Numerical Simulation of Laminar Flow Through a Pipe using COMSOL Multiphysics

[1]Hoai Nguyen, [2]Thang Hoang
[1,2]Department of Mechanical Engineering, College of Technology, University of Danang, Viet Nam
[1]Email: nhoai@dct.udn.vn
[2]Email: waveletthang@gmail.com

Abstract— In our contemporary life, it should be noted that using softwares to simulate fluid dynamic has become increasingly popular. There are many closed-source software packages used to model as well as simulate the fluid behavior, and COMSOL Multiphysics is one of the most powerful tools which many researchers prefer. In this paper, this software is therefore carried out to model laminar flow through a pipe with the purpose of understanding the dynamic structural of the laminar flow and their interactions. Also, at three-dimensional, an accurate time finite element methodology is conducted to perform the computations. The results indicate that the relationship between the aero-dynamics parameters such as: velocity, pressure depends heavily on the position and shape of pipe that flows go through. The physical mechanisms of these phenomena are analyzed, which can be a basis for predicting the structure of dynamic flows in real-world operation.

Index Terms— Mumerical Simulation, Laminar Flow, Turbulence Flow, COMSOL Multiphysics, Incompressible Flow, CFD, Finite Volume Method

◆

1 INTRODUCTION

Pipe flow under pressure is used for various different purposes. A fundamental understanding of fluid flow is completed necessary to with chemical engineering [1]. In the chemical and manufacturing industries, large flow networks are necessary to reach continuous transport of products and raw materials from various processing units. This requires a detailed understanding of fluid in pipes. Energy input to the gas or liquid is needed to make it flow through the pipe. This energy input is needed because there is frictional energy loss due to the friction between the fluid and the pipe wall and internal friction within the fluid. In pipe flow substantial energy is lost due to frictional resistances.

One of the most common problem in fluid mechanics is the estimation of this pressure loss. Determining the appropriate size pump will be easier than when pressure losses is computed [2]. Knowledge of the magnitude of frictional losses is of great importance since it decides the power requirements of the pump forcing the fluid through the pipe [3].

When a fluid flows through a pipe, the internal roughness of the pipe wall will generate local eddy currents within the fluid adding a resistance to flow of the fluid. The velocity profile in a pipe will point out that the fluid elements in the center of the pipe will move at a higher speed than those closer to the wall. Therefore, friction will turn up between layers within the fluid. This movement of fluid elements relative to each other is associated with pressure drop, called frictional losses [4]. Pipes with smooth walls such as glass, copper, brass and polyethylene have only a small effect on the frictional resistance. Pipes with less smooth walls such as concrete, cast iron and steel will create larger eddy currents which will sometimes have a significant effect on the frictional resistance. Rougher the inner wall of the pipe, more will be the pressure loss due to friction [5].

Velocity of flow will be proportional to flow rate. Smaller pipe causes a greater proportion of the liquid to be in contact with the pipe, which creates friction. Pipe size also affects velocity. Given a constant flow rate, decreasing pipe size increases the velocity, which increases friction. The friction losses are cumulative as the fluid travels through the length of pipe. The greater the distance, the greater the friction losses will be. Fluids with a high viscosity will flow more slowly and will generally not support eddy currents and therefore the internal roughness of the pipe will have no effect on the frictional resistance [6]. This condition is known as laminar flow.

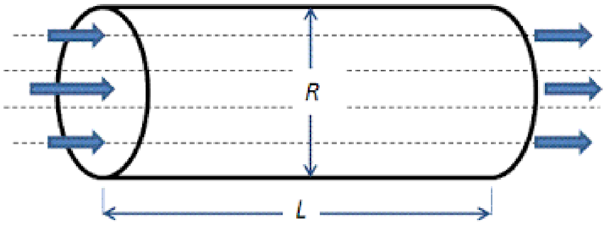


Fig. 1. Laminar flow through pipe

2 THEORETICAL BASIC LAMINAR FLOW THROUGH A PIPE

2.1. The average velocity of laminar flow through a pipe

In fully developed laminar flow, each fluid particle moves at a constant axial velocity along a streamline and the velocity profile u(r) remains unchanged in the flow direction. There is no motion in the radial direction, and thus the velocity component in the direction normal to flow is everywhere zero. There is no acceleration since the flow is steady and fully developed.

