American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-5, Issue-12, pp-155-166
www.ajer.org
Research Paper Open Access

The Fluid Structure Interaction of a Peristaltic Pump: Basics and Analysis

MdFayaz Ahamed[1],Sharmin Atique[2], MdAbulKhair Munshi[3], Tuomas Koiranen[4]

[1]*Department of Computational Engineering, Lappeenranta University of Technology, Finland*
[2]*Department of Basic Science, National Institute of Textile Engineering and Research, Bangladesh*
[3]*Department of Mathematics, National University, Bangladesh*
[4]*Department of Chemical Engineering, Lappeenranta University of Technology, Finland*

ABSTRACT:*The research work models the squeezing of the hose and computes the fluid motion of a peristaltic pump. The simulations have been conducted by using COMSOL Multiphysics FSI module.The simulations are performed for three different cases of the structure with different material properties and boundary loads. The numerical solutions are obtained by Finite Element Method solving the coupled partial differential equations of Navier-Stoke equations, stress-strain equations and Arbitrary Lagrangian-Eulerian method. The model captures total displacement of the hose or tube, and velocity magnitude of the fluid motion. A clear understanding and review of many mathematical and physical concepts are discussed with their applications in real field. In order to solve the problems and work around the resource constraints, a thorough understanding of mass balance and momentum equations, Finite Element method, Arbitrary Lagrangian-Eulerian method, two-way coupling method, and basic understanding to setup simulation for a fluid-structure interaction problem is also discussed.*

Keywords- *Fluid Structure Interaction (FSI),Finite Element Method(FEM), Arbitrary Lagrangian-Eulerian method (ALE), COMSOL Multiphysics,and two-way coupling.*

I. INTRODUCTION

This research work has been carried out to investigate the Fluid-Structure Interaction analysis of a peristaltic hose pump hose by using COMSOL Multiphysics. A peristaltic pump is a type of positive displacement pump that transports a variety of fluids by alternating compression and relaxation of the hose or tube drawing content in and propelling product away from the pump. Peristaltic pumps are useful for pumping abrasive, corrosive or delicate fluids that may cause damage or contaminate rotors or gears [1].The main advantage of peristaltic pumps is that no seals, valves, rotors or other internal parts ever touch the fluid. They are widely used for the fluid transport in chemical processing, waste and water treatment, petrochemical, mining and pharmaceutical, biomedical, and food processing industries.

Fluid-Structure Interaction (FSI) is a multi-physics phenomenon and defined well by Zienkiewicz and Taylor [2]. It occurs in a system where a solid structure may deform due to fluid flow surroundings or inside it, and vice versa. This deformation changes the boundary condition of a fluid system. This can also happen the other way around where the structure changes the fluid flow properties. That is the deformation of fluid flow and structure change has a nonnegligible influence on each other and this kind of multiphysics problem can be described by the relations of continuum mechanics.Fluid–structure interactions can be stable or oscillatory. In oscillatory interactions, the strain induced in the solid structure causes it to move when the stress is reduced, and the structure returns to its former state only for the process to repeat. A flying aircraft or a running car is an example of FSI. It appears in many natural phenomena and mechanical systems. FSI plays a very important role in the design and analysis of many engineering systems.

A peristaltic pump design probably largely depends on FSI. The factors that may affect the pump performance are pump speed, inner tube diameter, tube material, degree of tube occlusion, and back pressure. Flow rate is normally increased by the increase of inner tube diameter, pump speed, and tube occlusion. On the other hand, enhanced fluid viscosity and density decreases the flow rate. Applied load by rotors may lead to tube failure which is one of the most common drawbacks of a peristaltic pump performance [1].

FSI problems and multiphysics problems are often very complex and governing equations expressing the physical phenomenon of FSI problems in general are nonlinear partial differential equations. It is very difficult then to have analytical solutions of such problems. Therefore, they have to be solved by means of numerical solution or experiments. Here, the modelling and simulations have been carried out by using COMSOL Multiphysics FSI module.

The main objective of this work is to study the squeezing of the tube and compute the fluid flow motion. Also, this research work investigates the limitations of a two-way coupled FSI analysis of peristaltic pump. The simulations of structural mechanics and fluid dynamics are analysed separately to have a clear understanding of the result. This model is a validation and extension of the former work of COMSOL Multiphysics model of peristaltic pump[3].

II. BASIC CONCEPTS AND THEORY

2.1 Background

This study is an extension of COMSOL Multiphysics model [3] of a peristaltic pump solved with COMSOL multiphysics 4.2. This peristaltic pump model is an amalgamation of structural mechanics (to model total displacement of the hose) and fluid dynamics (to compute fluid velocity field). Therefore, it is an example of a fluid-structure interaction problem. Also, some other analysis characteristics are discussed in this model.

In general, Fluid-Structure Interaction problems are multiphysics problems which are very difficult to solve by analytical approach. Therefore, they are to be analysed either by using numerical simulations or experiments. Advanced discretization methods and availability of modern softwares in fields of computational fluid dynamics (CFD) and computational structural dynamics (CSD) have made this numerical solution possible. COMSOL multiphysics is one of the user friendly softwares for a multi-physics problem. Some other popular softwares in this field are ANSYS, Abaqus, ADINA, and so on. There are mainly two approaches for solving FSI problems using these softwares namely monolithic approach, and the partitioned approach.

