



INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X

Impact factor: 6.078

(Volume 6, Issue 4)

Available online at: www.ijariit.com

Setup for simulation and analysis of Pulsatile Flow

Nagasai Panuganti

nagasaipanuganti@gmail.com

Vasavi College of Engineering, Hyderabad, Telangana

ABSTRACT

In Fluid Dynamics, we can classify the flows in different ways, but we have two flows that remain pristine, these are Pulsatile flow and Continous Flow. Pulsatile flow is the one in which the flow is represented by cycles, whereas the Continous Flow is represented by a straight line. In this paper, Pulsatile flow is feed to the Inlet boundary as a varying sinusoidal wave with a constant amplitude and angular velocity. The feasibility of the model can be increased as we have can use a cosine wave instead of a sinusoidal wave according to the requirement. This setup gives a detailed explanation as to how to set up for the simulation and analysis of the Pulsatile flow. The results of the following setup can be obtained by using the Post-Processing in Ansys Fluent.

Keywords— Pulsatile Flow, Continous Flow, Amplitude, Angular Velocity, Sinusoidal wave, Cosine wave, Setup, Ansys Fluent

1. INTRODUCTION

In Fluid dynamic, a flow with periodic variations is known as Pulsatile Flow, or as Womersley flow. The flow profiles are first derived by John R. Womersley in his work with blood flow in arteries. The cardiovascular system of chordate animals, patients suffering from Artery Stenosis, Chronic and Intrapartum Hypoxia are a very good example where a pulsatile flow is found, but a pulsatile flow is also observed in hydraulic systems and engines, as a result of rotating mechanisms, which pumping the fluid. The flow characteristics of pulsatile flow depend on the Womersley Number.

This paper will illustrate the Modelling, Setup, and Solution of the two-dimensional laminar Fluid Flow in a partially filled rectangular tank using ANSYS Fluent.

This Paper gives details of how to:

- Set up and solve a transient surface problem using the segregated CFD Solver.
- Solution setup for the transient model.
- Boundary conditions for the Pulsatile flow.
- Define a User-Defined Function.
- Projecting Animated contours.
- Velocity profile and Pressure-Gradient Vs Time graphs.
- Comparing the obtained Fluent Results with Analytical solution of the arbitrary flow.

2. PROBLEM DESCRIPTION

Consider incompressible, laminar, fully-developed transient flow with constant thermophysical properties in an infinitely long circular pipe of radius R with Zero swirl and Reynolds number as 1118. Consider the density to be 1.225 kg/m³ and viscosity 1.7899*10⁻⁰⁵ kg/m-s, taking these values to be constant throughout the flow of the pipe. Inlet velocity is in the form an arbitrary flow i.e a sinusoidal flow.

Now, let us use a simple formula for obtaining the dimensions of the rectangular laminar geometry.