Now, consider a ring-shaped differential volume element of radius r, thickness dr, and length dx oriented coaxially with the pipe, as shown in Fig.2.

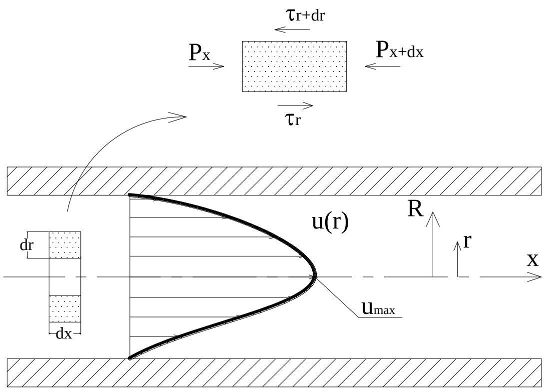


Fig.2. Free-body diagram of a ring-shaped differential fluid element of radius r, thickness dr, and length dx oriented coaxially with a horizontal pipe in fully developed laminar flow

The volume element involves only pressure and viscous effects and thus the pressure and shear forces must balance each other. The pressure force acting on a submerged plane surface is the product of the pressure at the centroid of the surface and the surface area. A force balance on the volume element in the flow direction gives:

$(2\pi r dr P)x-(2\pi dr P)x+dx+(2\pi r dx\tau)r-(2\pi dx\tau)r+dr=0$ (2-1)

which indicates that in fully developed flow in a horizontal pipe, the viscous and pressure forces balance each other. Dividing by 2πdrdx andrearranging,

$$r\frac{P_{x+dx}-P_x}{dx}+\frac{(r\tau)_{r+dr}-(r\tau)_r}{dr}=0 \qquad (2\text{-}2)$$

Taking the limit as dr,dx →0 gives

$$r\frac{dP}{dx}+\frac{d(r\tau)}{dr}=0 \qquad (2\text{-}3)$$

Substituting τ =-μ (du/dr) and taking μ= constant gives the desired equation,

$$\frac{\mu}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right)=\frac{dP}{dx} \qquad (2\text{-}4)$$

The quantity du/dr is negative in pipe flow, and the negative sign is included to obtain positive values for τ. (Or, du/dr =- du/dy since y =R-r.) The left side of Eq.2-4 is a function of r, and the right side is a function of x. The equality must hold for any value of r and x, and an equality of the form f(r) = g(x) can be satisfied only if both f(r) and g(x) are equal to the same constant. Thus we conclude that dP/dx = constant. This can be verified by writing a force balance on a volume element of radius R and thickness dx (a slice of the pipe), which gives Fig.3:

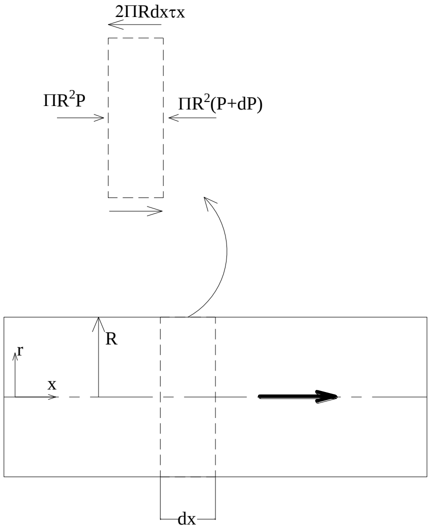


Fig.3. Free-body diagram of a fluid disk element of radius R and length dx in fully developed laminar flow in a horizontal pipe.

$$\frac{dP}{dx}=-\frac{2\tau_w}{R} \qquad (2\text{-}5)$$

Here tw is constant since the viscosity and the velocity profile are constants in the fully developed region. Therefore, dP/dx = constant. Equation 2-4 can be solved by rearranging and integrating it twice to give:

$$u(r)=\frac{1}{4\mu}\left(\frac{dP}{dx}\right)+C_1\ln r+C_2 \qquad (2\text{-}6)$$

The velocity profile u(r) is obtained by applying the boundary conditions ∂u/∂r = 0 at r = 0 (because of symmetry about the centerline) and u = 0 at r = R (the no-slip condition at the pipe surface). We get:

$$u(r)=-\frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1-\frac{r^2}{R^2}\right) \qquad (2\text{-}7)$$

