Analysis and Enhancement of Hydraulic Ram Pump using Computational Fluid Dynamics (CFD)

Piyush B. Shende
Student
Department of Mechanical Engineering
K.D.K.C.E Nagpur, India

Dr. S. K. Choudhary
Professor
Department of Mechanical Engineering
K.D.K.C.E Nagpur, India

A. P. Ninawe
Assistant Professor
Department of Mechanical Engineering
K.D.K.C.E Nagpur, India

Abstract

This paper presents the analysis of the hydraulic ram pump performance using computational fluid dynamic(CFD). The study specifies that mass flow through the waste valve of hydram is more, so to reduce it an enhancement is done and a new design is created. These are based on a systematic study of hydraulic ram pump and testing of on hydraulic ramp pump model. For which we consider literatures reviews & some of them are used for the analysis.
Keywords: Hydraulic Ram Pump, Computational Fluid Dynamic (CFD), Waste Valve, Enhancement

I. INTRODUCTION

An analysis of Hydraulic ram pump presents some of the most challenging problems faced by engineers in domestic use, agriculture use, industry use, because simulations need to address the moving parts in an appropriate manner. In this way, computational fluid dynamic (CFD) tools have become essential in aiding in the pump development. Hydraulic ram pump become more and more complex in order to address the higher output. Therefore flow characteristics are constantly evolving in order to keep CFD in pace with pump innovations. Flow characteristics help to optimize various parameters of a pre-existing.

In this study, a commercial CFD program FLUENT is used. In order to envisage flow domain more in pump model, FLUENT's moving and deforming grid feature is used. A flow analysis is performed for this purpose. Flow simulations for Hydram can provide valuable design information to engineers. These simulations allow for the effect on efficiency, mass flow rate and flow properties to be predicted based on changes in port and waste valve design, valve lift timing, or other parameters. Flow in pump produces a complex three-dimensional phenomenon involving turbulence, unsteadiness, etc. Moreover, the geometry is complex. Both experimental and numerical approaches have been reported and have contributed to the understanding of the highly complex flow interactions that occur in an hydram. CFD techniques are very useful tools for design and analysis of complex pump geometries. Although most of the current CFD research activities are on steady flows, there is an increasing interest in unsteady flow computations. The equations of fluid dynamics can be solved numerically with the aid of modern computers. The set of techniques and procedures to achieve this has lead to the development of what is known as CFD. Although it is not possible to leave out real experiments, numerical study using CFD allow the determination of certain trends and give an insight into the physics of complex problems that would be impractical to study experimentally. The present study tries to validate CFD prediction in selected pump geometries. But their reason will be explained in detail in Conclusion section. All of CFD, in one form or another, is based on the fundamental governing equations of fluid dynamics—the continuity, momentum, and energy equations. These equations speak physics. They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based on mass conservation, momentum equation, energy equation and turbulence equation. The governing equations can be obtained in various different forms. For most hydrodynamic theory, the particular form of the equations makes little difference. However, for a given algorithm in CFD, the use of the equations in one form may lead to success, whereas the use of an alternate form may result in oscillations (wiggles) in the numerical results, incorrect results, or even instability. Therefore, in the world of CFD, the various forms of the equations are of vital interest. In turn, it is important to derive these equations in order to point out their differences and similarities, and to reflect on possible implications in their application to CFD. Conservation laws can be derived by considering a given quantity of matter or control mass and its extensive properties, such as mass, momentum and energy. This approach is used to study the dynamics of solid bodies, where the control mass (sometimes called the system) is easily identified. In fluid flows, however, it is difficult to follow a parcel of matter. It is more convenient to deal with the flow within a certain spatial region that is called a control volume (CV), rather than in a parcel of matter which quickly passes through the region of interest. This method of analysis is called the control volume approach that is also used in FLUENT solver.

II. MODEL DESIGNING

The collected data is used to design the model. Designing software is Pro-E.
The domain is created in Pro-E software and imported into CFD software for the final results.

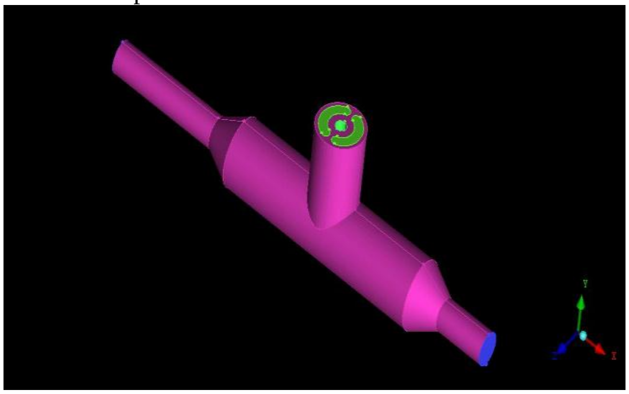

Fig 1: Domain imported into CFD

A. Meshing

In this study unstructured mesh is used for the geometry. ICEM pre-processor has a capability of generating unstructured mesh. There are two main advantage of unstructured meshing, one it occupies a very less memory in computer and another one is it requires very less time to compute and solve the equation to get solution. Takes less time to analysis delivered appropriate and accurate result.

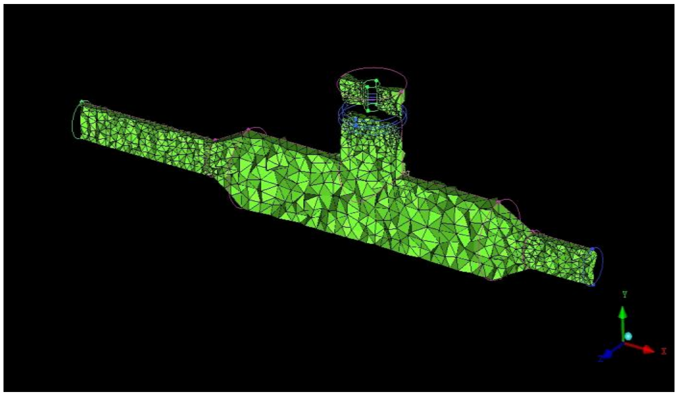

Fig. 2: Meshed geometry

The geometry is being meshed with the help of triangular cell and quadrilateral cell. The no. Of elements 132554. All of CFD, in one form or another, is based on the fundamental governing equations of fluid dynamics—the continuity, momentum, and energy equations.

B. Boundary Condition

Table – 1

Velocity inlet	***Inlet port***
	Velocity inlet or mass flow inlet
	Values can be added for different heads.
Outlet	*Mass outlet*
Wall	*Rest of parts of geometry are named as wall boundary condition (stationary).*

III. ANALYSES FOR EXISTING DESIGN

The dynamic grid approach is used to treat the moving piston in the computational area. In other words, the grid generation approach was used to treat the moving piston as a moving solid body in the computational domain without generating completely new grids. Piston moves upward and downward position considering open condition and closed condition of waste valve. The model structure is unstructured grid and to setup boundary condition for moving piston. Total number of computational cells was used about 132554. The above mentioned solver setting is used to get appropriate and accurate flow pattern. Following figures shows the flow pattern after analysis. Figures present the velocity vectors, pressure vectors for the open and closed condition for different heads.