$$Re = \frac{2UR}{V}$$

Where,

Re = Reynold's Number

U= velocity of the arbitrary flow

R= Radius of the Pipe

V=kinematic Viscosity

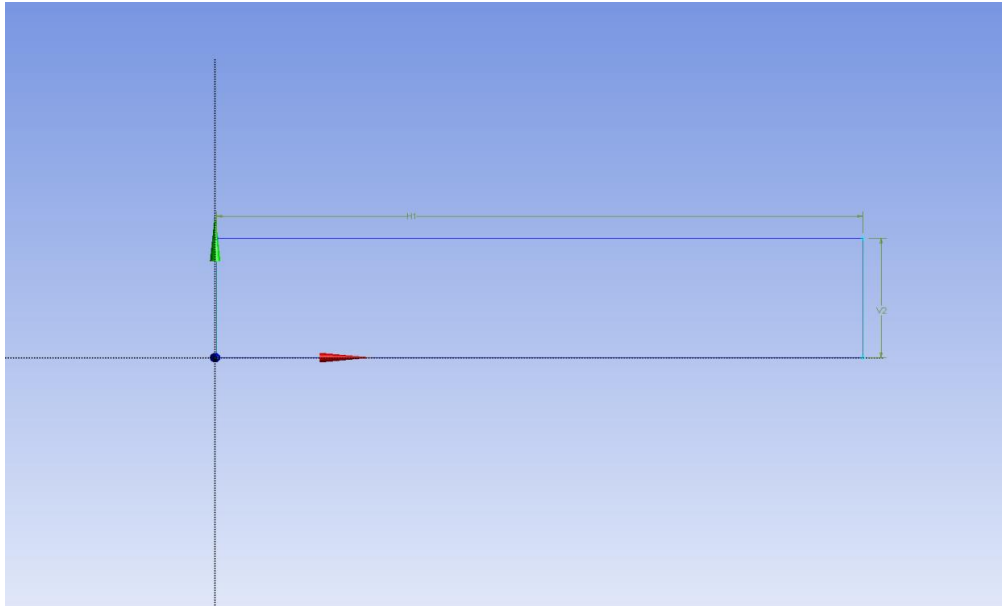


Fig. 1: Rectangular Laminar body Sketch

Length = 100mm
Radius = 10mm
Density = 1.225 kg/m³
Kinematic viscosity = 1.7899*10⁻⁰⁵ kg/m-s

3. STEPS FOR SETUP

3.1 Sketch

- (a) Sketch the 2D representation of the rectangular tank with the given dimensions in Space Claim / Design Modeller.
- (b) Create Surface from the Sketch.

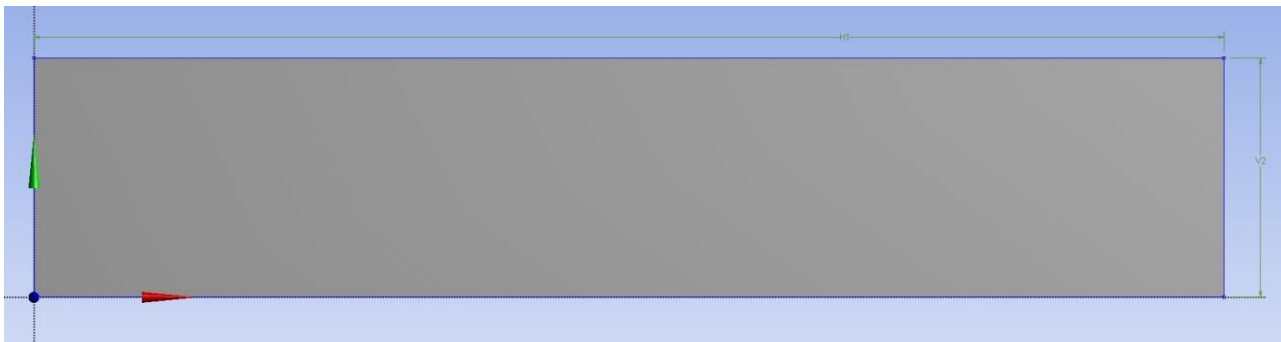


Fig. 2: Surface from Sketch

3.2. Meshing

- (a) Click on Mesh option in the model option in the Project drop-down.
- (b) Select the Face option and select the geometry.
- (c) Now, update the Geometry to get the following results.

