International Journal for Modern Trends in Science and Technology
Volume: 02, Issue No: 11, November 2016
http://www.ijmtst.com
ISSN: 2455-3778

Computational Estimation of Flow through the C-D Supersonic Nozzle and Impulse Turbine Using CFD

Snigdha Panday[1] | B. Joga Rao[2] | Dr.T.Jayanand Kumar[3]

[1]PG Scholar, Department of Mechanical Engineering, GIET (A), Rajahmundry, Andhra Pradesh, India.
[2]Assistant Professor, Department of Mechanical Engineering,, GIET (A), Rajahmundry, Andhra Pradesh, India.
[3]Professor & Head, Department of Mechanical Engineering,, GIET (A), Rajahmundry, Andhra Pradesh, India.

To Cite this Article
Snigdha Panday, B. Joga Rao, T.Jayanand Kumar, Computational Estimation of Flow through the C-D Supersonic Nozzle and Impulse Turbine Using CFD, *International Journal for Modern Trends in Science and Technology*, Vol. 02, Issue 11, 2016, pp. 24-32.

ABSTRACT

In this paper, CFD analysis of flow within, Convergent – Divergent rectangular super sonic nozzle and super sonic impulse turbine with partial admission have been performed. The analysis has been performed according to shape of a super sonic nozzle and length of axial clearance and the objective is to investigate the effect of nozzle-rotor interaction on turbine's performance. It is found that nozzle-rotor interaction losses are largely dependent on axial clearance, which affects the flow within nozzle and the extent of flow expansion. Therefore selecting appropriate length of axial clearance can decrease nozzle-rotor interaction losses. The work is carried in two stages:1) Modeling and analysis of flow for rectangular convergent divergent super sonic nozzle. 2) Prediction of optimal axial gap between the nozzle and rotor blades by allowing the above nozzle flow.

In the present work, using a finite volume commercial code, ANSYS FLUENT 14.5, carries out flow through the convergent divergent nozzle study. The nozzle geometry is modeled and grid is generated using ANSYS14.5 Software. Computational results are in good agreement with the experimental ones.

KEYWORDS: *Convergent – Divergent Nozzle,Nozzle-rotor interactions , Optimal axial clearence, Modeling using ANSYS FLUENT 14.5*

The work is carried in two stages:

1. Modeling and analysis of flow for rectangular convergent divergent supersonic nozzle.
2. Prediction of optimal axial gap between the nozzle and rotor blades by allowing the above nozzle flow.

In the present work, using a finite volume commercial code, ANSYS FLUENT 14.5, carries out flow through the convergent divergent nozzle study. The nozzle geometry is modeled and grid is generated using ANSYS14.5 Software. Computational results are in good agreement with the experimental ones.

The objective of present work is
1. Modeling and meshing of nozzle geometry.
2. Validate the CFD results of nozzle flow both theoretically and experimentally.
3. Modeling and meshing of nozzle and turbine blades is considered as a case of partial admission type.
4. Validate the CFD results of nozzle and turbine blades with experimental data.

This thesis aims to predict the following:

- Estimation of velocity at nozzle exit as a case of supersonic (or) not.
- Estimation of nozzle and turbine rotor gap under static condition of rotor.
- Flow visualization.

I. Fluid Governing Equation

The fundamental equations of fluid dynamics are based on the following universal law of conversation. They are:

1) Conservation of mass.
2) Conservation of momentum.
3) Conservation of energy.

Continuity Equation:

where 'ρ' is the density, 'V' is the fluid velocity. For an incompressible flow, the density of each fluid element remains constant.

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial\tau_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+\rho f_x$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+\rho f_y$$

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z}+\rho f_z$$

Unsteady Convective Pressure Diffusive Source

Momentum Equation:

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

$$\frac{\partial\rho}{\partial t}+\nabla.(\rho V)=0$$

Energy Equation:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)V\right]=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)$$
$$+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{xx})}{\partial x}$$
$$+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}$$
$$+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho f\cdot V$$

Equation of State:

(Pv= MRT)

II. Methods of Prediction

Experimental Investigation:

The most reliable information about the physical process is often given by actual measurement. It involves full scale equipment, which can be used to predict how identical copies of the equipment would perform under the same conditions. They are often expensive and time consuming. The resulting information however must be extrapolated to full scale and general rules for doing this are often available.

