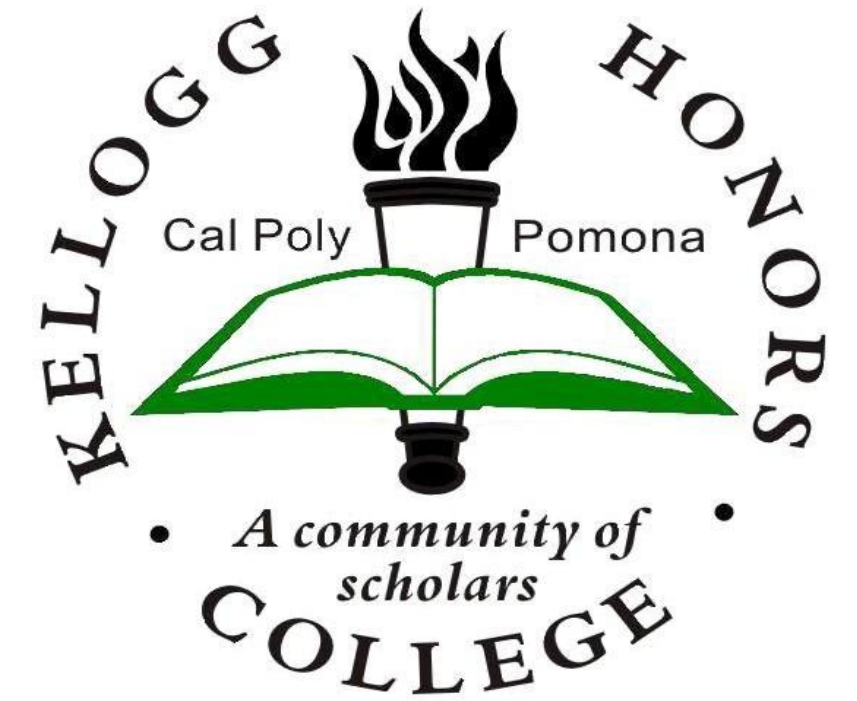


Computational Simulation of Blood Flow in Stenosis Artery

Colin Danahy, Mechanical Engineering