Therefore, the velocity profile in fully developed laminar flow in a pipe is parabolic with a maximum at the centerline and minimum (zero) at the pipe wall. Also, the axial velocity u is positive for any r, and thus the axial pressure gradient dP/dx must be negative (i.e., pressure must decrease in the flow direction because of viscous effects)

The average velocity is determined :

$$V_{avg}=\frac{2}{R^2}\int_0^R u(r)r\,dr=\frac{-2}{R^2}\int_0^R\frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1-\frac{r^2}{R^2}\right)r\,dr$$
$$=-\frac{R^2}{8\mu}\left(\frac{dP}{dx}\right) \qquad (2\text{-}8)$$

Combining the last two equations, the velocity profile is rewritten as:

$u(r)=2V_{avg}(1-r^2/R^2)$ (2-9)

This is a convenient form for the velocity profile since V_{avg} can be determined easily from the flow rate information. The maximum velocity occurs at the centerline and is determined from: Eq.2-9 by substituting r = 0

$U_{max} = 2V_{max}$ (2-10)

Therefore, the average velocity in fully developed laminar pipe flow is onehalf of the maximum velocity.

2.2. Pressure Drop of laminar flow through a pipe

A quantity of interest in the analysis of pipe flow is the pressure drop ΔP since it is directly related to the power requirements of the fan or pump to maintain flow. We note that dP/dx = constant, and integrating from x = x1 where the pressure is P1 to x = x1 + L where the pressure is P2 gives:

$$\frac{dP}{dx} = \frac{P_2 - P_1}{L} \qquad (2\text{-}11)$$

Substituting Eq.2-11 into the Vavg expression Eg. 2-8

$$\Delta P = P_1 - P_2 = \frac{8\mu L V_{avg}}{R^2} = \frac{32\mu L V_{avg}}{D^2} \qquad (2\text{-}12)$$

The symbol Δ is typically used to indicate the difference between the final and initial values, like $\Delta y = y_2 - y_1$. But in fluid flow, ΔP is used to designate pressure drop, and thus it is $P_1 - P_2$. A pressure drop due to viscous effects represents an irreversible pressure loss, and it is called pressure loss ΔP_L to emphasize that it is a loss (just like the head loss h_L, which is proportional to it).

In practice, it is found convenient to express the pressure loss for all types of fully developed internal flows as Fig.4

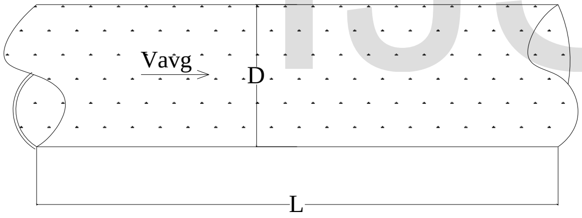


Fig.4. The relation for pressure is one of the most generalrelations in fluid mechanics, and it is valid for laminar or turbulent flows, circular or noncircular pipes, and pipes with smooth or rough surfaces

Pressure loss:

$\Delta P_L = fL\rho V^2_{avg}/2D$ (2-13)

where $\rho V^2_{vag}/2$ is the dynamic pressure and f is the Darcy friction factor

$f = 8\tau_w/\rho V^2_{avg}$ (2-14)

3 METHOD

COMSOL is the dominant physics simulation software in which Finite element method (FEM) and Partial differential equation are solved. The abilities of the software expand into the following eight add-on modules. Those are AC/DC, Chemical Engineering, Heat Transfer, and Structural Mechanics. Model libraries and supporting software such as Livelinks for SolidWorks and CAD have developed by the company [7].

COMSOL has various convenient features that have made this software be beneficial to the many engineers. It has developed in such a way that it is very easy to use for the simulation and modelling of real-world multiphysics. As a result, COMSOL has been a leading provider and developer of technical computing software. COMSOL is now the primary tools for engineers, researchers, and lecturers in the education and high tech product designs fields.

COMSOL simulation is a fundamental apparatus for the development of a new product. A various applications are included such as chemical, mechanical, electrical, and fluid. While talking about the need to couple the physics affecting a system; COMSOL simulation helps by providing an integrated simulation platform. COMSOL has been the unique simulation power offered which allows the present day‟s researchers and engineers to design the products in a short interval of time with low price. In conclusion, design challenges between physical effects interactions can be solved by the COMSOL Software.