Theoretical Calculations:

It works out on the consequences of mathematical model rather than those of an actual physical model. If the methods of classical mathematics are to be used for solving these equations then there is a little hope for predicting many phenomenon of practical interest. It is often referred as an analytical approach. In this simplification, assumptions are used in order to make the problem tractable. These solutions often contain infinite series, special functions and transcendental equations for Eigen values so that the numerical evaluation may be the formidable task.

Computational Methods:

In the computational approach a limited number of assumptions were made and high speed digital computer is used to solve the resulting governing fluid dynamic equations. The development of numerical methods and the availability of large digital computers hold the promise implicating the mathematical model which can be worked out on almost any practical problem.

Advantages:

a) No restriction to linearity
b) Complicated physics can be treated.
c) Time evolution of flow can be obtained.
d) It has potential of providing the information which cannot be obtained by any other means.
e) Computational can provide investigations which can be performed with remarkable speed. Designer can study the implications of hundreds of differential configurations in minimum time and choose the optimum design.
f) It gives detailed and complete information. It can also provide the values of all the relevant variables such as pressure, velocity, temperature, concentration, turbulence etc throughout the domain of interest.

Disadvantages:

i) Truncation errors
ii) Boundary condition problems
iii) Computer costs.
iv) Computer storage and speed.

Applications of CFD:

a) Aerodynamics of aircraft & vehicles: lift drag
b) Hydrodynamics of ship
c) Metrology: weather prediction
d) Turbo machinery: Flows inside rotating passage, diffusers.
e) Hydrology and oceanography: flows in rivers, estuaries and oceans

Problem solving steps: Once you have determined the important features of the problem we want to solve, we will follow the basic procedural steps shown below:

1. Create the model geometry and grid.
2. Start the appropriate solver for 2d or 3d modeling.
3. Import the grid.
4. Check the grid
5. Scale the grid
6. Select the solver formulation.
7. Choose basic equations to be solved: laminar or turbulent (or inviscid), chemical species or reaction, heat transfer models etc. Identify additional models needed: fans, heat exchangers, porous media etc.
8. Specify the material properties.
9. Specify the boundary conditions.
10. Adjust the solution control parameters.
11. Initialize the flow field.
12. Calculate the solution.
13. Examine the results.
14. Save the results.

III. Modeling of the Components

Modeling of Super Sonic Nozzle:

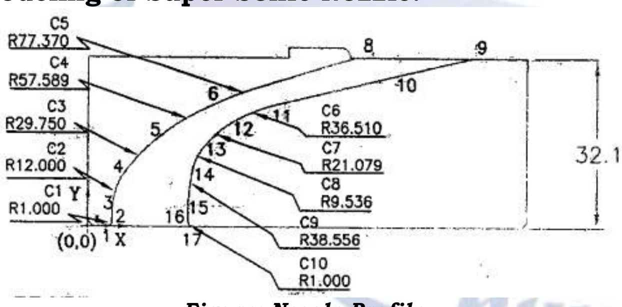

Figure: Nozzle Profile

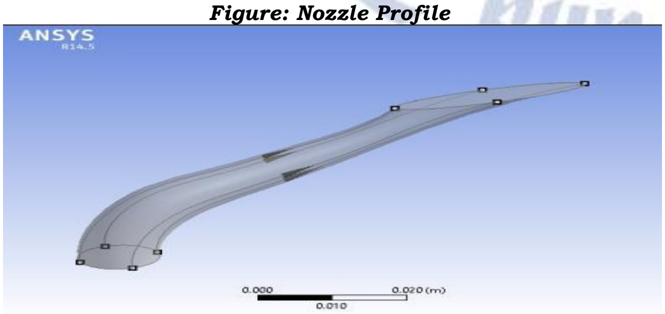

Figure: Solid model of nozzle profile.