Mentor: Dr. Jaehoon Seong

Kellogg Honors College Capstone Project



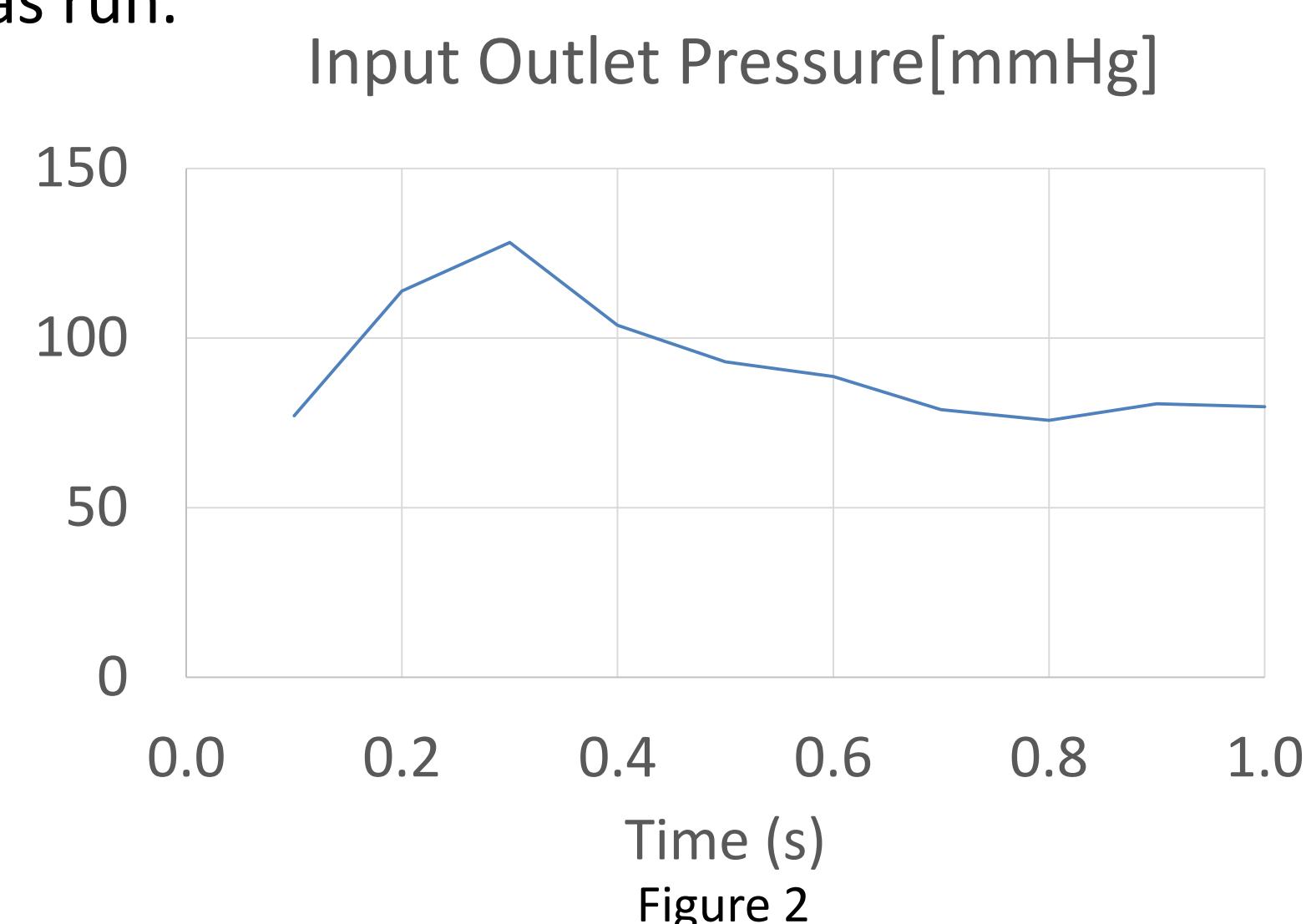
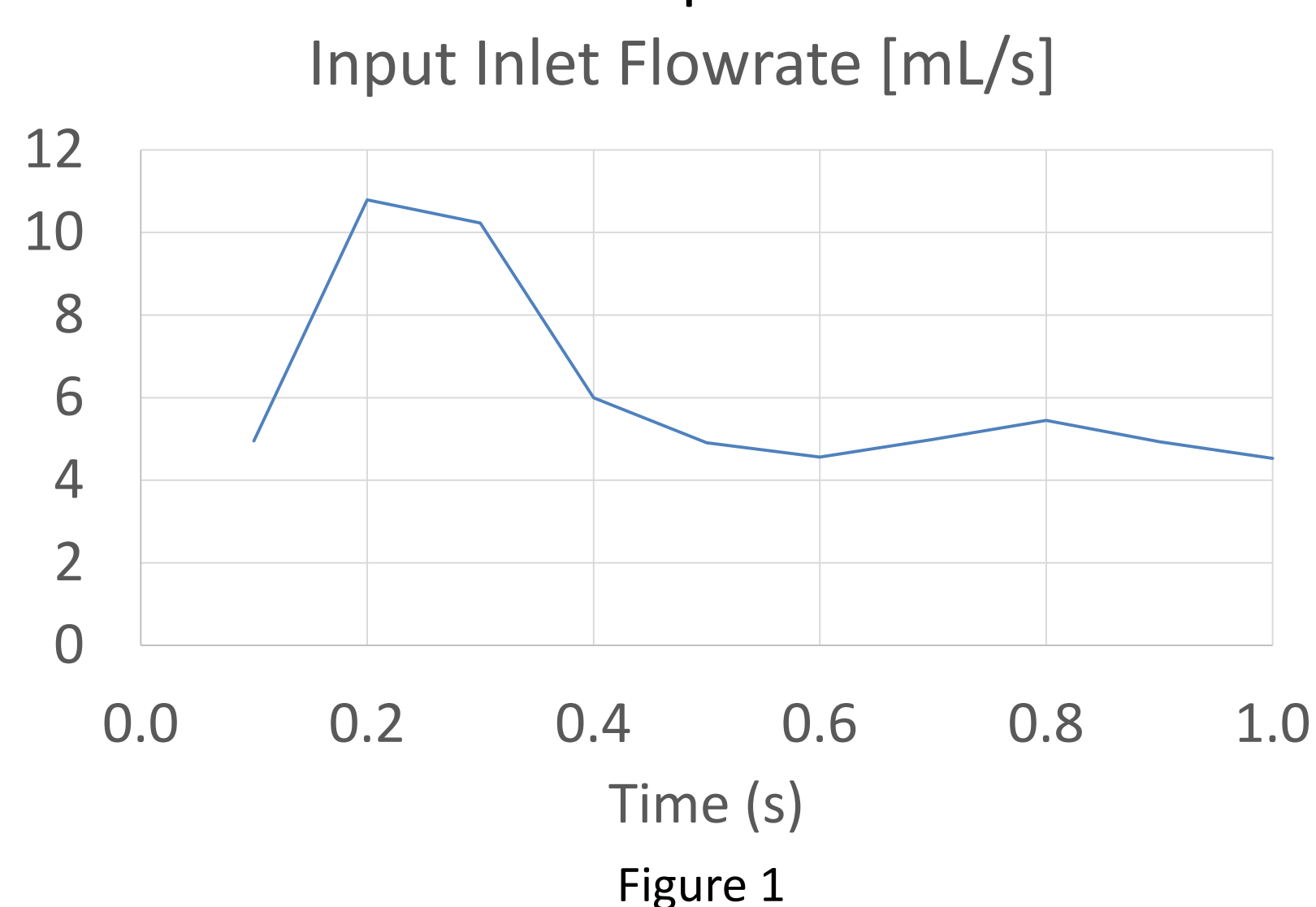
Introduction

