Flow Analysis of Water Jet Nozzle For Different Convergent Angles

G Srikanth Reddy[1], Basawaraj S Hasu[2]
[1, 2] Dept of Mechanical Engineering
[1, 2] AVN Institute of Engineering & Technology, Hyderabad, India

Abstract- *The water jet technology is the newest method for efficient cutting systems. The water jet nozzle uses the application like using high pressure water jet for cutting the materials efficiently than other conventional techniques which in case increases higher reduction of machining time. It uses water and abrasive material to cut hard materials like granite and metals. It is the preferred method when the materials being cut are sensitive to the high temperatures generated by other methods.*

Keywords- CFD Analysis, Nozzle design, Velocity and Pressure distribution, Abrasive material, Industrial application.

I. INTRODUCTION

Abrasive water jet (AWJ) machining is a process which is capable of cutting any material, with a good surface finish. The main characteristics of AWJ consist of negligibly low cutting force and very less thermal effects. The focus tube in a water jet cutting system accelerates the abrasive particles used for cutting. The performance of the abrasive jets increases as the fastness of the ABJ increases. Abrasive jets are made up of two continuous phases and a solid particle phase, which makes them much more complex than plain water jets.

The velocity distribution is a important parameter in abrasive water jet in terms of precision cutting. The most replaceable part in a water jet cutting head is the focusing tube, which helps the abrasive particles to mix with water and air. Erosion occurs due to high velocity and shear stress developed on the focusing tube wall. As a result, jet coherence decreases and the diameter of the focusing tube increases, which are undesirable in precision cutting.

II. METHODOLOGY

A water jet machine is used for material cutting purpose using the principle making higher velocity of water strikes the material to cut into two pieces. As per the working principle the machine needs to send the mixture of abrasive particles and water with greater velocity at outlet the nozzle must be more convergent providing best angular velocities and the nozzle needs to be designed with higher precision.
The main overview of the paper is to be designing of a water jet nozzle with changing of different parameters and conducting an computational fluid analysis on the nozzle to know the best design of which providing with greater velocities and pressure.

III. MODELING PART

The jet in the potential core region is applied to cutting, and jetting in the diffused droplet region is applied to dust-laying and aspirating; whereas the jet in the main region, the focus of this study, is applied to cleaning and surface finish.

A nozzle shown in Figure1 is widely used in engineering projects; however higher velocity, higher dynamic pressure, lower pump pressure and lower cost are necessary considerations. The optimization of nozzle structure offers a simple solution to these.

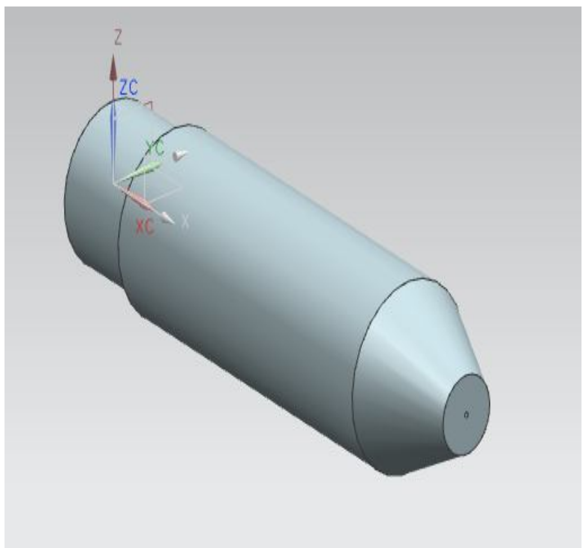


Figure 1. Isometric view of water jet nozzle

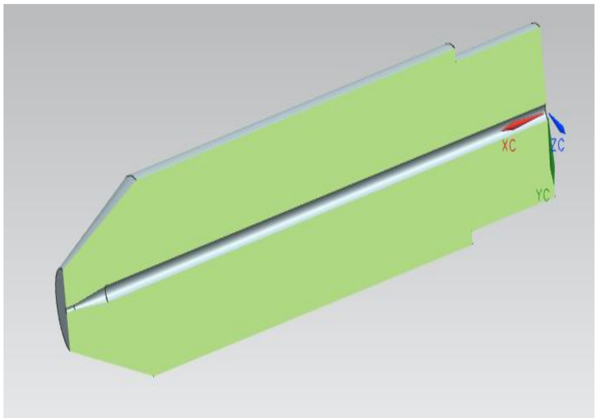


Figure 2. Sectional vie showing the outer and inner part of the fluid entrance and discharge sections.