In monolithic approach, the governing equations of fluid flow and displacement of the structure are solved simultaneously by a single solver.In other words, monolithic solution method directly runs fluid and structure equations by a unified algorithm. The interfacial conditions of fluid and structure are implicit in this solution approach. This process is better and more accurate for a multi-physics problem, but it may demand more resources and computational memory to develop and maintain such a specialized algorithm [4]. The mesh is connected here by non-conforming mesh methods. Here, all the numerical solutions are obtained by monolithic approach.

Partitioned approach is another way for solving FSI problems. Details about portioned method can be found in the literature [5-12]. In this method, sub programs are solved individually so that the structural solution does not change at the same time when fluid flow solution is computed. The governing equations of fluid flow and structural displacement are solved separately according to numerical solver and mesh discretization.Software modularity is preserved in partitioned approach as an existing flow and structural solver are coupled. Here, information is interchanged at the interface according to the coupling technique applied [13]. The interfacial conditions of fluid and structure are explicit in partitioned approach. A motivation of this approach is to combine fluidic and structural algorithm and to decrease the computational time. Hence, a partitioned method can be used to compute FSI problems with sophisticated fluid and structural physics [4]. A conforming mesh method is mainly used here to connect the mesh.

2.2 Coupling

A coupling is said to be one-way if the motion of a fluid flow is affected by structural deformation and vice versa. Ship propeller is an example of this kind. In this coupling method, fluid flow is calculated first up to the desired convergence. After that resulted fluid flow calculation is interpolated to the structural model at the interface. Then, the structural model calculation is iterated until the desired convergence is achieved.

In a two-way coupling, fluid flow is affected by structural deformation and at the same time structural deformation is affected by fluid flow. Wind power point is an example of a two-way coupling. Here, converged solutions of the fluid flow influence the solid body deformation when the first time step runs. Then the result of fluid flow calculation is interpolated to the structural mesh at the interface as one-way coupling and the result of the structural solver is obtained from the fluid flow solution considering it as a boundary condition. Hence, the mesh of the structure is displaced and the displacement values are interpolated to the fluid flow mesh [14]. The process is iterated until the desired accuracy is achieved.

2.3 Governing equations

The governing equations of fluid flow are the mathematical representation of the conservation law of physics [15]. In other words, governing equations describe physical behavior of any fluid flow problem. The entire fluid dynamics problem is based on three basic physical principles which are given below.

1. The mass of a fluid is conserved.
2. The rate of change of momentum is equal to the sum of the forces on a fluid particle, that is F=ma (Newton's second law).
3. The energy of a fluid is conserved which means the rate of change of energy equals the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics).

2.3.1 Mass conservation law and continuity equation. The mass conservation principle states that mass of an object never change which implies that mass can neither be created nor destroyed, although it can be rearranged. The equation of continuity is obtained from the mass conservation law. The following differential equation is the mass balance equation for an unsteady and compressible fluid.

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \cdots\cdots\cdots\cdots\cdots\cdots 2.1$$

In more compact vector notation the above equation can be written as follows

$$\frac{\partial \rho}{\partial t}+\mathrm{div}(\boldsymbol{\rho}\mathbf{u})=0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.2$$

Where ρ= the density of the fluid [kg/m^3]

u =the velocity of the fluid [m/s]

The first term of the above equation is the rate of change of density which is mass per unit volume. The second term represents the net flow of mass out of the element through its boundaries [15].

For an incompressible fluid like water the density ρ is constant. Hence, the first term of the equation 2.2 becomes zero and it is only left with second term. The equation is then reduced to

$$\mathrm{div}(\mathbf{u})=0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.3$$

Or in detailed notation,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.4$$

2.3.2 Momentum equation. According to the Newton's second law of motion, the rate of change of momentum is equal to the sum of the forces on a fluid particle.There are two types of body forces acting on a fluid particle namely surface and body forces [15].

As conservation of momentum for a compressible fluid flow was derived separately by Navier and Stokes, therefore this equation is usually called Navier-Stoke's equation [16]. The components of Navier-stokes equation in the three dimensions for an incompressible fluid flow can be expressed by the following partial differential equations.

For x component,

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\ F_x \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.5$$

For y component,

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+F_y \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.6$$

For z component,

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\ F_z \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.7$$

Where, p= static pressure

ρ =fluid density

μ = dynamic viscosity of the fluid

F_x = external forces acting along fluid element in x-axis

F_y = external forces acting along fluid element in y-axis

F_z = external forces acting along fluid element in z-axis

2.4 Finite element method

Finite element method (FEM) was first introduced by R. Courant in 1943. He used Ritz method of numerical analysis and minimization of variational calculus to get approximate solution to a vibrating structure. The first book on the FEM by Chung and Zienkiewicz was published in 1967.From the 1950's to the 1970's, it was developed by engineers and mathematicians into a discretization method for solving partial differential equations [17]. Most of the commercial FEM softwares developed in 1970s.

