Modelling and Parametric fluid flow Analysis (CFD) of convergent nozzle used in pelton turbine

#1Devidas D. Barale, #2Prof. C. Limbadri, #3Dr. R. R. Arakerimath

[1]Department of Mechanical engineering, Pune university, G.H Raisoni college of engineering & Management, Wagholi, Pune, India.
[2]Department of Mechanical engineering, Pune university, G.H Raisoni college of engineering & Management, Wagholi, Pune, India.
[3]Department of Mechanical engineering, Pune university, G.H Raisoni college of engineering & Management, Wagholi, Pune, India.

Abstract— A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enter) an enclosed chamber or pipe via orifice. A numerical study has been carried out to analyze the performance and flow characteristics of the convergent nozzle under operating pressure ratio and with different nozzle profiles. This paper aims to find out high outlet velocity from nozzle to increases the efficiency of pelton turbine.

Keywords: **ANSYS FLUENT, Convergent Nozzle.**

I. INTRODUCTION

Nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic energy. The values of temperature, pressure and velocity should be available at every section of the nozzle so as to design the nozzle shape at different diameter. It is used as a means of accelerating the flow of a gas passing through it to a supersonic speed. It is widely used in some types of steam turbine. The work carried out in two stages:
1. Modeling of nozzle for different diameter i.e 14mm, 18mm, 22mm, 24mm.
2. Analysis of flow and predict the best suited nozzle among the nozzle considered. Analysis has been done in ANSYS FLUENT.

1.1Problem statement: For maximum power generation by using pelton turbine it is required to provide regulate flow of water to the runner in a pelton turbine also it required high velocity of water through nozzle. This paper aims to find out high outlet velocity from nozzle.

1.2 The objectives of this study is:
1. To predict the flow behavior of nozzle.
2. To investigate the best suited nozzle for pelton turbine.
Recently some researchers worked on flow analysis of convergent –divergent nozzle are discussed bellow Nikhil D. Deshpande et.al[1] - They Works On theoretical & Cfd analysis Of De Laval Nozzle and they conclude that the results obtained by Computational Fluid Dynamics (CFD) are almost identical to those obtained theoretically.

Vishal Gupta et.al[2**]**It is found from numerical simulation of Pelton turbine at best efficiency point for different shapes that the circular jet are the most efficient giving highest efficiency. The spreading of water over the bucket is found to be more uniform for circular jet. The sharp edges of the jet other than circular shape give poor efficiency of the turbine because of uneven distribution of water on bucket surface.

Gutti Rajeswara Rao et.al[3] – They works on the effects of Mach number and Nozzle pressure ratios (NPR) on Mass flow rate. And they conclude that if mach no. increases then velocity also increases. Different nozzle ratios have different Mach numbers

C. Satheesh et.al[4]- They works on the Converging-diverging nozzles with divergence angles of 0.076°, 0.153°, 0.306° and 0.612° And they conclude that The overall optimum divergence angle was 0.306° in the range considered which had the highest efficiency of 70% at a pressure drop of 7.3MPa.

G. Satyanarayana et.al[5] - They works on the CFD analysis has been done on convergent divergent nozzle of different cross section like rectangular, square, and circular And they conclude that from result that rectangular nozzle give high exit velocity , high pressure drop and high temperature as compared to square and circular.

II. POSITION AND APPLICATION OF NOZZLE IN PELTON TURBINE

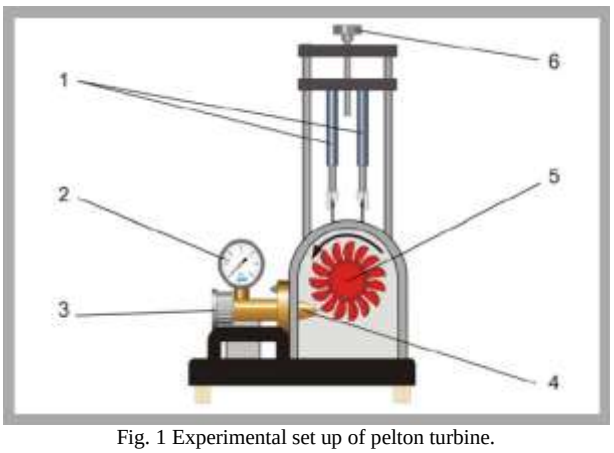

Fig. 1 Experimental set up of pelton turbine.

