CFD Analysis of Hemodynamic System of Human Body

Gaurav Devdikar[1] Pratik Mahure[2] Sourabh Lolge[3] Yashwant Hardikar [4]
Prof. R.S. Sewane[5]

Abstract: - Hemodynamic system of human body is complex to understand. Cardiovascular diseases which affects this system, are the main cause of illness and premature failure worldwide. This class of diseases that affect the heart and blood vessels accounts for approximately 40 percent of deaths, with a combined direct / indirect economic cost of approximately € 196 billion per year. Computational Fluid Dynamics has shown a great potential in diagnosing these complex diseases in the near future. This paper studies the fundamentals required for setting up a CFD simulation of different blood vessels. The paper reviews and draws conclusions from different case studies of implementation of CFD analysis in Medical Diagnosing field.

Keywords: -Hemodynamic system, Computational Fluid Dynamics, Navier – Stokes equation, Systolic , Diastolic , Blood Pressure, Arteries, Gmsh Finite Element meshing software, Carotid Bifurcation, Time averaged Wall shear stress, Aneurysm, Cardio Pulmonary Bypass.

I. Introduction

This seminar report gives an overview on recent developments in the field of computational Hemodynamics. The report reviews the methods, benefits and challenges associated with adoption and translation of computational fluid dynamics (CFD) modelling mainly within cardiovascular medicine. CFD, a specialist area of mathematics and a branch of fluid mechanics, is used routinely in a diverse range of safety –critical engineering systems, which increasingly is being used in cardiovascular systems. CFD modelling has already revolutionised research and development of devices such as stents, valve prostheses, and ventricular assist devices. Combined with cardiovascular imaging, CFD simulation enables detailed characterisation of complex physiological pressure and flow fields and the computation of metrics which cannot be directly measured, for example, wall shear stress. CFD models are now being translated into clinical tools for physicians to use across the spectrum of coronary, valvular, congenital, myocardial and peripheral vascular diseases. The adoption of CFD modelling signals a new era in cardiovascular medicine. While potentially highly beneficial, a number of academic and commercial groups are addressing the associated methodological, regulatory, education- and service-related challenges.

II. Literature review

1. Image-based computational hemodynamics evaluation of atherosclerotic carotid bifurcation models

- Widely accepted treatment for carotid artery stenosis includes stenting as well as carotid endoarter- ectomy (CEA), despite complications associated with distal embolism. This paper explains in detail the procedure for computational analysis of carotid bifurcation in Human body. The virtual platform incorporates high-resolution three-dimensional angiography results with Computational Fluid Dynamics modelling to determine clinically related indicators .

2. Using Computational Fluid Dynamics Analysis to Characterize Local Hemodynamic Features of Middle Cerebral Artery Aneurysm Rupture Points

- Although rupture of cerebral aneurysms typically occurs at the fragile wall at the apex or pole, some aneurysms rupture through the body or the neck. The purpose of this study was to clarify the association between aneurysm rupture points and hemodynamic features through the use of computational fluid dynamics (CFD) analysis. This study highlights the relationship between the local hemodynamic features and the rupture points observed during the microsurgical clipping. CFD may determine a rupture point of aneurysms using the feature of markedly low WSS.

3. Three-dimensional numerical simulation of blood flow in the aortic arch during cardiopulmonary bypass

- Blood flow was numerically simulated using the finite element method in the following representative case: a curved arterial cannula was inserted into the anterior wall of the distal ascending aorta 2 cm below the orifice of brachiocephalic artery Perfusion was performed. Computational grids, consisting of 1,493,297 tetrahedral elements, were generated. Computational fluid dynamics could be applied in the future to assess an individual's risk of stroke. Further multiple specific cases needed to be simulated.

III. Hemodynamics

- Hemodynamics or hæmodynamics is the dynamics of blood flow. The circulatory system is controlled by homeostatic mechanisms, much as hydraulic circuits are controlled by control systems. Hemodynamic response continuously monitors and adjusts to conditions in the body and its environment. Thus, hemodynamics explains the physical laws that govern the flow of blood in the blood vessels.
- Blood flow ensures the transportation of nutrients, hormones, metabolic wastes, O_2 and CO_2 throughout the body to maintain cell-level metabolism, the regulation of the pH, osmotic pressure and temperature of the whole body, and the protection from microbial and mechanical harms.Blood is a non-Newtonian fluid; best studied using rheology rather than hydrodynamics. Blood vessels are not rigid tubes, so classic hydrodynamics and fluids mechanics based on the use of classical viscometers are not capable of explaining hemodynamics. The study of the blood flow is called hemodynamics. The study of the properties of the blood flow is called hemo-rheology.

A. Blood

Blood is composed of plasma and formed elements. The plasma contains 91.5% water, 7% proteins and 1.5% other solutes. The formed elements are platelets, white blood cells and red blood cells, the presence of these formed elements and their interaction with plasma molecules are the main reasons why blood differs so much from ideal Newtonian fluids.