A. For Head 1m 1:

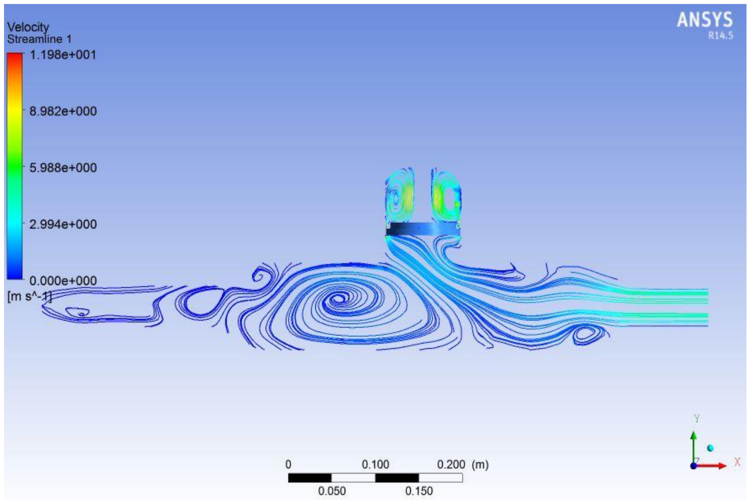

Fig. 3: Velocity Flow for Open Condition

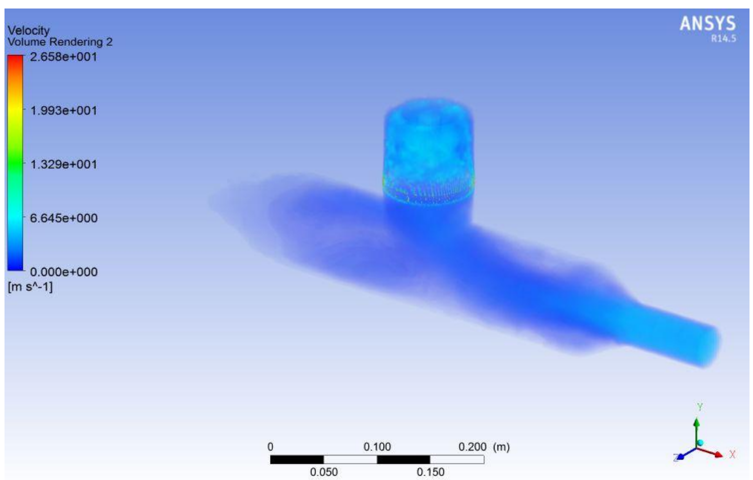

Fig. 4: Velocity Vector for Open Condition

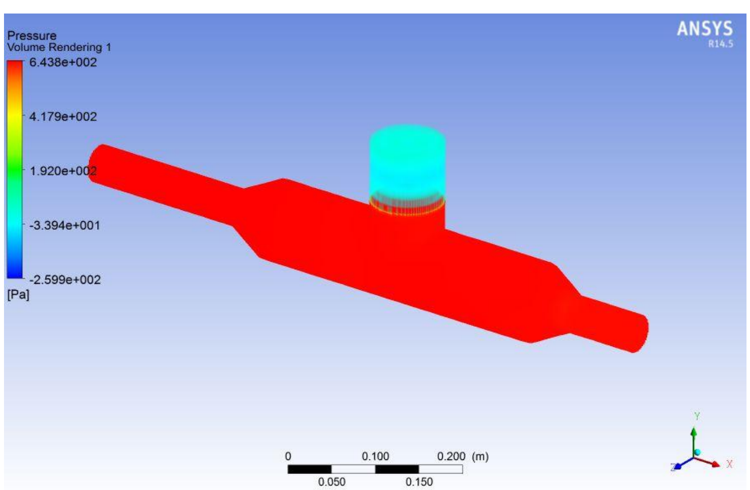

Fig. 5: Pressure Vector for Open Condition

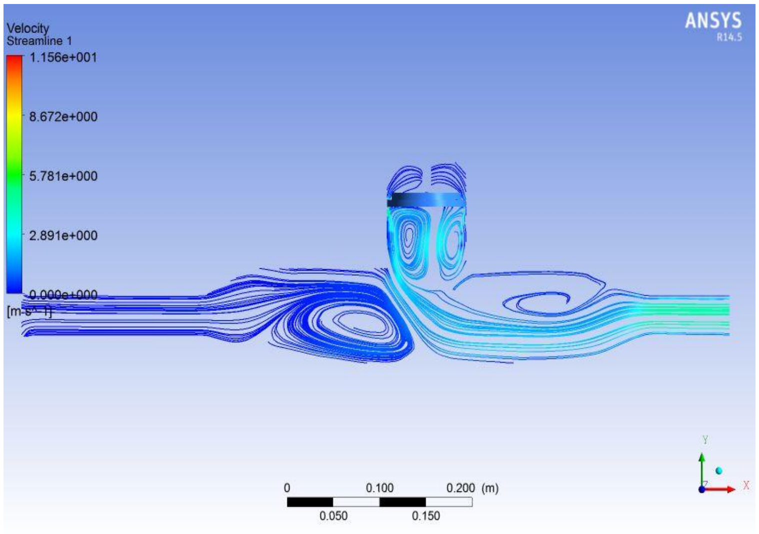

Fig. 6: Velocity Flow for Closed Position

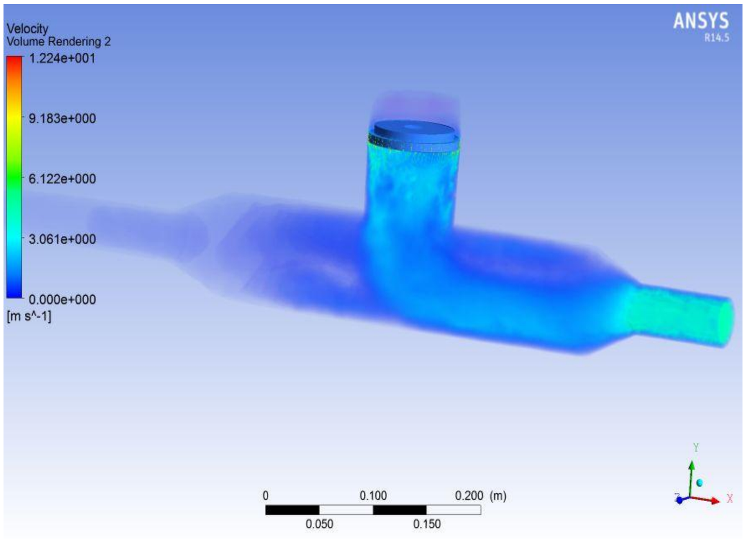

Fig. 7: Velocity Vector for Closed Condition

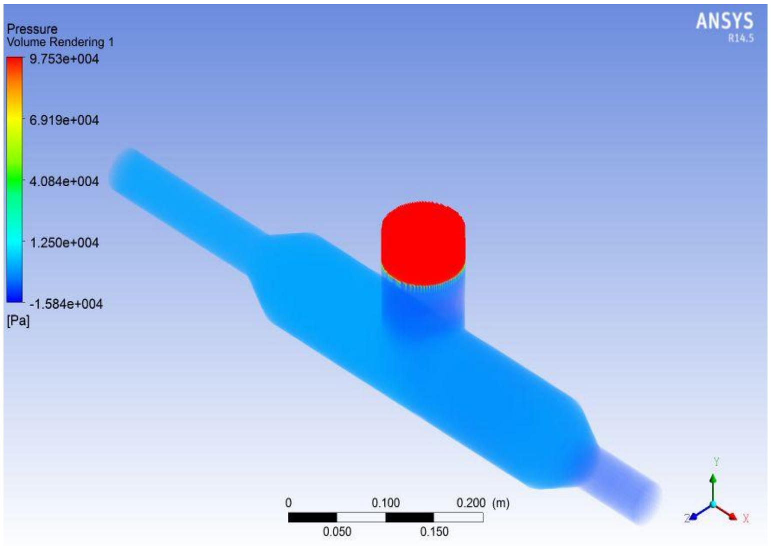

Fig. 8: Pressure Vector for Closed Condition

B. For Head 2m:

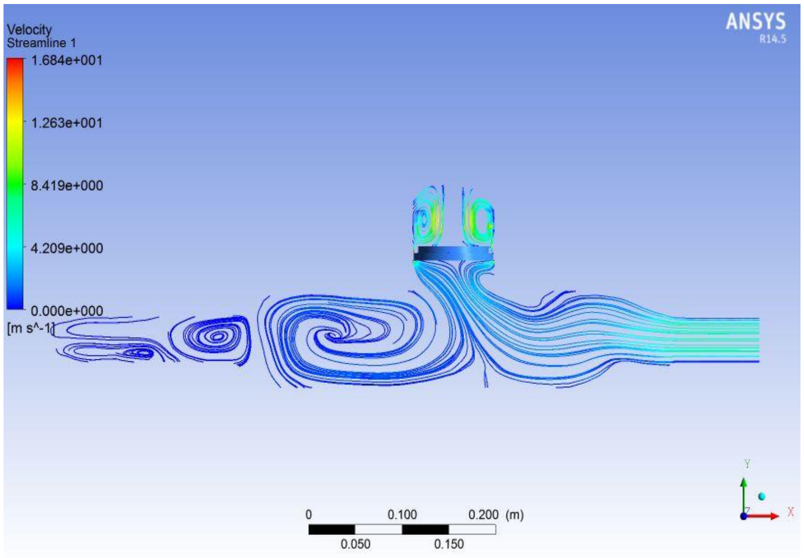

Fig. 9: Velocity Flow for Open Condition

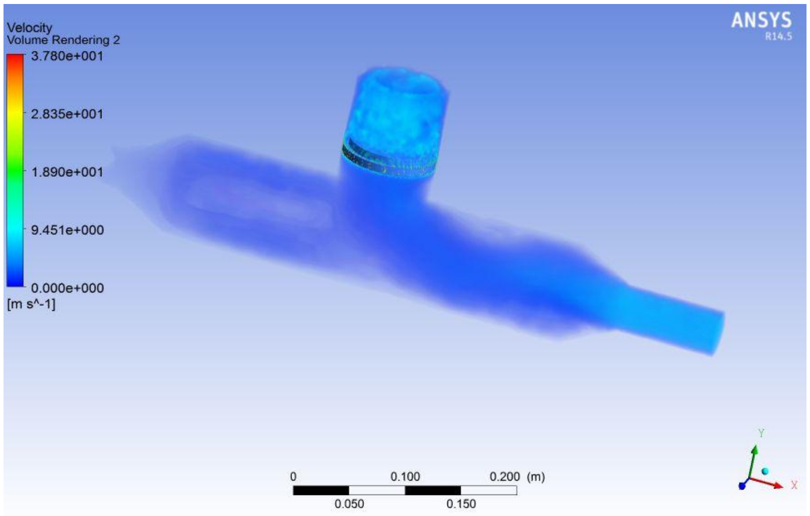

Fig 10: Velocity Vector for Open Condition

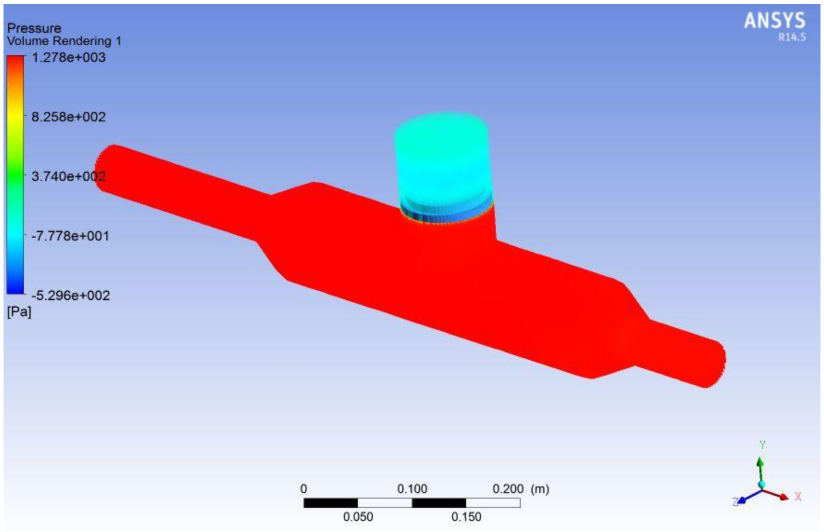

Fig 11: pressure vector for open condition

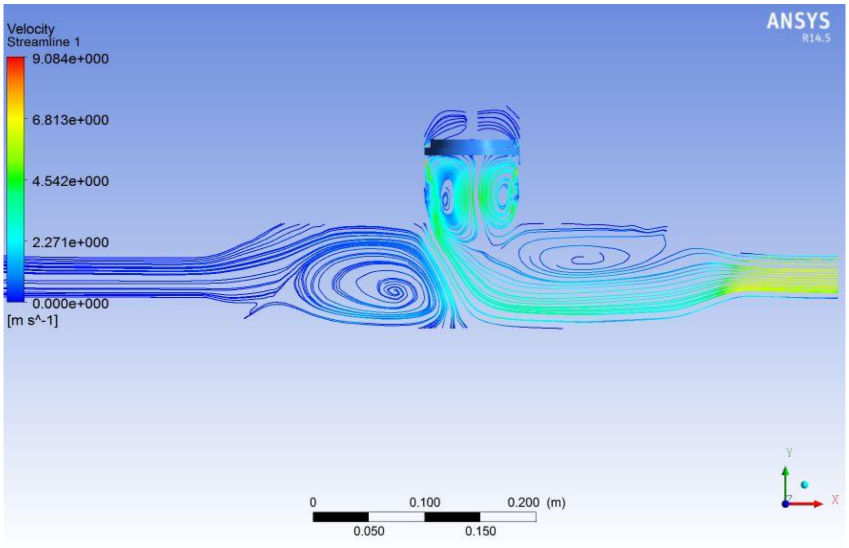

Fig 12: Velocity Flow for Closed Condition

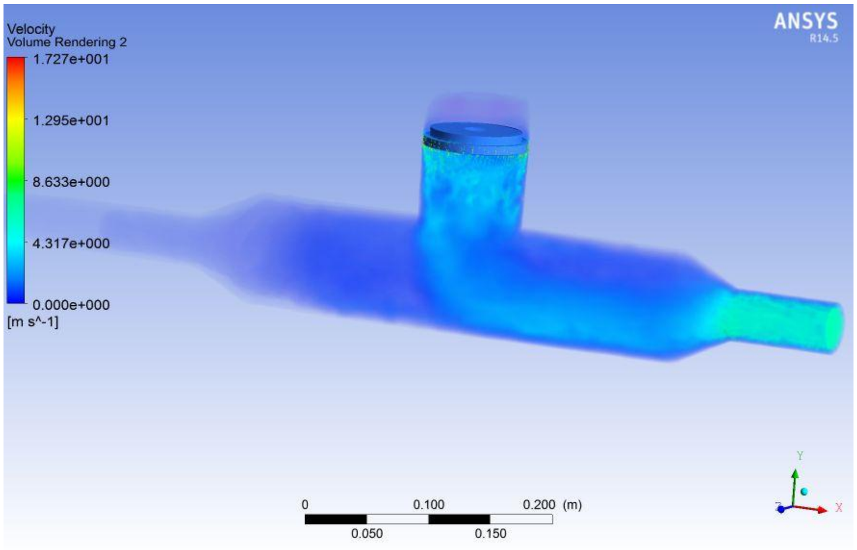

Fig. 13: Velocity Vector for Closed Condition

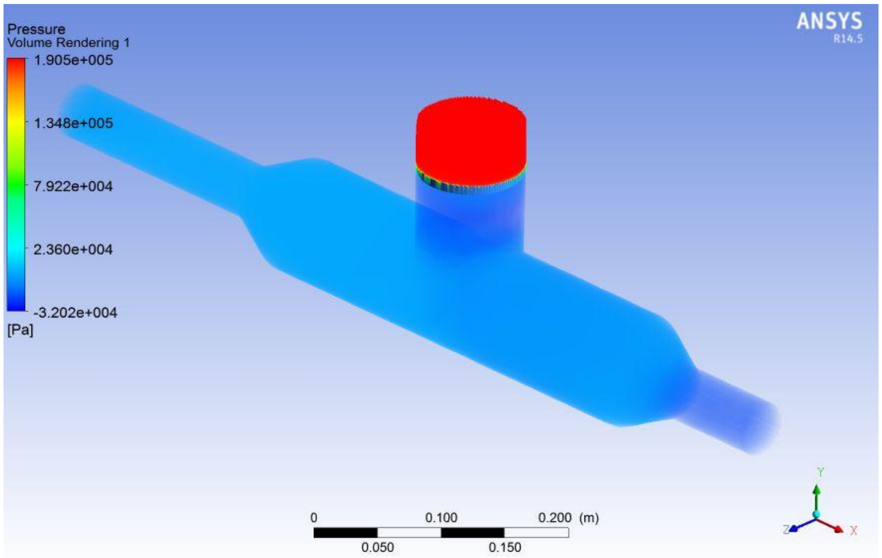

Fig. 14: Pressure Vector for Closed Condition

C. For Head 3m:

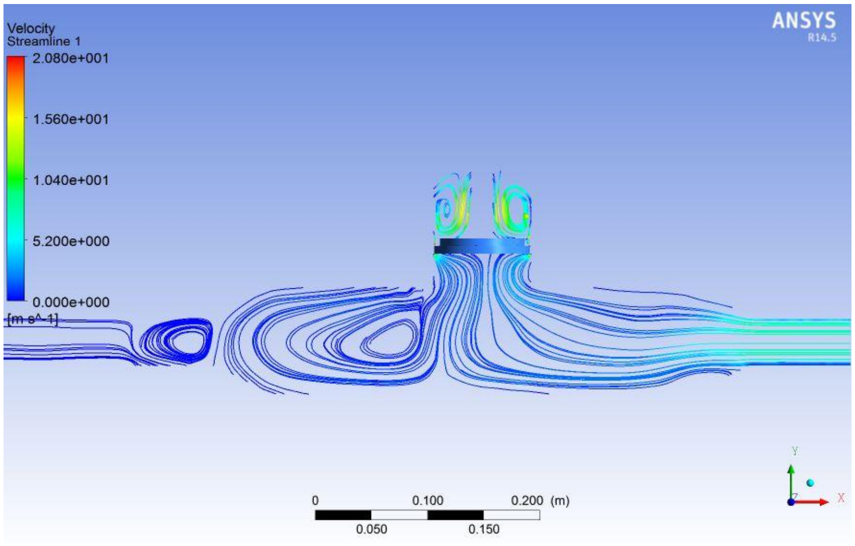

Fig. 15: Velocity Flow for Open Condition