The purpose of this advisor-guided independent research course was to come up with a working computer model of arterial stenosis. Arterial stenosis is the narrowing of a blood vessel and can be caused by atherosclerosis or trauma. Based on the organ that is being supplied by the stenosis artery the patient could suffer from pain to more severe issues depending on the organ. The idea behind this research stemmed from me wanting to relate the same principles I had learned in my fluid mechanics course relating to pipe-flow to a blood vessel since I recognized the same principles of systematic fluid structure and the similar geometrical figures in each of the systems. I also wanted to focus my knowledge of fluid mechanics in a biomedical engineering setting in order to understand the mechanical principles of how the body functions and how to come up with a potential solution to this medical issue.. Analyzing these results we can hopefully be able to implement better biomedical devices and medical operations that could better diagnose the stenosis and possibly prevent any restenosis or problems for the patient. Various fluid mechanic principles will be utilized to analyze the results from the simulation including the Navier-stokes equation. Analysis and simulation will be performed using ANSYS Fluent 16.1

Method

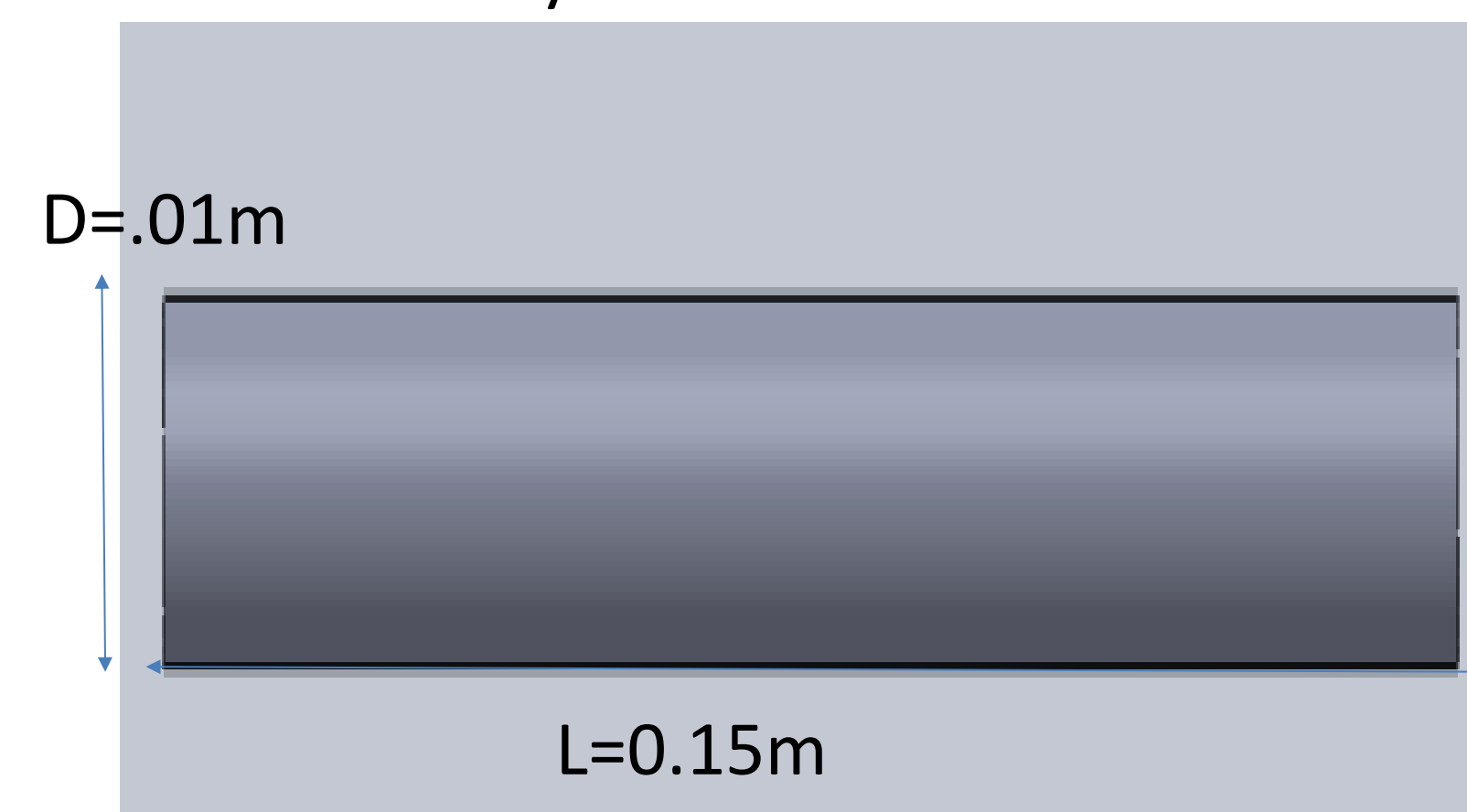
The models of the computational study was adopted from a journal publication entitled "XXXXXXXXXXXX [1]" Load Ansys Workbench 16.2. The diameter created for the straight tube was a diameter of .01m and a length of .15m and the diseased artery had the same length and diameter but a reduction of half the diameter in the middle. A coarse mesh was created for the model in ANSYS and with in the mesh module the inlet, outlet, and fluid zone of the model were specified for the simulation. Once the model was complete a transient simulation(varying properties over time) was selected and the properties of the blood, density and viscosity, were entered into the setup. Input profiles, Fig.1 and Fig.2, were selected for the input and output conditions and were both obtained from patient information found from similar studies.