3.1. Tutorial COMSOL

The Steady Incompressible flow tutorial in COMSOL Multiphysic version 5.2a is solved. Steady incompressible Navier-Stokes backstep geometry models help to examine the physics of the geometry in the absence of external forces. In the geometry below in Fig.5, fluid enters from the narrower region towards the wider region making a velocity profile of parabolic structure.

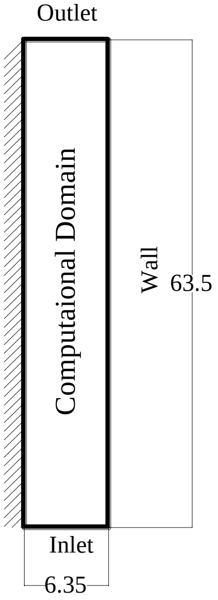


Fig.5. Computational domain

The velocity components of fluid u = (u, v) is computed in the x and y axis and the pressure p in the region is defined by

---

- *Hoai Nguyen is currently teaching in Department of Mechanical Engineering, College of Technology, The University of Danang, Viet Nam, PH: +84-934566516. E-mail: nhoai@dct.udn.vn*
- *Thang Hoang is currently teaching in Department of Mechanical Engineering, College of Technology, The University of Danang, Viet Nam, PH: +84-1208948867. E-mail: waveletthang@gmail.com*

the geometry of the above figure. The stationary incompressible Navier-Stokes equations are used by the Partial Differential Equation (PDE) model of this application. The equations are as follows:

$$\rho(u*\nabla)u=\nabla[-pI+u\{\nabla u+(\nabla u)^T\}]+F \qquad (3\text{-}1)$$

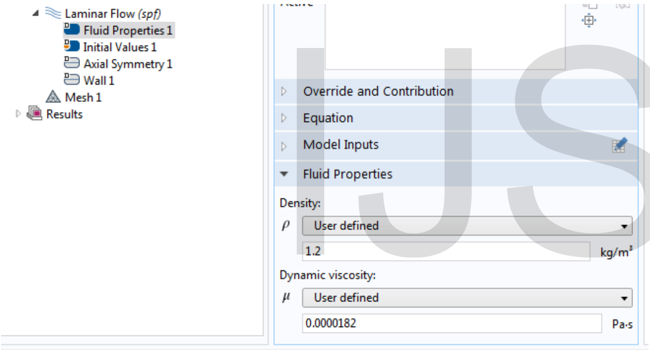
$$\rho\nabla*u=0 \qquad (3\text{-}2)$$

The first equation is the balance of momentum from Newton`s second law. The second equation is the equation of continuity, where zero on the right-hand side states that fluid is incompressible. The pattern of the flow depends only on the Reynolds number. The properties are as follows:

Dynamic viscosity (μ) = 0.0000182 Pa.s

Density (ρ) = 1.2 kg/m3

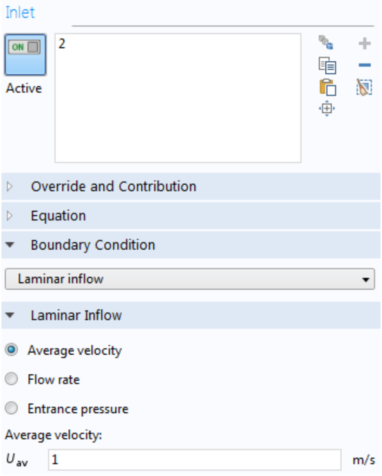
First of all, software for the analysis of COMSOL Multiphysics is opened, and the selection of laminar flow model is done for the simulation. Under fluid flow, single phase 24 flow and laminar flow were selected respectively. After the selection of flow model, study method was chosen as a stationary in which the field variables do not change according to the time. Then the geometry of the backflow is drawn. Here, a property of air is taken as fluid. It is shown in Fig.7.