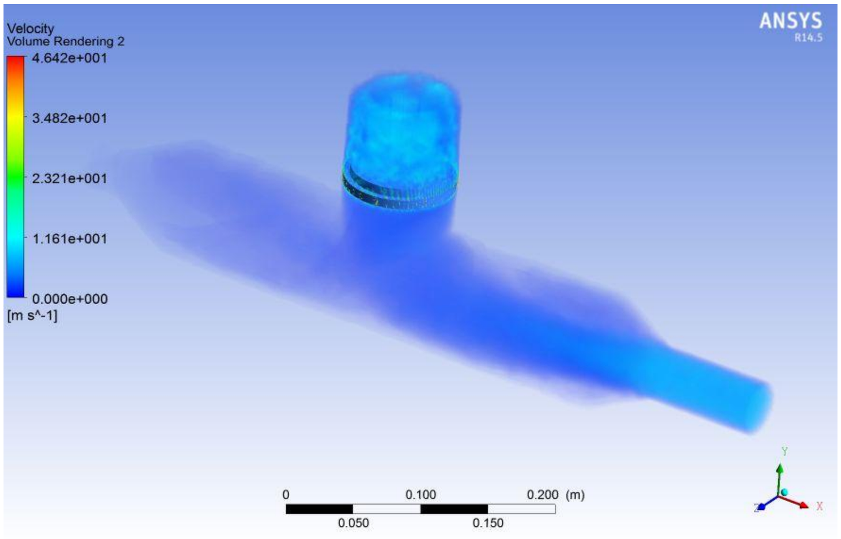

Fig. 16: Velocity Vector for Open Condition

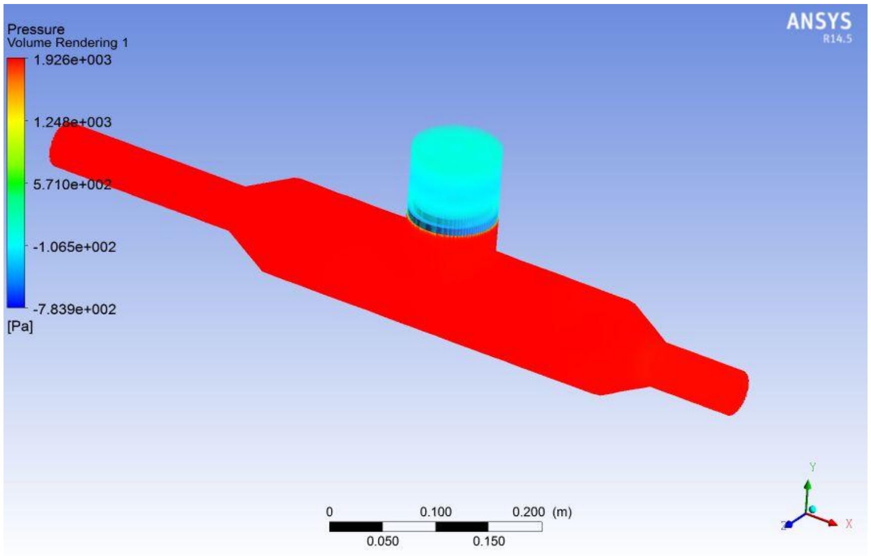

Fig. 17: Pressure Vector for Open Condition

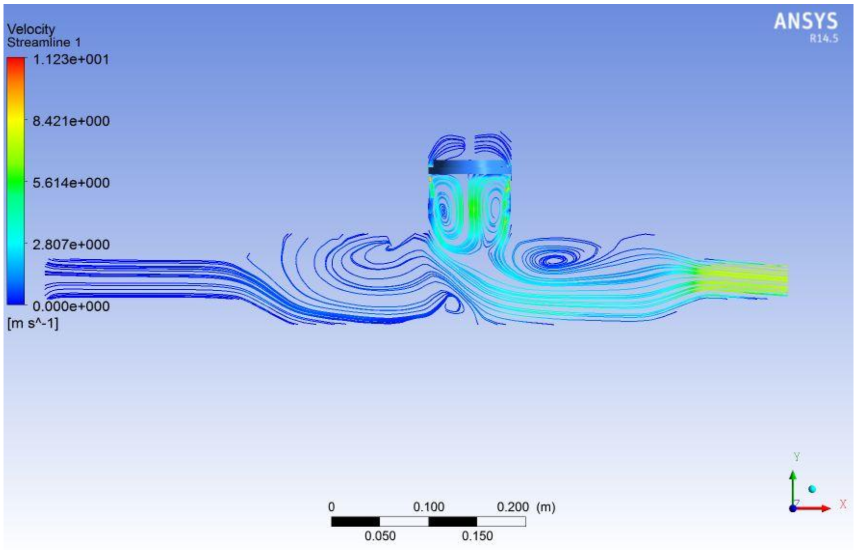

Fig. 18: Velocity Flow for Closed Condition

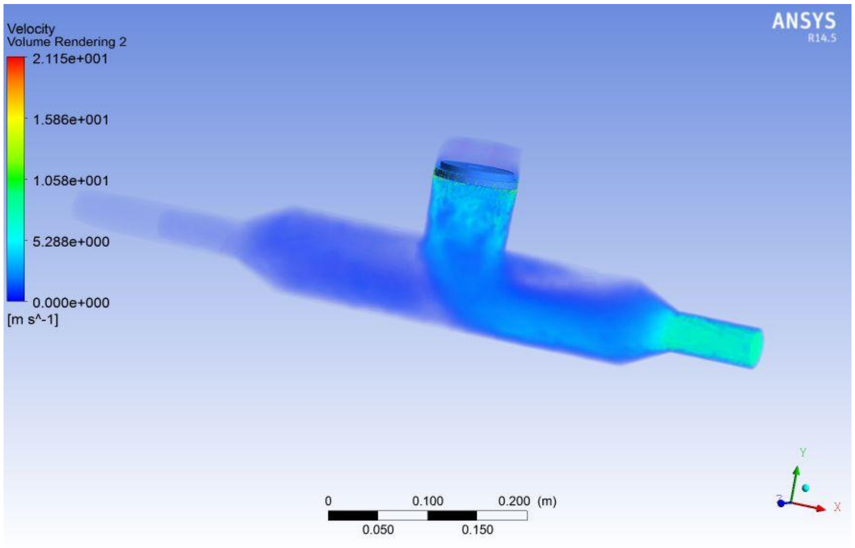

Fig. 19: Velocity Vector for Closed Condition

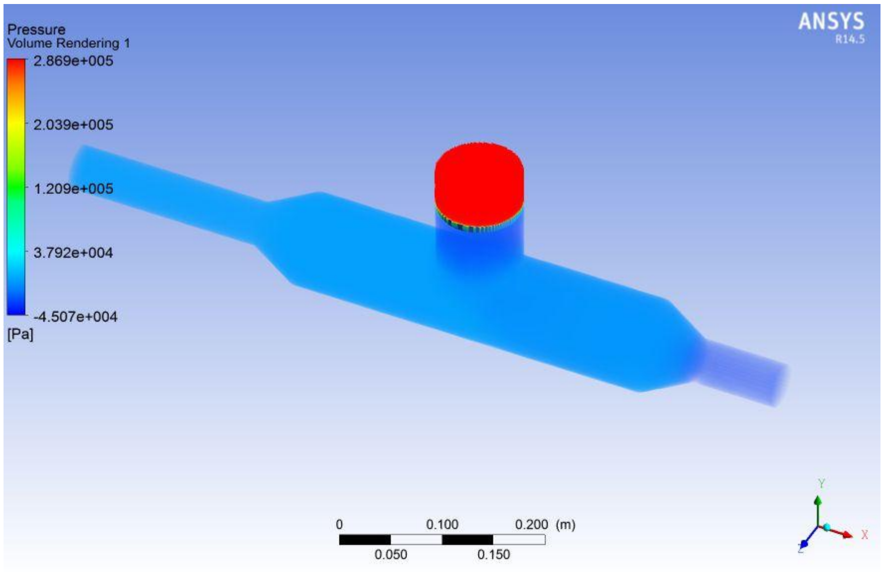

Fig. 20: Pressure Vector for Closed Condition

D. For Head 4m:

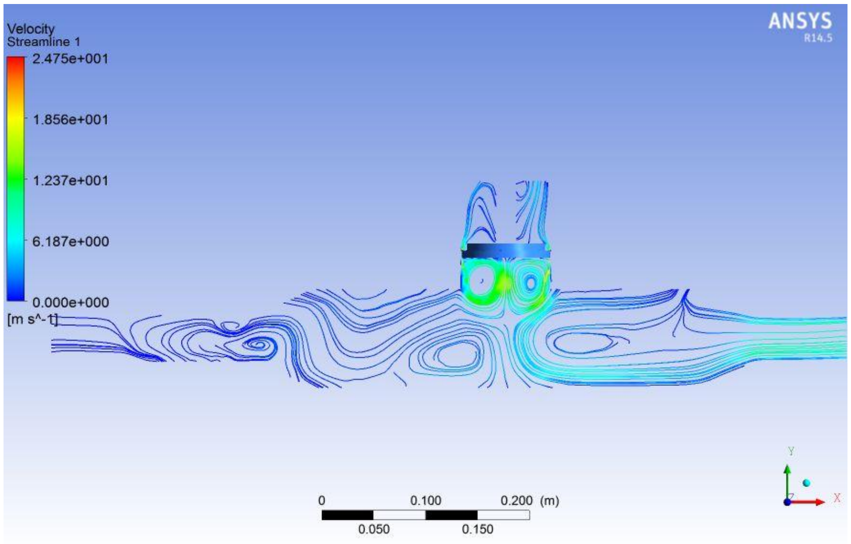

Fig. 21: Velocity Flow for Open Condition

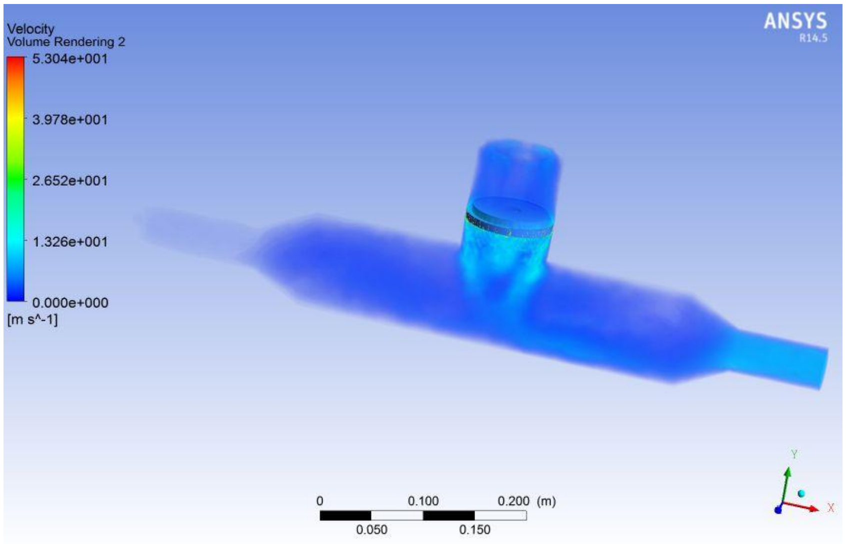

Fig. 22: Velocity Vector for Open Condition

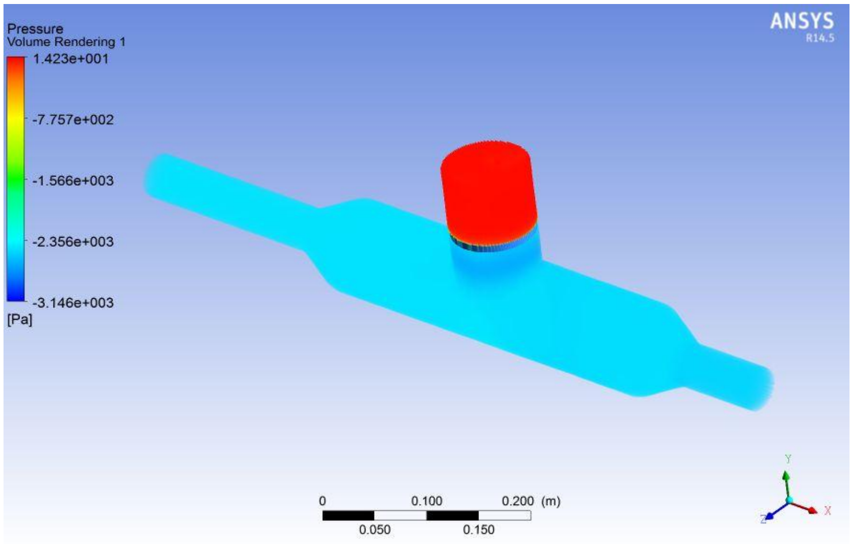

Fig. 23: Pressure Vector for Open Condition

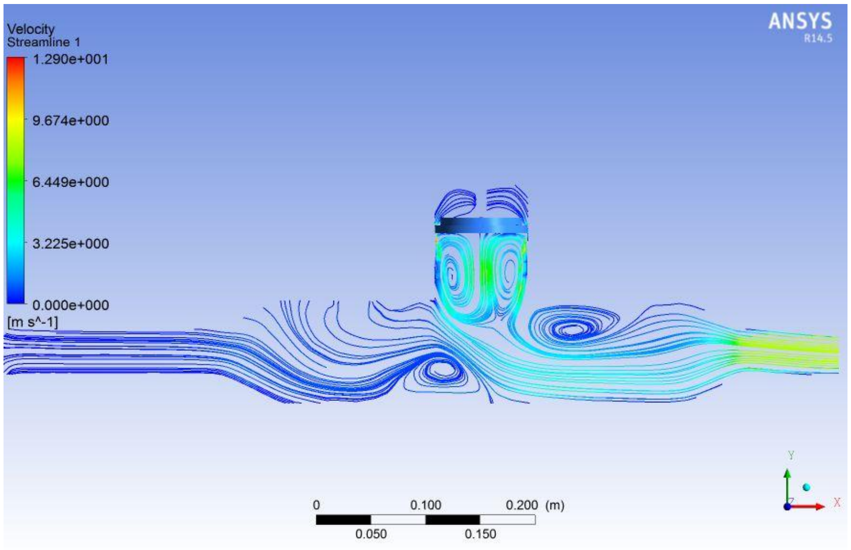

Fig. 24: Velocity Flow for Closed Condition

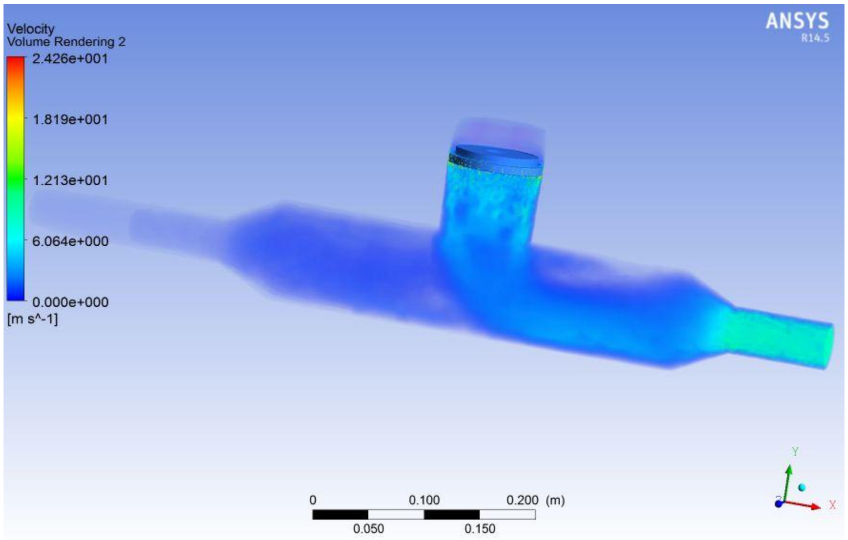

Fig. 25: Velocity Vector for Closed Condition

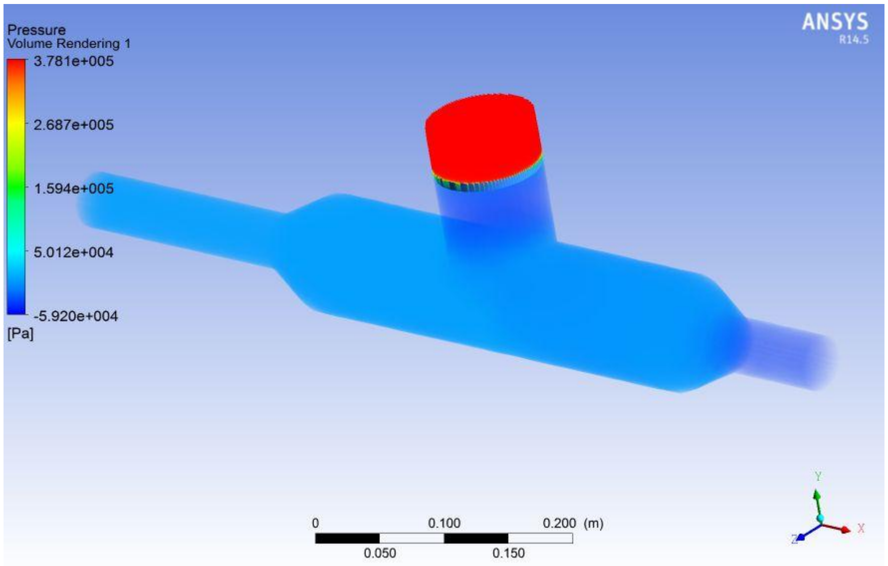

Fig. 26: Pressure Vector for Closed Condition

E. For Head 5m:

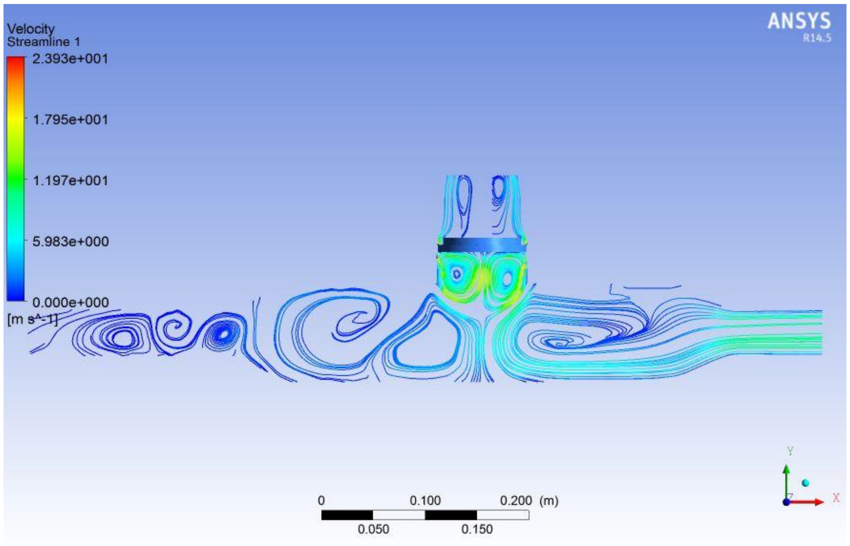

Fig. 27:Velocity Flow for Open Condition

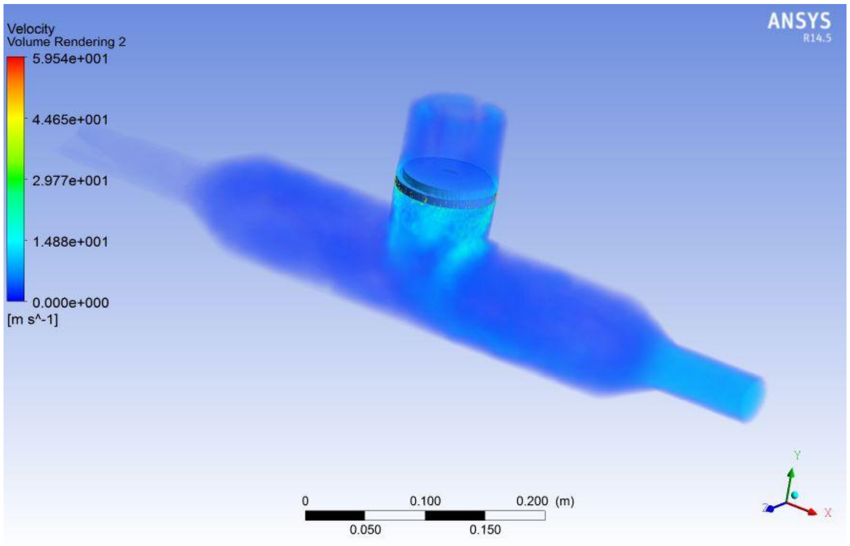

Fig. 28: Velocity Vector for Open Condition

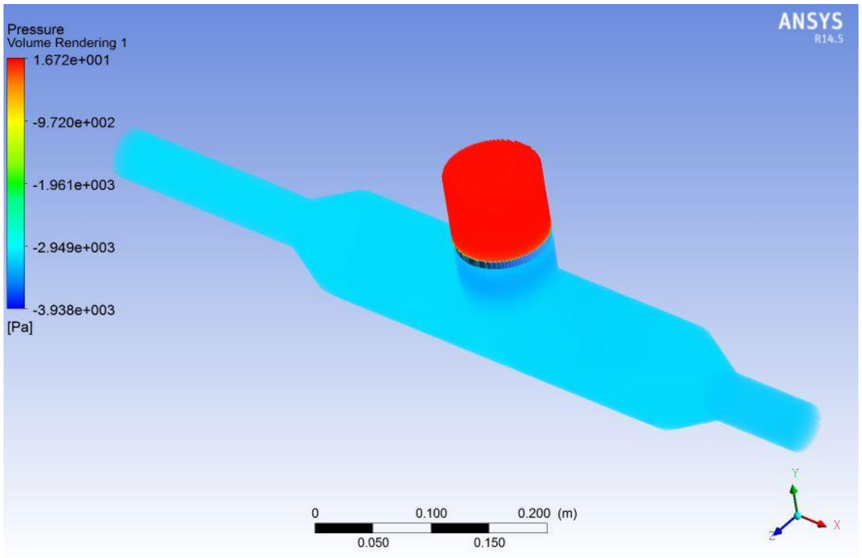