The finite element method is a numerical procedure for solving equations that govern the problems found in nature. In general, the natural behavior or physics of any problem can be described by differential or integral equations. This is why the FEM is understood in mathematics as a numerical technique for solving partial differential or integral equations. Also, a space and/or time dependent analysis for one or more variables representing the physical behavior of system can be evaluated by the FEM. In terms of structural analysis, the FEM is a powerful method for computing the displacements, stress-strain behavior of any structure under certain loads [18]. Hence, the FEM is convenient to use in the area of FSI as both the fluid domain and structural domain can easily be discretized by the means of same discretization method.

The FEM technique is widely used in new industrial product design and existing product failure analysis. A company may analyze the reliability of their product by the FEM method prior going into production. Also, it allows a company to modify their product structure to meet the product failure by specifying Von Mises Stress. The technique is now being a standard tool for structural analysis with the help of increased computational memory and commercially available softwares [19].

The FEM technique uses a complex system of points which are called nodes to make a grid called mesh [20]. The mesh is made according to the size of the structure or solid and the elements of mesh can be formulated as triangles and quadrilaterals in two dimensions (2D) or tetrahedral and hexahedra in three dimensions (3D). The mesh elements hold material properties of a structure or solid and evaluate the displacement or deformation of the structure under certain loads and boundary conditions. Nodes of the mesh are assigned according to the load or stress level applied or experienced by the solid body. In general, nodes are denser in the area of large stress applied than little or no stress area. A polynomial expansion expresses the displacements of a structure within each mesh element. As the exact analytical solution of this polynomial is more complex and generally unknown, the FEM only provides only an approximation to the exact solution [18].

The FEM analysis starts with identifying “the analysis type” and “a conceptual model” of any problem to select the appropriate structural model and computational approach. The analysis type usually identified based on physical phenomenon, static or dynamic nature, material properties, and accuracy level of a problem. For instance, an analysis can be structural static, modal, transient dynamic, steady-state thermal, or transient thermal.

The next step is to select a “mathematical model” for numerical analysis of a structure which includes three basic characteristics which are “the geometric selection of the element of a structure” by means of its geometrical components (points, lines), “the mathematical expression of the basic physical laws” governing the behavior of the structure (the force-equilibrium equations and the boundary conditions) and “the specification of the material properties and the loads” acting on the structure [18].After selecting a “mathematical model”, the step continues by specifying “a proper numerical method” for the problem, for instance, The Finite Element Method, the Finite Volume Method.

2.5 Arbitrary Lagrangian-Eulerian (ALE) method

The numerical simulation of multi-physics problems in fluid dynamics and nonlinear solid mechanics often requires dealing with strong deformation of the continuum under consideration, while allowing for a clear delineation of free surfaces and fluid-fluid, solid-solid, or fluid-structure interfaces [21]. The Arbitrary Lagrangian-Eulerian (ALE) method is a combined kinematic approach of the use of the classical Lagrangian and Eulerian reference frames. It is a method of an automatic and continuous re-joining of mesh which is usually done in such a way that the mesh is displaced according to the displacement of the moving body [22].

In Lagrangian type methods, each individual node of the computational mesh follows the associated material particle motion and usually used in structural analysis. This methods are efficient and quite suitable for handling problems of small deformation and where boundary condition nonlinearities do not change with the course of deformation [23]. The main weakness of this methods is that it cannot follow large mesh distortions and element entanglements.

On the other hand, Eulerian formulation methods are normally used to remedy the large mesh distortion and element entanglements problems and mainly used in Fluid dynamics. In this methods, the computational mesh is considered to be fixed and the material particle moves with respect to the grid. In the Eulerian description large distortions in the continuum motion can be handled with relative ease, but the resolution of flow details may not be obtained [21].

It is believed, therefore, that an Arbitrary Lagrangian-Eulerian (ALE) formulation method combining the best features of both the Lagrangian and Eulerian methods is important for accurate simulation of general finite strain deformation and metal forming problems.

In general, ALE is a finite element formulation method in which the reference frame (computational mesh) is not a *priori* fixed in space, or attached to the body [23]. In other words, in an ALE formulation, the finite element mesh need not stick to the material point or be fixed in space, but may be moved arbitrarily with respect to the material body. A proper ALE formulation may be changed to a Lagrangian formulation if the arbitrary mesh motion is chosen to be the same as the material motion. On the other hand, if we choose to fix the computational mesh, an ALE formulation should reduce to Eulerian formulation.

ALE describes the motion of fluid in a moving reference frame whose velocity is almost arbitrary with the sole constraint that the velocity on the fluid–solid boundary must equal to that of the boundary. In essence, reference frame velocity is usually neither the fluid particle velocity such as in a pure Lagrangian approach nor zero in a pure Eulerian approach [24]. After introducing a reference frame which moves with some velocity, the modified Navier–Stokes equation for ALE of a viscous incompressible fluid flow can be expressed as the following way.

$$\rho\left(\frac{\partial u_i}{\partial t} + \left(u_j - \hat{u}_j\right)\frac{\partial u_i}{\partial x_j}\right) - \frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right)\right) + \frac{\partial p}{\partial x_i} = 0 \cdots\cdots\cdots\cdots 2.8$$

$$\frac{\partial u_i}{\partial x_j} = 0;\ i,j = 1,2,3, \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 2.9$$

Where u_j = the components of the fluid flow velocity

$\hat{u}_j$ = the components of the domain velocity

p= pressure

ρ =fluid density

μ = dynamic viscosity of the fluid.

