

# Modelling and Simulation of Nozzle Used In Hydraulic Turbine

**Kiran Rambhau Wankhede**  
Guided by **Prof.P.V.Kulkarni**

*Master of Technology*

*Department of Mechanical Engineering, Deogiri Institute of Engineering and Management Studies,  
Aurangabad*

*Affiliated to Dr. Babasaheb Ambedkar Technological University, Lonere, Raigad, Maharashtra, India*

Submitted: 15-10-2021

Revised: 26-10-2021

Accepted: 28-10-2021

**ABSTRACT**— This project mainly focused on design of hydraulic nozzle by applying CATIA software. And simulation applying computational fluid dynamics analysis by using formulation method. The design can be altered till the best performance and desired output is obtained. CFD can provide the solution for different operating condition and geometry configuration in less time and found very useful design and development. In Hydraulic turbine the energy of flowing water needs to be converted into kinetic energy.

**Index Terms**-ANSYSFLUENT.ConvergentNozzle.

## I. INTRODUCTION

A Nozzle is a device designed to control the direction or characteristics of a fluid flow as it exits an enclosed chamber or pipe via an orifice. The total energy at the end of the pipe consist of pressure energy and kinetic energy. Nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic

energy. Nozzle is regulated by the governer via mechanical-hydraulic or electro-hydraulic controls. Hydraulic Turbine is a device that converts the energy in a stream of fluid into mechanical energy .

□ Types of Hydraulic turbine:

1. Impulse turbine
2. Reaction turbine
3. Pelton wheel turbine
4. Francis turbine
5. Kaplan turbine

Nozzle is specific component used in all these turbine.

□ Types of Nozzles

1. Aero spike nozzle
2. Liquid nozzle
3. High speed gas nozzle
4. Propelling nozzle
5. Magnetic nozzle.

❖ We are study on Liquid nozzle.



Fig 1. Hydraulic Nozzle

## II. AIM AND OBJECTIVES

- Aim  
 Modelling and Simulation of Hydraulic turbine Nozzle by using CATIA Software and ANSYS Fluent.
- The Objective of study is\_
  - To predict the flow behaviour of through Nozzle
  - To investigate the best suited nozzle for Hydraulic Turbine.

## III. THEORETICAL FORMULATION OF NOZZLE

Function of nozzle is to regulate the flow of water to the runner in an impulse turbine runner. Nozzle is used to high velocity of turbine .Main application of nozzle covers majority of the engineering, chemical and medical fields. Throughout the years, engineer’s have found many ways to use the force that can be imparted by a jet

of fluid on a surface diverting the flow. For examples, Fireman can make use of the kinetic energy stored in a water jet to deliver water above the level in a nozzle to extinguish fires in high-rise buildings.

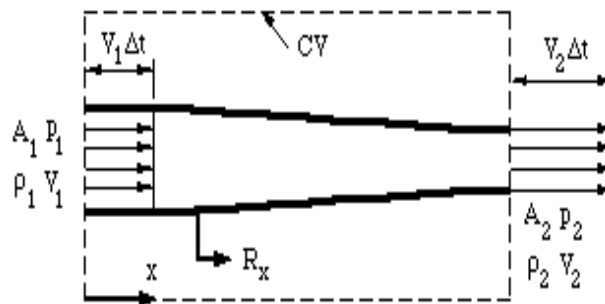
### 3.1. POSITION OF HYDROULIC TURBINE:

The equations used below are for one dimensional nozzle flow. It corresponds to the idealization and simplification of flow equations.

Nomenclature of symbols used is as follows:

- P – Pressure (Pa)
- V – Velocity (m/s)
- g – Gravitational acceleration (m/s<sup>2</sup>)
- z – Height (m)
- A – Area (m<sup>2</sup>)
- γ – Adiabatic index (Cp/Cv)
- h – Enthalpy (J)
- ρ – Density (kg/m<sup>3</sup>)
- m– Mass flow rate (kg/s)

Fig.5 1D flow nozzle



### 3.2. Continuity Equation used in nozzle:

Now we apply the principle of mass conservation. Since there is no flow through the side walls of the duct, what mass comes in over A<sub>1</sub> goes out of A<sub>2</sub>,

$$\text{Volume flow in over } A_1 = A_1 V_1 \Delta t$$

$$\text{Volume flow out over } A_2 = A_2 V_2 \Delta t$$

Therefore,

$$\text{Mass in over } A = \rho A_1 V_1 \Delta t$$

$$\text{Mass out over } A = \rho A_2 V_2 \Delta t$$

So,

$$\rho A_1 V_1 = \rho A_2 V_2$$

Where,

- ρ = mass density of the fluid,
- V<sub>1</sub>& V<sub>2</sub> = Flow velocity at inlet and outlet.
- A<sub>1</sub>& A<sub>2</sub> = cross-sectional area at inlet and outlet.