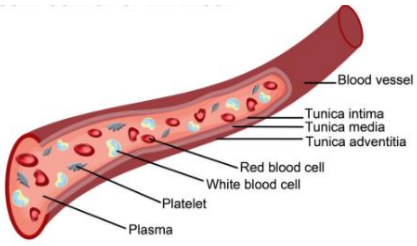


Fig 3: Composition of Blood

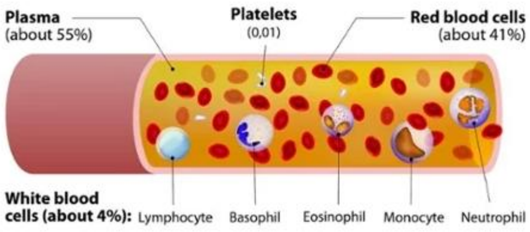


Fig 4: The elements of Blood

1) ***Physics of Blood:*** The main functions of the blood are the transport, and delivery of oxygen and nutrients, removal of carbon dioxide and waste products of metabolism, distribution of heat and signals of immune system. The blood flow resistance is influenced by the complicated architecture of the vascular network and flow behaviour of blood components - blood cells and plasma. At a macroscopic level the blood appears to be a liquid material, but at a microscopic level the blood appears to be a material with microscopic solid particles of varying size - various blood cells

2) ***Blood Density:*** Density is a fluid property and is defined as: ρ = Mass/Volume Usually blood density is assumed to be similar (equal) to water density at T=300 K and P=105 Pa ρ = 1060 [Kg/m3 .

3) ***Blood Viscosity :*** For viscous flow if the relationship between viscous forces (tangential component) and velocity gradient is linear then the fluid is called Newtonian and the slope of the line is a measure of a fluid property called viscosity (dynamic). Sy= µ dv/d.Also a cinematic viscosity can be defined by: υ = µ/ρ. Typical values are: µblood = 0.0035 [kg/ms] (υblood = 3.3 10-6 [m2/s]).

4) ***Blood :Type of fluid***

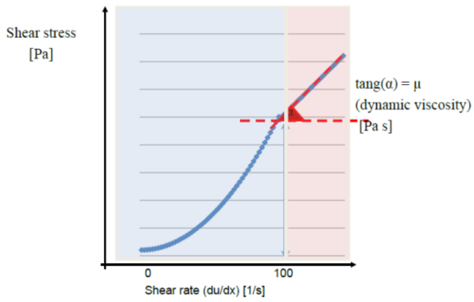


Fig 5 Graph indicating type of fluid

Figure shows that blood is Newtonian and Non Newtonian as well.But other theories consider blood as Non Newtonian and is studied using Rheology.

B. Cardiovascular Hemodynamics

This section essentially deals with dynamics of blood flow in the cardiovascular system.

1) ***Cardiovascular System***

Cardiovascular system is one of the most important systems of the human body. It is defined as an organ system that permits blood to circulate and transport nutrients (such as amino acids and electrolytes), oxygen, carbon dioxide, hormones, and blood cells to and from the cells in the body to provide nourishment and help in fighting diseases, stabilize temperature and pH, and maintain homeostasis. It is also referred to as Circulatory system.

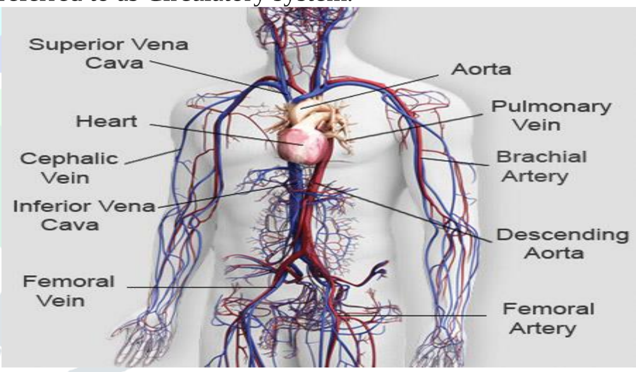


Fig 6 : Human Cardiovascular System

2) ***Cardiovascular System in engineering point of view***

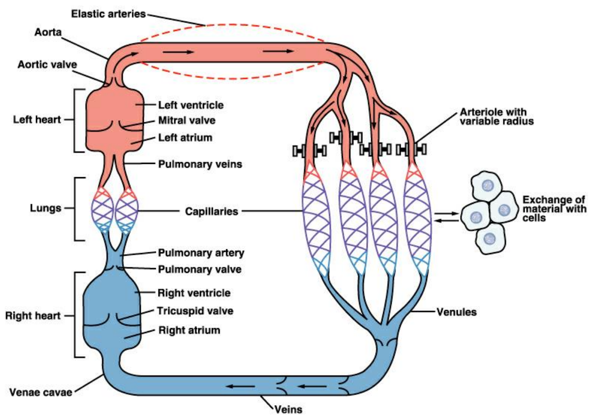


Fig 7 : Exploded view depicting functioning of cardiovascular system

As a CFD engineer, one should have knowledge about:
Basic understanding of the circulatory system working
Main parts and their functions.Characteristics of the blood e.g.: Blood density ,viscosity etc.

3) Important Parameters

- Diameter and Wall shear stress
- Pressure
- Velocity
- Flow
- laminar vs. turbulent
- Resistance
- Viscosity
- Energy
- Area
- Volume

- **Diameter and Wall shear stress**

Vessel type	Diameter (mm)	Wall Thickness	Function
Aorta	25	2	Pulse dampening and distribution
Large Arteries	1.0-4.0	1 mm	Distribution of arterial blood
Small Arteries	0.2-1.0	1 micron	Distribution and resistance
Arterioles	0.01-0.2	6 micron	Resistance (pressure & flow regulation)
Capillaries	0.006-0.01	0.5 micron	Exchange
Venules	0.01-0.20	1 micron	Exchange, collection and capacitance
Veins	0.2-0.5	0.5mm	Capacitance function(blood volume)
Vena Cava	35	1.5 mm	Collection of venous blood