Density:
ρ User defined
1.2 kg/m³
Dynamic viscosity:
μ User defined
0.0000182 Pa·s

Fig.6. Insertion of Fluid Properties in Laminar Flow

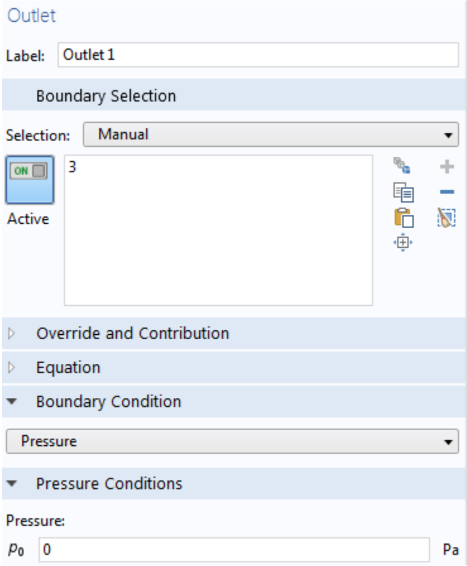
Inlet: Average velocity was V = 1 m/s used by COMSOL in this model and the same velocity were used to solve this tutorial. Inlet Properties is shown in Fig.8.



ON 2
Laminar inflow
● Average velocity
○ Flow rate
○ Entrance pressure
Average velocity:
U_{av} 1 m/s

Fig.7. Inlet Properties

Outlet: At outlet, pressure was selected as a boundary condition and was zero as a suppress backflow. Outlet properties is shown in Fig. 9



Label: Outlet 1
Selection: Manual
ON 3
Pressure
Pressure:
p_0 0 Pa

Fig.8. Outlet Properties

Meshing: After inserting all properties, meshing is another step for the COMSOL simulation of the laminar flow under fluid analysis. Meshing is defined as the representation of geometric shapes expressed as a set of finite elements. In this flow analysis, all of the meshes created were physics controlled and automatically generated.

Label: Mesh 1
Sequence type:
Physics-controlled mesh
Element size:
Normal

.9. Mesh properties

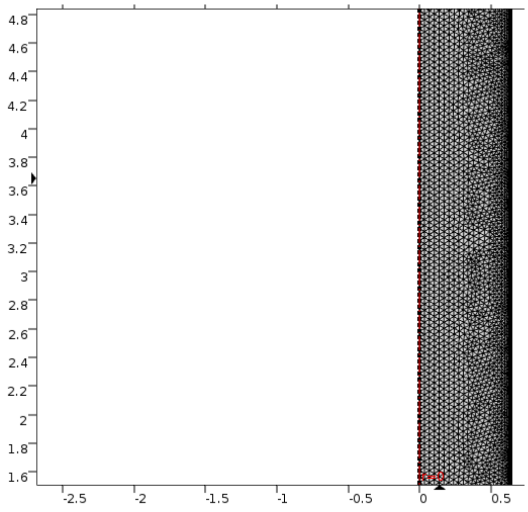


Fig.10. After mesh

4 RESULT AND DISCUSSION

4.1. Velocity of laminar flow through a pipe

The velocity magnitude is shown in the figure below. Different colour seen in the legends represents the difference in velocity according to the position of a pipe. The maximum . average and minimum velocity is also located in a pipeline figure below. The average velocity is increasingly reduced when laminar flow near wall. The maximum velocity occurs at the centerline of pipe. This pointed out that the fluid elements in the center of the pipe will move at a higher speed than those closer to the wall. It is perfectly consistent with the proven theoretical basis.

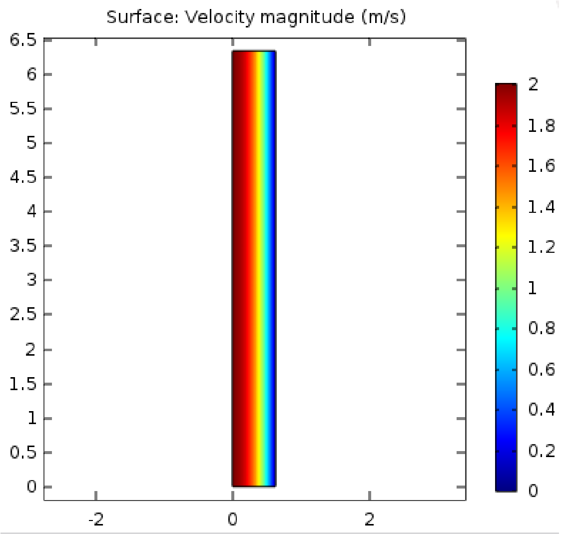


Fig.11. Velocity simulation of laminar flow through a pipe (2D)