This equation is called the continuity equation for steady one-dimensional flow.

### 2 Bernoulli’s equation

$$P + \rho + \rho gh = P + \rho + \rho gh$$

Where, P<sub>1</sub> and P<sub>2</sub> are the inlet and outlet pressure of the fluid V<sub>1</sub> and V<sub>2</sub> are the inlet and outlet velocity of the fluid. As the pressure decreases in the nozzle with increase in the velocity because they both are inversely proportional. Their relation can be deduced by the Bernoulli's equation.

### 3.3 Mass flow rate

$$\text{Volume flow rate } Q = AV \dots \quad \text{m}^3/\text{s}$$

$$\text{Mass flow rate } m = \text{density} \times \text{Volume flow rate}$$

$$= \rho \times A \times V \quad \text{Kg/s}$$

Where,

$V$  or  $Q$  = Volume flow rate,  
 $\rho$  = mass density of the fluid.

$v$ =Flow velocity of the mass elements,  $A$  = cross-sectional vector area/surface,

#### IV. CFD BASED SIMULATION OF NOZZLE

##### 4.1. Simulation

ANSYS Software used for simulation

Steps of Simulation

1. Modelling from CATIA.
2. Meshing using ANSYS CFD Software.
3. General-1] Solver type-pressure based It's used for Dense Material
4. Energy Equation On Viscous Model

Standard K-E Model for non-Compressible

5. Boundary Condition.

6. Axis

Axis-Symmetry for cylindrical part

7. Solution Method.

##### 4.2. Methodology

###### 4.2.1. Study on Liquid Nozzle

Modeling of nozzle for different dia.20mm, 18mm, 22mm, 16mm. The modeling of the nozzle was done using CATIA Software in Part design .The Dimension of Nozzle is presents in the table given bellow. In this work solid work simulation is used for a meshing of nozzle. It creates sufficient smooth meshing as shown in fig. Bellow. Selections of Nozzles are the convergent Nozzles Because of operating pressure ratio and with different nozzle profiles.

###### 4.2.2. Dimensions And Modeling Nozzle:

Table I (Nozzle Dimensions)

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	60
2	Inlet diameter (mm)	40
3	Outlet diameter (mm)	20

###### 1] Analytical Design of Nozzle in CATIA



Fig.6. Isometric View of Nozzle 20mm Dia



Fig.7. 2D View of Nozzle 20mm Dia

Table 2. (Nozzle Dimensions)

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	60
2	Inlet diameter (mm)	40
3	Outlet diameter (mm)	18



Fig.8. Isometric View of Nozzle 18mm Dia

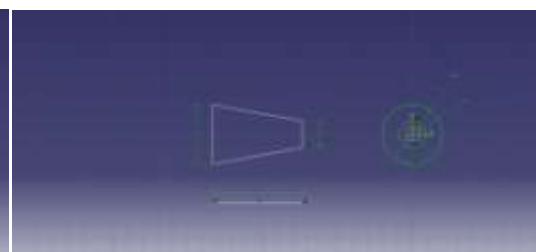


Fig.9. 2D View of Nozzle for 18mm Dia

Table 3 (Nozzle Dimensions)

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	60
2	Inlet diameter (mm)	40
3	Outlet diameter (mm)	22



Table 4. (Nozzle Dimensions)

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	60
2	Inlet diameter (mm)	40
3	Outlet diameter (mm)	16

