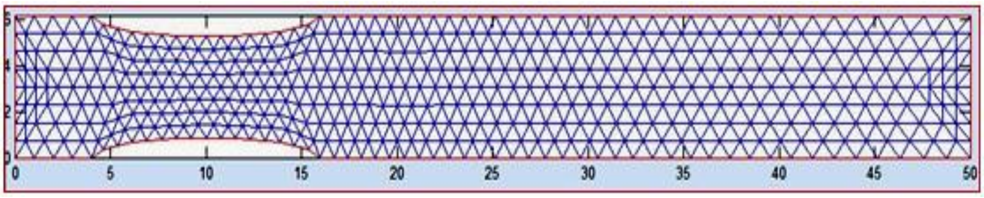
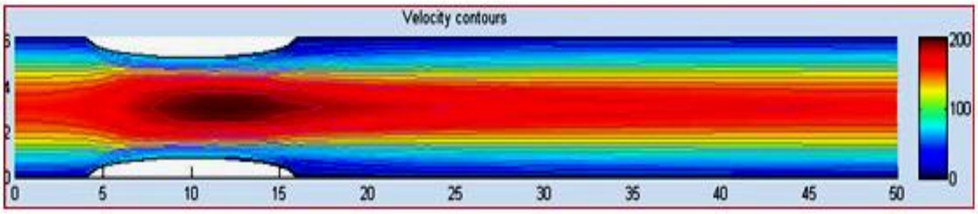


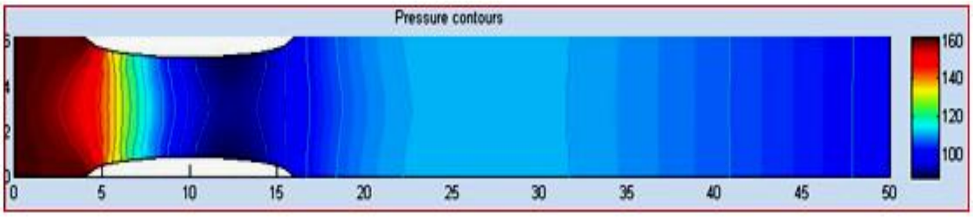
Figure 14: Flat Artery with Rigid Walls and Symmetric Stenosis