Table 1 **Diameter and Wall shear stress**

- **Velocities and other characteristics**

Velocities : Aorta: 30-50 cm/s
Capillaries: 0-1 cm/s
Blood mass: 8% of body mass

Volumes (percent of total blood volume): Systemic: 83%
Arteries: 11%
Capillaries: 5%
Veins: 67%
Pulmonary: 12%
Heart: 5%

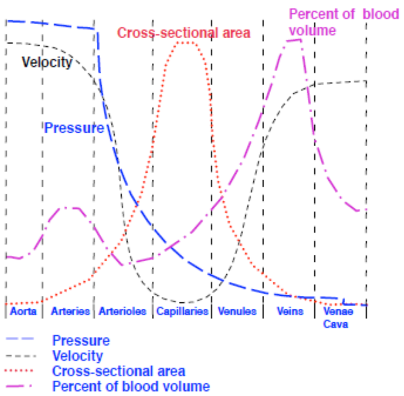


Fig 8: Diagrammatic representation of different parameters vs. blood tubes

- **Velocity vs. Area**

• Flow is constant throughout
• Flow =∫velocity * area

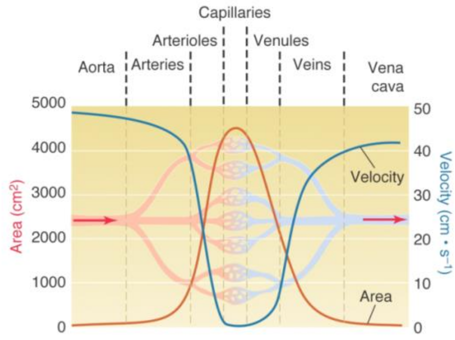


Fig 9: Diagrammatic representation of velocity vs. area for different blood tubes

IV. Computational Hemodynamics

Computational Hemodynamics is defined as mathematical modelling (with the help of programming) of dynamics of blood flow. Mathematical modelling is done on the basis of the parameters mention in section 3.Computational Hemodynamics involves different numerical methods. Hemodynamic system can be compared with Electric circuit.

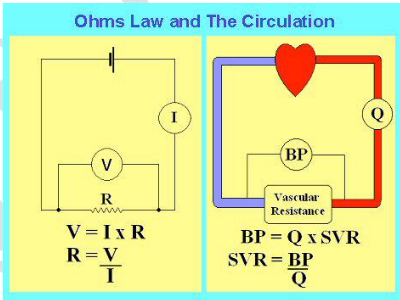


Fig 10: Diagram depicting analogy between cardiovascular system and electrical system

CFD analysis differs from numerical modelling or is a type of computational modellingCFD is simply the process of obtaining numerical solutions to accepted classical equations (Navier-Stokes, *ca* 1827) Numerical modelling usually involves the development of equations of assumed form and containing free constants derived from experimental data.

V. Simulation

As discussed, Finite Element Analysis and Computational Fluid Dynamics can be used to develop different blood flow simulation.

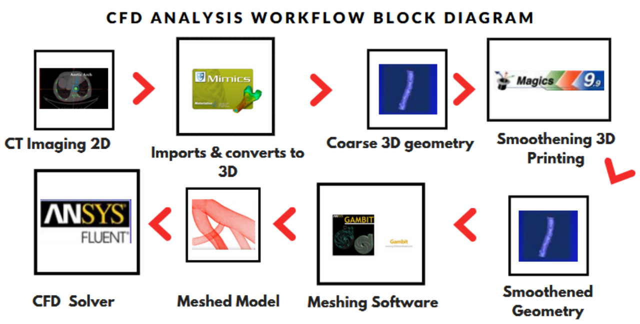
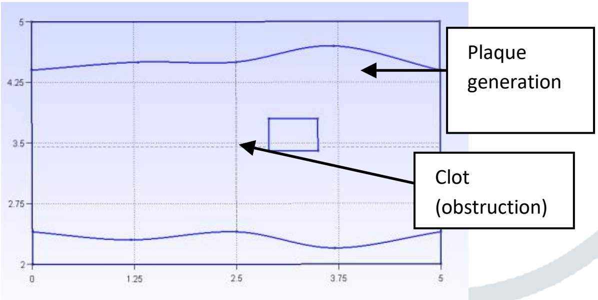


Fig : CFD Analysis Workflow Block Diagram

As discussed in the above block diagram, the geometry is obtained through CT (Computed Tomography) imaging, cleaned and pre processed to make the geometry ready for analysisFor any CFD Analysis, complex components are assumed to have some basic shapes. Thus, making the analysis easier. Assumptions should be such that solver will give the results closer to the actual. Hence, we have assumed a square shape obstruction as clot in the artery.

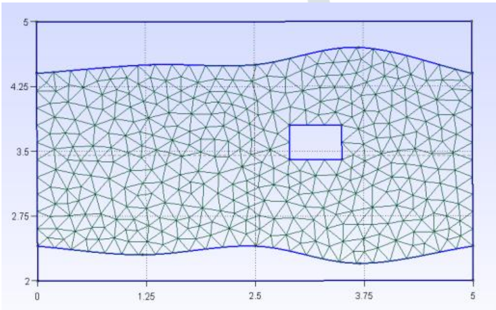
1) ***Problem Statement :*** To develop a 2D simulation of blood flowing through an artery and observing the effects caused by presence of some obstacle in the path of flow.

2) ***CAD model:***



- A rectangular shaped artery is modelled. Splined curves are used to signify plaque generation.