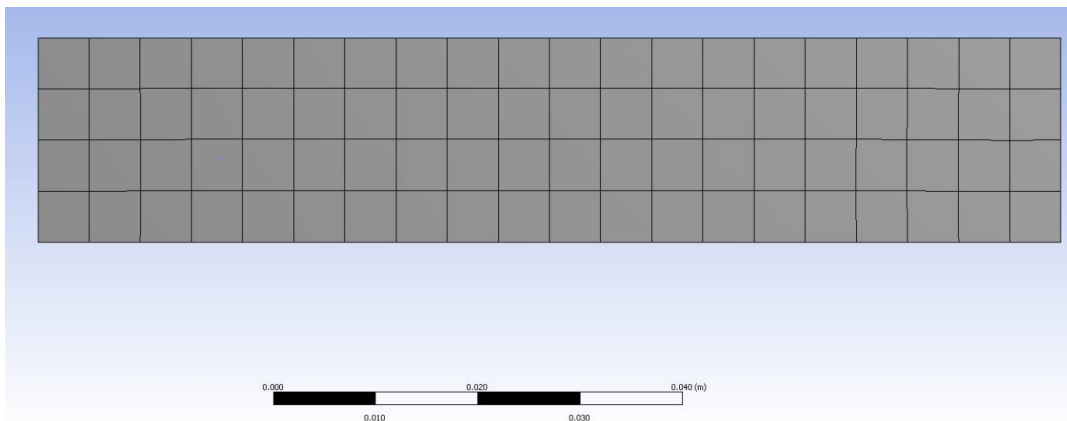


Fig. 3: Default Mesh

- (a) Select the edges option and then select the edges in the geometry to perform edge meshing.
- (b) Select the type of definition you would like to give to your mesh and then specify that particular value.
- (c) Under the details of the Mesh, press Sizing Option and change the relevant settings into fine, coarse, or medium concerning the problem.
- (d) For a better mesh, right-click on the mesh option and select Mapped Face Meshing.
- (e) Update your Mesh.

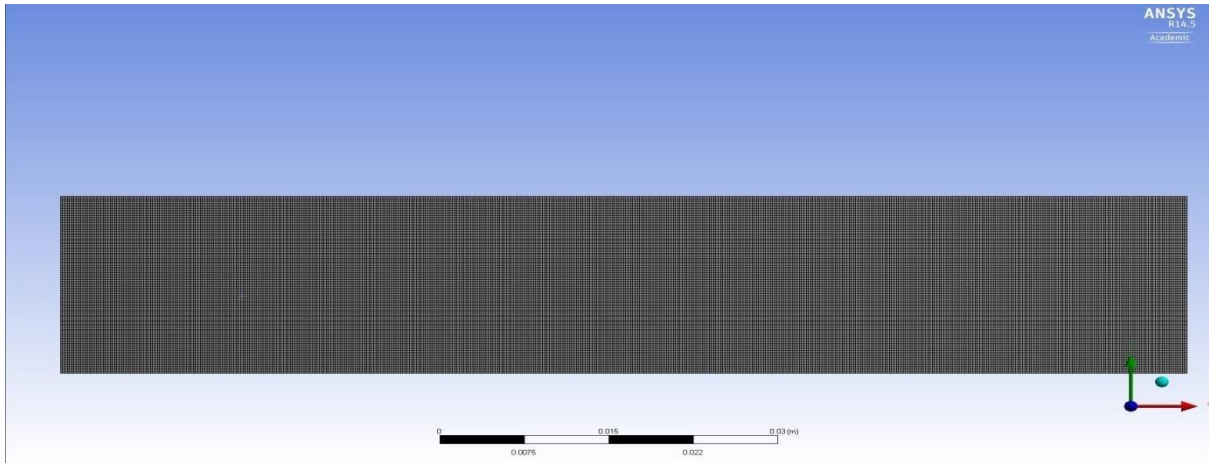


Fig 4: Fine Mesh Display

Statistics of the Mesh:

Nodes: 50601

Elements: 50000

3.3. Named Selections

- (a) Naming the Surfaces of geometry is required for defining the Boundary Conditions
- (b) Using Create Named Selection Options, Inlet, outlet, top, and bottom of the pipe are named.