1 spring balance, 2 manometers, 3 adjustment of the nozzle cross-section, 4 needle nozzle, 5 Pelton wheel, 6 adjustment of the band brake.

The function of the needle jet (or nozzle) is to regulate the flow of water to the runner in an impulse turbine runner. Nozzle is used to provide high velocity of turbine wheel. The needle jet is regulated by the governor via mechanical-hydraulic or electro-hydraulic controls. The shape is designed for rapid acceleration at the exit end and for assuring a uniform water jet shape at openings. The needle valve/nozzle assembly is placed as close to the runner as possible to avoid jet dispersion due to air friction.

III. THEORETICAL FORMULATION OF NOZZLE

The equations used below are for one dimensional nozzle flow. It corresponds to the idealization andsimplification of flow equations. Nomenclature of symbols used is as follows:

P – Pressure (Pa)
T – Temperature (K)
V – Velocity (m/s)
g – Gravitational acceleration (m/s2)
z – Height (m)
A – Area (m2)
Cp – Specific heat at constant pressure (J/kg K)
Cv – Specific heat at constant volume (J/kg K)
γ – Adiabatic index (Cp/Cv)
h – Enthalpy (J)
R – Specific gas constant (J/kg K)
ρ – Density (kg/m3)
– Mass flow rate (kg/s)

3.1Continuity Equation used in nozzle

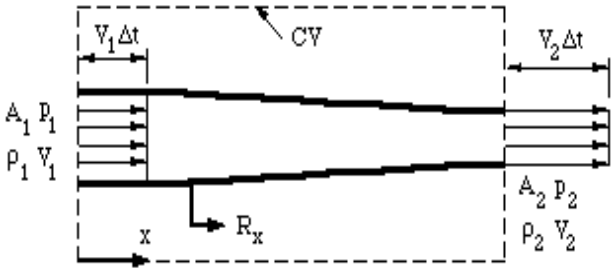

Fig.2 1D flow nozzle

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_1 goes out of A_2,

volume flow in over $A_1 = A_1 V_1 \Delta t$

volume flow out over $A_2 = A_2 V_2 \Delta t$

Therefore,

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

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_1& V_2 = Flow velocity at inlet and outlet.

A_1& A_2 = cross-sectional area at inlet and outlet.

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

3.2 Bernoulli's equation

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho v_2^2 + \rho g h_2$$

Where,P_1 and P_2 are the inlet and outlet pressure of the fluidV_1 and V_2 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

Volume flow rate Q = AV…….m^3/s

Mass flow rate m = density × Volume flow rate

= ρ × A× V…….Kg/s

Where

V or Q = Volume flow rate,

ρ = mass density of the fluid.

v = Flow velocity of the mass elements,

A = cross-sectional vector area/surface,

IV. CFD BASED SIMULATION OF NOZZLE

I have selected four different nozzle i.e 14mm, 18mm, 22mm, 24mm. I have chosen this nozzle for checking the effect on performance of pelton turbine also for checking trial and error.

In this project the designing and analysis of convergent nozzle geometry is done in the CFD (Computational Fluid Dynamics software). Firstly the design of nozzle is made in the surface modeling or CATIA software and then the nozzle geometry is further analyzed in fluent software in order to analyze the flow inside the convergent nozzle and to get the view of the behavior of fluid inside the convergent nozzle.

4.1 Modeling

The modeling of the nozzle was done using surface modeling and file was saved in standard format. The dimensions of the de Laval nozzle are presented in the table given below.

Table I Nozzle Dimensions

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	13
2	Inlet diameter (mm)	32
3	Outlet diameter (mm)	14

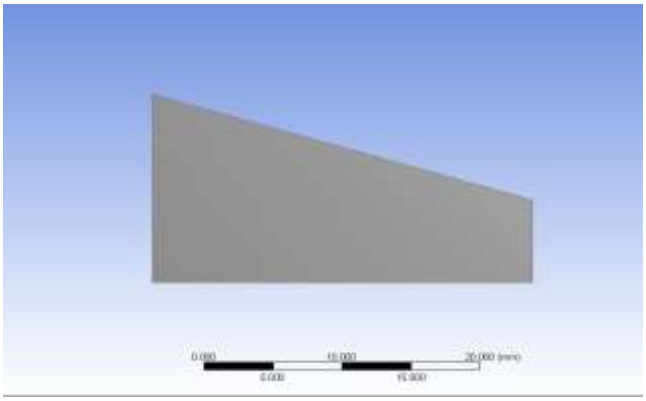

Fig.3 Surface model of nozzle

Table II Nozzle Dimensions

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	13
2	Inlet diameter (mm)	32
3	Outlet diameter (mm)	18