Fig. 29: Pressure Vector for Open Condition

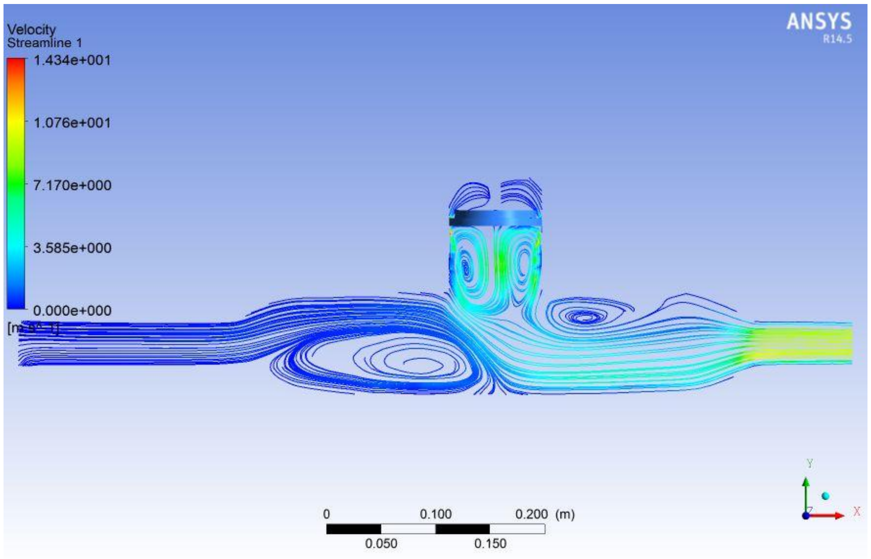

Fig. 30: Velocity Flow for Closed Condition

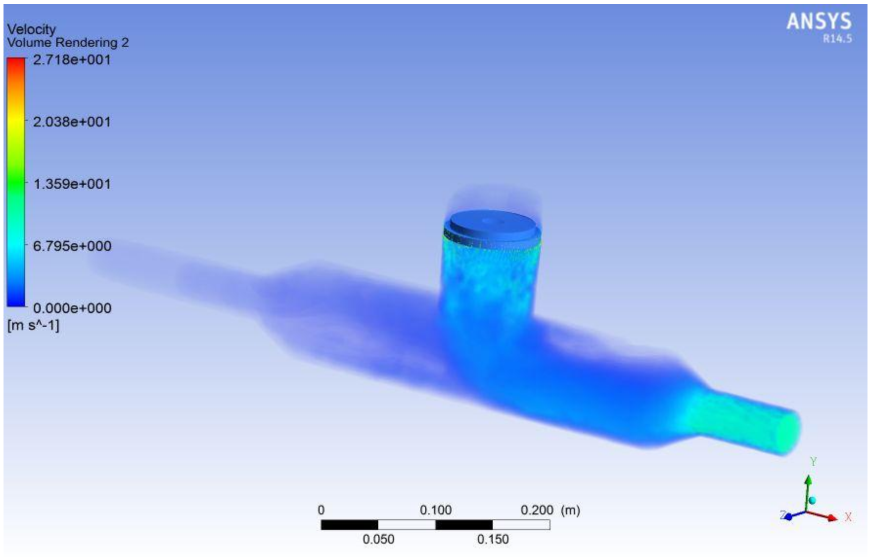

Fig. 31: Velocity Vector for Closed Condition

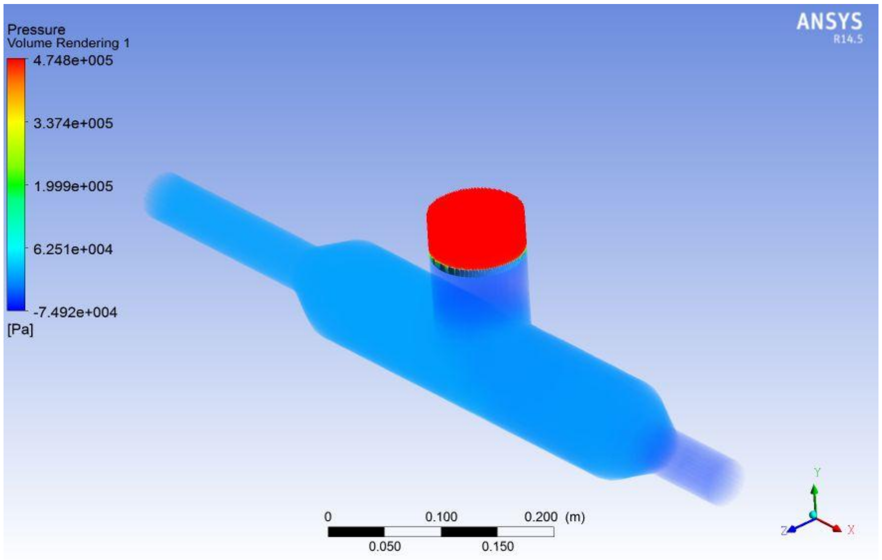

Fig. 32: pressure vector for closed condition

The mass flow rate through waste valve calculated using the CFD for head 4m is 0.016 kg/sec as shown in following fig –

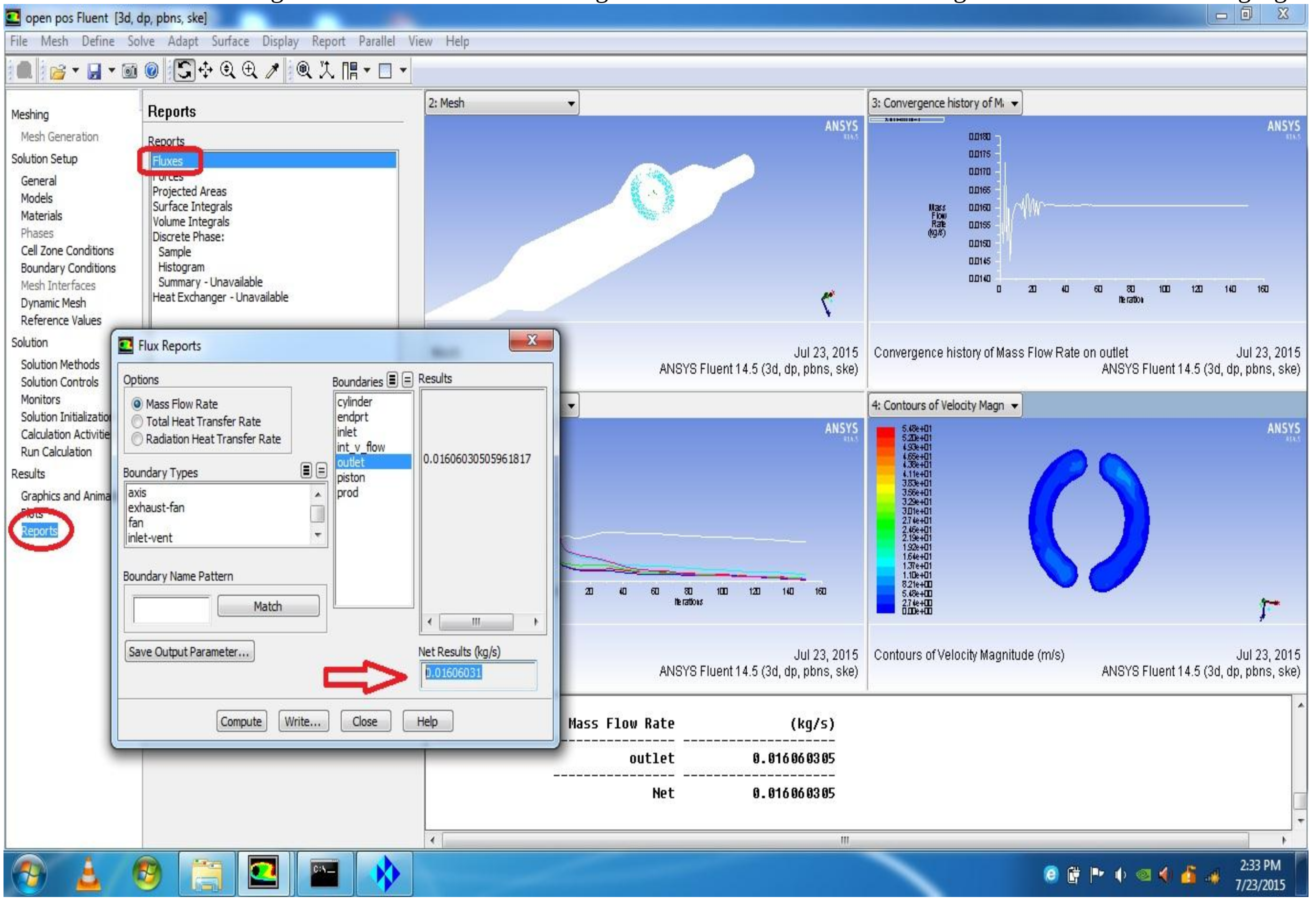

Fig. 33: Mass Flow Rate through Waste Valve for Head 4m

IV. ANALYSIS OF ENHANCEMENT APPROACH

Present CFD analysis reports the stream line flow, velocity vectors, pressure vectors for different heads 1m, 2m, 3m, 4m, and 5m. Water flows through the pipe with some amount of kinetic energy this energy is utilized to perform the rest work of lifting the amount of water . As the water enters the working domain the velocity varies with the varying pressure in the pump body. The flow is probably turbulent as the flow is heavily unsteady. All the analysis is redone by changing the design of waste valve and same procedure is make for the analysis of the pump. The changed model is shown as follow-