Fig.12. Isometric View of Nozzle 16mm Dia

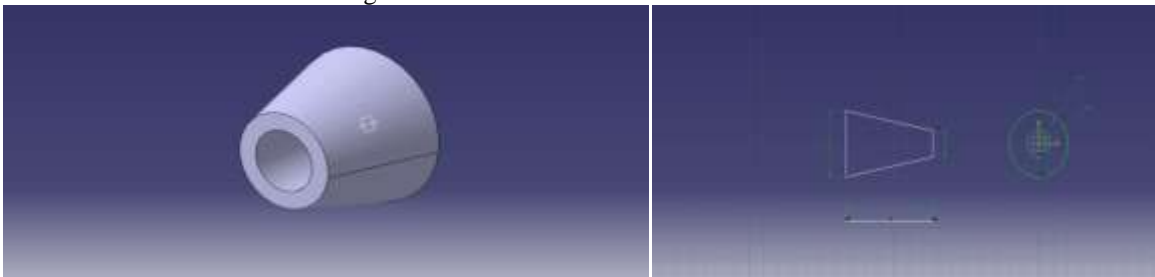


Fig.12. Isometric View of Nozzle 16mm Dia

Fig.13. 2D View of Nozzle for 16mmDia

**4.2.3. Meshing**

After modeling of the nozzle, its meshing was done Using ANSYS CFD software

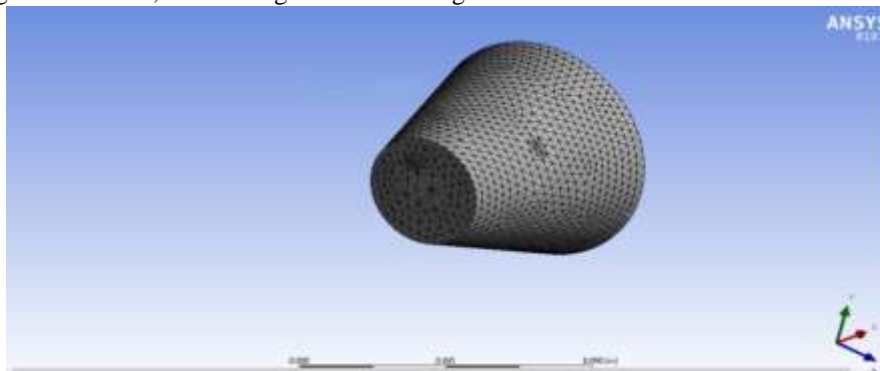


Fig. 7 Meshing the model

4.2.4. Solution Method

Table V

General	Solver type- Pressure based. It is used for dense material.
Model	Energy equation: on Viscous model: Standard K-E model for non-compressible flow
Material/Cell zone	Density (kg/m <sup>3</sup> )=998.2 Specific heat CP (j/kg k)= 1.4 Thermal Conductivity(w/mk)=0.6 Viscosity(kg/ms)= 0.001003
Boundary condition	Inlet velocity = 30m/s Water temperature = 288.16k Mass flow rate = 0.0074 m <sup>3</sup> /s
Axis	Axi-symmetric For cylindrical part.
Solution method	SimpleC (simple with correction)
Solution Initialization	Hybrid Iter Scalar-0 1 1.000000e – 00 2 9.623980e- 04 3 1.255136e – 04 4 4.024191e – 05 5 6.985084e – 06 6 1.998794e – 06 7 3.888076e – 07 8 1.067647e – 07 9 2.177220e – 08 10 5.921898e – 09 Hybrid initialization is done.
Run calculation	No. of iteration = 500

V. RESULTS AND DISCUSSION

The axis was mirrored. Following are the contour plots that were obtained –

**Nozzle Diameter is 20mm**

1. Velocity Contours: The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The velocity at the exit of nozzle is 135.6416m/s

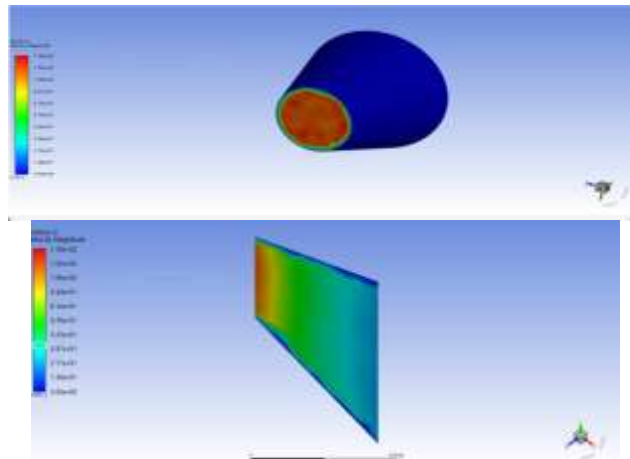


Fig.15. 3D Form And Plate Form Contours of Velocity Magnitude (m/s)

2. Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 8073053 Pa.