III. RESEARCH METHODOLOGY

The model of a peristaltic pump here is setup in two dimensional (2D) axes symmetric wizard. The fluid domain of the hose in both cases is made up as a rectangle of height 0.1 meter (m) and width of 0.01m. Another adjacent rectangle of same height and width of .005m is considered to be the solid domain.

The model is built based on two-dimensional (2D) axisymmetric wizard of COMSOL Multiphysics. A Nylon tube of 0.1m long with inner radius of 0.01m and outer radius of 0.015m is used to define topology of the model. Here, the inner part of the tube is fluid domain and outer part is solid domain. A time and position dependent force density is applied in radial direction along outer wall of the tube. Although force density can be taken from real peristaltic pump data but for simplicity here we took them from Normal distribution. The force is applied along positive z-direction of outer surface of the tube where rotor starts squeezing the tube at t=0.1 second(s) and its full engagement occurs at t=0.5s.Similarly disengagement of the rotor starts at t=1.0s and ends at t=1.4s.The process of load distribution is controlled by smoothed unit step function (Heaviside).This is a model of full cycle of t=1.5s.The model is first built for water and then it is tested for sulfuric acid at temperature 20 degree Celsius.

The structural deformation analysis is done by using Structural Mechanics Module of COMSOL Multyphics where we assume that the material is linear, elastic and geometrically non-linear. A hyper elastic material model is also set up to evaluate the squeezing of the tube. In addition to that the axis symmetric 2D model is tested when two forces applied like a real peristaltic pump.This simulates the situation when first roller is about to disengage and the second roller is starting the pumping action.

We solved our model as a coupled problem. The reaction forces from the fluid on the tube are implicit but the forces from the tube on the fluid are explicit. Hence, one can solve first for structural mechanics and then for the fluid dynamics to reduce computational memory consumption. Here, we used two-way coupling method of simulation which leads to more computational work but more accurate result.

The material for fluid domain is water in all simulation cases. It is also tested for sulfuric acid at temperature 20 degree Celsius. The material for solid domain is nylon in all simulation cases.

3.1 Boundary Conditions

In case of structural analysis, the top and bottom ends of the tube are assumed to be constrained along both coordinate axes. The boundary condition at the tube's outer surface is described by a time and space dependent load. For fluid flow, the boundary conditions at the inlet and outlet assume that the total stress is zero, that is:

$$\mathbf{n}\cdot\left[-pI + \mu\left(\nabla u + (\nabla u)^T\right)\right] = 0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 3.3$$

The mesh is fixed and supposed not to displace in r-direction at the symmetric axis and in z-direction at the top and bottom of the tube. A non-slip FSI boundary is set up for the inner wall of the tube.

IV. RESULTS AND DISCUSSION

Total displacement and Von Mises stress is presented in structural analysis of all simulation cases. The fluid velocity magnitude and velocity field are also presented here at different time state. In addition, the figure of total pumped fluid, inflow and outflow rate, accumulated flow, total stress are analyzed in 2D axis symmetric cases .

4.1 Case 1:2D axis symmetric

The following Figures 4.1 and 4.2 show total displacement of the hose of the peristaltic pump of water and sulfuric acid inside it, respectively. Here, deformations are shown at 0.7 second (s) when rotor reaches its full force engagement. That is, when rotor applies maximum pressure at the tube or hose is 0.7s and deformation is shown at that time only.The color represents the deformation of the tube material. Total displacements of the tube are shown in the figure 4.1 and figure 4.2in case of water and sulfuric acid respectivelyunder the pressure applied by rotor on tube wall. Here the tube is rotationally symmetric with respect to z-axis. Also, the velocity magnitudes and velocity fields of water and sulfuric acid are shown in the figure 4.3 and figure 4.4 respectively.Thevelocity fields are mostly affected by tube deformation at the time t=0.7s, when the rotor is fully engaged for a while.

