Available online www.ejaet.com

European Journal of Advances in Engineering and Technology, 2023, 10(4): 26-29

Research Article ISSN: 2394 - 658X

Studying and Modelling Velocity and surface pression in Laminar Flow in Pipes Using COMSOL Multiphysics

Papa Touty Traore[1], Hawa Diatara Ndiath[1], Moussa Dieng[1] and Issa Diagne[1]

[1]Physics Department, Cheikh Anta Diop University, Dakar, Senegal - 11000
papatoutytraore@gamil.com

ABSTRACT
This article studies some important works carried out on numerical analysis and approach modeling and the numerical tools used in the analysis of the important parameters to be considered in the laminar flow, such as friction losses, heat transfer, etc. laminar flow in pipes. This review focuses on laminar flow simulations.
The objective of this work is to study the evolution of the fluid velocity and the pressure of the fluid in the tube in the laminar regime as a function of the diameter of the tube.

Key words: Laminar, velocity, pression, COMSOL Multiphysics

---

1. INTRODUCTION

Flow in pipes [1] and ducts has long been the subject of much research. The results of this work have a great impact on practical application by giving the design engineer essential quantitative data for the construction of effective fluid management systems.
In this present work, the three-dimensional laminar flow [2] in a tube is modeled. The objective lies in the analysis of the evolution of the velocity and pressure in the section of the tube.
The results of this study make it possible to take into account the dimension of the sections of the pipes in the design in relation to the flow velocity of the fluid.
This simulation is performed under the multiphysics comsol [3] environment

2. MATHEMATICAL MODEL

Laminar flow occurs in a cylindrical tube 300mm high and 12.5mm radius. Liquid water is the fluid used. The fluid inlet speed in the tube is 0.05m/s. Fluid temperature is 300K.

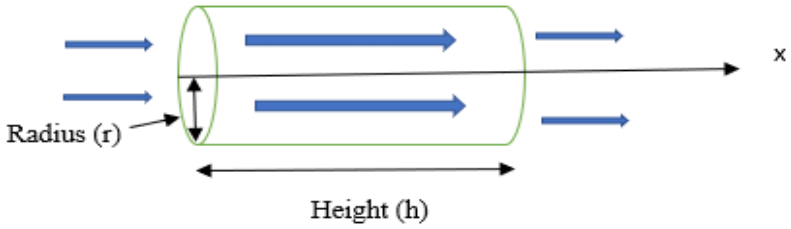

Fig. 1. Laminar flow through pipe

Equations
The stationary incompressible Navier-Stokes equations [4] are used by the Partial Differential Equation (PDE) model of this application. The equations are as follows:

The first equation is the balance of momentum from Newton`s second law [5]. 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 [6]:

$$\boldsymbol{\rho(U.\nabla) = \nabla.\nabla[-pl + K] + F} \quad \textbf{(1)}$$
$$\boldsymbol{\rho\nabla.U = 0} \quad \textbf{(2)}$$

Boundaries condition
Wall :
$\boldsymbol{U = 0}$
(3)
Inlet :
$$\boldsymbol{[-p.L + p.(\nabla U + (\nabla U)^{\tau}]n = -\widetilde{p_o}.n} \quad \textbf{(4)}$$
$$\boldsymbol{\widetilde{p_o} \leq p_o} \quad \textbf{(5)}$$
$$\boldsymbol{U = -U_o.n} \quad \textbf{(6)}$$

Laminar inflow
$$\boldsymbol{L_{entr}\nabla_{\tau}.[-pl + p(\nabla_{\tau}.U + (\nabla_{\tau}U)^{\tau})] = -p_{entr}.n} \quad \textbf{(7)}$$
Outlet

$\boldsymbol{L_{entr}}$ entrance Lenght
$\boldsymbol{U = U_{avr}}$ average velocity
$\boldsymbol{p_o}$ Pression