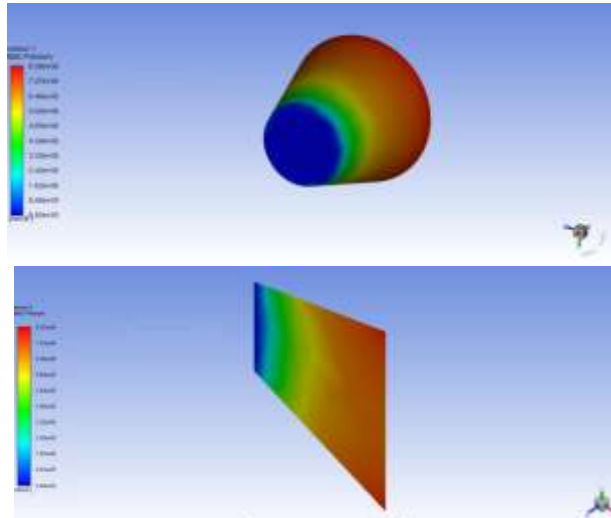


Fig. 17. 3D Form And Plate Form Contours of Static Pressure (Pascal)

**Nozzle Diameter is 18mm**

1. Velocity Contours: The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The velocity at the exit of nozzle is 162.9529 m/s

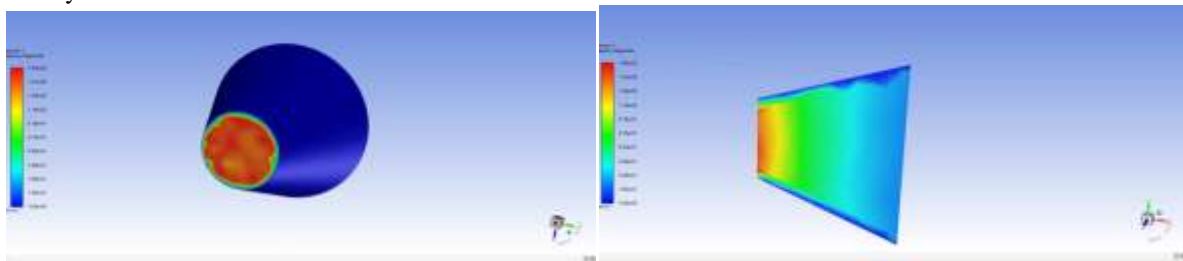


Fig. 15. 3D Form And Plate Form Contours of Velocity Magnitude (m/s)

2. Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 1.22559e+07 Pa.

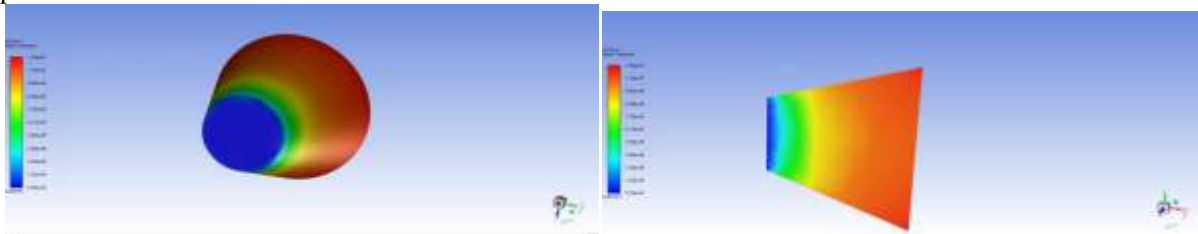


Fig. 17. 3D Form And Plate Form Contours of Static Pressure (Pascal)

**Nozzle Diameter is 16 mm**

2. Velocity Contours: The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The velocity at the exit of nozzle is 210.2285 m/s

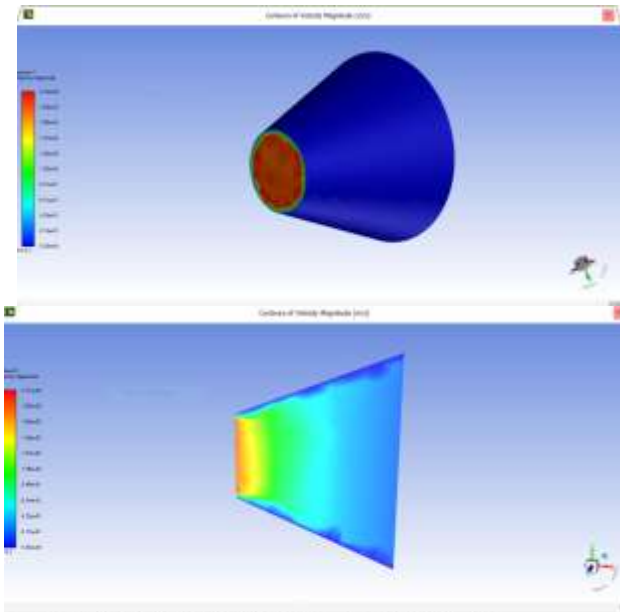


Fig.15. 3D Form And Plate Form Contours of Velocity Magnitude (m/s)

3. Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is  $1.975562e+07$  Pa.