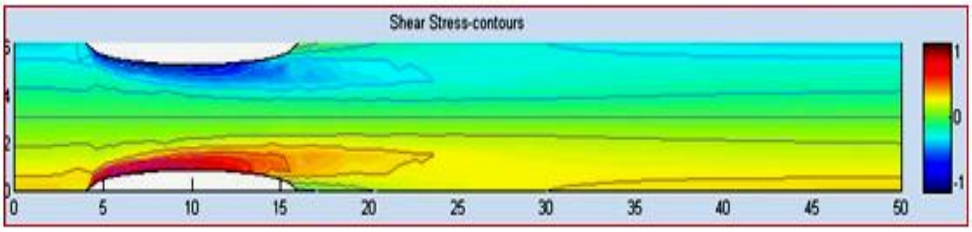
(a) Mesh of the Geometry



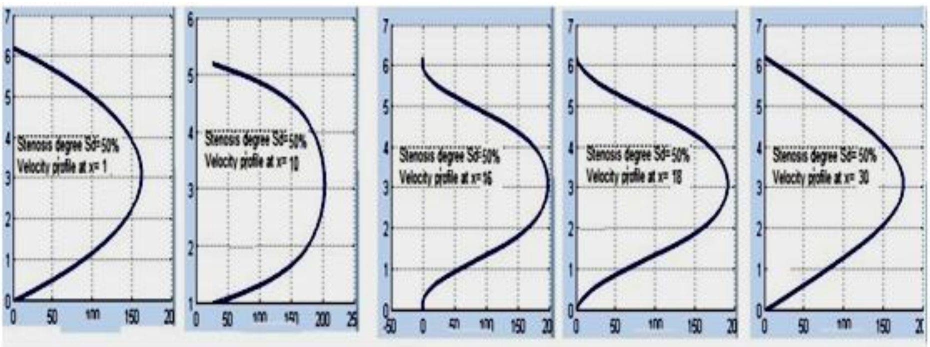
(b) Velocity Distribution and Contours



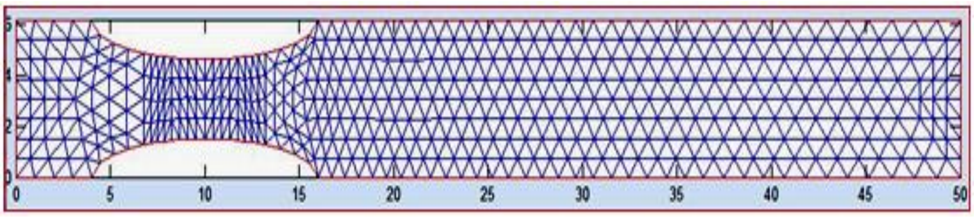
(c) Pressure Contours



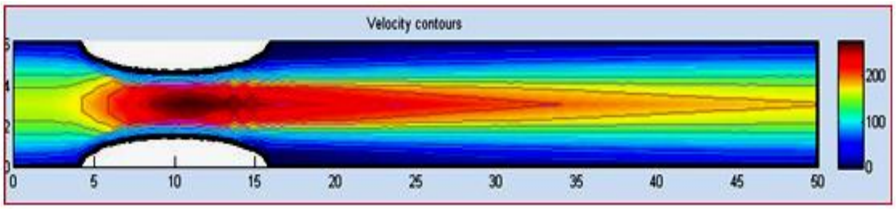
(d) Shear Stress Contours



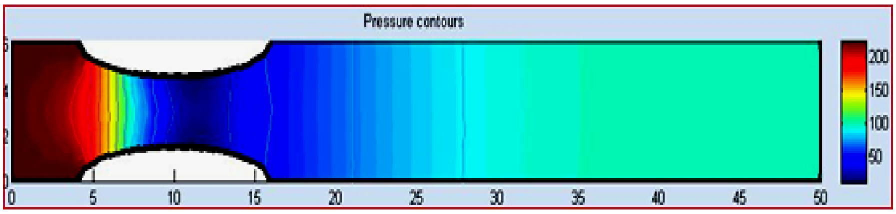
(e) Velocity Profiles at Point Positions (x= 1, 10, 16, 18, and 30)
Figure 15: Stenosis Artery with Re= 200 and Stenosis Degree = 50%



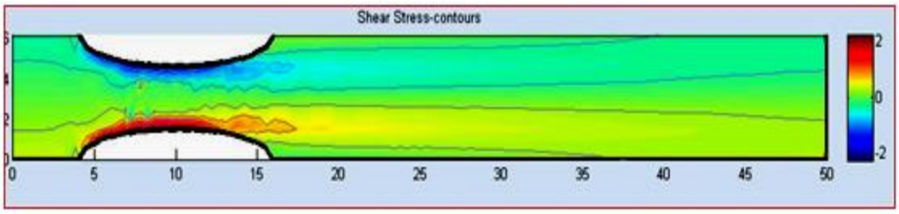
(a) Mesh of the Geometry



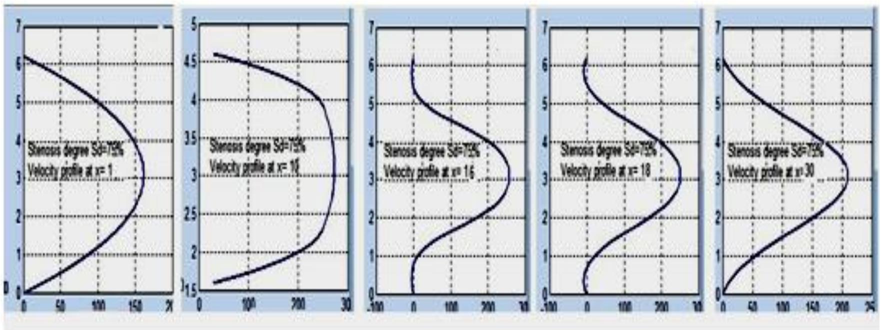
(b) Velocity Distribution and Contours



(c) Pressure Contours



(d) Shear Stress Contours



(e) Velocity Profiles at Point Positions (x= 1, 10, 16, 18, and 30)

Figure 16: Stenosis Artery with Re= 200 and Stenosis Degree = 75%