IV.FLOW ANALYSIS OF WATER JET NOZZLE

1. COMPUTATION FLUID DYNAMICS:

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations:

- Conservation of mass
- Conservation of momentum
- Conservation of energy
- Conservation of species

The results of CFD analyses are relevant in:

- Conceptual studies of new designs
- Detailed product development
- Troubleshooting
- Redesign

CFD analysis complements testing and experimentation. Reduces the total effort required in the experiment design and data acquisition.

2. HOW DOES CFD WORK?

FLUENT solvers are based on the finite volume method.

- Domain is discretized onto a finite set of control volumes (or cells).
- General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes.

Partial differential equations are discretized into a system of algebraic equations. All algebraic equations are then solved numerically to render the solution field.

3. CFD ANALYSIS ON NOZZLE FOR 13 DEGREES OF ANGLE:

The CFD analysis on nozzle is conducting Ansys 16 .0 version.

First on opening Ansys workbench fluid flow(fluent) is chosen for doing analysis. After that created a fluid flow in the first step and a geometry is selected to conduct analysis, For this there are two steps either created the model in Ansys workbench or else you can import an external geometry which is created in other cad software's.

In this paper 2nd step had been chosen and created fluid model in Nx-cad and then imported into Ansys workbench.

FINITE ELEMENT MODEL:

Now the generated fluid model is converted into Ansys xt file format for conducting CFD analysis on the nozzle. The Figure.3 shows the 3d fluid model which is imported into as external geometry

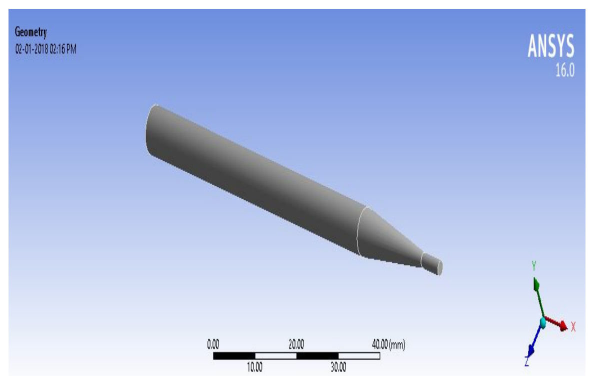


Figure 3. shows the 3D fluid model

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

After meshing, the number of Nodes are 8024 and Elements are 6480, the meshed fluid model can seen in Figure.4.

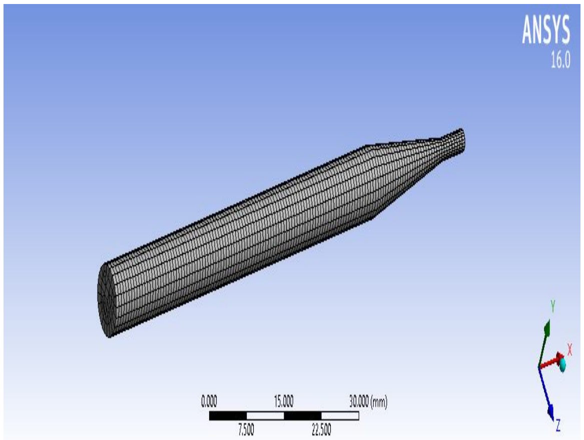


Figure 4. Meshed fluid model.

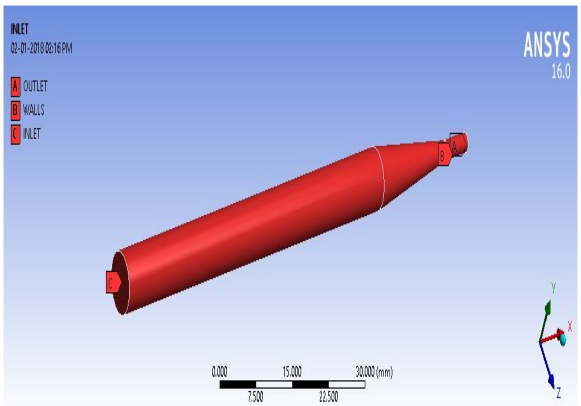


Figure 5. Inlet selection

V. BOUNDARY CONDITIONS

The boundary conditions are taken up by doing theoretical calculations, from the theoretical calculations considering convergent angle as 13 degrees and applying in the momentum equation

The two velocities as 28m/s and 142 m/s according to the theoretical calculations. Now considering the same inlet velocity as 25 m/s and pressure as constant temperature and applied on the fluent model.