3.4. General

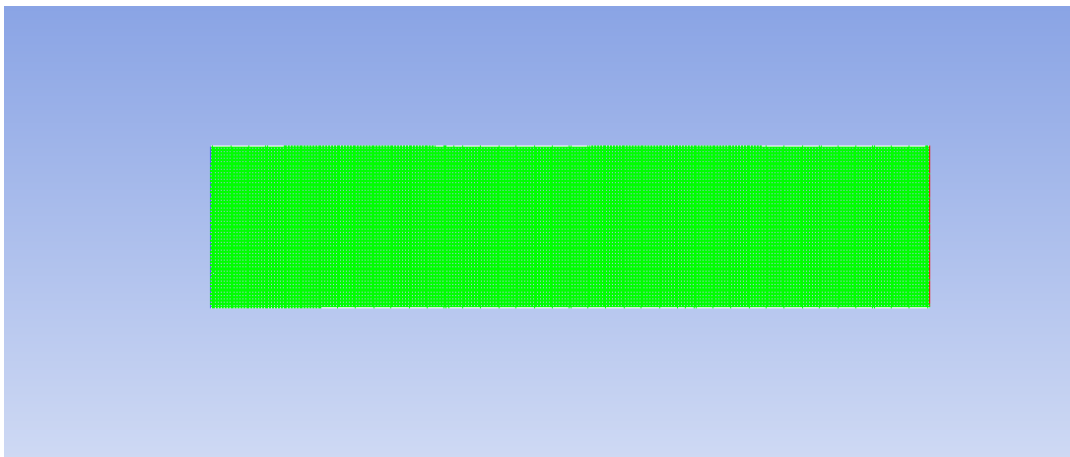


Fig 5: Imported Mesh in Fluent

- (a) Once the Mesh is imported into the Fluent, Check the Mesh by clicking on Check Mesh Option from Task Page.
- (b) Under General, Select Transient Condition for Unsteady Flow conditions for the given problem.
- (c) Click on Report Quality to check the Minimum Orthogonal Quality and Maximum Aspect Ratio.
- (d) Check on pressure-based in the Type drop-down.
- (e) Check on Planar In 2D-space drop-down.

3.5. Model

- (a) Double click viscous option and select Laminar.
- (b) Check Multiphase, Energy, Radiation, Heat Exchanger, Species, Discrete Phase, Solidification & Melting, and Acoustics as Off.
- (c) We do this to check only for the laminar case.
- (d) 3.5: Materials
- (e) Under Fluids option, consider air or specify to create your fluid with particle density and kinematic viscosity.
- (f) For Air, we have a pre-specified value for the Density and kinematic viscosity.

3.6. User-Defined Function

- (a) For a Pulsatile flow, the flow is Arbitrary. So, to give a sinusoidal wave function as an input to the rectangular laminar channel, we need to give a function.
- (b) Open a note pad and copy the code given below:

3.6.1. Code

```
#include"udf.h"
DEFINE_PROFILE(axial_velocity,thread, position)
{
face_t f;
real t= CURRENT_TIME;
begin_f_loop(f, thread)
{
F_PROFILE(f,thread,position)= 1+(0.1*sin(0.2*t));
}
end_f_loop(f,thread)
}
```

- (a) We need to execute the above code which is in the form of $1+A \sin(\omega t)$, where " ω " is the angular velocity, and " t " is time period one wave function.

(Note: The translation motion of the tank is given in the function as $f(x) = A \sin \omega t$. Since the equation cannot be assigned from the parameters in the Fluent Database, an User Defined Function is interpreted to the ANSYS Fluent. The UDF can be written in any text application and saved in the .c format for the ANSYS Fluent to get interpreted with. It is created concerning unsteady velocity, thread, and position. Once the UDF is compiled, it is assigned to the Boundary Conditions.)



- (b) As mentioned above, we need to follow the steps to interpret the code.
- (c) After the above process, and Interpreted UDFs window pops up, there browse the ".txt" file in which you have saved your code.
- (d) Click on the Interpret option for the file to get interpreted.