Description of laminar flow
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 along the direction of flow. There is no motion in the radial direction, and so the velocity component in the direction normal to the flow is zero everywhere. There is no acceleration since the flow is steady and fully developed [7].
Presentation of Comsol multiphysics
COMSOL is the dominant physical simulation software in which the finite element method (FEM) and partial differential equation are solved. The capabilities of the software extend to the following eight add-ons. These are AC/DC, Chemical Engineering, Heat Transfer [8] and Structural Mechanics. Model libraries and supporting software such as Live links for SolidWorks and CAD have been developed by the company. COMSOL has various handy features which have made this software will be beneficial for many engineers. It has developed in such a way that it is very easy to use for simulation and modeling of real-world Multiphysics. Therefore, COMSOL is one of the leading suppliers and developers of technical computer software. COMSOL is now the main tool for engineers, researchers and lecturers in the field of education and fields of high technology product design. COMSOL simulation is a fundamental device for the development of a new product. Various applications are included such as chemistry, mechanics, electricity, and fluids.
While talking about the need to couple the physics affecting a system; COMSOL simulation helps by providing an integrated simulation platform [10]. COMSOL has been offering unique simulation power that allows today's researchers and engineers to design products in a short time span at low cost. In conclusion, the design challenges between physical effects interactions can be solved by the COMSOL software.

3. RESULTS AND DISCUSSION

The following figures simulation represent the results after

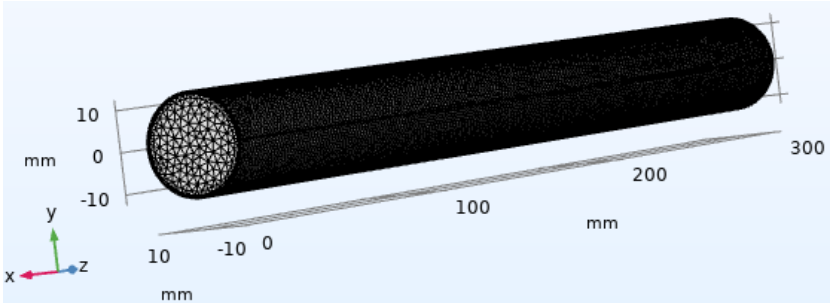

Fig. 2 Mesh cylinder

The figure 2 shows the pipe after mesh.The mesh method used is the finite volume with a very low error step.
The results was valided by the works done by

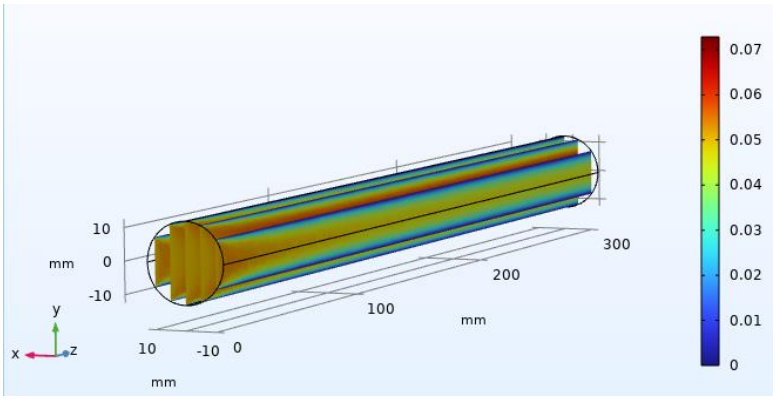

Fig. 3a Velocity magnitude with 5 slice (3D)

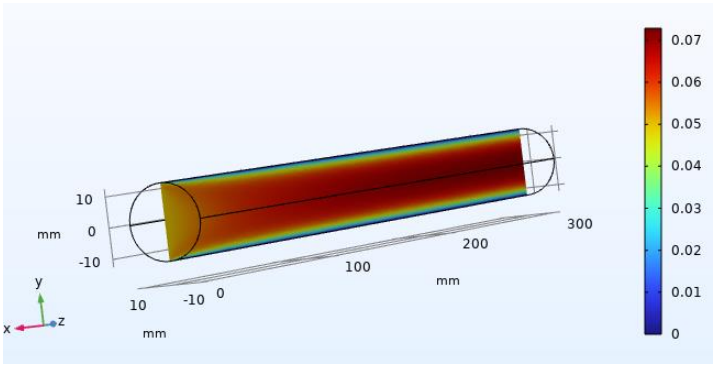

Fig. 3b Velocity magnitude 1 slice (3D)

The figure 3 and 4 shows how the velocity changes inside the pipe.The difference beteween the two figures is figure 3a shows the magnitude of velocity with 5 slice and the figure 3b 1 slice.The different colors on both s diagram show how the velocity changes inside the pipe.We observe that , the magnitude of the velocity is higher in the center of the pipe relative at the wall.The figure 3b we observe clearly the distribution of the velocity through each axis.