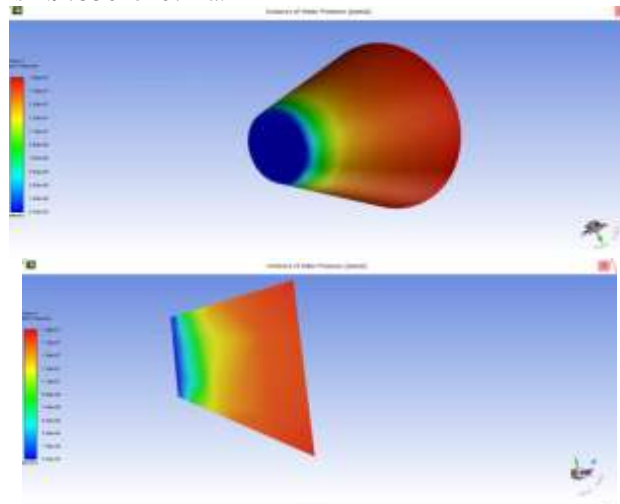


Fig. 17. 3D Form And Plate Form Contours of Static Pressure (Pascal)

**Nozzle Diameter is 22 mm**

4. Velocity Contours: The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The velocity at the exit of nozzle is 109.1571 m/s

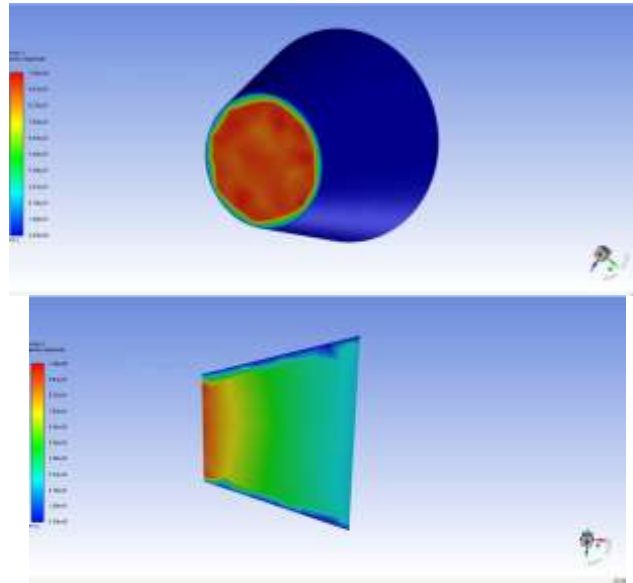


Fig.15. 3D Form And Plate Form Contours of Velocity Magnitude (m/s)

5. Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 5476251 Pa.

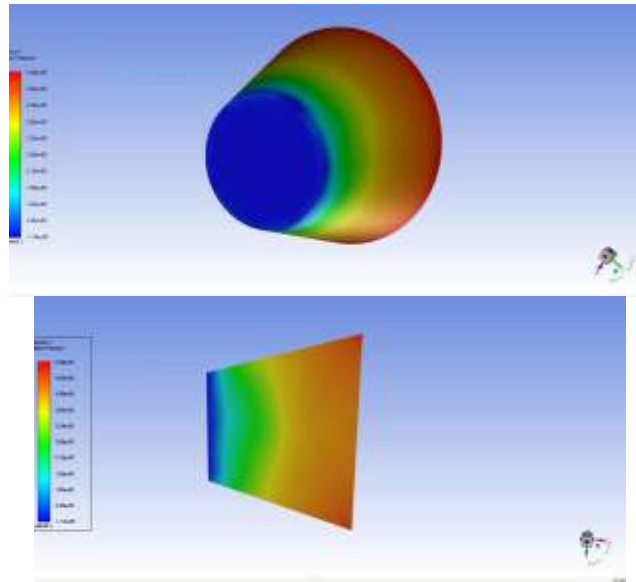


Fig. 17. 3D Form And Plate Form Contours of Static Pressure (Pascal)

**5.1. Summary of CFD results**

Result based on the CFD analysis :