RESULTS FOR 13 DEGREES OF FLUENT MODEL:

The results are plotted for the fluent model with 13 degrees of convergent angle.

From the figure.6. the convergent history on y axis and the iterations on x axis, which the solution is converged at 79th of iteration that means the results are been perfectly converged for flow analysis of high pressure water inside the nozzle.

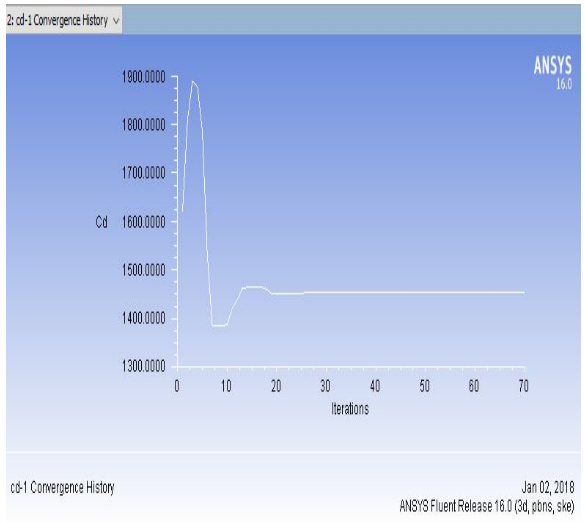


Figure 6. Convergence history for 13 degrees of fluent angle model.

The figure.7 shows the pressure output can seen clearly the color coding, From the figures it can see that at inlet the pressure is 34.2 MPa and the outlet pressure is 32.78 MPa. So, there is no such difference in outlet and inlet pressures.

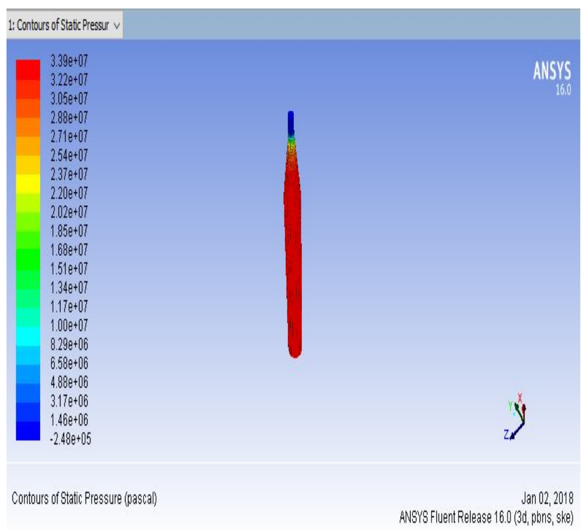
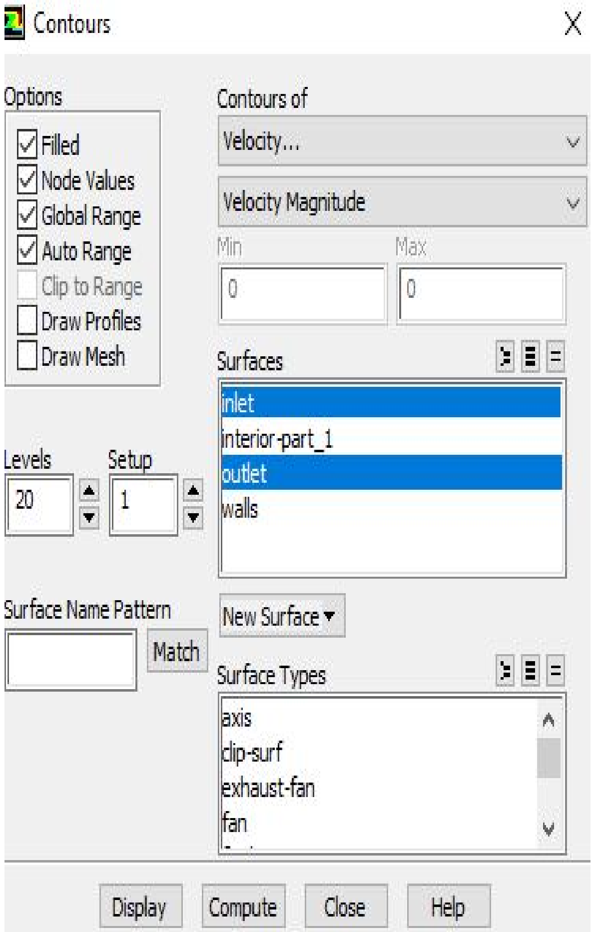


Figure 7. Pressure results at inlet and outlet.