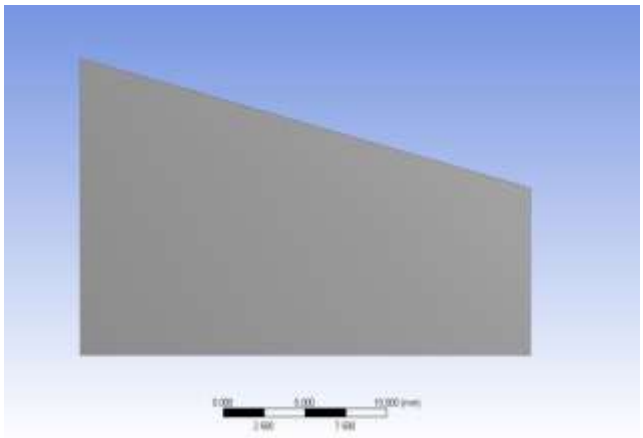

Fig.4 Surface model of nozzle

Table III Nozzle Dimensions

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	13
2	Inlet diameter (mm)	32
3	Outlet diameter (mm)	22

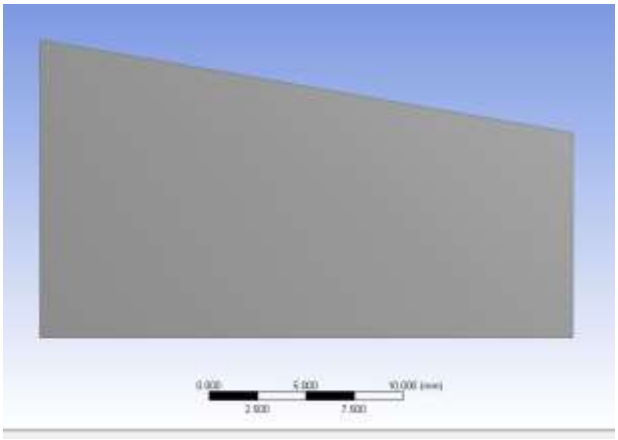

Fig.5 Surface model of nozzle

Table IV Nozzle Dimensions

Sr.No.	Parameter	Dimensions
1	Total nozzle length (mm)	13
2	Inlet diameter (mm)	32
3	Outlet diameter (mm)	24

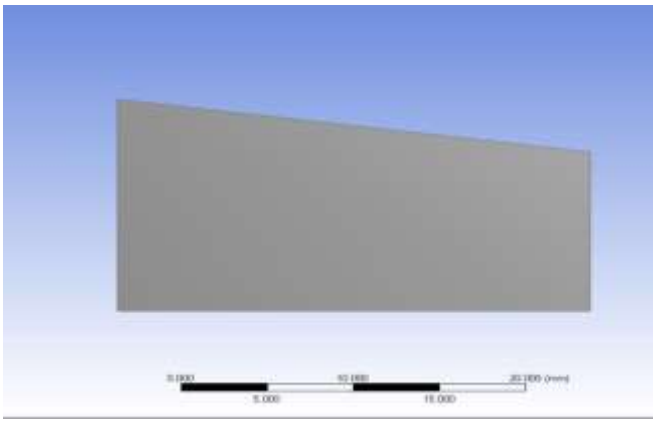

Fig.6 Surface model of nozzle

4.2 Meshing

After modeling of the nozzle, its meshing was doneUsing ANSYS CFD software.

Mesh quality: Orthogonal quality range from 0 to 1, where value close to zero corresponds to low quality.

Minimum orthogonal quality = 9. 69576e – 01
Maximum aspect ratio = 1.90672e + 00
Meshing type - structural mesh.