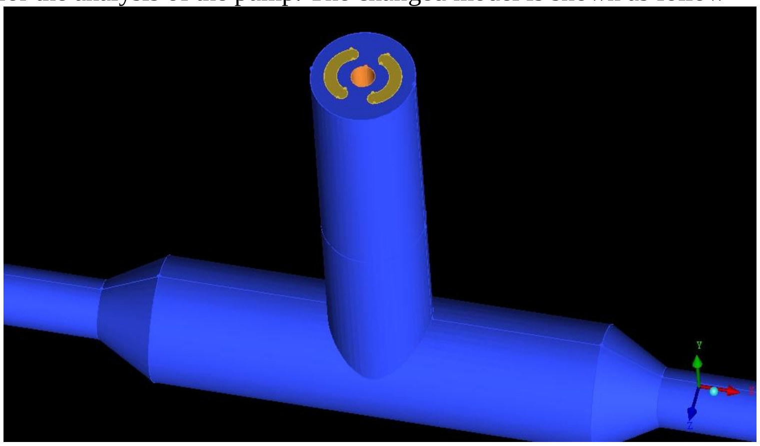

Fig. 34:New Working Domain

Import the above geometry into ICEM software for the meshing and used FLUENT as solver. Applying same solver setting which have used for validated geometry. The meshed geometry and stream line flow, velocity vector and pressure vector are drawn as shown in figures -