Now the procedure is followed for seeing the velocity at outlet which the inlet as 25 m/s. On applying 25 m/s the velocity magnitude at discharge is seen as 255 m/s and as net velocity is 45 m/s, so it can say the nozzle can useful for industrial applications but as our project we should extend by changing different parameters of angels.



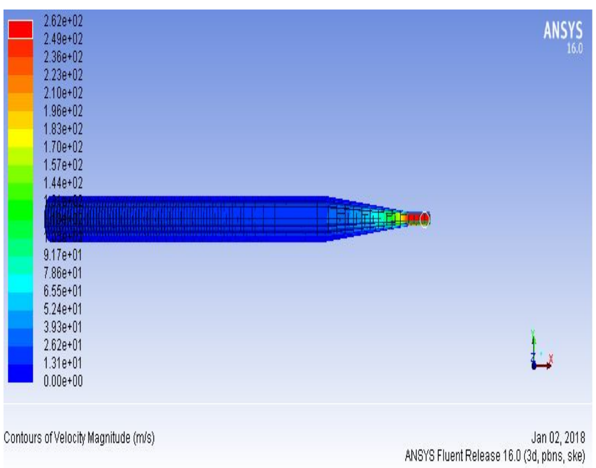


Figure 8. Velocity results at inlet and outlet for fluent angle 13 degrees.

Now creating the streamline with input we can clearly estimate whether the results converge or not perfectly. This streamlines are created by choosing velocity input as shown in Figure.9

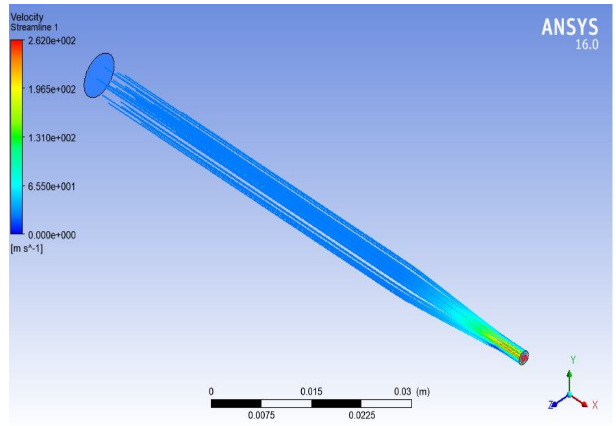


Figure 9. Velocity input

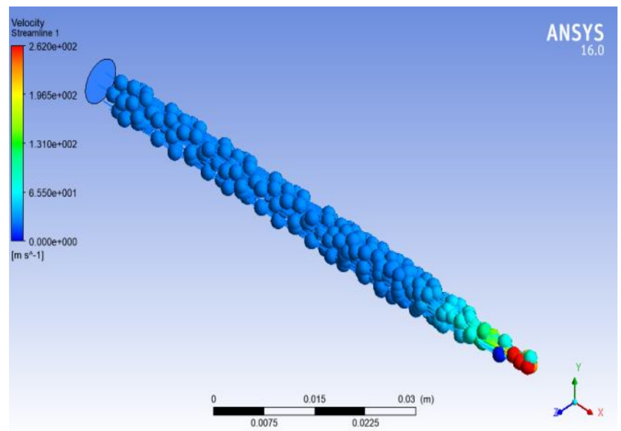


Figure 10. Bubble structures showing the stream line of water discharging from the nozzle.

CFD ANALYSIS ON NOZZLE FOR 14 DEGREES OF ANGLE:

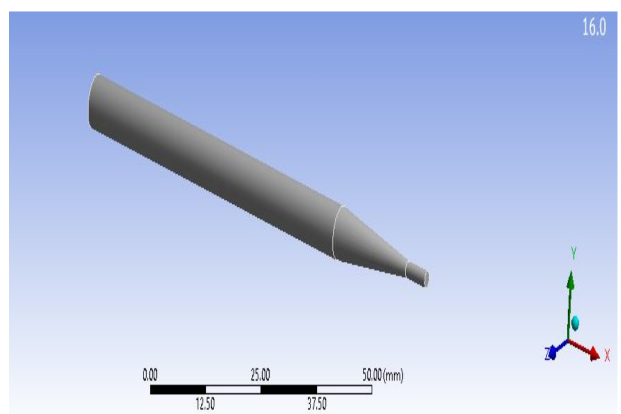


Figure 11. Ansys model which is used for analysis for 14 degrees.