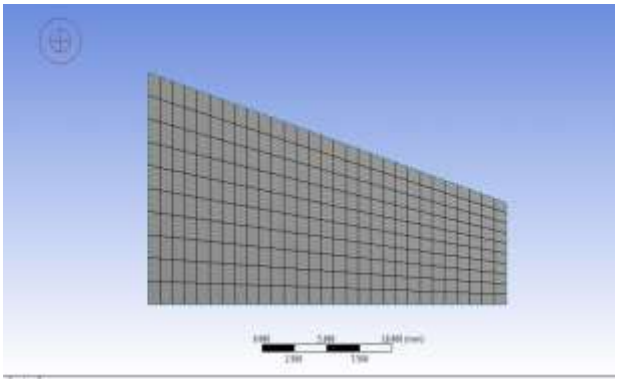

Fig. 7Meshing the model

4.3. 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/m3)=998.2 Specific heat CP (j/kg k)= 4182 Thermal Conductivity(w/mk)=0.6 Viscosity(kg/ms)= 0.001003
Boundary condition	Inlet velocity = 3.77 m/s Water temperature = 300k Mass flow rate = 0.0074 m^3/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 –
5.1. Nozzle Diameter is 14mm

1) Velocity Contours: The velocity is minimum atthe inlet and goes on increasing till the nozzle exit. The velocity at the exit of nozzle is 19.75m/s

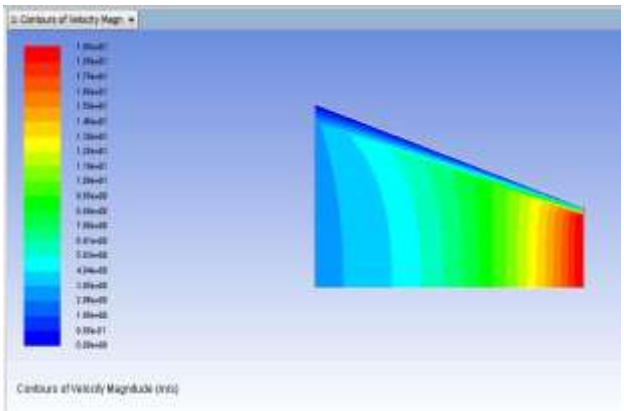

Fig. 8 Contours of Velocity Magnitude (m/s)

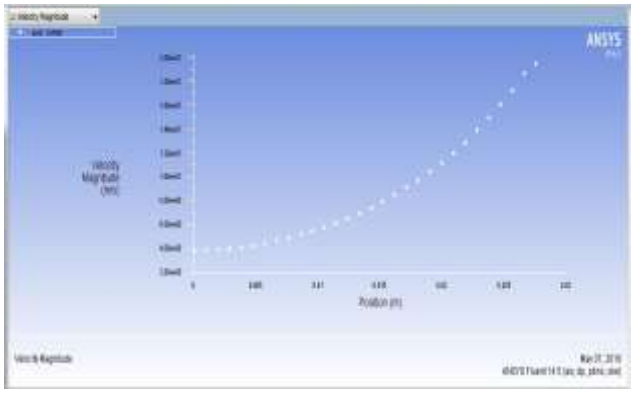

Fig. 9 Contours of Velocity Magnitude (m/s)

2) Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 2×10^4Pa.

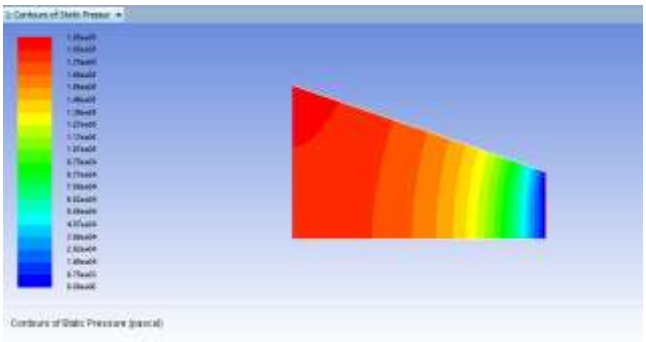

Fig. 10 Contours of Static Pressure (Pascal)