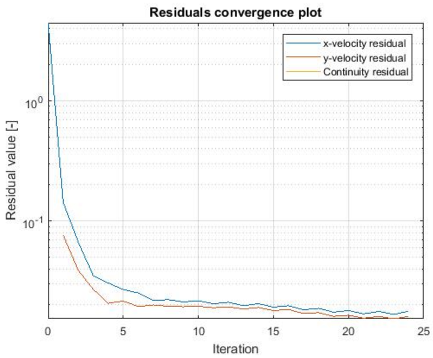
3) ***Meshed model***



- Geometry is meshed using Gmsh finite element mesh generator.

‘

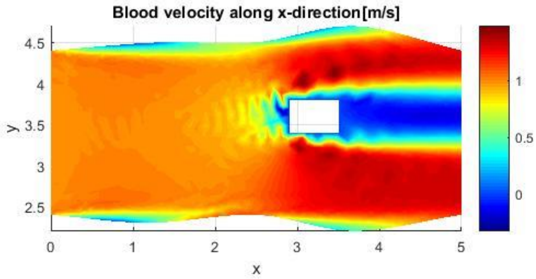
4) ***Solution***



- Mathematical model consisting of Navier – Stoke’s equation is solved by using MATLAB solver.

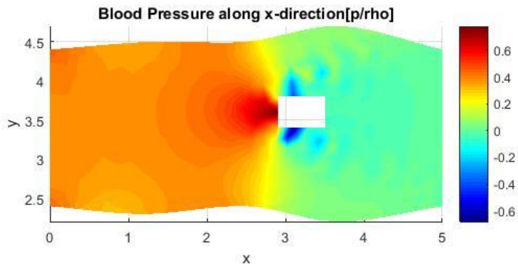
5) ***Simulation Results***

- Velocity Contour



from the fig it can be seen that there is the variation of velocity of blood before and after the clot, in addition to that the velocity is less in the direction of clot and is more in the neighbouring sides of the clot, hence the variation of flow of blood can be analysed through CFD

- Pressure Contour



from the fig it can be seen that the pressure entirely reduces after the clot. The pressure becomes very high near the starting section of clot due to the obstruction provided by it, so the variation of pressure can also be analysed by CFD.

6) ***Conclusion:*** From the analysis and study it can be concluded that CFD can be used to study the variation and behaviour of blood as it flows through the veins and arteries and is a useful tool to predict the health condition .

VI. Case Studies of CFD application in medical field

A. Respiratory Flow in Lung Bifurcation

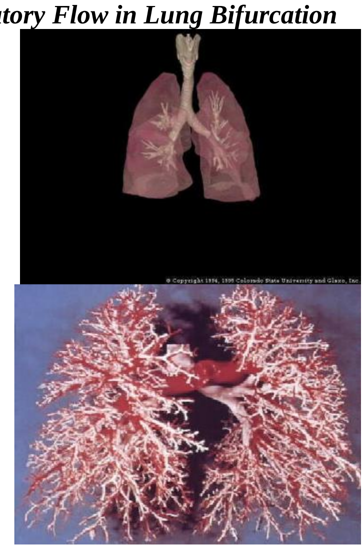


Fig 11: Actual Lungs Bifurcation

1) ***Problem Statement***: Flow in the airways of the lungs is not understood well enough by medical science and cannot satisfactorily measured.

2) ***Mathematical model*** :Air flow at low speeds is well predicted by the incompressible form of the Navier Stoke EquationsAir at low speeds is a constant-property ,Newtonian fluidFlow is generally laminar in small airwaysTo assume rigid walls of the Bifurcation is the first approximation.

3) ***Application of Navier Stokes Equation***

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla)\vec{u} = -\frac{\nabla p}{\rho} + \nu \nabla^2 \vec{u} + \frac{\vec{F}}{\rho}$$

4) ***Surface of selected model :***

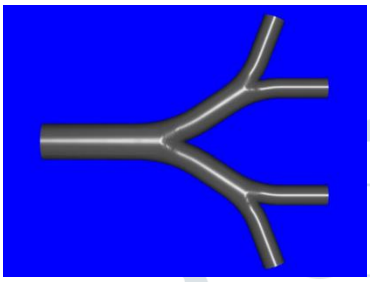


Fig 12: In Cad software Bifurcation Airway is designed considering rigid walls

5) ***Discretization of the Geometry:***

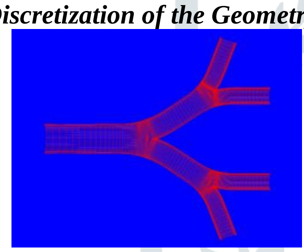


Fig13: Meshing and grid generation

6) ***Algorithm:*** Finite Volume formation. To apply conservation laws. Implicit Time stepping and Newton iteration to converge each time step.

7) ***Visualisation of Result :***

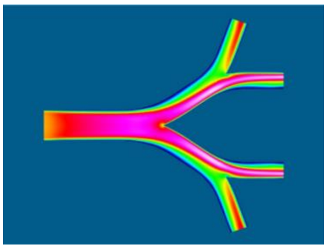


Fig 14: Simulated model

8) ***Asymmetric Area simulation:***

Fig 15: Simulated model of asymmetric airway

9) ***Conclusion*:** CFD analysis helped to understand actual air flow in the Bifurcation. Respiratory diseases including blockages in airways of the bifurcation can now be analysed.

B. Atherosclerotic carotid bifurcation

Atherosclerosis is a chronic disease of artery vessels whereby the artery walls thicken causing the lumen size for blood transport to narrow. Carotid artery stenosis (CAS) is a narrowing of the carotid arteries, the two major arteries that carry oxygen-rich blood from the heart to the brain. It is caused by a build up of plaque (atherosclerosis) inside the artery wall that reduces blood flow to the brain. Its symptoms are weakness or numbness in arms and legs, difficulty speaking which lead to stroke and extreme effects can be permanent brain injury.

