A study of internal pipe flows and the internal carotid artery

Submitted by:

Emirhan Ilhan, Parikshit Upadhyaya

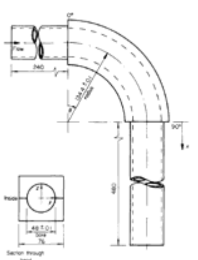
Project Course in Scientific Computing(SF 2567)

Introduction

The study of internal flows is helpful in the field of medicine. Certain diseases such as atherosclerosis are closely related to the blood flow dynamics in blood vessels which can be analyzed as internal flow in a pipe. Since the internal carotid artery is the largest artery in the human body and the blood flow is fastest through it, it is paramount to understand the flow around that region. The first part of this project is about verifying and validating a 90° bent pipe problem whereas the second part of the project is about the numerical investigation of this physical problem and the implications of the results of this investigation. ANSYS Fluent has been used for the numerical simulations.

90° Bent Pipe Model:

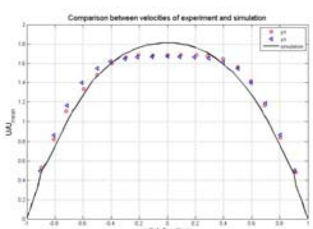
We have opted to solve a 90° bent pipe problem first to validate that a model that we create similar to that of the internal carotid artery model actually works. It was not possible to validate the internal carotid artery model directly as there are no available experimental/numerical results for the geometry that was analyzed in this study. Having said that, there is experimental results for similar problems. One such [1] example can be seen below:



The inlet boundary condition is given as a fully-developed laminar flow with 10.5 mm/s mean velocity. The walls of the pipe have a no-slip boundary condition. The working fluid is water. A non-obvious modeling detail is about how to implement a fully-developed flow boundary condition at the inlet. Fully developed flow implies that the velocity profile does not change in the fluid flow direction. In reality, a constant velocity profile becomes fully-developed after a certain entrance length and this is length is a function of Reynolds number. In our model, we could imply an artificial fully-developed profile. Instead, we have put a 1.5 m artificial inlet length before our 240 mm inlet so that the flow could find time to become fully-developed on its own.

Results:

Two things are given for this case as the experimental results in the paper. The first one is the velocity profile obtained in a cross section 0.58 diameters upstream of the bent inlet plane. A comparison of the numerical simulation results with these experimental results is given in the figure below:

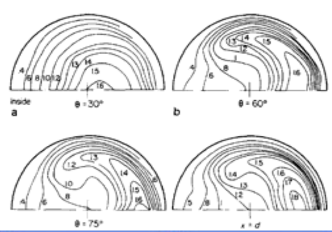


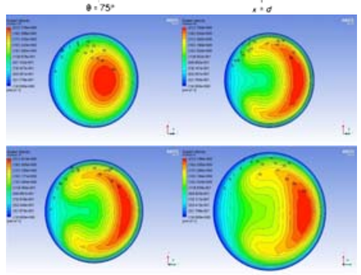
The numerical simulation results capture the behavior of these profiles well apart from the centerline. However, a theoretical analysis of fully developed laminar flows actually dictate that the velocity profile should be parabolic.

For a geometry shown in the figure above, one can obtain the following by solving continuity and momentum equations:

$$u=\frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\left(\frac{y}{a}\right)^2-\left(\frac{y}{a}\right)\right]$$

Therefore, it is probable that the mismatch between the numerical results and the experimental results around the centerline is caused by LDA measurement errors near the higher velocity parts of the cross section. The second comparison we'll make is about the velocity distributions at different angles of the bent. Below are the figures from the experiment and the simulation at different cross sections:

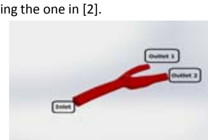




One can see that the results are almost identical. At 30°, we have a flow that has slightly moved to the right which is due to the fact that the pipe has just started to turn. At 60°, this displacement becomes more prominent. At 75°, this speed goes down a little bit just like it is in the experiment. At x = d, the overall magnitude of the velocity distribution increases also like it is in the experiment.

Internal carotid artery model:

Now we move on to the actual purpose of this project, the internal carotid artery model. We required a geometry for this investigation and we ended up using the one in [2].

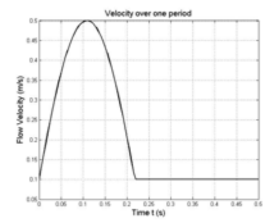


In the figure, inlet diameter is around 6.3 mm whereas outlet 1 diameter is around 4.5 mm and outlet 2 diameter is about 3 mm. The working fluid in this problem is blood. Since blood is not a predefined fluid in ANSYS database and actually a non-Newtonian fluid, we need some work to do. Here, we use Carreau's model to model the blood flow. [3]

$$\mu=\mu_\infty+(\mu_0-\mu_\infty)[1+(\lambda\dot{\gamma})^2]^{(n-1)/2}$$

$$time\ constant \rightarrow \lambda = 3.313s, \qquad power-law\ index \rightarrow n = 0.3568$$
$$zero\ strain\ viscosity \rightarrow \mu_0 = 0.056, infinite\ strain\ viscosity \rightarrow \mu_\infty = 0.0035$$