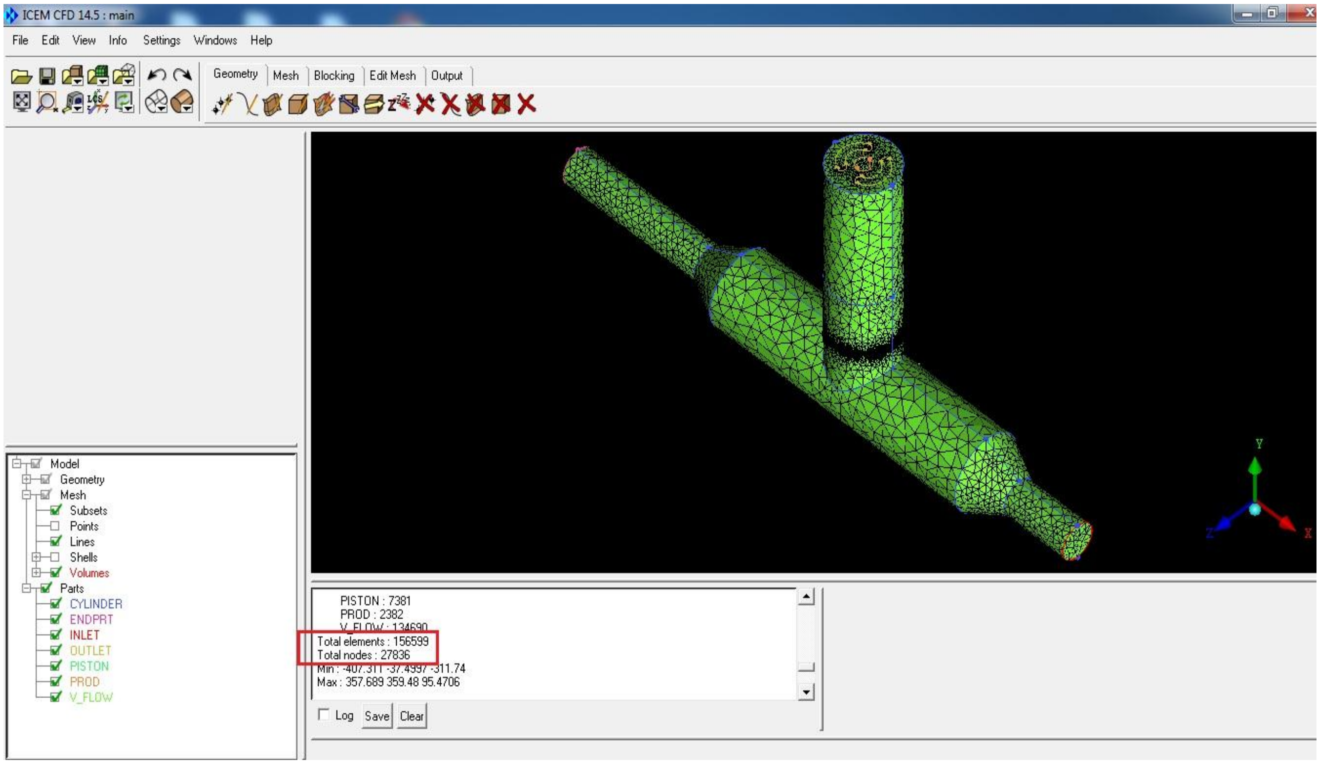

Fig. 35: Mesh Geometry

A. For Head 1m:

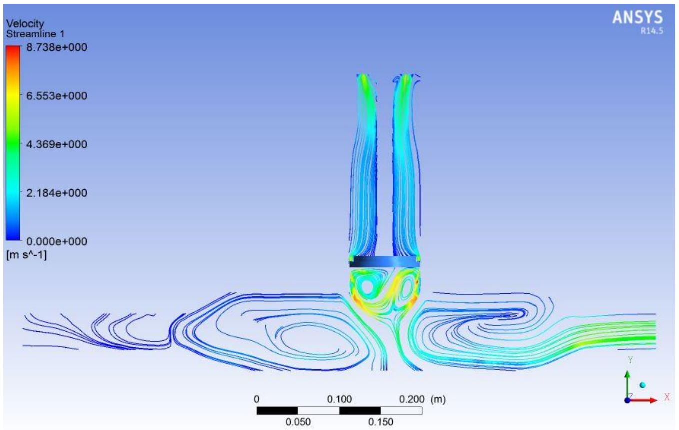

Fig. 36 Velocity Flow for Open Condition

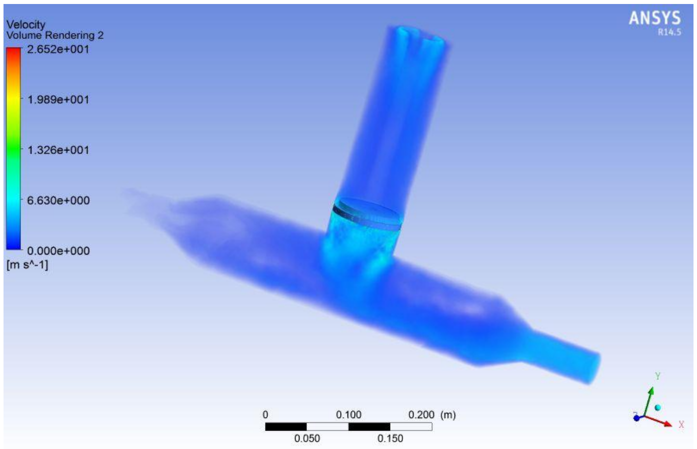

Fig. 37: Velocity Vector for Open Condition

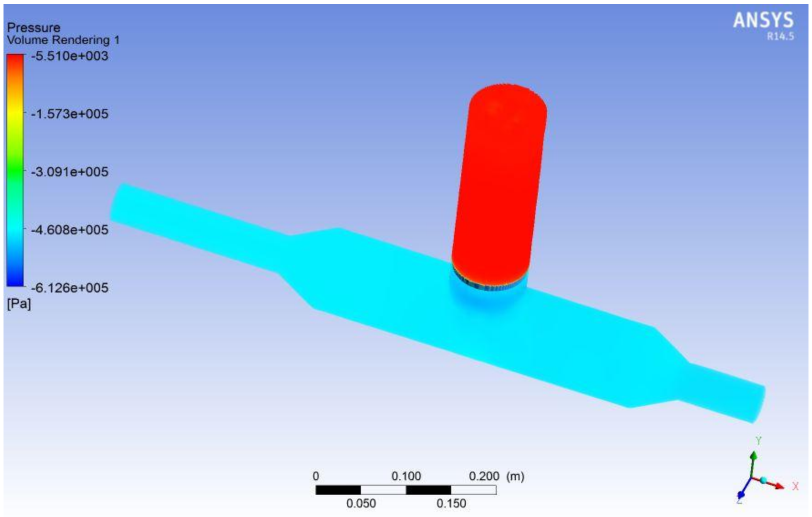

Fig. 38: Pressure Vector for Open Condition

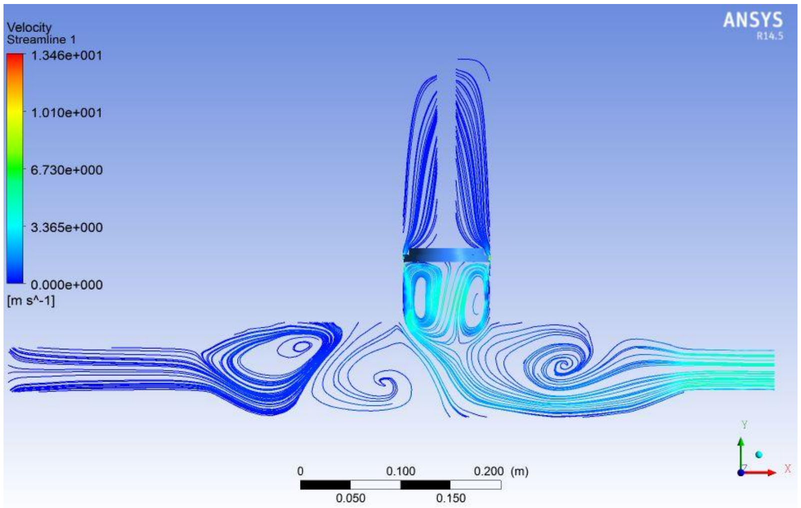

Fig. 39: Velocity Flow for Closed Condition

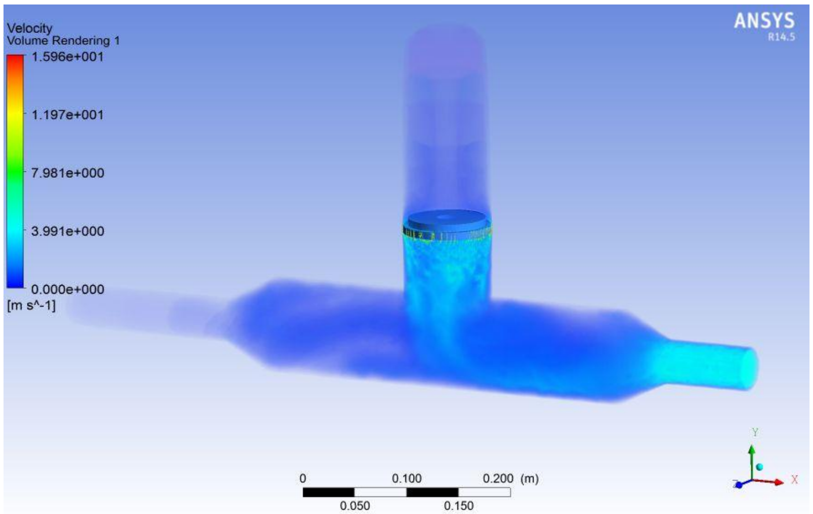

Fig. 40: Velocity Vector for Closed Condition

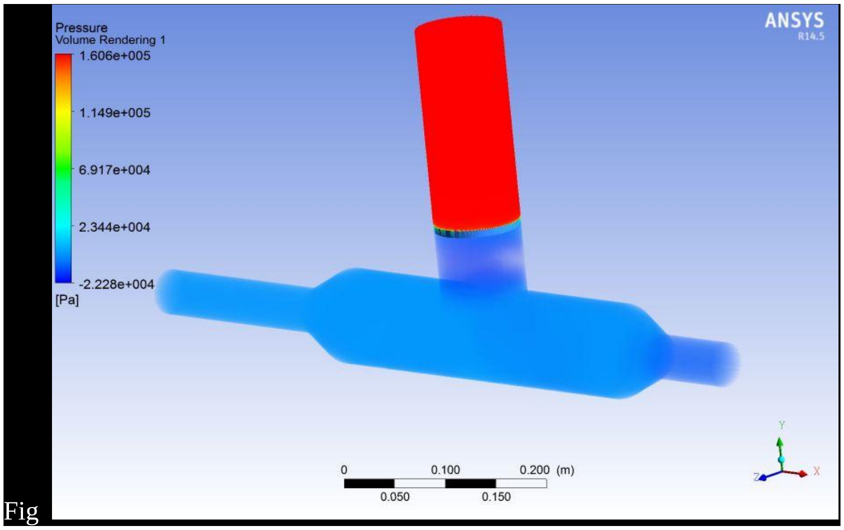

Fig. 41: Pressure Vector for Closed Condition

B. For Head 2m:

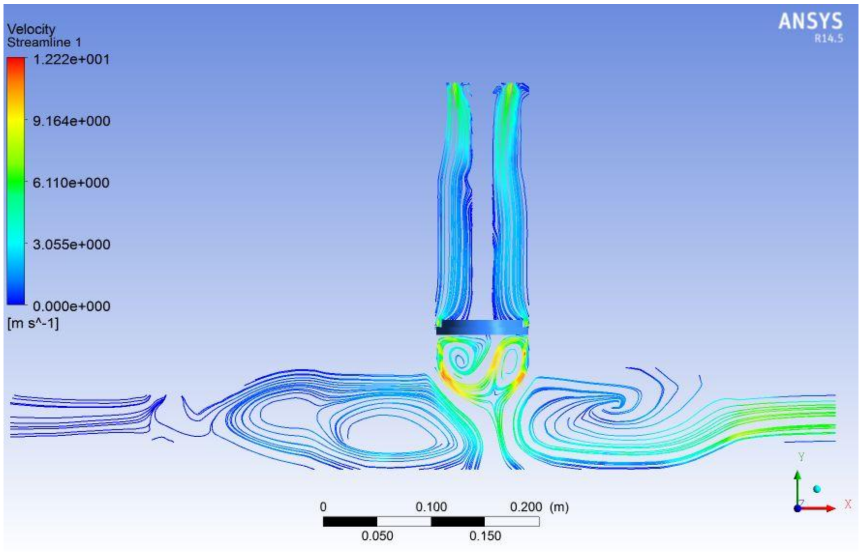

Fig. 42: Velocity Flow for Open Condition

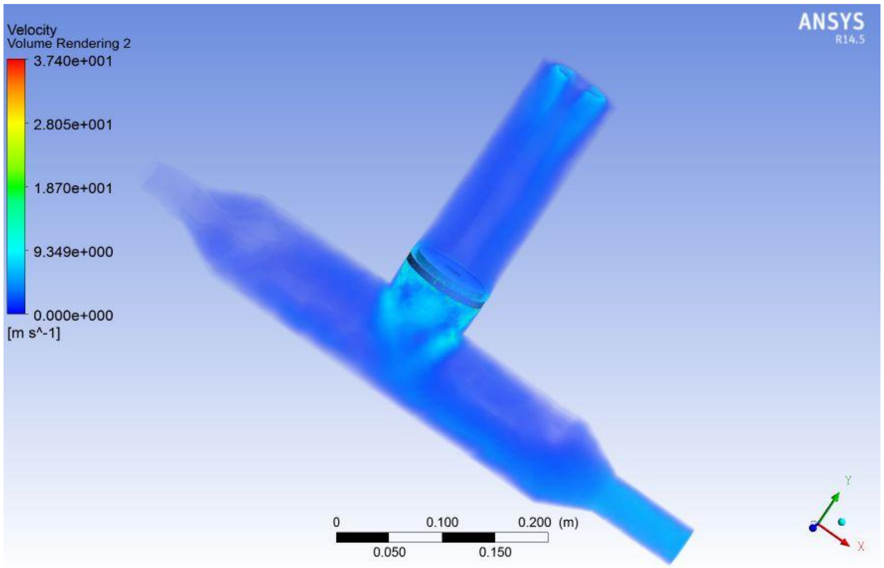

Fig. 43: Velocity Vector for Open Condition

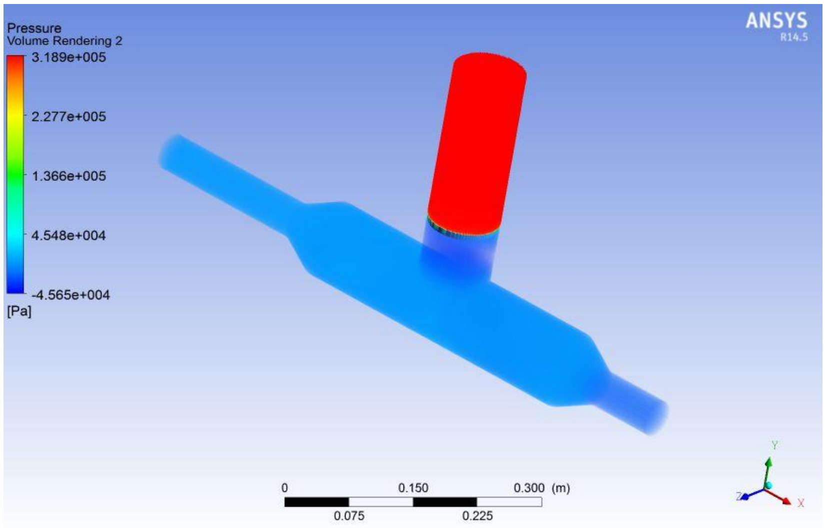

Fig. 44: Pressure Vector for Open Condition

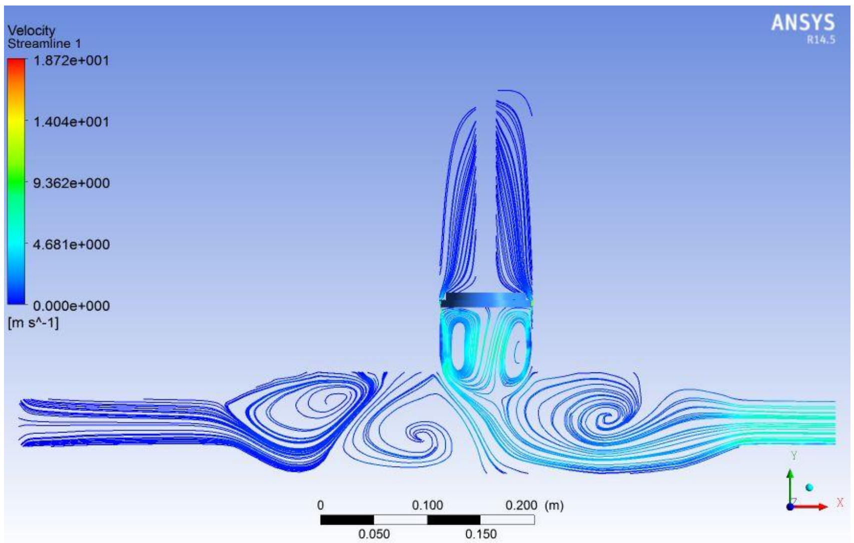

Fig. 45: Velocity Flow For Closed Condition

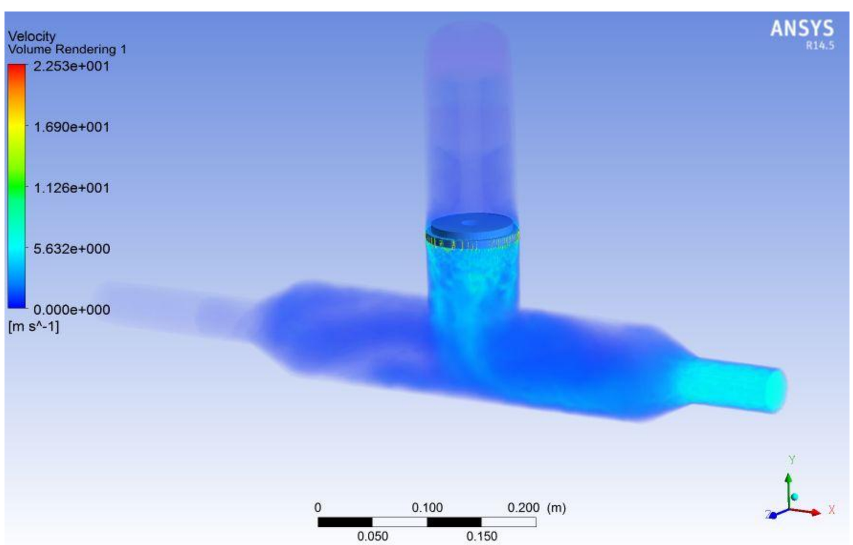

Fig. 46: Velocity Vector for Closed Condition

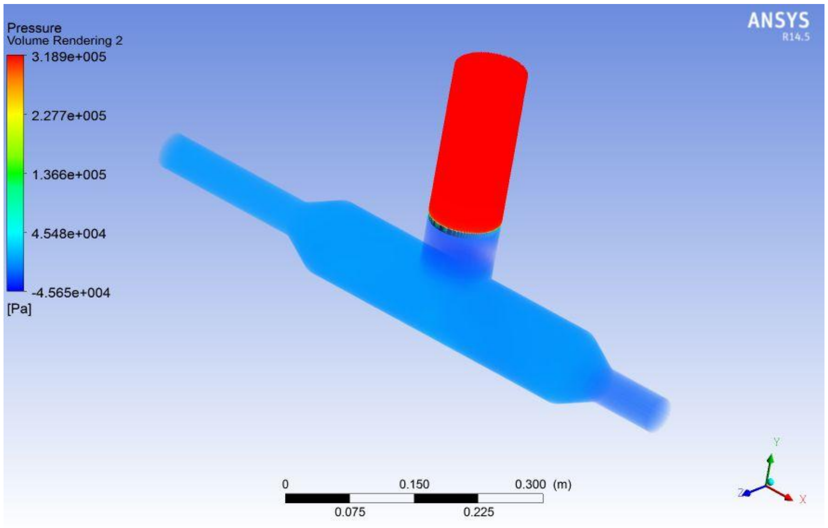

Fig. 47: pressure vector for closed condition

C. For Head 3m:

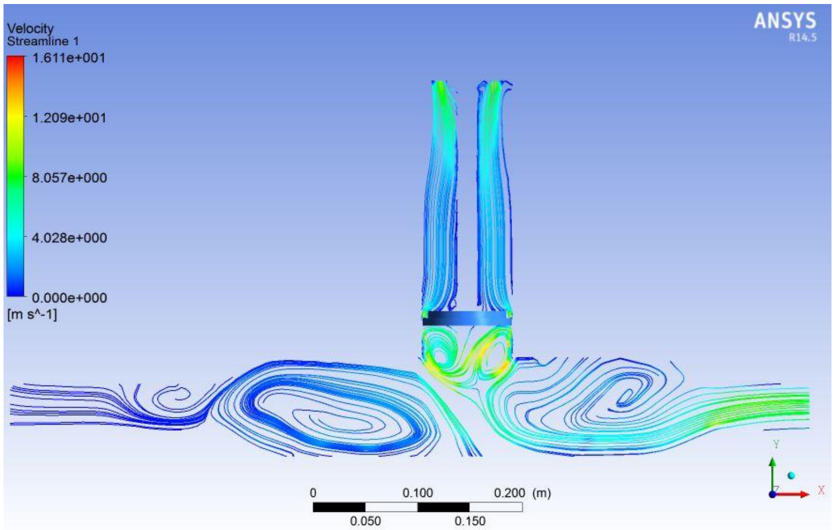

Fig. 48: Velocity Flow for Open Condition

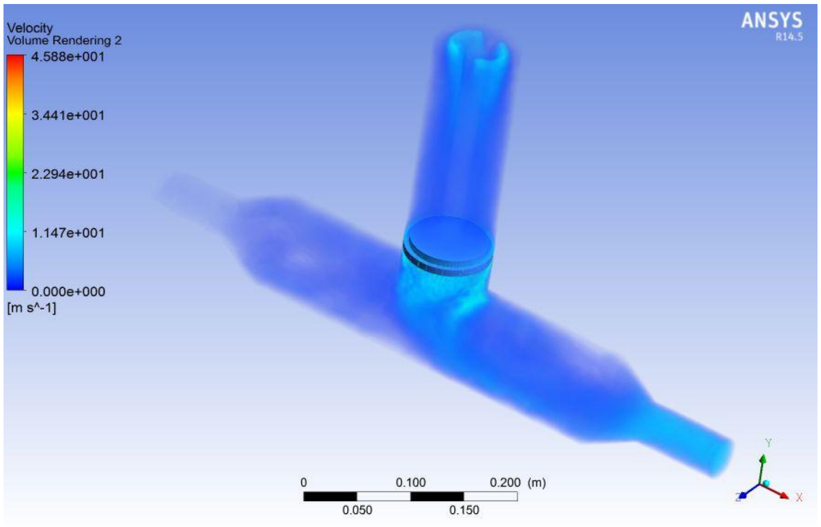

Fig. 49: Velocity Vector for Open Condition

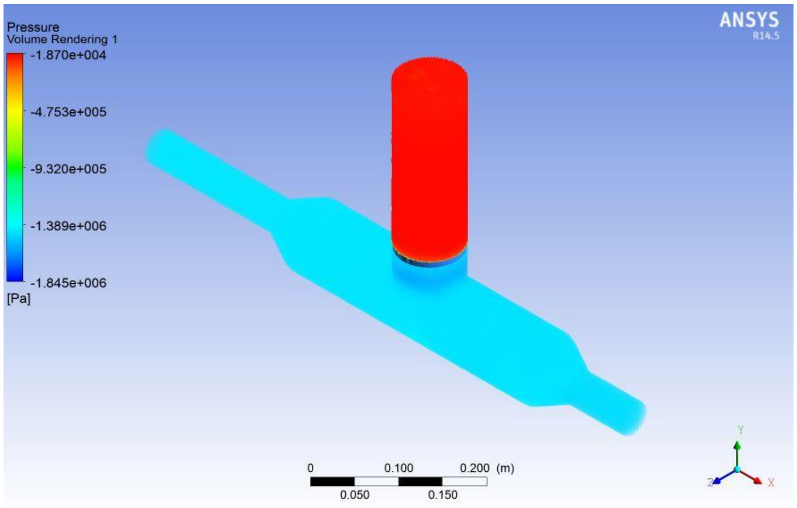

Fig. 50: Pressure Vector for Open Condition

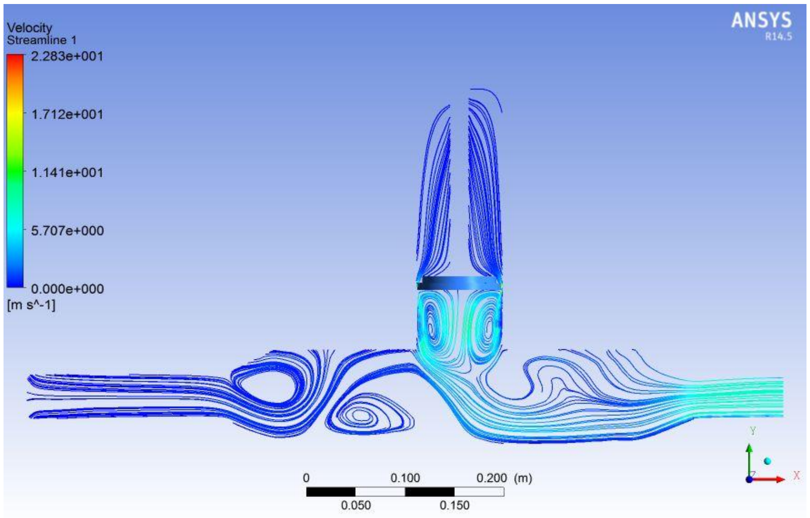

Fig. 51: Velocity Flow for Closed Condition

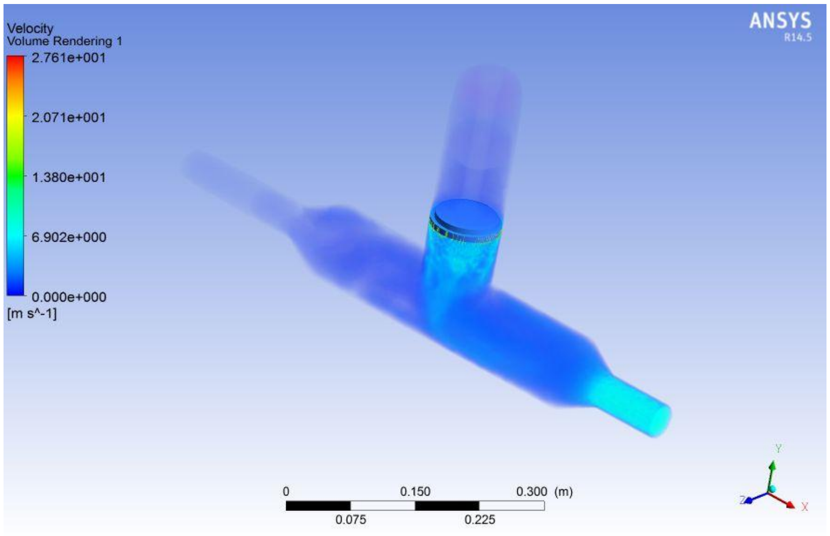

Fig. 52: Velocity Vector for Closed Condition

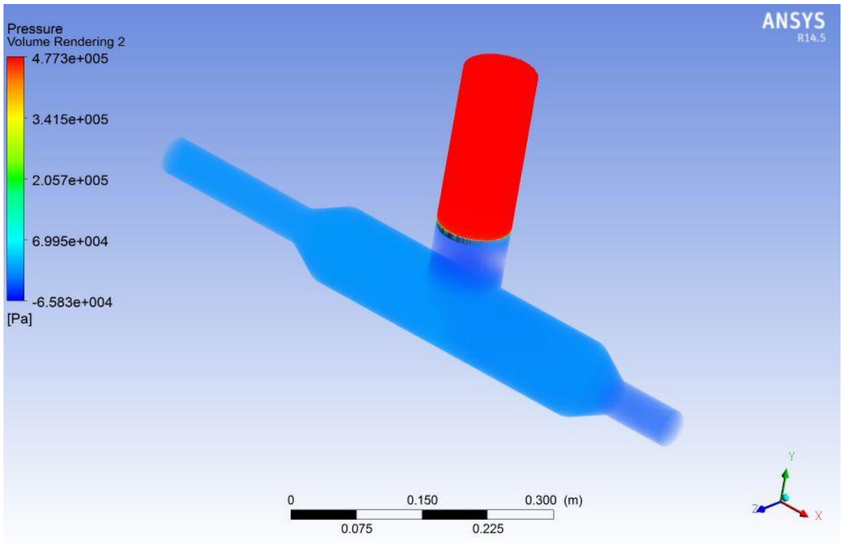

Fig. 53: Pressure Vector for Closed Condition