Before running the model, surface monitors within the simulation were created to record important blood flow information, pressure, velocity and flowrate, at the inlet, midsection, and outlet of the artery. The solution then had to be initialized and then the simulation was run by selecting a time step size of 0.1s, 30 time steps, and 20 iterations for each of the time steps and the simulation was run.

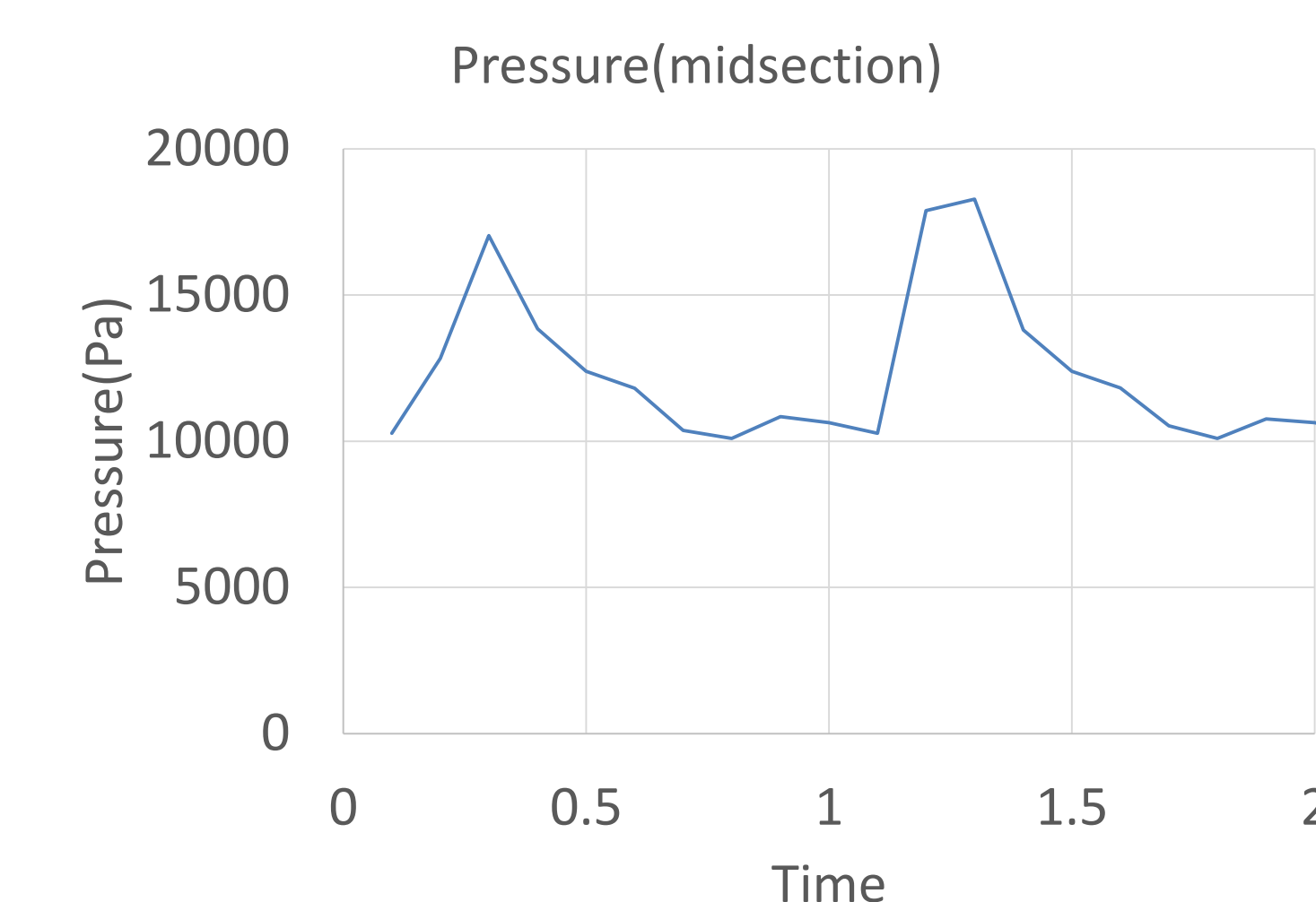
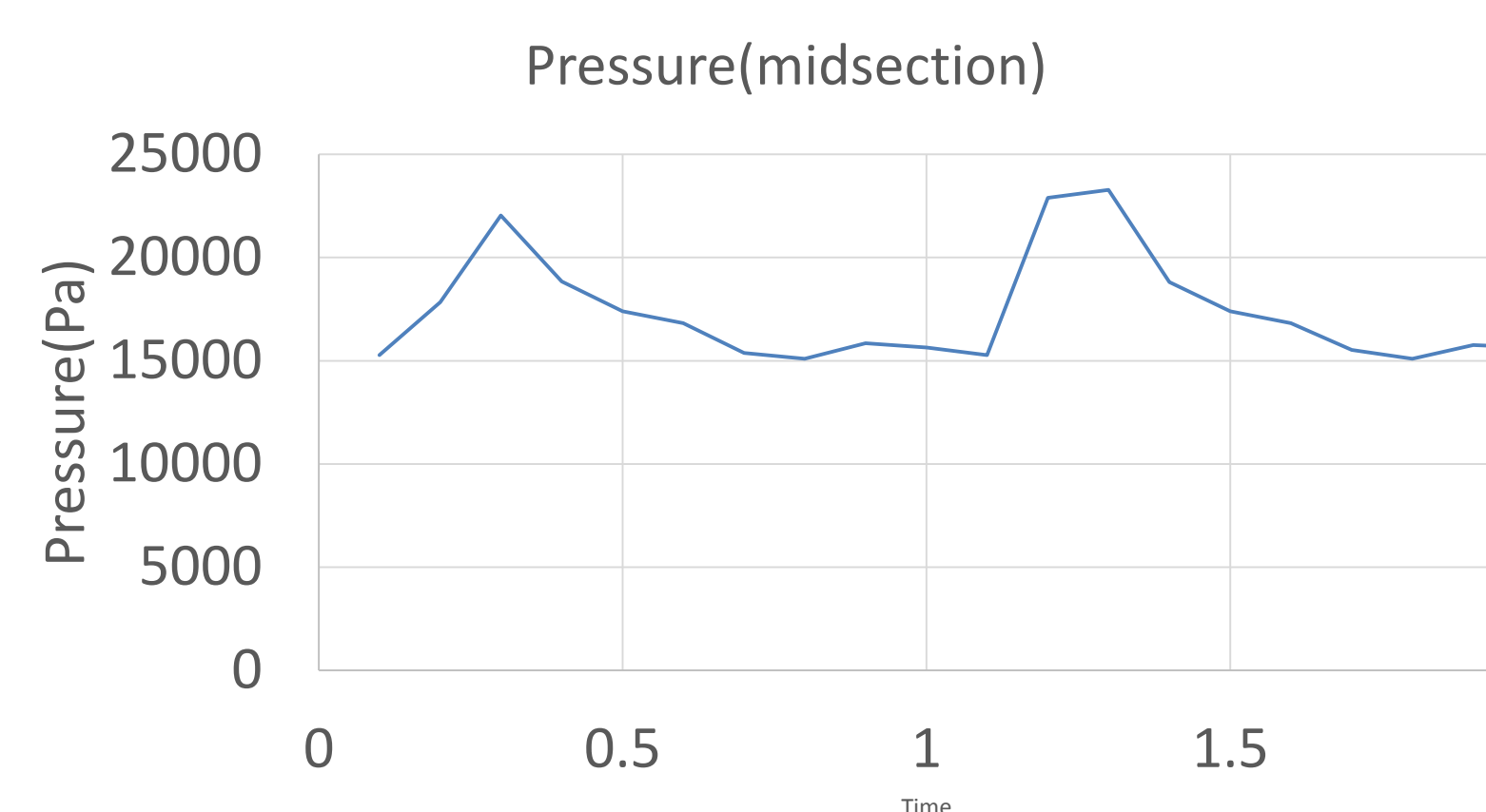
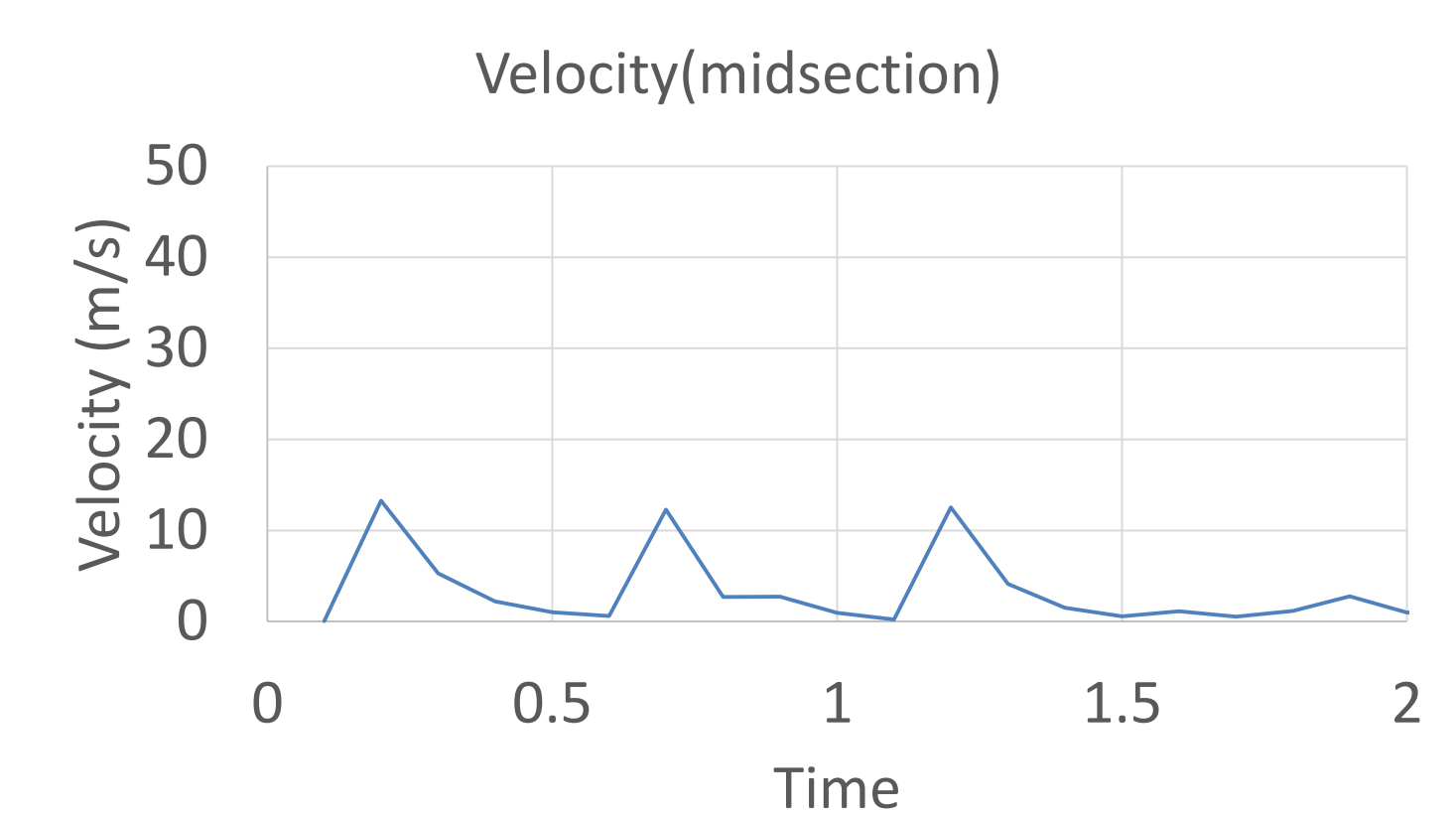