These values and Carreau's model can be used to define blood as a non-Newtonian fluid in ANSYS database. As boundary conditions, we have a pulsatile cyclic flow at the inlet. The implementation of this boundary condition is different from just implementing a constant velocity boundary condition. We have used a velocity profile proposed by [4]. In this profile, one period corresponds to 120 beats per minute. The maximum velocity in a cycle is 0.5 m/s whereas the minimum velocity is 0.1 m/s. This proposed velocity profile looks as follows:



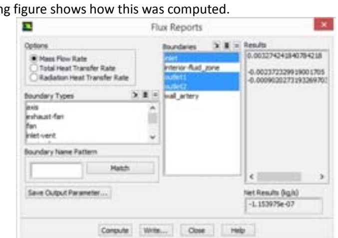
This is obviously going to produce different results compared with a steady velocity flow boundary condition. The stress values are expected to be the highest around the peak of this cycle. We can also observe different flow behaviors around at different times because this is a transient process. A user defined function is implemented to model this flow behavior. Normal resting blood pressure in an adult is approximately 120/80 mmHg. So, we will use the average of these two pressures as our outlet boundary conditions where 100 mmHg is equal to 13.332 kPa.

Results:

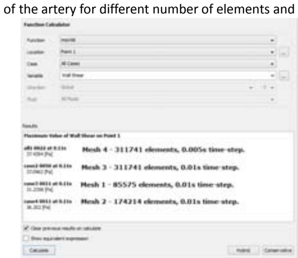
We have used different meshes and time-steps for solving this problem. Our meshes ranged from 85575 elements to 311741 elements and our time-steps were 0.01 and 0.005. It was not possible to go over 512K elements since we used a student edition of ANSYS. A summary of different meshes can be seen in the table below.

	# of Elements	*Time − step*
Mesh 1	85575	0.01
Mesh 2	174214	0.01
Mesh 3	311741	0.01
Mesh 4	311741	0.005

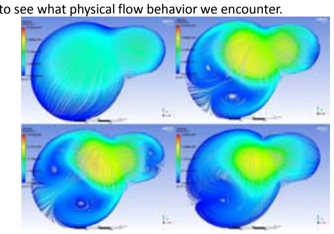
To verify our results, we first checked if mass was conserved or not. The following figure shows how this was computed.



As one can see, the mass difference between the inlet and the outlets is almost zero. This means that the mass is numerically conserved. Following figure shows the Wall Shear Stress at $0.11s$ at a critical region at the bifurcation of the artery for different number of elements and time-steps.



We look into the effect of the bifurcation of the artery as a function of time and try to see what physical flow behavior we encounter.



We have four time instances at a cross section of the bifurcation in the streamline figure above. From left to right: (a) t = 0.05s, (b) t = 0.11s, (c) t = 0.2s, (d) t = 0.35s. As we can see from (a), we start off with a relatively constant velocity. Not much is happening at this time instance. When the velocity reaches peak as seen in (b), we see large vortices generated in the larger branch. This part of the branch is especially important since these circulation zones are perfect zones for plaque formation. (c) shows us the time instance when the flow velocity starts to decrease. The circulation region in the large artery is still present and effective. However, a small circulation region is also generated in the small region. This is due to the fact that when the flow rate is high, the small artery is filled with fluid so there is not a possibility of a slow velocity region in that artery. (d) shows us the time instance in our velocity pulse where the flow speed is constant. At this instance, we see that the circulation area in the small artery is, once again, nonexistent. This is due to the fact that the flow becomes so slow that the velocity distribution over the artery is again quite uniform. The circulation area is a little bigger in the large branch and its movement is very limited. This means that the flow distribution in the large artery is suitable for plaque formation. This time instance is especially important to analyze since the constant velocity portion of the pulse makes up more than 50% of the entire period of the pulse.

References:

[1]M. M. Enayet, M. M. Gibson, A. M. K. P. Taylor, M. Yiannekis. "Laser-Doppler measurements of laminar and turbulent flow in a pipe bend". International Journal of Heat and Fluid Flow, 1982.

[2]Retrieved from: https://grabcad.com/library/carotid-bifurcation

[3]W. Y. Chan, Y. Ding, J. Y. Tu. "Modeling of non-Newtonian blood flow through a stenosed artery incorporating fluid-structure interaction" ANZIAM J. 47 (EMAC2005) pp. C507-C523, 2007. Retrieved from: http://journal.austms.org.au/ojs/index.php/ANZIAMJ/article/viewFile/1059/1001

[4]SINNOTT, Matthew. CLEARY, Paul W. & PRAKASH, Mahesh. An investigation of pulsatile blood flow in a bifurcating artery using a grid-free method. *Fifth International Conference on CFD in the Process Industries CSIRO, Melbourne, Australia* 2006