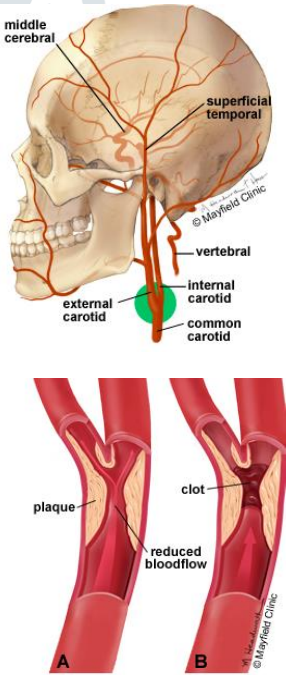


Fig 16: Carotid bifurcation atherosclerosis

1) ***Problem Statement:*** While a strong correlation exists between severe CAS and stroke incidence, using anatomical stenosis geometry as a single indicator of risk of stroke is poorly justified. The hemodynamic burden (effect on blood flow transportation) caused by the existence of the plaque was not investigated, which is an important practical indicator in helping the treating physician for carrying out risk stratification and arranging appropriate treatments.

2) ***Inspiration for CFD :*** The reports have shown that frequent occurrence of localized atherosclerotic plaques in curvature, bifurcation , and branching of arterial vessel regions suggest that fluid dynamics and vessel geometry may have an influence in plaque formation. Numerical examinations until now which has established that the magnitude and gradient of blood flow near the vessel wall ,or the wall shear stress ,is a source of pathogenesis of CAS.

3) ***Methodology:*** Patient-specific realistic carotid bifurcation models were reconstructed from MR image data and their corresponding stenosis severity were anatomically assessed. The numerical accuracy of the proposed modelling approach was evaluated through comparison of numerical and experimental results on the basis of two standard idealized carotid bifurcation models. Then, a computational hemodynamics approach was proposed and performed on these patient specific carotid bifurcation models. The hemodynamics performance and computational efficiency were examined for clinical diagnostic purposes.

4) ***Importing model from MR imaging*:** A 3D model is exported from Mimics software which was used for reconstructing the computational model as .stl file (stereo lithography).

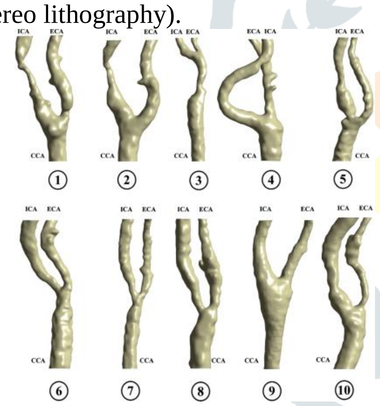


Fig 17: Ten patient specific carotid models

Short Form	Description
CCA	Common carotid artery
ICA	Internal carotid artery
ECA	External carotid artery
ST -AHCB	Standard model with 63%ICA stenosis averaged tuning fork shaped model (no.9 in Fig 17)
TF -AHCB	Standard model with plain, healthy tuning fork shaped model (no.10 in Fig 17)

Table II: Technical termsST -AHCB and TF –AHCB are considered for comparison with other 8 models.

5) ***Computational mesh generation for CFD* analysis :** The carotid bifurcation CFD models were generated with a structural hexahedral mesh using a multiblock O grid method within ICEM CFD (ANSYS,US).A structured mesh reduces numerical diffusion and requires lower cell counts.

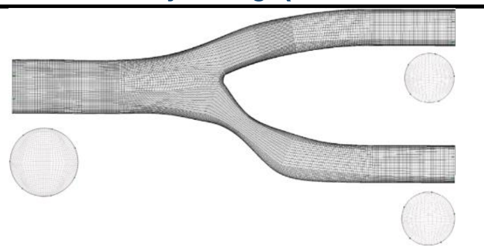


Fig 18: Meshed model of ST-AHCB

6) ***Boundary conditions and flow modelling:*** **Inlet Boundary condition:** Holdsworth boundary condition is applied at the inlet of each model using temporary varying Wimberley velocity profile. This type of velocity profile describes the characteristic of oscillatory flow in a tube arising in the solution of the linearized Navier Stokes equations.

$$u_1(r,t) \approx \frac{k_s}{4\mu}(r^2 - a^2)(1 + \cos\omega t).$$

Analytical solution given by Zamirwhere t is time; r is the radial coordinate; a is the radius of artery; ks is the steady-state part of the pressure gradient ; J0 is the Bessel function of order zero of the first kind; ω are angular frequencies after Fourier Transformation; and Λ ¼ i3=2α is the complex frequency parameter.**Outlet Boundary condition:** The downstream peripheral vascular impedance (DPVI) model was adopted for the outlet boundary condition. Its main purpose was to establish a feasible and efficient hemodynamics framework through connecting two porous domains with different transient permeability configurations to outlets of carotid bifurcation.