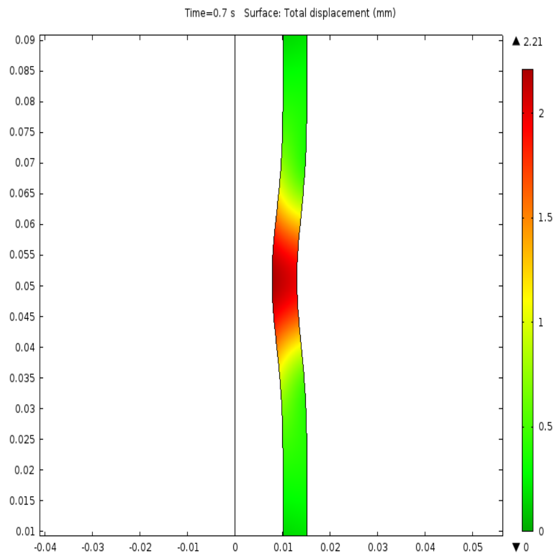

Figure4.1. Total displacement for water

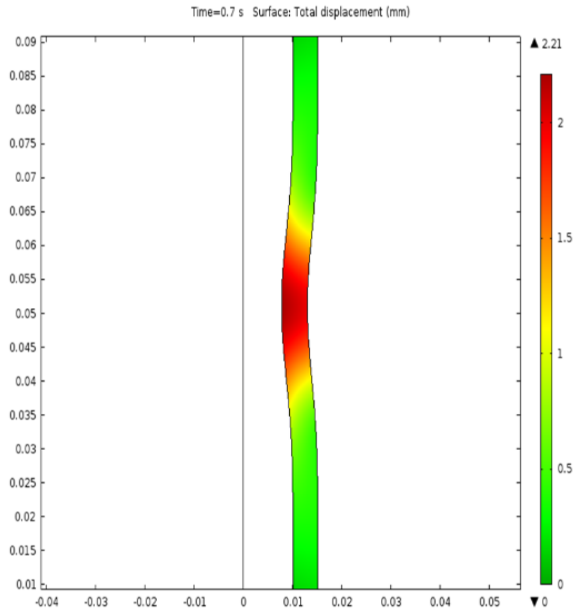

Figure 4.2. Total displacement for sulfuric acid

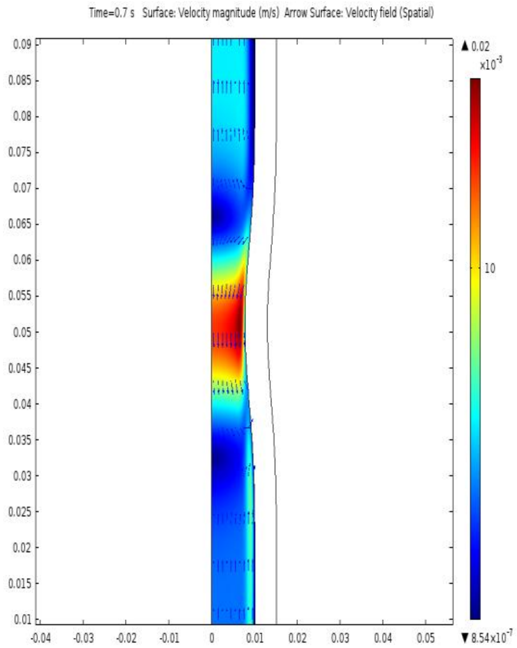

Figure 4.3. Velocity field of water

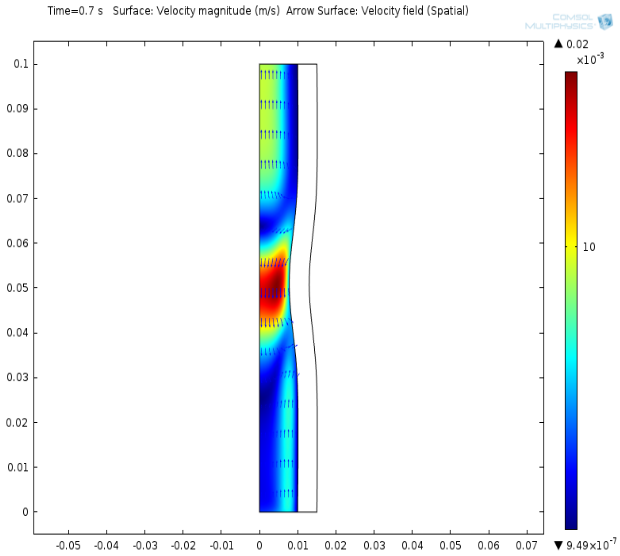

Figure 4.4. Velocity field of sulfuric acid

4.2 Case 2: Linear elastic vs hyper elastic

A linear, elastic and geometrically non-linear material is taken as the outer surface of the tube or hose (solid subdomain) and structural deformation is analyzed. The following figures 4.5 and 4.6 depict the total displacement of the tube under the pressure applied by rotor on tube wall and velocity field affected by tube deformation at t=0.7s respectively.

A hyper, elastic and geometrically non-linear material is also taken as the outer surface of the tube or hose (solid subdomain) and structural deformation is analyzed.

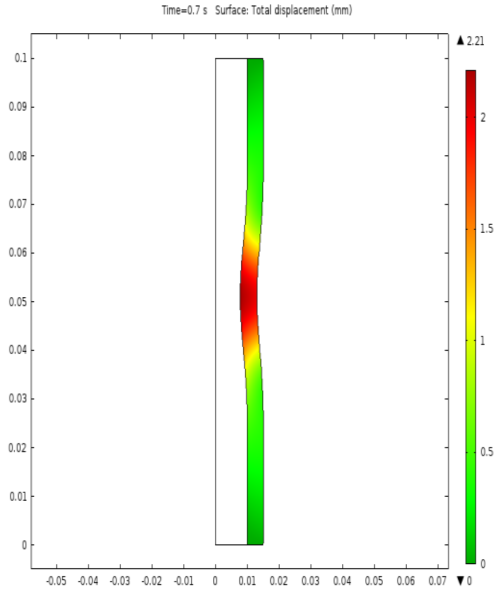

Figure 4.5. Total displacement (liner elastic material)