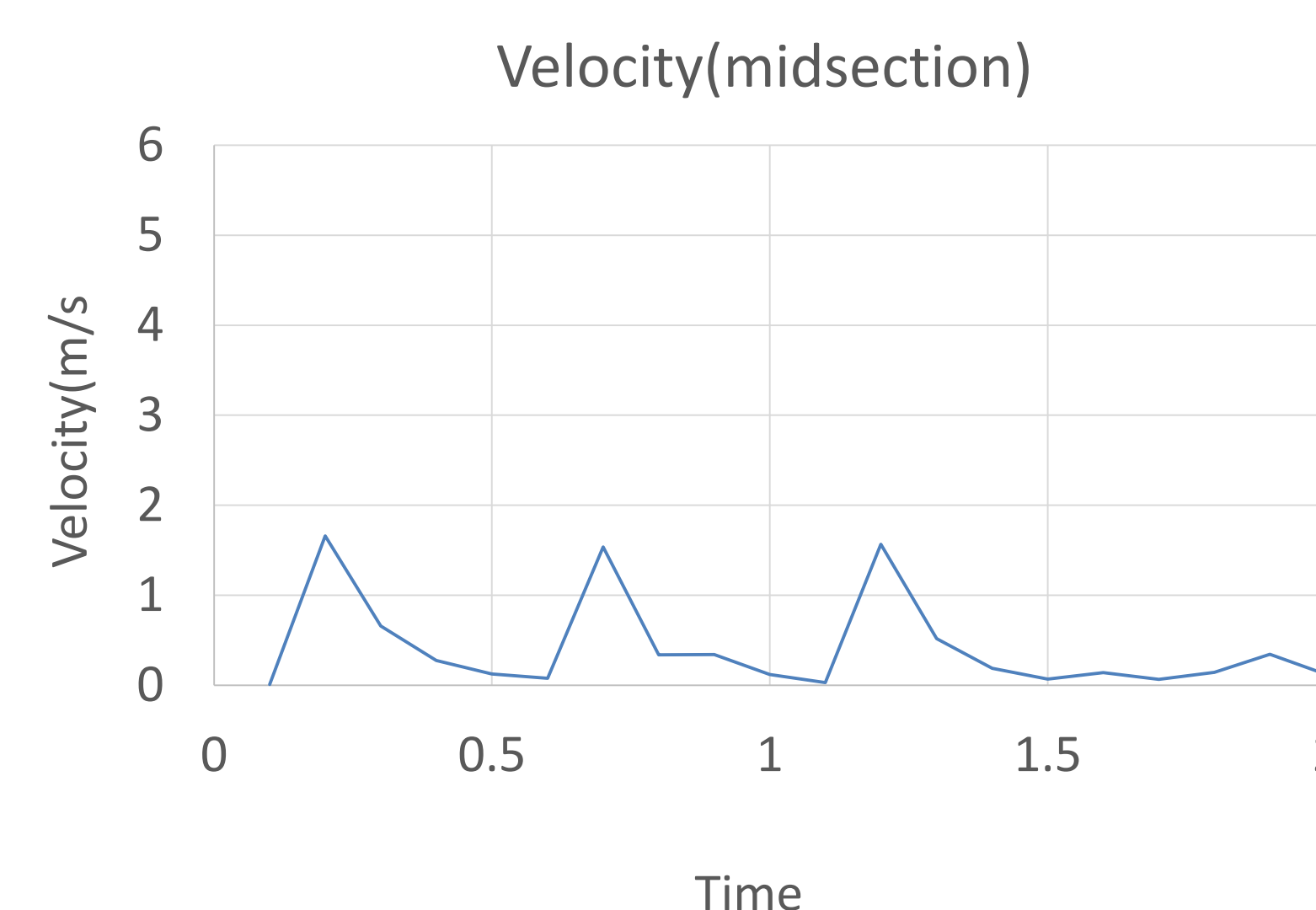
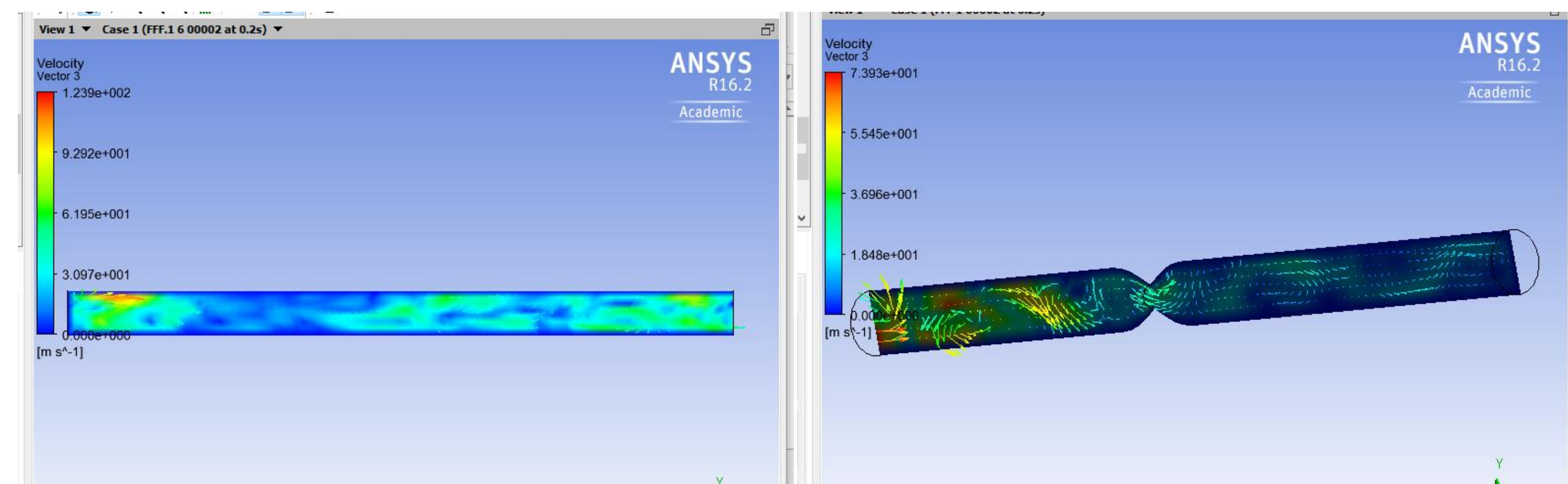
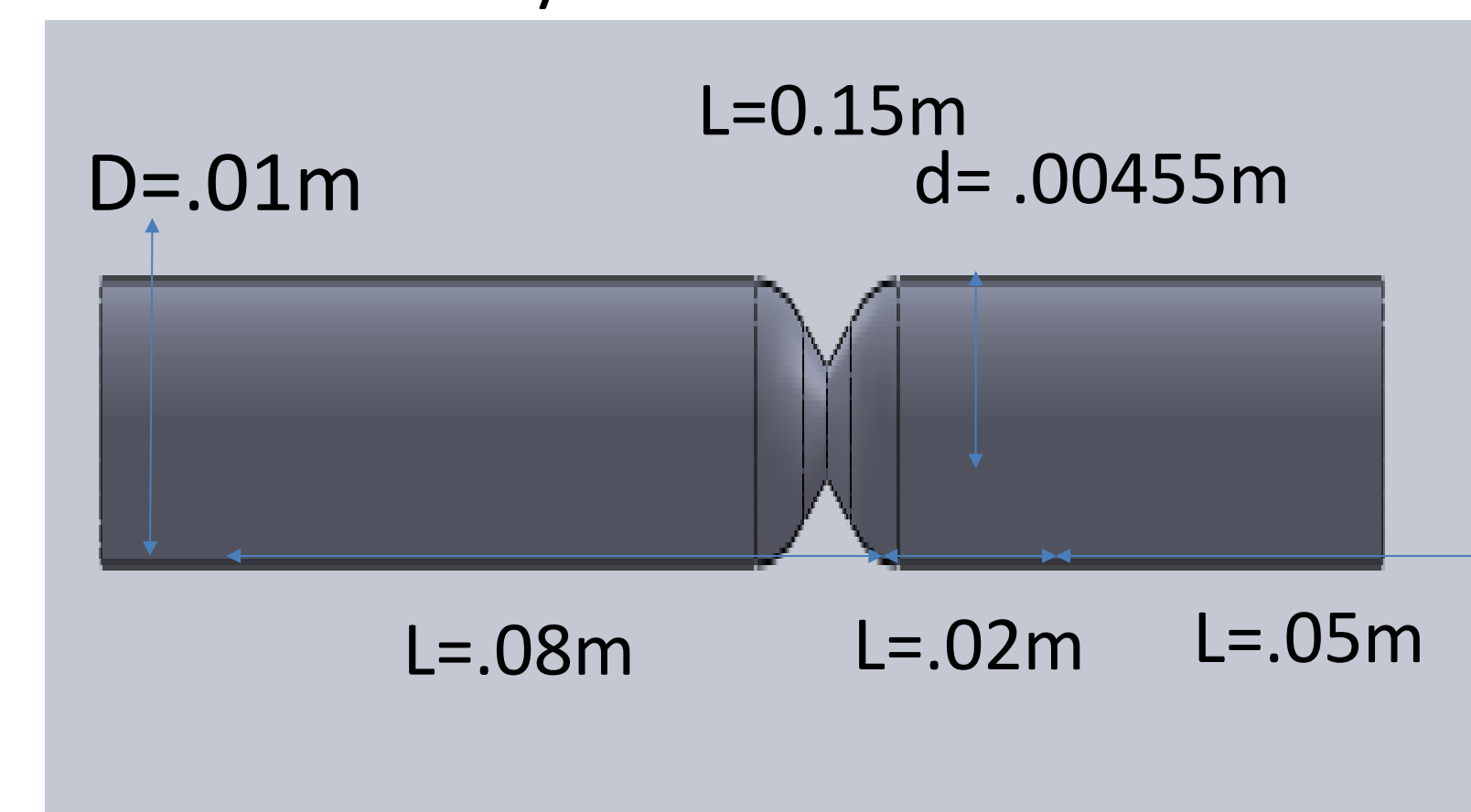


Models

Normal Artery



Diseased Artery



Results

AS can be seen from the graphs in the diseased artery the pressure at the midsection is lower than the non-diseased artery while the velocity is higher

References

- [1] Mehdi Jahangiri , Mohsen Saghafian, Mahmood Reza Sadeghi. "Numerical simulation of hemodynamic parameters of turbulent and pulsatile blood flow in flexible artery with single and double stenosis." Journal of Mechanical Science and Technology.09 August 2015. Web. <http://link.springer.com/article/10.1007/s12206-015-0752-3>
- [2] Bhaskaran,Rajesh."FLUENT-Bifurcating Artery".Confluence.Cornell,5 January 2015. 10 January 2016