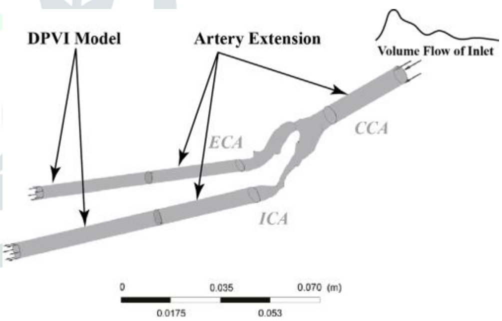


Fig 19: A schematic diagram of the DPVI modelling method

Flow modelling : The shear thinning and viscoelasticity of blood are closely related to its microscopic structures such as deformation, aggregation and alignment of red blood cells which mainly determine the rheological behaviour of blood. The studies show that shear thinning effects are dominant and hence viscoelasticity can be ignored for prediction of velocity. In this study Carreu –Yasuda model is used for identifying shear thinning behaviour.

$$\frac{\eta - \eta_\infty}{\eta_0 - \eta} = [1 + (\lambda\gamma)^a]^{(n-1)/a},$$

$\eta_\infty = 3.7 \times 10^{-3}$ Pa s, $\eta_0 = 3.14 \times 10^{-2}$ Pa s, $\lambda = 2.517$ s, $n = 0.5736$, $a = 2$, and $\rho = 1060$ kg m^{-3}.

7) ***CFD simulation:*** Each computational mesh was imported into ANSYSCFX, which uses a finite volume method to solve the unsteady incompressible Navier–Stokes equations. Vessel walls were assumed to be rigid for all simulations. For the simulation, two full cardiac cycles were

required to damp out the initial transient errors. Therefore, a total of three full cardiac cycle calculations were performed with the last cycle used for data analysis.

Two widely adopted flow indicators Time averaged wall shear stress (TAWSS) and Oscillatory shear index (OSI)are used to evaluate total shear stress exerted on the wall throughout cardiac cycle.

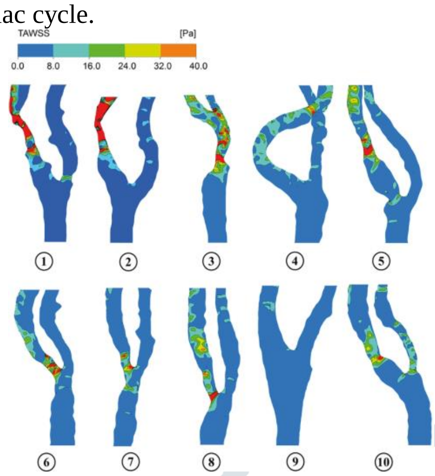


Fig 20: Time-averaged WSS(wall shear stress) distributions of the ten studied carotid bifurcation models.

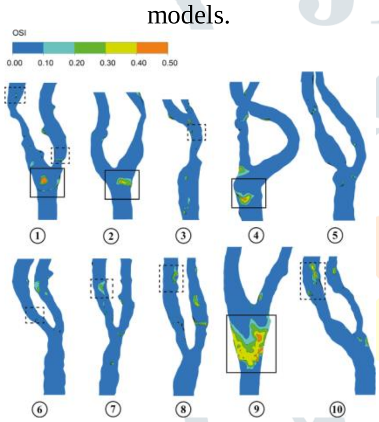


Fig 21: OSI distributions of the ten studied carotid bifurcation models. Box indicate high OSI and dotted indicate disturbed flow induced by stenotic lesions.

8) *Results:*

Image based anatomical evaluations alone can determine artery stenosis burden but cannot provide detailed evidence to quantify the stenosis severity. As TAWSS based analysis of the stenotic carotid bifurcation models can capture the abnormally high WSS lesion sites as well as the disturbed flow regions quantified by high OSI, therefore, the risk of direct endothelial injury and the consequent progression of plaque could be assessed and predicted. It was found that the presence of plaque increases WSS due to accelerated blood flow, which can cause plaque erosion and thrombosis which lead to stroke. Error was in the range of 16 to 25 % which is commendable achievement. Unlike conventional methods which need to insert a probe for obtaining results, CFD analysis is accurate, safe and efficient.

C. *CFD analysis for detecting rupture points in the body:*

Aneurysm: An aneurysm is a localized, abnormal, weak spot on a blood vessel wall that causes an outward bulging, likened to a bubble or balloon. Aneurysms are a result of a weakened blood vessel wall, and may be a result of a hereditary condition or an acquired disease.

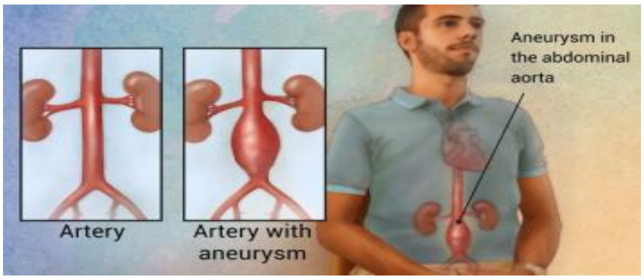


Fig 22: Aneurysm

1) ***Problem Statement:*** Although rupture of cerebral aneurysms typically occurs at the fragile wall at the apex or pole, some aneurysms rupture through the body or the neck. The purpose of this study was to clarify the association between aneurysm rupture points and hemodynamic features through the use of computational fluid dynamics (CFD) analysis.

2) ***Methods:*** Twelve ruptured middle cerebral artery bifurcation aneurysms were analyzed by 3-dimensional computed tomographic angiography and CFD. Rupture points were evaluated on intra operative videos by 3 independent neurosurgeons. Wall shear stress (WSS) was calculated at the rupture point, aneurysm dome, and parent artery. Intra-aneurysmal flow patterns were evaluated with cross-sectional velocity vector planes that included the rupture points.