Modeling of Blade Profile:

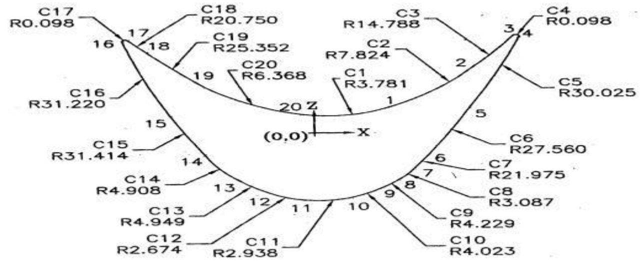

Figure: Blade Profile.

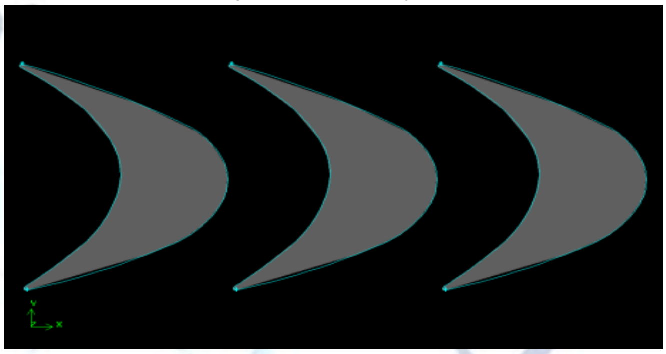

Figure: Series of Rotor Blade.

Modeling of Nozzle and Turbine Blades as Partial Admission Case:

As analysis is to be carried out for different axial gap between rotor blades and nozzle, length of ordinate is fixed. Rotor blades and nozzle with different axial gaps are modeled separately as shown in figures for 3mm, 4mm and 5mm axial gaps respectively.

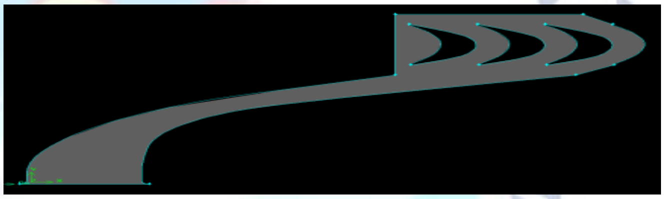

Figure: Solid model as a partial admission case of Nozzle and Rotor blades with 3mm axial gap.

Figure: Solid model as a partial admission case of Nozzle and Rotor blades with 4mm axial gap.

Figure: Solid model as a partial admission case of Nozzle and Rotor blades with 5mm axial gap.

Mesh generation:

Mesh Generation for 3D C-D nozzle:

A 3D Model is created in ANSYS FLUENT 14.5 as shown in figure. To generate the structured grid as shown in figure with hexahedral cells ANSYS FLUENT 14.5 is used. This mesh is in good agreement with turbulence model and good results are obtained otherwise if they aren't in good agreement then they won't match with the experimental ones.

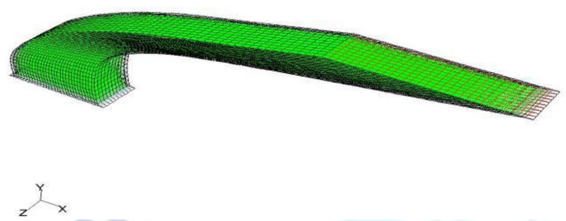

Figure: Meshed View of 3 D Nozzle Profile.

Mesh Generation for 2D C-D nozzle:

A 2D nozzle profile is modeled in ANSYS FLUENT 14.5. To generate the structured grid as shown in figure with quadrilateral elements ANSYS FLUENT 14.5 is used. This mesh is in good agreement with turbulence model and good results are obtained otherwise if they aren't in good agreement then they won't match with the experimental one.

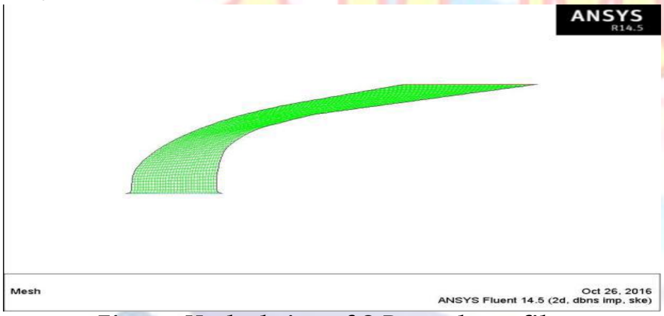

Figure: Meshed view of 2 D nozzle profile.