5.2. 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 12.1156m/s

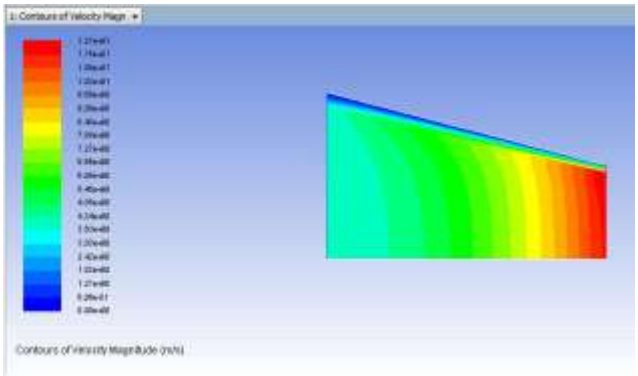

Fig. 11 Contours of Velocity Magnitude (m/s)

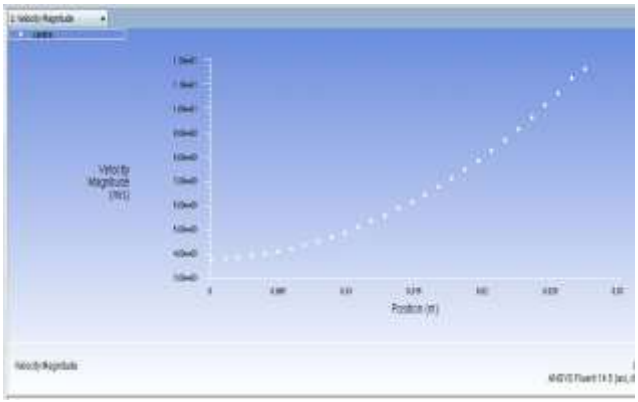

Fig. 12 Contours of Velocity Magnitude (m/s)

2) Pressure Contours: The pressure is maximum atthe inlet and goes on decreasing till the outlet. The static pressure at the outlet is 5×10^3 Pa.

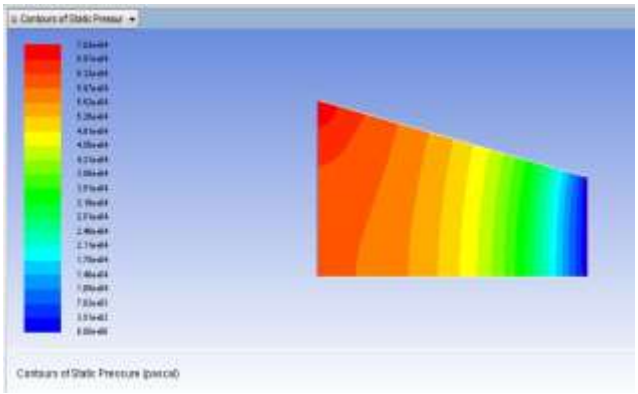

Fig. 13 Contours of Static Pressure (Pascal)

5.3. Nozzle Diameter is 22mm

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 8.1775m/s

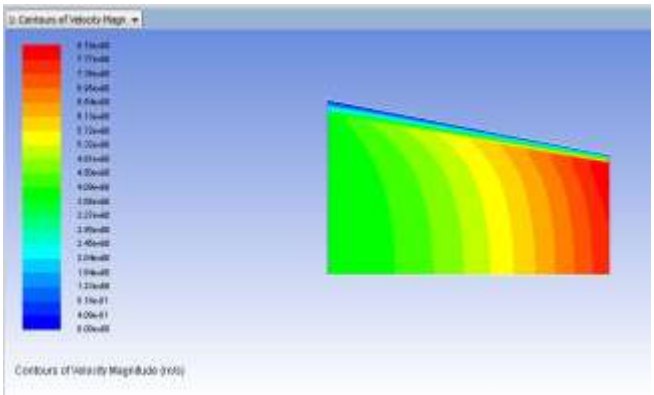

Fig. 14 Contours of Velocity Magnitude (m/s)

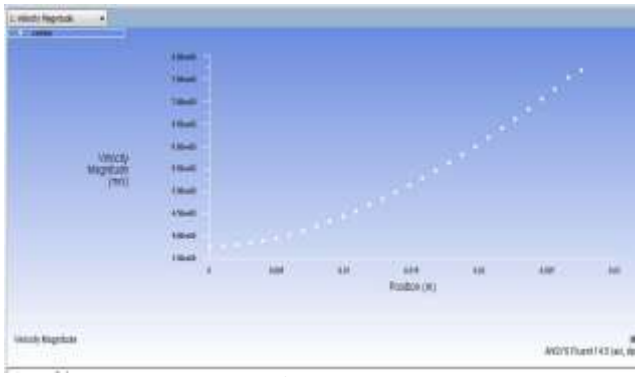

Fig. 15Contours of Velocity Magnitude (m/s)

2) Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 2×10^3Pa.