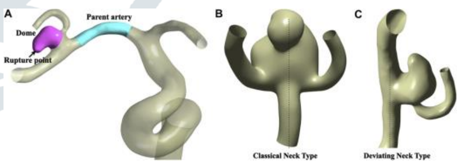


Fig 23: Model of Cerebral Artery with Dome and Rupture point

3) ***Numerical Modelling***: For the fluid domain, 3D incompressible laminar flow fields were obtained by solving the continuity and Navier-Stokes equations. Numerical modeling was performed using a commercially available CFD package (ANSYS CFX12.1; ANSYS Inc., Canonsburg, PA, USA). Vessel walls were assumed to be rigid, and no slip boundary conditions were applied at the walls. Blood was assumed to be an incompressible Newtonian fluid with a blood density of 1056 kg/m3 and a blood dynamics viscosity of 0.0035 N/m2/s. Because patient-specific flow information was not available, pulsatile boundary conditions were based on the superposition blood-flow waveforms of the common carotid artery as characterized by Doppler ultrasound in normal human subjects for transient analysis (9). Traction-free boundary

conditions were applied at outlets of the MCA and the anterior cerebral artery (20). The time steps were 0.005 seconds. To reduce initial transients, we computed 3 cardiac cycles, and data of the third cardiac cycle were analyzed. Hemodynamic results of the flow field were examined at end-diastole.

4) ***Results:*** The mean WSS at the rupture point (0.29 Pa) was significantly lower than that at the dome (2.27 Pa) and the parent artery (8.19 Pa) ($P < .01$). All rupture points were located within the area of WSS ≤ 11.2% of the WSS at the parent artery. WSS at the rupture point was correlated with the minimum WSS at the dome ($r = 0.64$, $P < .05$), but not with aneurysm size ($r = 0.26$) or the aspect ratio ($r = 0.16$). Flow patterns revealed that all rupture points were located in lower-velocity area, which was associated with complex flow patterns and/or deviating necks.

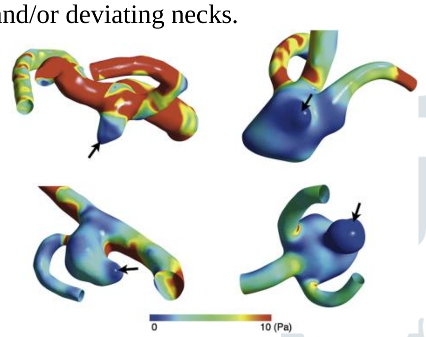


Fig 24: Wall shear stress distribution for 4 representative ruptured middle cerebral artery aneurysms. Arrows show the rupture points.

5) ***Conclusion:*** CFD can be efficient tool for detecting rupture point in the body. Compared to microsurgical clipping, it can be safe and more accurate method.

D. *CFD analysis for determining effects of temporary setups during surgical operations*:

Cardiopulmonary bypass is a technique that temporarily takes over the function of the heart and lungs during surgery, maintaining the circulation of blood and the oxygen content of the patient's body

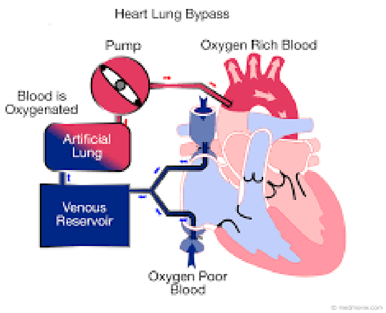


Fig 25: Cardio Pulmonary Bypass

1) ***Problem Statement:*** During Cardio Pulmonary Bypass (CPB) techniques used during cardiac surgery patient may undergo stroke. Hence to analyse this situation and to find out where WSS is high which subsequently leads to stroke are the main objectives.

2) ***Methodology*:** It was assumed that a 22 Fr curved aortic cannula would be inserted via the anterior wall of the distal ascending aorta, 2 cm below the orifice of brachiocephalic artery. The tip of the cannula was angled slightly cephalad. This was done by moving the cannula 308 counter-clockwise away from the long axis of the aorta. The cannula was then fixed perpendicular to the aortic wall where it was inserted.

By using the Amira advanced 3D visualization and volume modelling software (Mercury Computer Systems, Inc., Chelmsford, MA, USA), the centreline data of the aorta and its branches was determined from the CTscans using the successive region growing method [6]. The cross sections were superimposed perpendicular to the centreline data. Based on the centreline and diameter information, by using a sweeping technique, the solid geometry of the aorta and its branches were generated in form of the non uniform rational B-spline (NURBS) curves.

Finally, based on the reconstructed geometry described above, computational grids with tetrahedral cells (called elements in finite element method) were generated using the advancing front method for the purposes of CFD analysis. The reconstructed grids consisted of 265,856 nodes and 1,493,297 tetrahedral elements.

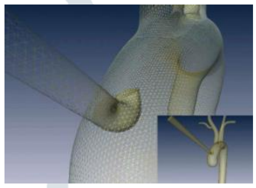


Fig 26: Computational Grid

3) ***Numerical Modelling*:** For the purposes of this study, therefore, the blood was assumed to be an incompressible, homogeneous and Newtonian fluid, with a constant viscosity of 0.0035 Pa. The density of blood was taken to be 1050 kg m3. The vessel walls were considered to be rigid, and a no-slip condition was applied at the walls. At the ends of the branches and the distal end of the aorta, natural outflow boundary conditions were applied. In a CFD analysis for an incompressible fluid, because the density is taken constant, the calculated pressures are relative to the pressure boundary conditions, rather than absolute. Thus the wall pressure (the pressure perpendicular to the wall) is expressed by the relative pressure value. In the present model, the pressures at the ends of the vessels were assumed to be the base reference pressure.