The named selections are created by creating new selections as inlet, walls and outlet sections.

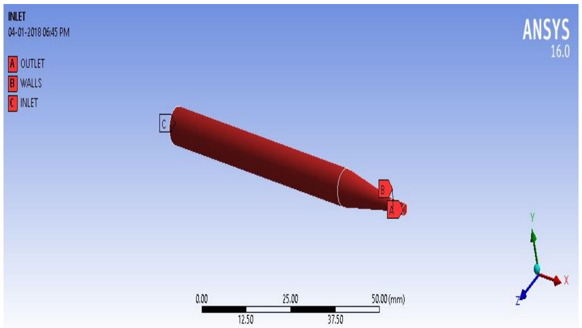


Figure 12. Inlet, walls and outlet sections

RESULT FOR 14 DEGRESS OF CONVERGENT ANGLE:

But the pressure outlet varies with -1mpa than the convergent angle with 13 degrees. The results of pressure inlet are 33.44 MPa. This result can be seen clearly in Figure.13.

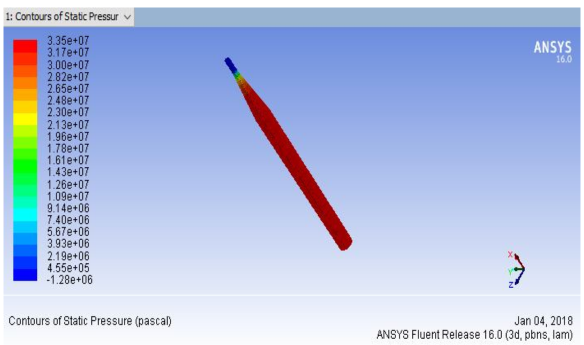


Figure 13. The results of pressure inlet are 33.44 MPa.

CFD ANALYSIS ON NOZZLE FOR 15 DEGREES OF ANGLE:

The same above process is used for designing the fluid model with convergent angle 15 degrees. The Fig.14 shows clearly the angle.

The Ansys procedure is same for this fluid model even the boundary conditions are also similar like 15 degrees angle input.

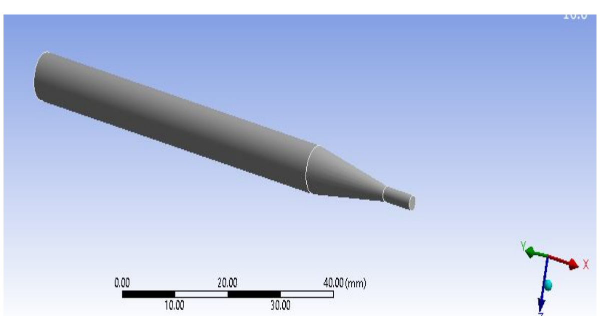


Figure 14.Fluid model with convergent angle 15 degrees

The named selections are created by creating new selections as inlet, walls and outlet sections. The Figure.15 shows clearly about the named selections.

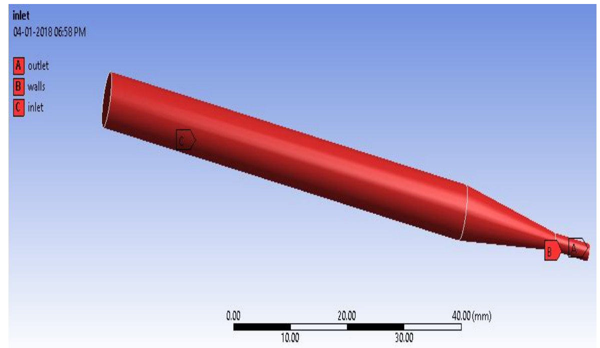


Figure 15 The named selections

RESULT FOR 15 DEGRESS OF CONVERGENT ANGLE:

From the results it seen that the velocity outlet is similar to the convergent angle 15 degrees. The velocity outlet is 255 m/s. The velocity gradient can be seen in figure. From all the results it is observed that changing of angle is not affecting a great change of velocities and pressures for the nozzle. As per the objective of the project changing of different convergent angles the results varies like 255 m/s and pressure values as on average 34 MPa.
To change this affect the inlet diameter had changed from 6.4mm to 7mm and the boundary conditions are followed to get the results.