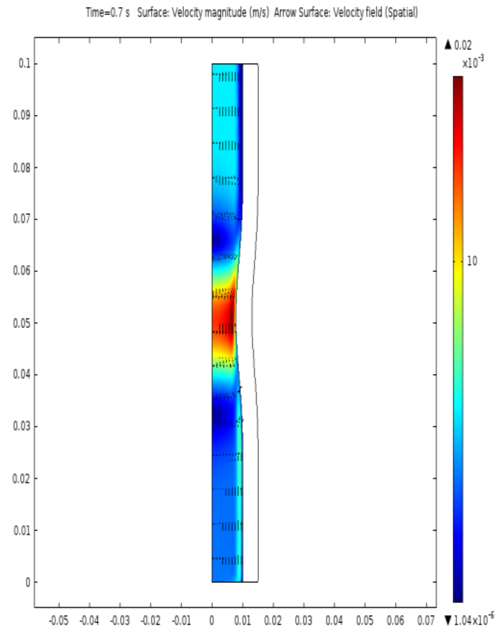

Figure 4.6. Velocity field (linear elastic material)

As above, the figure 4.7 below shows to the total displacement of the tube and figure 4.8 shows velocity magnitudes when rotor applies maximum pressure on the tube.

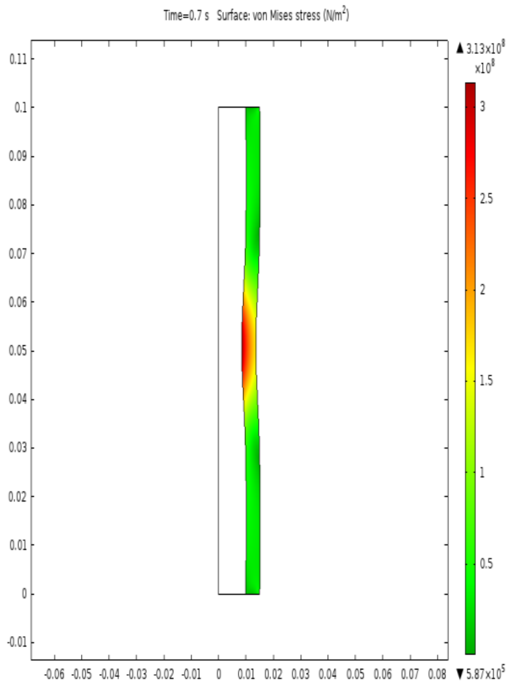

Figure 4.7. Total displacement (hyper elastic material)

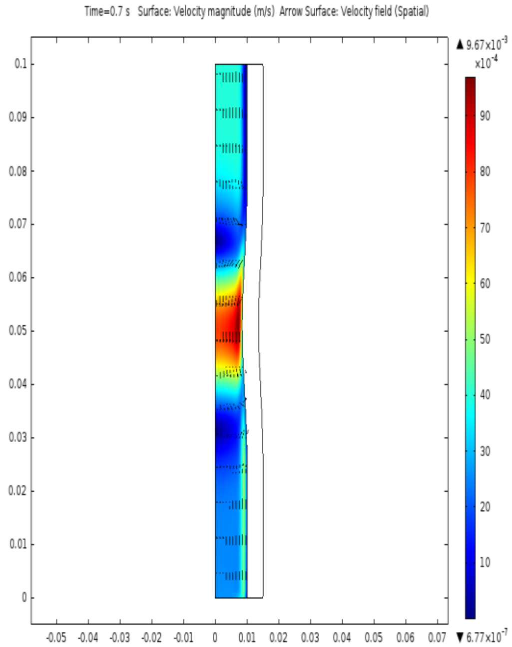

Figure 4.8. Velocity field (hyper elastic material)

4.3 Case 3: when two forces applied

A peristaltic pump usually has two rotors. The total displacement of the hose of the peristaltic pump at 0.7s when two rotors reach their full force engagement has been simulated (see figure 4.9). That is, the deformation is shown when two forces apply maximum pressure at the tube or hose like real peristaltic pump functionalities. The velocity field and velocity magnitude of the fluid domain are affected by the solid deformation (see figure 4.10).In addition,a three dimensional (3D) view of "inner" surface of the solid domain of the tube or hose is shown in the following figure 4.11 and 3D view of "outer" surface is shown in the figure 4.12 below.

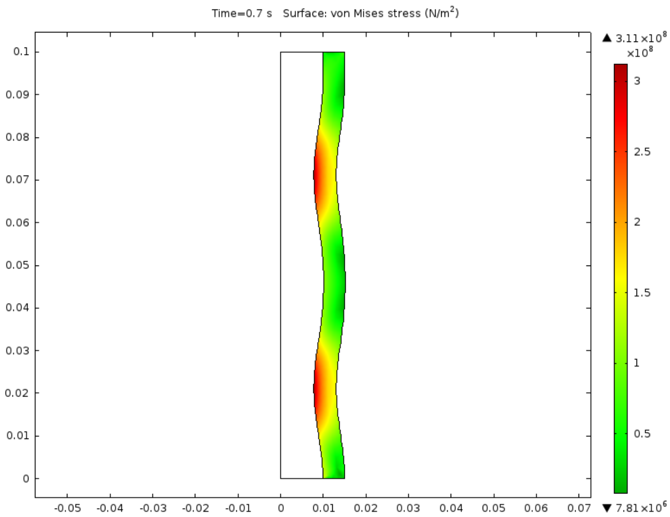

Figure 4.9. Total displacement (two forces)

