ABSTRACT

Monitoring the blood flow pattern inside the arteries over a period of time can give significant insights into the development of atherosclerotic plaques. However, the validity of the data collected over a period of time to perform blood flow analysis in the arteries should be determined beforehand. In this work, a validity check is performed for using the MRI data of a carotid vessel collected at different times to perform a blood flow analysis. The collected MRI data is used to reconstruct a three-dimensional (3D) geometry of the carotid bifurcation and this geometry is used in a finite element model to conduct a blood flow analysis. This process is repeated for the two different sets of MRI data of an individual, collected at two different periods in time. The results of the blood flow pattern inside the carotid for the two different cases are very similar. This work shows that the procedure is robust and precise to give highly repeatable results for the blood flow pattern.

KEY WORDS
Fluid-Structure Modelling, Blood Flow, Carotid Artery, Repeatability studies

1. Introduction

A lot of research is going on in understanding the implications of the physics of blood flow in arteries on the development of atherosclerotic plaques. To understand the blood flow phenomena in arteries, a fluid flow analysis should be performed on the arteries either experimentally or computationally. An analysis performed using a Computational Fluid Dynamics (CFD) software has been tested and proved to be efficient and accurate within the limits of assumptions made for the analysis. Such an analysis requires the information of the artery geometry in a solid model form. A very efficient and accurate way of obtaining the geometry information is using non-invasive in-vivo high resolution Magnetic Resonance Imaging. There has been extensive research and improvements in the field of high resolution Magnetic Resonance Imaging (MRI), that allows for clear identification of regions of plaque and also to differentiate various components in the plaque [1], [2]. As a result of this it has been possible to extract vessel geometry information for use in CFD software packages. A variety of analyses have been performed using the geometry information including a complete three-dimensional (3D) fluid flow analysis including the fluid-structure interaction to model the elastic nature of the vessel. Information about flow parameters like velocity, wall shear stress, oscillatory shear stress, wall shear stress temporal gradient etc have been documented at various regions of the arteries [3].

A study performed on a single patient, monitoring the changes in the fluid flow pattern over a period of time as the lesion develops can give significant understand on the phenomena of atherosclerotic plaques. The carotid bifurcation region of single individual was used to perform the repeatability test. Two separate sets of high resolution carotid MR images were taken on two different days. The MRI data thus obtained was used to construct the solid model of the geometry. This geometry was used for a CFD analysis. The results of the analysis show good repeatability.

2. Methodology

The data from the MR imaging is stored in DICOM (Digital Imaging and Communication in Medicine) format. The file of DICOM format consists of two parts: a header (containing information about the patient’s name, type of scan, image table position, the dimensions of the image) and the image data. A number of commercially available software can take these DICOM files for visualization, segmentation and 3D rendering of MRI data. One of such interactive tools is Materialise's Interactive Medical Image Control System (MIMICS). This software automatically reads the header data in the DICOM files and determines the sizes of each pixel and
arranges the MR images along their table positions. By default all images are scaled to a gray scale range between 0 – 4095 Houndsfield units. In order to extract the information of the carotid, segmentation of the MRI data needs to be done. Segmentation consists of Thresholding, Region Growing, 3D reconstruction. At the end of a 3D reconstruction a very coarse solid model is obtained a smoothing operation is performed to and after segmentation Smoothing operations.

During Thresholding, a range of gray scale values are selected such that the region to be segmented is best contrasted within that range. Since the MRI data available in this work was bright blood data, the range of 1350-4095 was selected. This range allows for the selection of bright colored pixels. Figure 1 shows the result of thresholding. As can be seen, regions of high brightness have been colored/selected.

As shown in figure 2, Mimics allows for smoothing and remeshing an STL or 3D object. This is needed in order to remove the artifacts from 3D reconstruction and raise the quality of the triangles so that the preprocessor of an FEA package can build a tetrahedron mesh from them.

The smoothing algorithm changes the position of a point according to the positions of its neighboring points. The importance of the other points can be raised or lowered by adjusting the Ratio-parameter. If this Ratio is low (0.01) the new position is mainly dependent on the old position of the point. If this ratio is 1, the dependency is spread over the points. The new position is still 50% dependent on the old position. With high values for the ratio, the new position is mainly determined by the position of the other points of the triangles. In this last case it is obvious that we talk about smoothing. This algorithm is exercised on every point of the part. The smoothing algorithm has the side effect that the part is shrinking. This factor will compensate this. This compensation step is similar to the first one except that the parameter has a negative sign - this means that point is actually moved in opposite direction (out of "smooth" position), so points "oscillate " forwards and backwards.