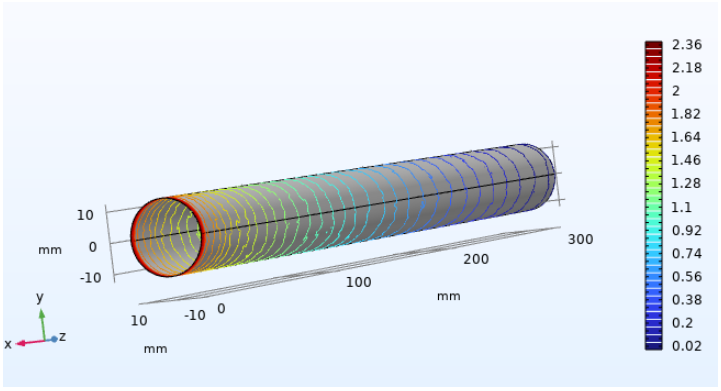

Fig. 4 Surface pression (3D)

La figure 3 shows how the pression changes at the surface of the pipe.The colors label show how the pression is at the surface of the pipe We notice that through the inlet of the pipe the pression is very important at the inlet and it's decreasing and tends to 0.02 atm at the outlet.That's shown by the differents colors.
We remark that the pression is important where exactly the velocity of the fluid is increasing. That means there is relationship between the velocity and the pression of the liquid.

4. CONCLUSION

In this work, the change of surface pression and the magnitude of the velocity of water through pipe were shown. The study was established in laminar flow under COMSOL Multiphysics software.
The surface pression is more important at the inlet relative to the outlet, the magnitude of velocity as well.
That means there is a relationship between the velocity and the pression in laminar flow.

REFERENCES

[1]. N. Yucel and N. Dinler, "Numerical Study on Laminar and Turbulent Flow through a Pipe with Fins Attached," An International Journal of Computation and Methodology, Vol. 49, No. 2, 2006, pp. 195-214.
[2]. Soundalgekar and V. G. Divekar, "Laminar SlipFlow through a Uniform Circular Pipe with Small Suction," Publication de l'Institut Mathematique, Nouvelle Serie, Vol. 16, No. 30, 1993, pp. 147-157.
[3]. KLO. Saheed Ismail, George T. Adewoye, Analyses and Modeling of Laminar Flow in Pipes Using Numerical Approach, Journal of Software Engineering and Applications, 2012, 5, 653-658.

[4]. Hyunjun Min, Jiho Shin, Jaeho Choi, and Hohyun Lee, Modeling and Simulating Flow Phenomenon using Navier-Stokes Equation, Proceedings of the International Conference on Industrial Engineering and Operations Management Rabat, Morocco, April 11-13, 2017
[5]. Xiaoping Liu Revisiting Newton's Second Law of Motion, ResearchGate July 30, Tuesday, 2019
[6]. Václav Uruba,, On Reynolds Number Physical Interpretation, AIP Conference Proceedings Volume 2000, 7 August 2018, Article number 0200193, 7th Meeting of Departments of Fluid Mechanics and Thermodynamics; Hotel Thermal Sturovo; Slovakia; 27 June 2018 through 30 June 2018; Code 138542.
[7]. T. Zhao and P. Cheng, "A Numerical Solution of Laminar Forced Convection in a Heated Pipe Subjected to a Reciprocating Flow," International Journal Heat Mass Transfer, Vol. 38, No. 16, 1995, pp. 3011-3022.
[8]. M. Z. Rad, A. Nouri-Broujerdi and J. Seume, "Numerical Computation of Compressible Laminar Flow with Heat Transfer in the Entrance Region of a Pipe," Proceeding of 2008 ASME Summer Heat Transfer Conference HT, Jack sonville, 10-14 August 2008.
[9]. A. Nouri-Broujerdi and M. Ziaei-Rad, "Simulation of Compressible Pipe Flow under Different Thermal Conditions," 2007 ASME-JSME Thermal Engineering Summer Heat Transfer Conference, Vancouver, 8-12 July 2007.