CFD ANALYSIS ON NOZZLE FOR 13 DEGREES OF ANGLE WITH 7MM INLET DIAMETER:

The same above process is used for designing the fluid model with convergent angle 13 degrees. The below fig 7.1 shows clearly the angle.

In changing the inlet diameter from 6.4 mm to 7 mm the fluid model is created can seen in figure.16.

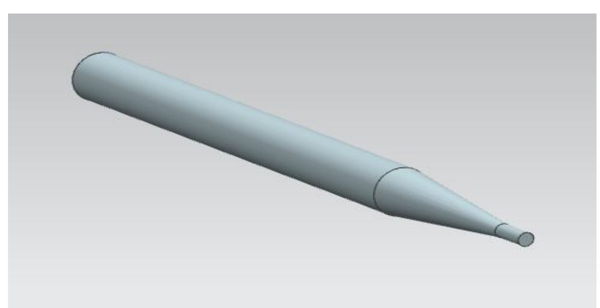


Figure 16. 3d Fluid model for 7 mm inlet diameter with 13 degrees angle.

RESULT FOR 13 DEGRESS OF CONVERGENT ANGLE WITH 7MM INLET DIAMETER:

Figure 17. shows the velocity gradient for the increased inlet diameter of 7mm.

Foe the inlet velocity 25 m/s the discharged velocity is about 318 m/s. So, by this result we can say the velocity is increased 50 percent more than the previous results.

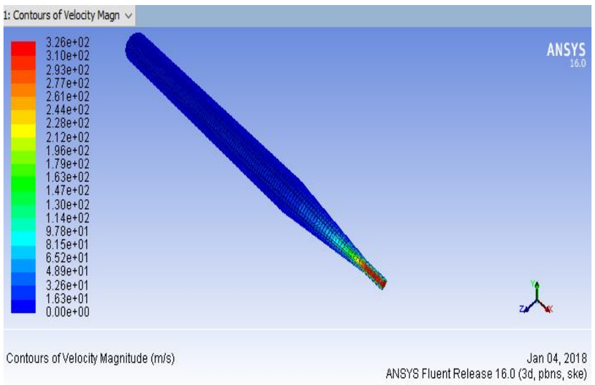


Figure 17. shows the velocity gradient for the increased inlet diameter of 7mm.

VI. RESULT

Convergent angle	13 degrees	14 degrees	15 degrees
Velocity (m/s)	255	255	255
Pressure (MPa)	34.21	33.44	34.12

INTELET DIAMTER 7MM		
Convergent angle	13 degrees	14 degrees
Velocity (m/s)	318	306
Pressure (MPa)	50.27	48.62

CFD analysis is performed on the nozzle for an inlet fluid velocity of 25m/s and the results are documented. From the above analysis it can be observed that the maximum velocity of 318m/s is flowing at the convergent side of the nozzle and a maximum pressure of 50.27 MPa acting at the divergent side of the nozzle.

VII. CONCLUSION

CFD analysis is the most important aspect to determine outlet velocity for a nozzle. For any flow analysis to perform, inlet parameters and outlet parameters (Boundary conditions) play the most crucial role. Creating a prototype and then performing flow analysis on the part to achieve desired exit velocity may or may not be realized. CFD flow analysis is performed on the nozzle for different inlet and outlet diameters in Ansys workbench to achieve satisfactory results.

REFERENCES

[1] Vimal Kumar Pathak and Sumit Gupta, “Study of Nozzle Injector Performance Using CFD”, International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.4, No.3, August 2015.

[2] Amir Khalid, “Computational Fluid Dynamics Analysis of High Injection Pressure Blended Biodiesel”, International Research and Innovation Summit (IRIS2017).

[3] Chirag J.Padhya, “CFD Analysis & optimization of fuel injector by changing its geometry”, IJIRST –International Journal for Innovative Research in Science & Technology, Volume 1 , Issue 7 , December 2014.

[4] K.S.Sai Krishna, Kasanagottu Shouri, “Fuel Flow Analysis of Injector Nozzle”, International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 5, September 2013.