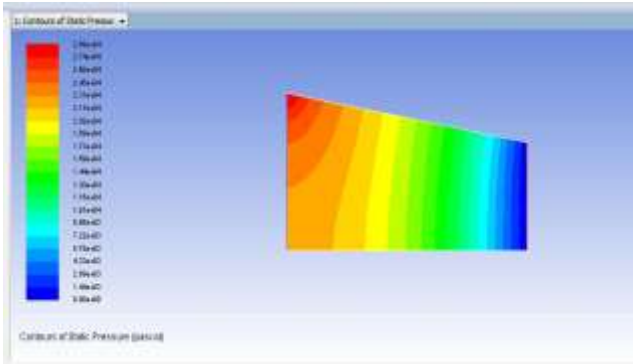

Fig. 16 Contours of Static Pressure (Pascal)

5.4. Nozzle Diameter is 24mm

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 6.89747 m/s

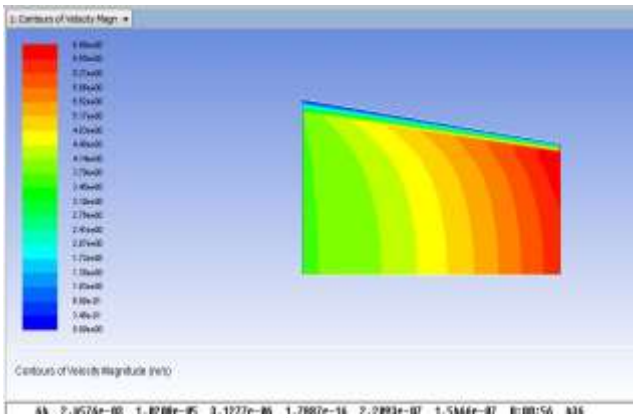

Fig. 17 Contours of Velocity Magnitude (m/s)

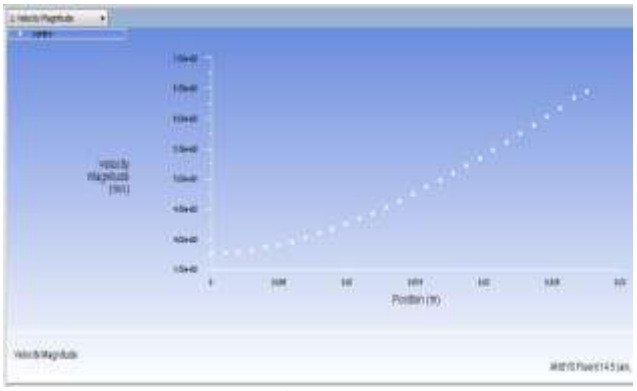

Fig. 18 Contours of Velocity Magnitude (m/s)

2) Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 1.4×10^3Pa.

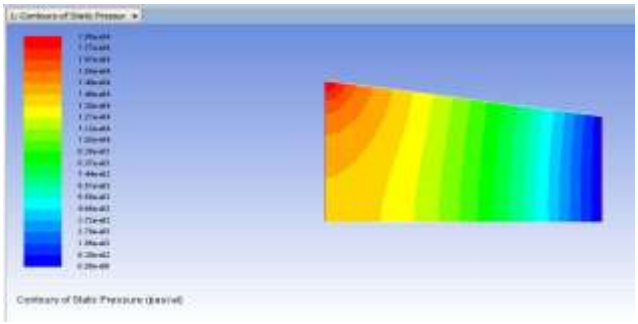

Fig. 19 Contours of Static Pressure (Pascal)

5.5 Summary of CFD results:

Result based on the CFD analysis Table VI

Dia. Of nozzle (mm)	Velocity (mm)	Inlet pressure (Pa)	Outlet pressure (Pa)	Mass flow rate (kg/s)
14	19.75	18×10^4	2×10^4	3038.73
18	12.11	6×10^4	5×10^3	3080.05
22	8.17	2.2×10^4	2×10^3	3104.10
24	6.89	1.4×10^4	1.4×10^3	3115.38