3.7. Boundary Conditions

- (a) Click on the "Boundary Conditions" option from the solution setup menu.
- (b) Click on the inlet option in the zone and then select "Velocity-inlet" as the type from the settings.
- (c) Now click on the edit option.
- (d) Change the "Velocity Specification Method" to "Components", "Reference Frame" to "Absolute".\
- (e) Change the "Initial Gauge Pressure" to "udf axial_velocity"
- (f) Change the "X-velocity" to "udf axial_velocity"
- (g) Let the Y-Velocity be constant.
- (h) Click on the outlet option from the zonal drop-down and select the Pressure-outlet type.
- (i) Click on the edit option and change the necessary parameters.
- (j) Click on the wall option in the zonal drop-down and then click on the edit option.
- (k) Wall window appears on the screen, here in this condition we are giving the "Stationary wall" and "No-Slip" shear conditions being the ideal cases to solve the problem, but if one would like to see the variation with respect to Knudsen number, we have to specify the shear by differentiating the analytical solution of the arbitrary flow.
- (l) Click on "OK".

3.8. Solution Methods

- (a) The panel of Solution Methods is expanded by clicking on it.
- (b) Set up the Scheme of Pressure-Velocity Coupling as SIMPLE.
- (c) In the Spatial Discretization window, we have to set "Gradient" to "Least Squares Cell-Based", "Pressure" Standard" and the "Momentum" to "first order Upwind", but later change to "second-order Upwind".
- (d) For initial value to converge we give first-order upwind but later we give second-order upwind, to get better results.
- (e) Click on the "Forst order Implicit" in "Transient Formulation"

3.9. Solution Controls

- (a) Click on the solution controls option from the solution, under "Under-Relaxation Factors" in the drop-down.
- (b) In the drop-down specify the values of "Pressure" as 0.3, "Density" as 1, "Body Forces" as 1, and "Momentum" as 0.7.
- (c) Check for the "equations", "Limits", "Advanced" options as per your requirement.
- (d) If you the values to be in default, click on the default option.

3.10. Residual Monitors

- (a) Click on the "Monitors" option from the "Solution" tab, Uner "Residuals, Static and Force Monitors", click on the Residuals.
- (b) Double click on the option then change the Continuity, X-velocity, and Y-velocity as 10^{-07}

(NOTE: If your residuals stay constant after a high number of iteration, maybe you have an initial solution close to the converged one. In that case, residuals can't reduce as much as you're asking. You can check this changing your initial solution.)

3.11. Solution Initialization

- Click on the "Solution Initialization" and then from the "Initialization Methods" select "Hybrid Initialisation" or "Standard Initialization".
- In the Hybrid Initialization, we just need to click on the initialize option.
- In the Standard Initialization, we give "compute from" as "inlet", "Reference Frame" as "Relative to Cell Zone" or "Absolute".
- And we need to specify the initial values of Gauge pressure, X-velocity, and Y-Velocity.
- Then click on initialize option to initialize the values at the inlet.

3.12. Calculation Activities

- Click on the "Calculation activities" in the solution. Then give "1" in the "Autosave Every (Time Step)" option, then click on edit option and then "Append file name with" as "time-step".
- Under "Solution Animations" click on the "create/edit" option, then the Animation sequence is changed to one.
- Then the "Active name" is given, set "Every" as "1", "when" as "Iteration" and click on define option, then Animation Sequence Window appears, there select the Contours from the Display type and click on edit option.
- Now, select the contours of "velocity" and "velocity magnitude".
- Check the options like "Filled", "Node Values", "Global Range", "Auto Range".
- Click on the compute option.

3.13. Run Calculations

- Click on the "Run Calculations" option and then click on the check case option.
- Set the 'Time-Stepping Method' as "Fixed". We need to calculate the time step size, Number of time steps, and Maximum.Iterations/Timestep.
- $t = \frac{2\pi}{w}$; "t" is the time period of the wave, "w" is the angular velocity.
- $\Delta t = \frac{t}{10}$; "Δt" is the time step size.
- $T = n * t$; T = total number of waves, n = number of waves.
- According to the requirement the Maximum.Iterations/Timestep is fixed.