The remeshing package in Mimics saves the
information of the solid model in STL format. The STL format stores the information of the solid model in the form of triangles. Since this information can be directly used in the FEA package of mesh generation, it is important to make sure that these triangles are uniform and conducive for FEA analysis. As mentioned above, the remeshing tool in Mimics provides the facility of identifying the bad triangles and algorithms for correcting them. In this work, the Height/Width (normalized) ratio of (0.3) was used to identify and correct the bad triangles in the geometry.

The geometry has been smoothened and remeshed. The geometry is then exported to a .lis file format that can be read as in input file by ANSYS.

Ansys is a multi-physics analysis software, that includes a flow analysis module called FLOTRAN. This software was chosen to do the flow analysis because of the built in capability to do a fluid-structure interaction problem without having to make use of 3rd party softwares to do the structure analysis and the coupling between fluid and structure. The governing equations solved by the Ansys, FLOTRAN packages are given in the following. The continuity equation for conservation of mass, which is given in the following in indicial notation.

\[ \dot{\rho}_o + \partial_i (\rho V_i) = 0 \]

The momentum equation in indicial notation, that are using in Ansys are as follows:

\[ \partial_o \rho V_i + \partial_j (\rho V_j V_i) = \rho g_j - \partial_i P + R_i + \partial_j (\mu \partial_j V_i) + T_i \]

Where \( \rho \), \( V \) denote density and velocity respectively. \( P \) denotes pressure and \( g \) denotes acceleration due to gravity. \( R \) and \( T \) denote distributed resistance and viscous loss terms respectively. Note that in the case of incompressible fluid density is assumed to be constant. Furthermore, Ansys solves the above equations for a relative pressure \( \rho_{rel} \) instead of absolute pressure. Hence a reference pressure needs to be specified in the problem. In this study the reference pressure was set equal to atmospheric pressure. Finally the Arbitrary Lagrangian-Eularian (ALE) scheme was used for advection. With the Eulerian framework it is not straightforward to solve problems involving moving boundaries or deforming domains. While such problems are more suitable for a Lagrangian framework, in practice the mesh distortions can be quite severe leading to mesh entanglement and other inaccuracies. A pragmatic way around this problem is to move the mesh independent of the fluid particles in such a way as to minimize the distortions. This is the ALE formulation which involves moving the mesh nodal points in some heuristic fashion so as to track the boundary motion/domain deformation and at the same time minimizing the mesh degradation.

Blood is treated as an incompressible, Newtonian fluid (this assumption has been proved to work well for large vessels. The density and viscosity of blood was assumed to be 1100 kg/m3 and 0.004 Pa.s respectively. There were no turbulence models used in the simulation. The boundary condition for the simulation consist of a uniform velocity at the inlet, a constant pressure at the outlet and no-slip condition along the walls of the carotid vessel. The value of velocity at the inlet was chosen such that a constant flow rate of 12 ml/s was maintained. The mesh used in one of the analysis problems is show in Figure 3b (with cross-sections from the MRI images shown in Figure 3a). This figure also shows the variation of wall shear stress in the carotid vessel.

![Figure 3a: MR Images of the neck section. a, b and c are samples from MR images of neck taken at respective location of sections shown in Fig. 3b.](image)
3. Results

The two sets of MRI data that have been used in this study have been labeled as “scan1” and “scan2”. These two sets of data contain the carotid MR images of a single person taken at different points in time. Mimics was used for visualization, segmentation and 3D reconstruction of solid models from the data sets. Figure 4 shows the solid models of scan1 and scan2 data, constructed using Mimics.

The above geometries were imported into Ansys to carry out flow analysis. The boundary conditions and assumptions made have been specified under methodology. The velocity pattern inside the carotid was found to be very similar in both cases. Figure 5 shows the velocity pattern at steady state, across a vertical cross section of both cases.
The maximum velocity achieved in both cases varied by about 2%. Note the similarity in the flow pattern at the bifurcation region. We see a recirculation region in the internal carotid artery in both cases. The regions of high velocity shown in red also match in both cases.

The following plot in Figure 6 shows the contour plot of Wall Shear Stress (WSS) at steady state, along the walls of the carotid. The maximum value of WSS differed by a value of 18%. However as can be seen in figure 6, this maximum value occurred in very small regions at the inlet or the outlet. In general qualitatively the WSS throughout the carotid is very close.

4. Conclusion

A robust, accurate and repeatable system of procuring MRI data and performing flow analysis has been developed. This method has been used to conduct steady flow as well as time dependent pressure pulse analyses. The results of the analyses have been found to be in comparison with existing literature. A repeatability study was also performed on the MRI data of a carotid extracted from the same person at two different points in time. The MRI data was used to construct a solid geometry using Mimics. The geometry was imported into Ansys to perform flow analysis. The solid model obtained from Mimics and the results of the flow analysis from Ansys are qualitatively very similar. Thus this study lays the foundation for further work to be done in understanding the changes in flow pattern in blood vessel over a period of time and their influence on vascular diseases.

References


Figure 6. Contour plot of wall shear stress