D. For Head 4m:

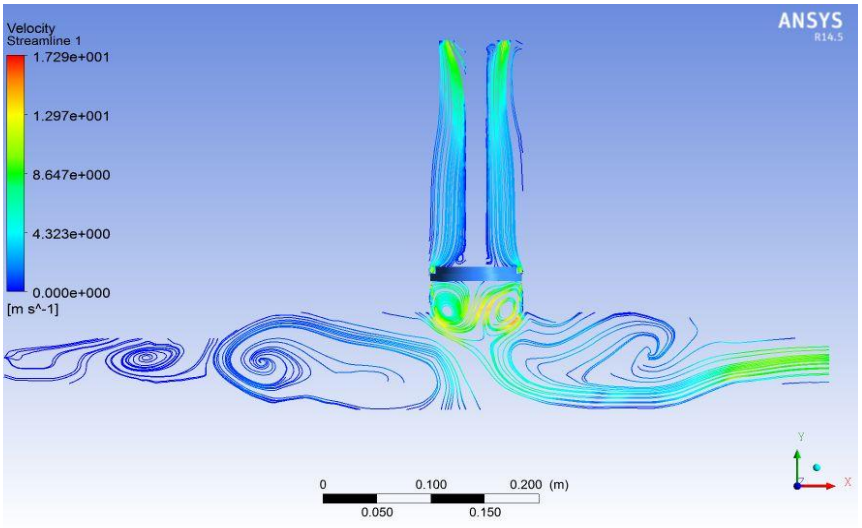

Fig. 54: Velocity Flow for Open Condition

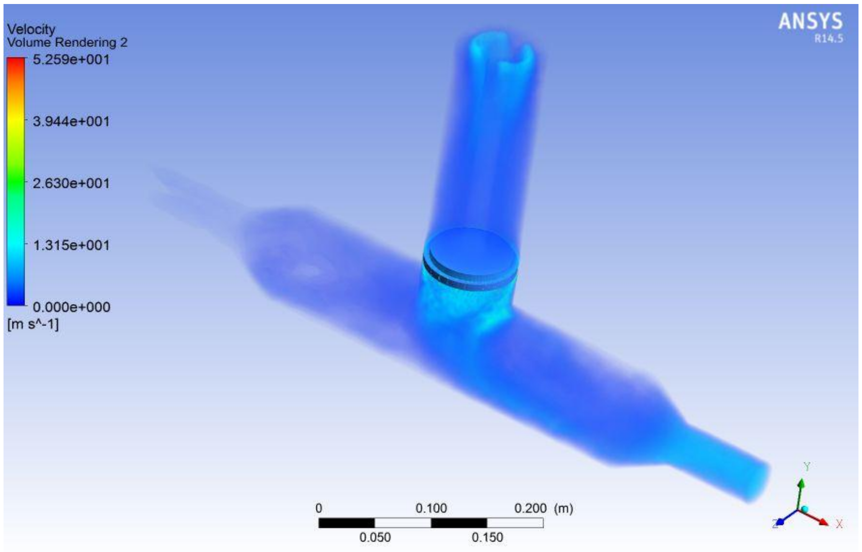

Fig. 55: Velocity Vector for Open Condition

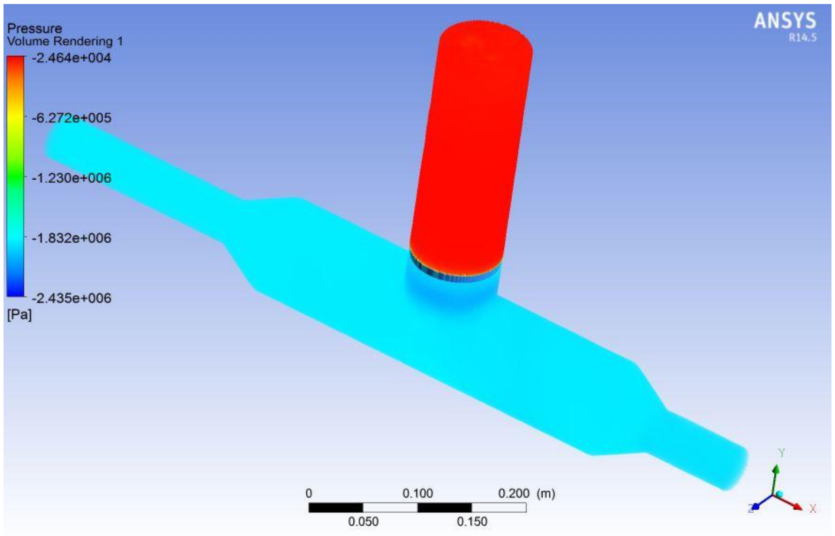

Fig. 56: Pressure Vector for Open Condition

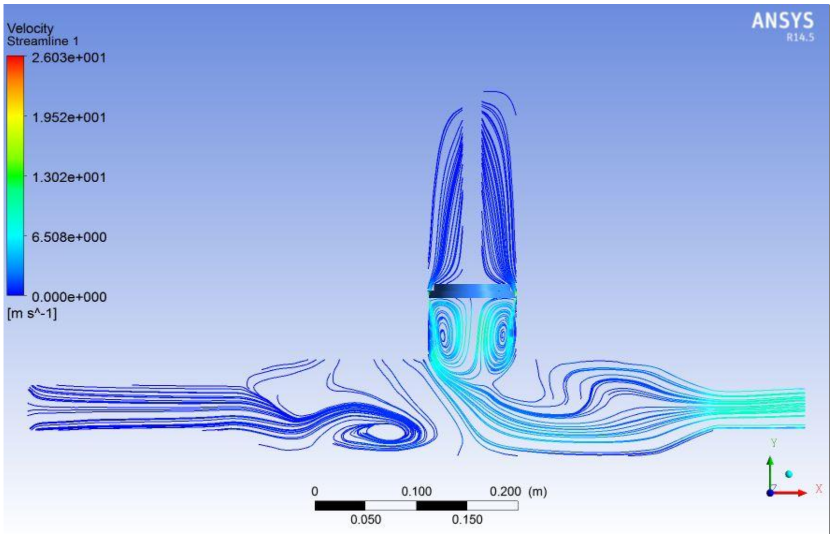

Fig. 57: Velocity Flow for Closed Condition

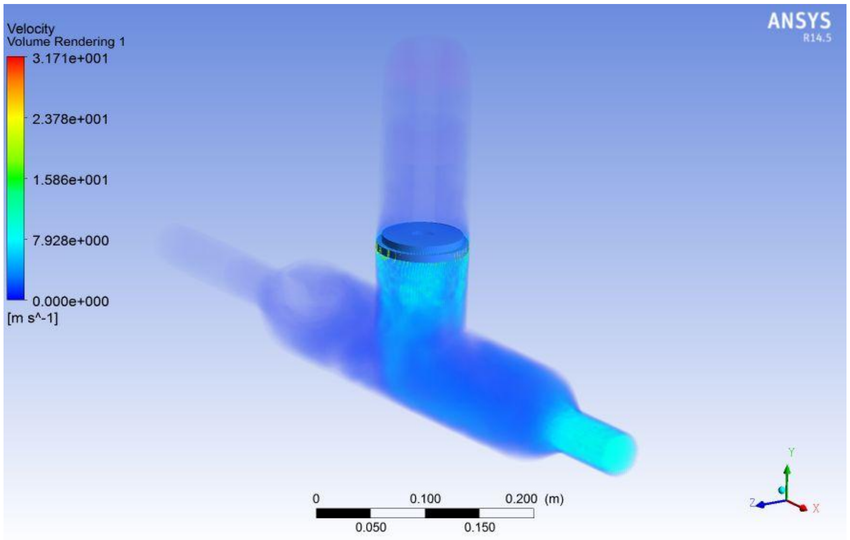

Fig. 58: Velocity Vector For Closed Condition

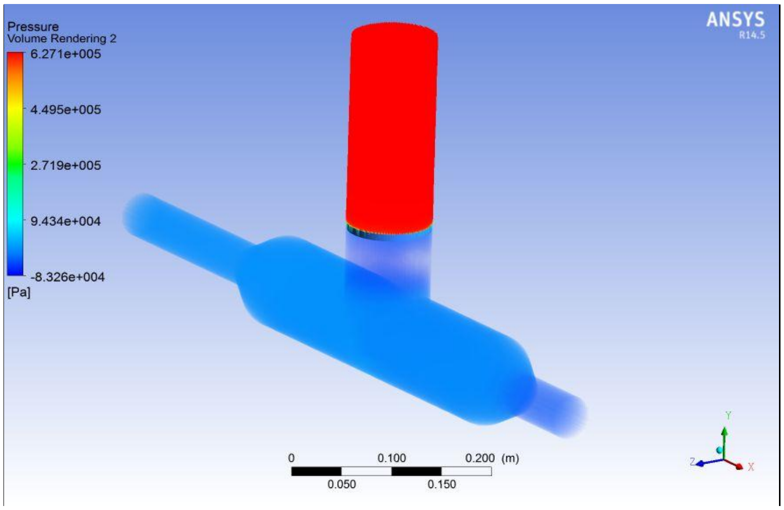

Fig. 59: Pressure Vector for Closed Condition

E. For Head 5m:

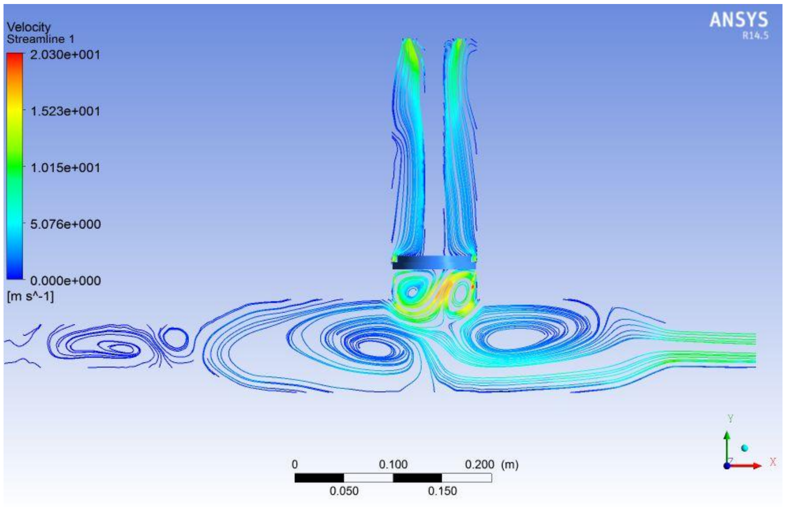

Fig. 60: Velocity Flow for Open Condition

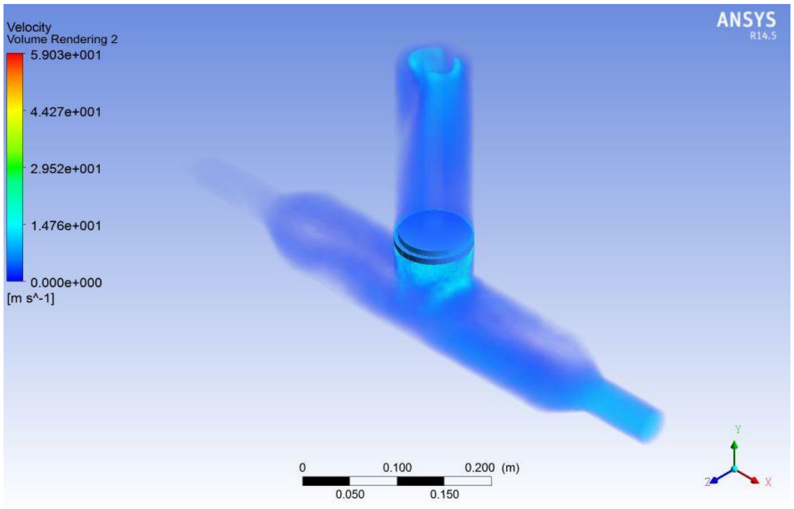

Fig. 61: Velocity Vector for Open Condition

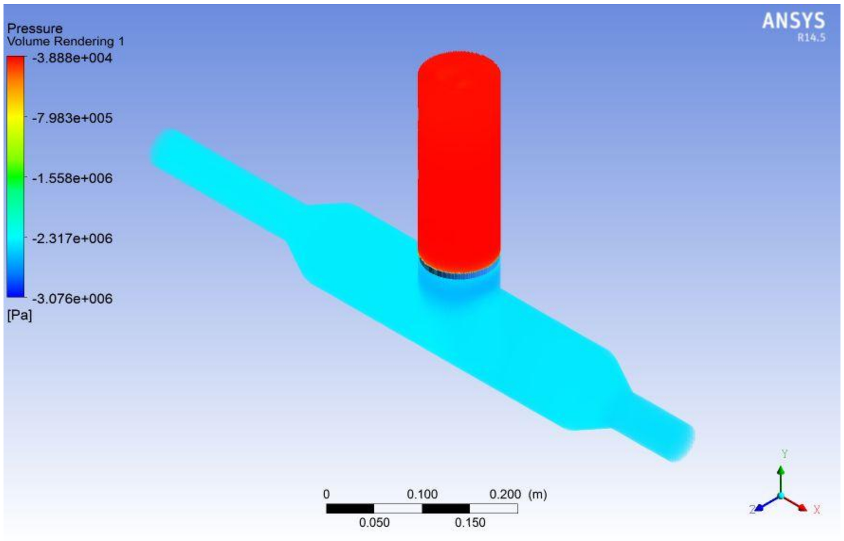

Fig. 62: Pressure Vector for Open Condition

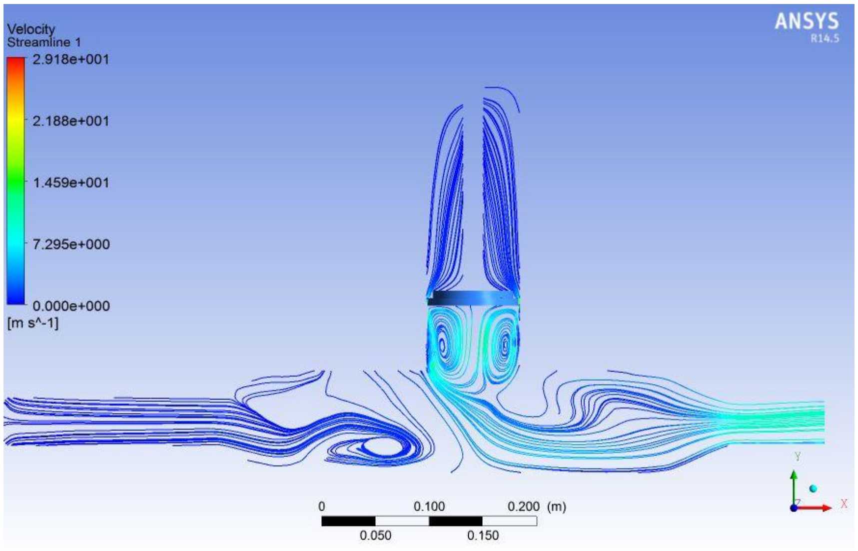

Fig. 63: Velocity Flow for Closed Condition

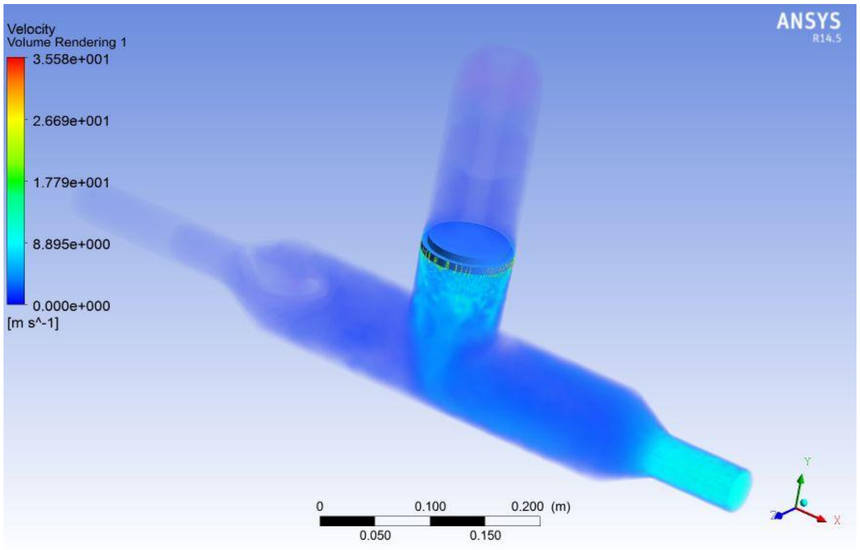

Fig. 64: Velocity Vector for Closed Condition

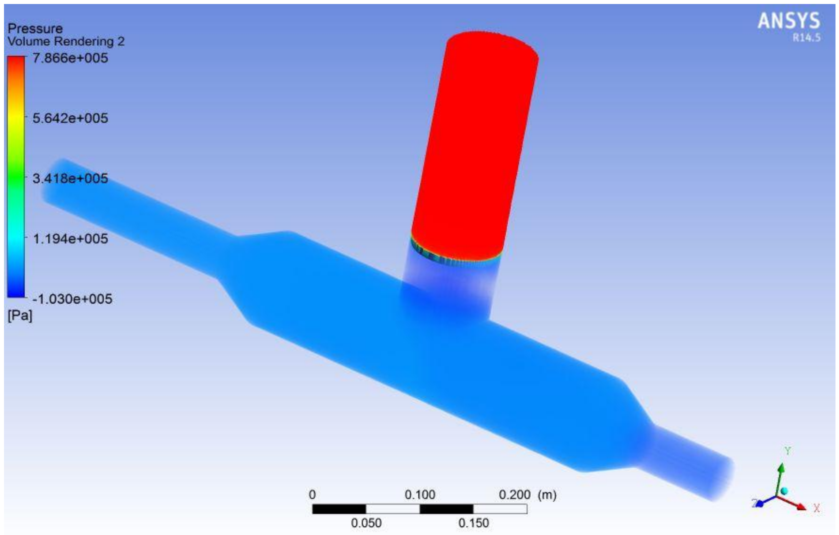

Fig. 65: Pressure Vector for Closed Condition

The value of mass flow rate through waste valve calculated with help of CFD after doing an enhancement work is numerically and comparatively less than that for the mass flow rate through waste valve of the past design model. These values of mass flow rate through waste valve is calculated for head 4m and it is found 0.0012 kg/seram increases as the wastage of water through the waste valve is decreases and the value is shown as follows : -

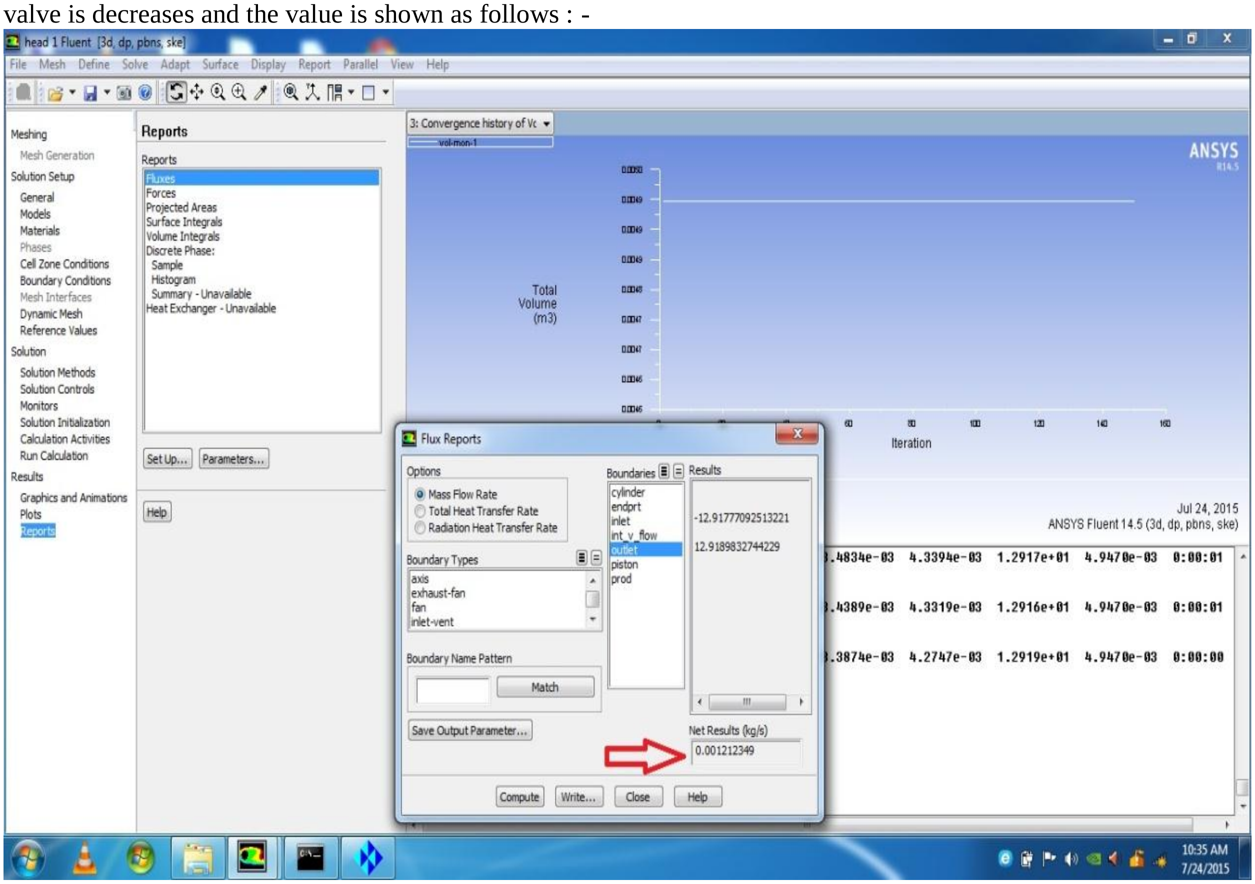

Fig. 66: Mass Flow Rate through Waste Valve of Enhancement Model

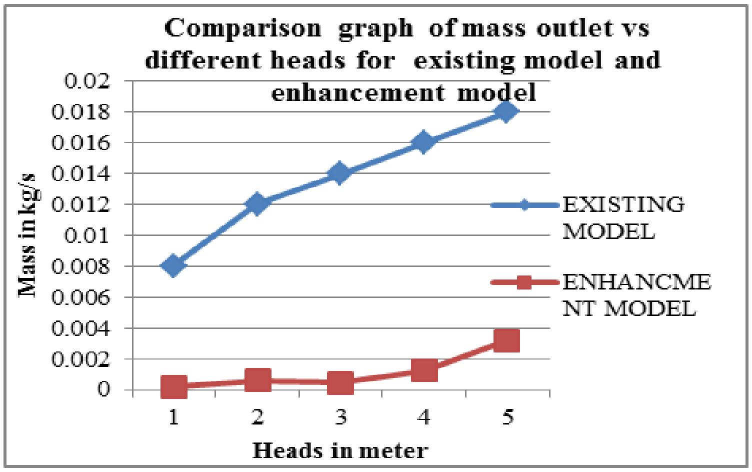