3.14. Post-Processing

3.14.1. Graph

- Open CFD-post
- Click on the "location" tab in the toolbar.
- Click on the "Line" option, Insert line window is opened, enter the desired name.
- Now in the domains, select "All Domains" and set method as Two points.
- Enter the coordinates of the points and enter the number of samples required.
- Click on "Apply", necessary changes are made.
- Click on the graph option and give the Title of the Graph.
- In the General setting, set "Type" to "XY". In the data, series give the location as the line name.
- Give the X and Y Axes Variables.
- Click on the "Apply" option.

3.14.2. Contours

- Open CFD-post
- Click on the "location" tab in the toolbar.
- Click on the "Plane" option, "Insert plane window" is opened, enter the desired name.
- Give the domain as "All Domains" and locations as "Plane 1"
- Select the variable for which you want to draw the contour.
- Click on the "Apply" option.

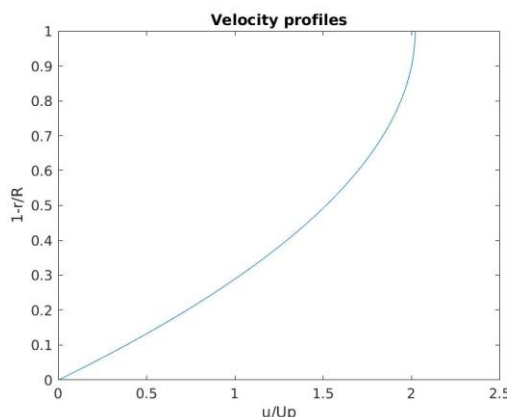


Fig. 6: Velocity Profile

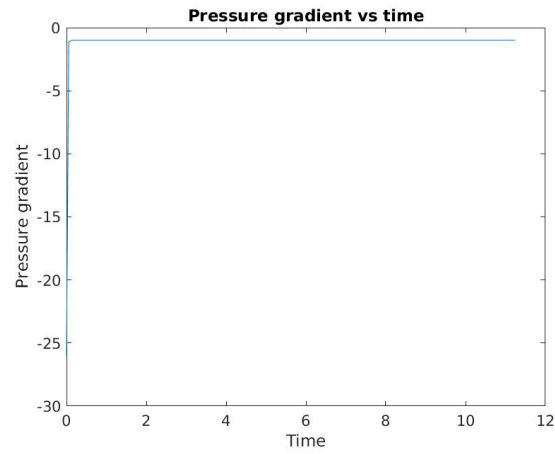


Fig. 7: Pressure.Gradient vs Time

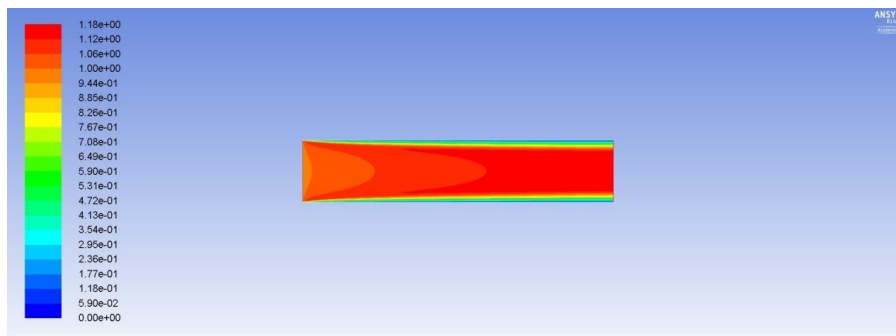


Fig. 8: Velocity contour

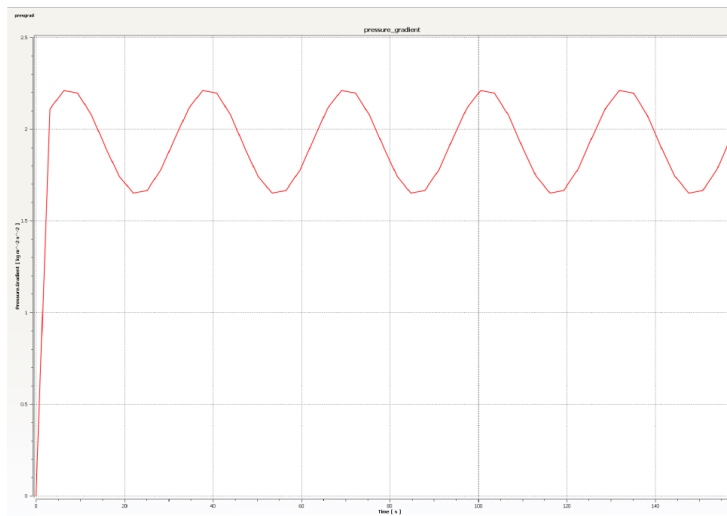


Fig. 9: Pressure Gradient Graph

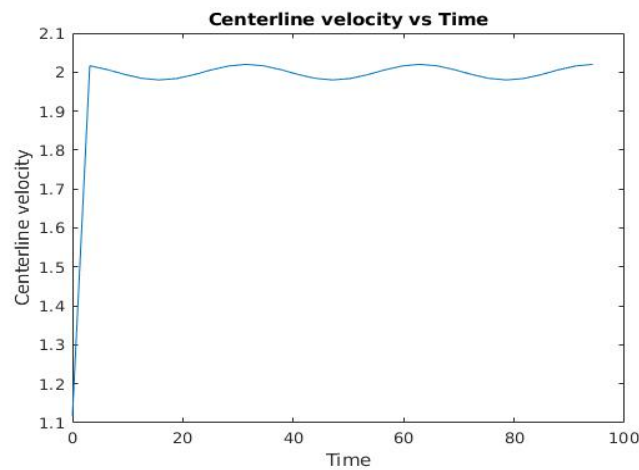


Fig. 10: CenterlineVelocity vs Time

4. CONCLUSION

In the above set up we use a rectangular lamina for the flow analysis. Commercially used ANSYS Fluent has been used to create and develop the above setup. Firstly, we create a rectangular laminar and the apply mesh to the lamina. We have considered the input at the boundary to be a sinusoidal