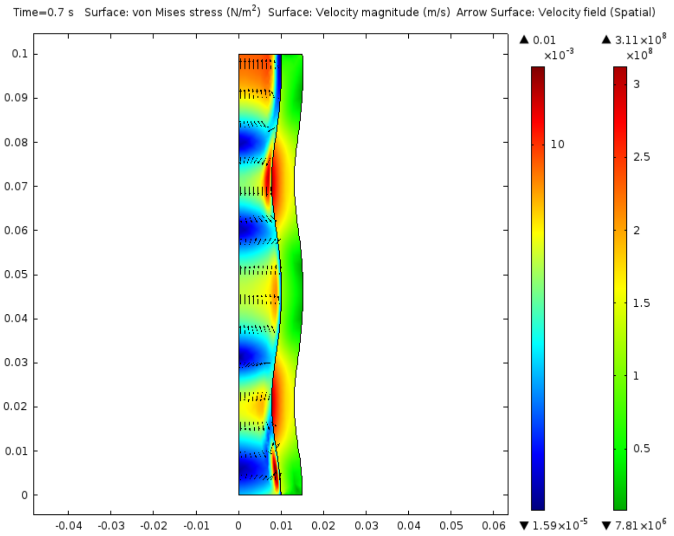

Figure 4.10. Velocity field (two forces)

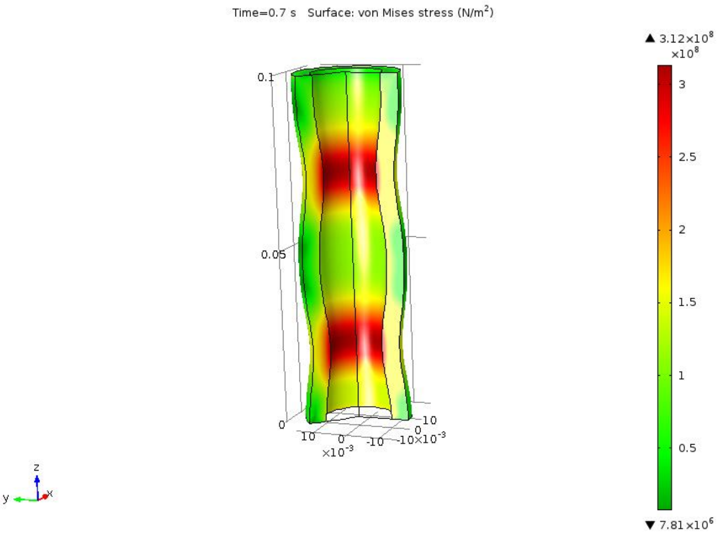

Figure 4.11. 3D view of inner surface of tube

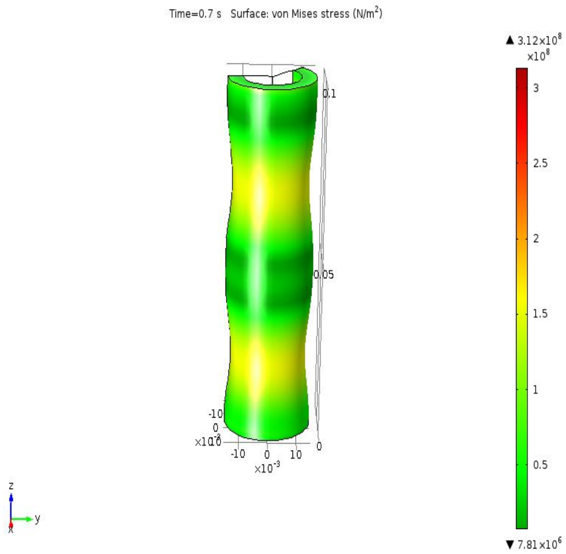

Figure 4.12. 3D view of outer surface of tube

V. CONCLUSION

A two-way coupled FSI analysis was developed to estimate stress and strains in the tube, and how it affects to the fluid flow pattern. The work is done by using COMSOL multiphysics FSI module. The structural and fluid flow responses of threedifferent cases have been plotted and observed that the results have very less variations among first two cases. The only difference is noticed in the fourth case (when two forces) and it gives a clear view of structural displacement with fluid velocity magnitude. The fluid flow rates and average pressure fluctuations are also evaluated in case 1. In addition, the model is working only with certain dynamic viscosity and density: In general, when viscosity is very less then till to certain extent of density, and not working at all when very high viscosity or density. That is, increasing fluid viscosity and density decreases flow rate and the model stops working because the force then was not enough to flow the fluid. The averaged pressure is increased with the increased viscosity and density.

Overall, this paper provides a clear understanding and review of many mathematical and physical concepts. In order to solve the problems and work around the resource constraints, a thorough understanding of mass balance and momentum equations, finite element concepts, ALE, and COMSOL Multiphysics simulation set up are understood. The concepts of FSI learnt and applied are not only limited to this model but also applicable to a wide range of other problems such as design of aircraft, sloshing in tanks and flutter of bridges.