Within the reconstructed model, a cardiopulmonary bypass flow of 4.1 l/min was simulated. The Navier—Stokes equations that govern fluid motion were solved by the finite element method using Acusolve CFD solver (ACUSIM Software, Inc., Mountain View, CA, USA) . Amira was used again for the 3D visualization of the results. The wall shear rate was determined by the velocity parallel to the wall within the grid adjacent to the wall, since a no-slip condition was applied to the wall. Wall shear stress was then determined from the product of the viscosity of the fluid and the wall shear rate. The average distance from the wall to the adjacent grid was 0.68 mm.

4) **Simulations:**

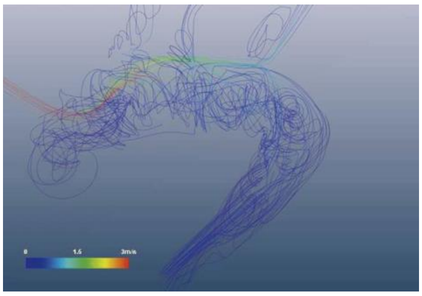


Fig 27: The wall pressure distribution on the aortic wall during cardiopulmonary bypass.

The wall pressure is the force applied perpendicular to the wall. The highest wall pressure was observed at the superior-posterior wall of the aorta where the jet flow impingement occurred. The highest wall pressure was 3104.8 Pa.

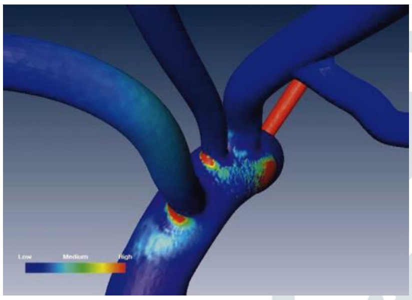


Fig 28: The shear stress on the aortic wall during cardiopulmonary bypass
(Posterior view)

The shear stress is the force applied parallel to the wall caused by flow viscosity. The maximum wall shear stress was 25.1 Pa.

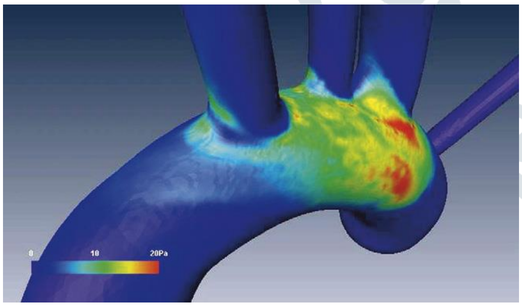


Fig 29: Streamlines of the turbulence flow within the aortic arch

The streamline of turbulence was visualized using colour velocity mapping. The high velocity vortex flow is eventually headed into the left subclavian artery.

5) **Conclusion and Results**: This study confirmed that blood flow during cardiopulmonary bypass can be simulated and visualized using the CFD techniques that are currently available. The present study showed that blood flow during CPB can be simulated using CFD techniques. It provides a good example of how computational fluid dynamics could be applied in the future to assess an individual's risk of stroke**,** as a result of CPB. Further multiple representative cases need to be selected.

VII. Technical Challenges

The CFD simulation for each carotid bifurcation model (mentioned section 5.2) in was performed by a HP XW6600 workstation, 6 CPU (E5440 2.83 GHz) were assigned for local parallel run mode, and the total simulation time is around 10 h.

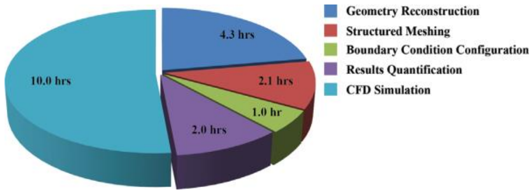


Fig 30: Time required for each step

This tells us that CFD simulation of Human body which is extremely complex gives effective results but is time consuming.
Geometry reconstruction discussed in each case study is very tedious task. The inlet and outlet conditions for blood vessels are not constant hence approximations have to be made.

.

VIII. Conclusion

Computational Fluid Dynamics is proving to be extremely important in Medical field day by day. Engineering and Medical streams are coming closer and in coming near future they will work together to cure complex diseases. Important parameters such as Wall shear stress, turbulence of blood, point of rupture in the body and artery blockage can now be easily detected. This report gives idea about properties of blood as a fluid in CFD domain, basic steps involved in CFD, governing equations and case studies of application of CFD in Medical field. If we are able to overcome technical challenges CFD will prove to be a helping hand in diagnosing the patients.

Acknowledgement

We wish to express our sincere thanks and gratitude to Prof. R S Sewane for her enthusiastic and valuable guidance. We would like to express our honour, respect towards HOD of our college Dr N.P.Sherje for providing us with this platform. We would also like to express our deep sense of gratitude to all those who directly or indirectly helped us in completing this paper.

References

[1] Jingliang Dong, "Image-based computational hemodynamics evaluation of atherosclerotic carotid bifurcation models", Computers inBiologyandMedicine43(2013)1353–1362

[2] Keiji Fukazawa1,"Using Computational Fluid Analysis to Characterize local Hemodynamic Features of Middle Cerebral Artery Aneurysm Rupture Points",(2014)

[3] Three-dimensional numerical simulation of blood flow in the aorticarch during cardiopulmonary bypass, Science direct,www.elsevier.com

[4] Rajat Mittal," Computational modelling of cardiac hemodynamics: Current status and future outlook" Journal ofComputationalPhysics305(2016)1065–10