wave with a specific amplitude and angular velocity. Later, we simulate the flow to find out various factors of the flow, such as velocity, pressure, etc. After conducting the analysis, Graphics, contours, and streamlines can be produced using the Post-Processing. For accurate results, the values can be exported and can be feed to other software for accurate results. The above-mentioned procedure for the simulation and analysis of Pulsatile flow opens a lot of scope in the field of hemodynamic.

5. REFERENCES

- [1] Tomas Prior, BSc; Edward Mullins, BSc; Phillip Bennett, Ph.D.; Sailesh Kumar, Ph.D.: Umbilical venous flow rate in term fetuses: can variations in flow predict intrapartum compromise?
- [2] Bchara Sidnawi, Sridhar Santhanam, Qianhong Wu: Analytical and Numerical Study of a Pulsatile Flow in a Porous Tube.
- [3] Morton H. Friedman, Vivian O'Brien, and Louis W. Ehrlich: Calculations of Pulsatile Flow through a Branch Implications For the Hemodynamics of Atherogenesis.
- [4] Nakai Y, Miyazaki Y, Matsuoka Y, Matsumoto M, Imanaka M, Ogita S: Pulsatile umbilical venous flow and its clinical significance.
- [5] Ghosh GS, Fu J, Olofsson P, Gudmundsson S: Pulsations in the umbilical vein during labor are associated with increased risk of operative delivery for fetal distress.
- [6] Prior T, Mullins E, Bennett P, Kumar S: Umbilical venous flow rate in term fetuses: can variations in flow predict intrapartum compromise
- [7] Successfully treated bilateral renal artery stenosis in a patient with coronary artery disease, Yasemin Gunduz, Ramazan Akdemir, Salih Sahinkus, and Mehmet Bulent Vatan.