VI. COMPARISON CHART

6.1 Pressure comparison chart

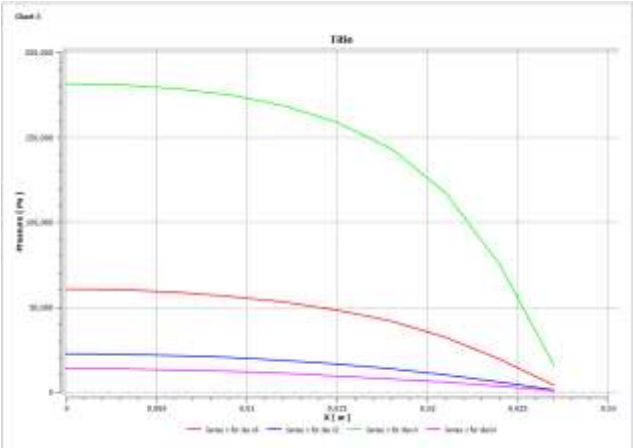

Fig 20 Pressure comparison chart

In the pressure comparison chart I have consider four different exist diameter of nozzle with the help of CFD analysis compare the pressure between them.

6.2 Velocity comparison chart

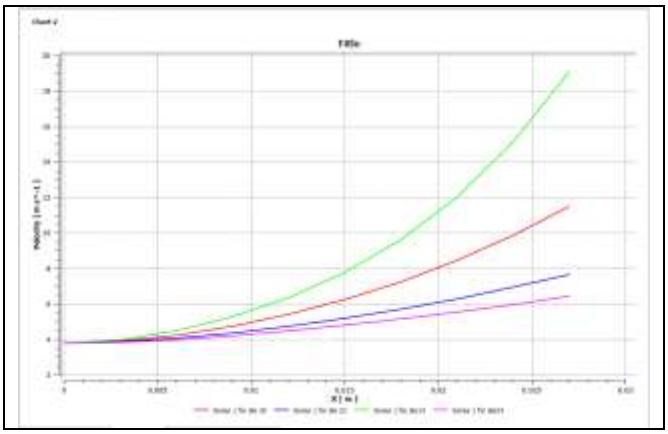

Fig. 21 Velocity comparison chart

In the velocity comparison chart 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 19.75m/s for minimum nozzle outlet diameter i.e 14mm.

VII. CONCLUSION

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 study is related to performance enhancement of pelton turbine by using high velocity convergent nozzle. On the basis of CFD analysis we get maximum exit velocity i.e 19.75m/sfor minimum nozzle outlet diameter i.e 14mm.

REFERENCES

[1]. Nikhil d. Deshpande, suyash s. Vidwans, pratik r. Mahale, rutuja s. Joshi, k. R. Jagtap theoretical & cfd analysis of de laval nozzle vol. 2 april 2014

[2]. Vishal gupta, dr. Vishnu prasad dr. Ruchi khare effect of jet shape on flow and torque characteristics of pelton turbine runner

[3]. GuttiRajeswaraRao,U.S.Ramakanth, A.Lakshman (2013) Flow Analysis in a Convergent-Divergent Nozzle Using CFD, IASTER (2013) 2347-5188.

[4]. C. Satheesh , A. rulmurugudesign and analysis of c-d nozzle increase the Efficiency using CFD, SJIF

[5]. G. Satyanarayana, ChVarun, S.S Naidu (2013) CFD analysis of convergent divergent nozzle. FASCICULE3 (2013)

[6]. Kunal pansari, s.a. kjilani analysis of the performance and flow characteristics of convergent divergent (c-d) nozzle ,ijaet july 2013.

[7]. Hunter, C.A., "Experimental, Theoretical, And Computational investigation Of Separated Nozzle Flows," AIAA Paper 1998-3107, 1998.

[8]. K.M. Pandey, Member IACSIT and A.P. Singh," CFD Analysis Of Conical nozzle For Mach 3 At Various Angles Of Divergence With Fluent software", International Journal of Chemical Engineering and Applications, Vol. 1,No. 2, August 2010, ISSN: 2010-0221.