Fig. 67: Comparison of Mass Outlet Vs Different Heads for Existing Model and Enhancement Model

V. CONCLUSIONS

Using CFD, analysis of existing model and enhancement model is done. The mass flow rate through the waste valve is comparitevely less in enhancement model as compare to existing model. The new design is good to get higher output as compare to the old model of hydram. New design of hydram gives better performance than the old one.

REFERENCES

[1] Shuaibu Ndache Moammad, "Design and Construction of a Hydraulic Ram Pump",Leonardo Electronic Journal of practices and Technologies,ISSN 1583-1078,Issue 11,July-Dec.2007,P.59-70
[2] Ma Chi* and Hu Yinde, "Hydraulic ram,a device lifting water without conventional energy",Zhejiang University of Technology.
[3] Dr. Abiy Awoke Tessema, "Hydralic Ram Pump System Design and Application",ESME fifth annualconferenceon manufacturing and process industry,September2000
[4] D. Tulapona,"The Use of Hydra ram for water pumping in Tanzania."
[5] Oyuko 0. Mbeche, "The Hydralic ram pump in Kenya"
[6] W.T. Weercoon &V.Liyanaj,"Hydralic ram pump technology and practices in Zambia"
[7] E.J. Schiller,"Practical aspects of Hydram operation"
[8] Larry Caretto, "One Equation Of Computational Fluid Dynamics",california state university
[9] Sajid Hussein Ali Ali- Abbasi, "Cfd Analysis Of Enhancement Of Turbulant Flow Heat Transfer In A Horizontal Circular Tube With Different Inserts "European scientific journal , may 2014,edition vol.10,no.15
[10] J.D.Anderson,Jr, "Governing Equation of fluid Dynamics " National air and space museum, Smithsonian institution, Washington, DC
[11] Dmitri Kuzmin, "Introduction to Computational Fluid Dynamics".