REFERENCES

[1] Elabbasi, N., Bergstrom, J., & Brown, S., "Fluid-Structure Interaction Analysis of a Peristaltic Pump", COMSOL conference in Boston, Boston, 2011.

[2] Zienkiewicz and R. L. Taylor, “The Finite Element Method, vol. 1, Butterworth-Heinemann”, 5th edition, 2000.

[3] Peristaltic Pump-COMSOL Multiphysics, "Retrieved from COMSOL Multiphysics Modeling Software",2011.

[4] Hou, G., Wang, J., & Layton, A., "Numerical Methods for Fluid-Structure Interaction — A Review", Commun. Compute. Phys., 2012, pp.337-377.

[5] M. Sch¨afer, G. Sieber, R. Sieber, and I. Teschauer, “Coupled fluid-solid problems: examples and reliable numerical simulation,” in Proceedings of the Trend in Computational Structural Mechanics, International Center for Numerical Methods in Engineering _CIMNE_, Barcelona, Spain, May 2001.

[6] K. C. Park, C. A. Felippa, and J. A. DeRuntz, “Stabilization of staggered solution procedures for fluid-structure interaction analysis,” Computational Methods for Fluid-Structure Interaction Problems, American Society of Mechanical Engineers, New York, NY, USA, 1977, T. Belytschko and T. L. Geers, Eds., vol. 26, pp. 95–124.

[7] C. A. Felippa and K. C. Park, “Staggered transient analysis procedures for coupled mechanical systems: formulation,” Computer Methods in Applied Mechanics and Engineering, vol. 24, no. 1, 1980, pp. 61–111.

[8] S. Piperno, C. Farhat, and B. Larrouturou, “Partitioned procedures for the transient solution of coupled aeroelastic problems, part I: model problem, theory and two-dimensional application,” Computer Methods in Applied Mechanics and Engineering, vol. 124, no. 1-2, , 1995, pp. 79–112.

[9] S. Piperno, “Explicit/implicit fluid/structure staggered procedures with a structural predictor and fluid subcycling for 2D inviscid aeroelastic simulations,” International Journal for Numerical Methods inFluids, vol. 25, no. 10, 1997, pp. 1207–1226.

[10] D. P. Mok, W. A. Wall, and E. Ramm, “Accelerated iterative substructuring schemes for instationary fluid structure interaction,” Computational Fluid and Solid Mechanics, K. J. Bathe, Ed.,Elsevier, 2001, pp. 1325–1328.

[11] F. Nobile, Numerical approximation of fluid structure interaction problems with application to haemodynamics, EPFL, Switzerland, Doctoral thesis, 2001.

[12] D. Peric and W. G. Dettmer, “A computational strategy for interaction of fluid flow with spatial structures,” in Proceedings of the 5th International Conference on Computational of Shell & Spatial Structures(IASS IACM’05), 2005.

[13] Richter, T.," Numerical methods for fluid–structure interaction problems", Institute for Applied Mathematics, University of Heidelberg, Germany, 2010.

[14] Raja, R. S., Coupled fluid structure interaction analysis on a cylinder exposed to ocean wave loading, CHALMERS UNIVERSITY OF TECHNOLOGY,Göteborg, Sweden ,Master’s Thesis in Solid and Fluid Mechanics ,2012.

[15] Versteeg, H. K., &Malalasekera, W., "An introduction to Computational Fluid Dynamics-Finite Volume Method", Essex: Longman Group Limited, 1995.

[16] Iguchi, M., &Ilegbusi, O. J.,"Momentum Transfer", Basic Transport Phenomena in Materials Engineering,Springer, 2014.

[17] Chen, P.," Elementary Finite Elements", Finite Element Methods and Their Applications, Springer Berlin Heidelberg, 2005, pp. 1-85.

[18] Oñate, E.," Introduction to the Finite Element Method for Structural Analysis", Structural Analysis with the Finite Element Method, Netherlands: Springer Netherlands, 2009, pp. 1-42.

[19] Cheng, L. L. Simulation of fluid flow and fluid-structure interactions in microdevices,National University of Singapore, Singapore, Bachelor thesis, 2007.

[20] Hanif , M. A., Oad, R. K., & Patel, R., Fluid Structure Interaction Analysis of Planar Buoy,Blekinge Institute of Technology,Sweden, Master’s Degree Thesis,2011.

[21] Donea, J., Huerta, A., Ponthot, J.-P., & Rodr´ıguez-Ferran, A., "Arbitrary Lagrangian-Eulerian methods", Encyclopedia of Computational Mechanics, 2004.

[22] Kjellgren, P., &Hyvarinen, J., "An Arbitrary Lagrangian-Eulerian finite element method”, In Computational Mechanics, 1998, pp.81-90.

[23] Gadala, M. S., & Wang, J.," A Practical Procedure for Mesh Motion in Arbitrary Lagrangian-Eulerian Method", Engineering with Computers, 1998, pp.223-234.

[24] Chen, H. Y., Zhu, L., Hou, Y., Liu, Y., & Kassab, G. S.), "Fluid–Structure Interaction (FSI) Modeling in the Cardiovascular System”, Computational Cardiovascular Mechanics, Springer Science+Business Media, 2010.