**Table No. 6**

Dia. Of nozzle (mm)	Velocity Outlet (m/s)	Max Pressure Inlet (MPa)	Max Mass flow rate (kg/s)
20	135.64	8.073	42590.96
18	162.95	12.255	41444.703
16	210.22	19.755	42245.8112



22	109.15	5.476	41470.451
----	--------	-------	-----------

In comparison table I have consider four different exist diameter of nozzle with the help of CFD analysis compare the velocity between them and we get maximum exit velocity i.e. 210.22m/s and maximum Inlet pressure i.e. 19.755MPa for minimum nozzle outlet diameter i.e. 16mm

### 5.2. Theoretical Analysis of Velocity

According to continuity Equation :

$$\rho A_1 V_1 = \rho A_2 V_2$$

1. For Dia 20

$$V_2=120\text{m/s}$$

2. For Dia 18

$$V_2=148.14\text{m/s}$$

3. For Dia 16

$$V_2=187.5\text{m/s}$$

6. For Dia 22

$$V_2=99.17\text{m/s}$$

### 5.3. Theoretical And Experimental ANSYS Velocity Analysis:

Table no. 7

Dia. Of nozzle (mm)	Theoretical Velocity(m/s)	Experimental ANSYS Fluent Velocity(m/s)
20	120	135.64
18	148.14	162.95
16	187.5	210.22
22	99.17	109.15

In this table shows theoretical velocity lesser than the experimental velocity because of the friction inside the nozzle. But both analysis shows minimum dia i.e.16 is maximum velocity (i.e.187.5m/s and 210.22m/s).

## VI. CONCLUSION

- Nozzle mainly used to convert the thermal and pressure energy into useful kinetic energy.
- Velocity, Pressure increases which nozzle like size.20mm, 22mm, and 18 mm. 16 mm. nozzle.
- Maximum power generation by using hydraulic turbine it is required to provide regulate flow of water to the runner in hydraulic turbine and also how to required high velocity of water through nozzle.
- In the last few decades, a lot of work has been done for optimization of jet quality of impulse turbine. Many authors have worked for finding out most efficient shape of nozzle.
- This project shows that how to the velocity as well as pressure values varies across the nozzle. Since the wall have zero velocity outlet

view can show that there is increase in velocity across the central axis.

- This study is related to performance enhancement of hydraulic turbine by using high velocity convergent nozzle. On the basis of CFD analysis we get maximum exit velocity i.e. 210.22 m/s for minimum nozzle outlet diameter i.e. 16mm.

## REFERENCES

- [1]. DevidasD.Barale, Prof.C.Limbari, Dr.R.R.Arakerimath, 2016; Modelling and parametric fluid flow Analysis and effect of convergent nozzle used in pelton turbine, JETIR, ISSN-2349-5162
- [2]. Tridib Kumar Mehata ,A detailed study on performance evaluation of multiple co-axial nozzle,2015;issn-2230-984
- [3]. Shofiya Shalini, D.Sundarapriyaa N. ,Experimental investigation of nozzle design, by simulation method and performance analysis in moppet engines.2018;;ISSN -2229-5518,vol.9

- [4]. Dr.Vishal Gupta, Pradip Kr. Kurmi, Manish Sharma, Numerical analysis of impulse turbines nozzle,2018;ISSN-330778039
- [5]. Vishnu Prasad, Vishal Gupta, Sarojragnekar,Performance analysis of nozzle used in impulse hydraulic turbine using CFD;ISSN-306034738
- [6]. Nikhil D. Deshpande, Suyash S. Vidwans,Pratik R.Mahale,Rutuja s. Joshi, K.R. Jagtap Theoretical and CFD analysis of de level nozzle vol.2 April 2014
- [7]. Gutti Rajeswara Rao, U.S. Ramakanth, A.Lakshman (2013) Flow Analysis in a convergent-Divergent nozzle using CFD, IASTER (2013) 2347-5188.
- [8]. Deepak Bisen, Prof. Shuneel Kumar Shukla, Dr. P.K. Sharma, Review Paper on Nozzle in Hydro-Turbine.IJATES. Vol. No. 02, Issue No. 8 Aug 2014, ISSN: 2348-7550.
- [9]. Malay S.Patel, D. Sulochan Mane and Manikant Raman, Concept and CFD analysis of De-Laval nozzle. IJMET\_07\_05\_024, Issue: 5 sep 2016, ISSN: 0976-6340.
- [10]. Kunal pansari, S.A.Jilani, Analysis of the performance and flow characteristics of Convergent-Divergent (C-D) Nozzle. IJAET issue: July 2013 ISSN: 22311963.