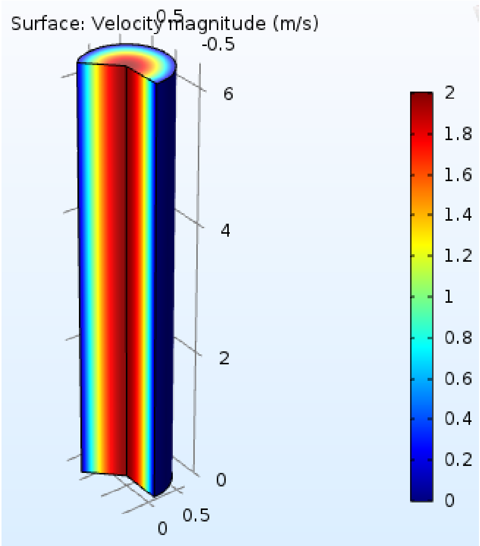


Fig.12. Velocity simulation of laminar flow through a pipe (3D)

4.3 Pressure of laminar flow through a pipe

The pressure in the different region of pipeline is represented by the variation in the colour shown in the figure below. The figure shows that the pressure is in decreasing order starting from inlet towards outlet.

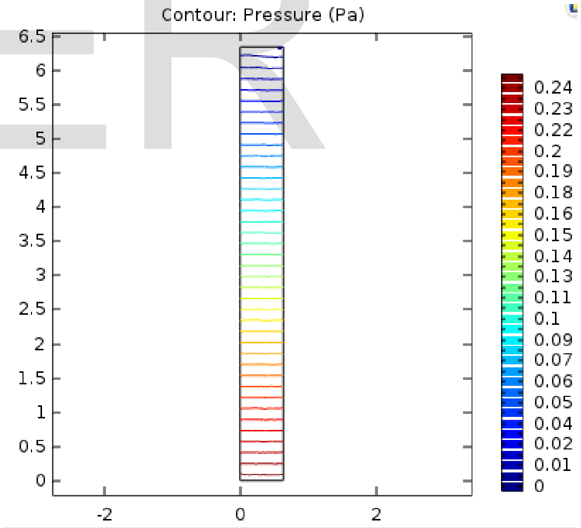


Fig.13. Pressure simulation of laminar flow through a pipe

4 CONCLUSION

In this paperm,The basic idea of this thesis is to use standard COMSOL for the observation of the laminar flow of fluids in a pipe and compare it with the results obtained in the theoretical basis. Hence, fluid flow was able to compare after the successful simulation but heat transfer did not converge to give a result.

Additionally, a finite element methodology has been adopted to visualize the dynamics of the flow characteristics inside the pipe. The average velocity from the COMSOL simulation was found 1 m/s. In particular, the maximum value of laminar flow velocity appeared at the center of the pipe. These values of velocity found experimentally and COMSOL simulation is the same. This also reinforces the authenticity of the theoretical basis for laminar flow through the pipe.

REFERENCES

[1] Goldstein, Richard, *Fluid mechanics measurements*, CRC Press, 1996.

[2] Olsson, A., Stemme, G., & Stemme, E., "Numerical and experimental studies of flat-walled diffuser elements for valve-less micropumps, *Sensors and Actuators A: Physical*, 84(1), 165-175, 2000.

[3] Rahman, S. U., N. M. Tukur, and I. A. Khan., "PC-Based Teaching Tools for Fluid Mechanics." *Conference on Enhancing Students Learning*, KFUPM, Saudi Arabia, 2002.

[4] Brunone, Bruno, et al., "Velocity profiles and unsteady pipe friction in transient flow", *Journal of water resources planning and management,* 126.4 (2000): 236-244.

[5] Antaki, George A., *Piping and pipeline engineering: design, construction, maintenance, integrity, and repair*, CRC Press, 2003.

[6] Martínez-Palou, Rafael, et al., "Transportation of heavy and extra-heavy crude oil by pipeline: A review." *Journal of Petroleum Science and Engineering,* 75.3 (2011): 274-282.

[7] Acharya, Saroj., *Analysis and FEM Simulation of Flow of Fluids in Pipes: Fluid Flow COMSOL Analysis*, (2016).