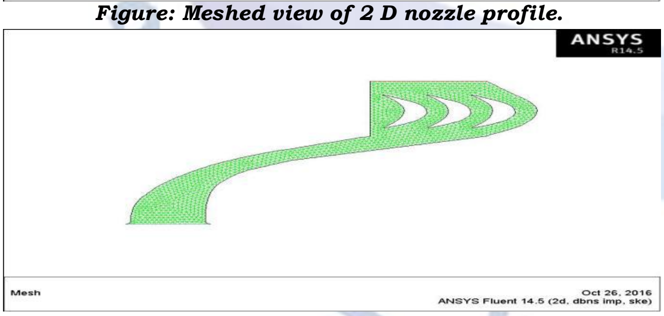

Figure: Meshed view as a partial admission case of Nozzle and Rotor blades with 3mm axial gap.

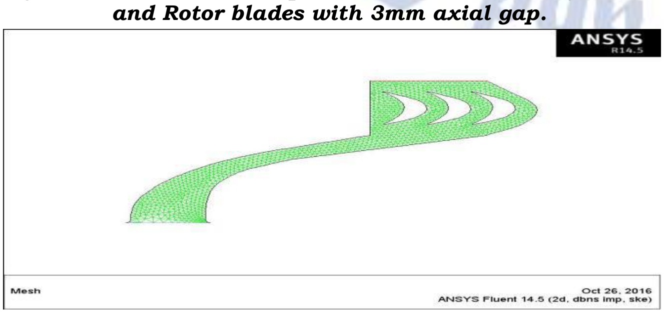

Figure: Meshed view as a partial admission case of Nozzle and Rotor blades with 4mm axial gap.

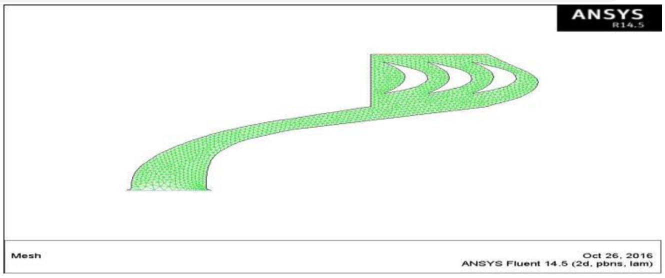

Figure: Meshed view as a partial admission case of Nozzle and Rotor blades with 5mm axial gap.

ANALYSIS OF C-D RECTANGULAR NOZZLE:

Solver: Material selection and operating condition defining:

The solver is defined first. Solver is taken as Coupled based and formulation as implicit, space as 2D and time as steady. Velocity formulation as absolute and gradient options as Green Gauss Cell based are taken. Energy equation is taken into consideration. The viscous medium is also taken. Then analysis is carried out using K-epsilon turbulence model. The selection of material is done. Material selected as gas. The properties of gas taken as follows:

Density of ideal gas is considered
Cp (Specific heat capacity) = 2034.6J/Kg.K
Thermal Conductivity = 0.0706 W/m-K
Viscosity = 6.07 e-5 (Kg/m-s)
Molecular weight = 23.05 (Kg/Kg-Mol)

The analysis is carried out under operating condition of Zero
Pascal. Gravity is not taken into consideration.

BOUNDARY CONDITIONS:

Nozzle Inlet:

Pressure inlet is taken as inlet for nozzle. The value ofpressure is 8101325 Pascal. Initial gauge pressure is taken as 7898681 Pascal. Temperature is taken as 1583K.

Nozzle Outlet:

The nozzle outlet is set as pressure outlet with a value of 13e5.

Controls set up:

The solution controls are set as listed below. The under relaxation factor was set as given.
Turbulence Kinetic Energy 0.8
Turbulence Dissipation rate 0.8
Turbulence Viscocity 1

Discretization Equation is selected as given:
Flow (Second order up wind)
Turbulence Kinetic Energy (1st order upwind)
Turbulence dissipations rate (1st order upwind)

Initialization:

Solution initialization is done. Initial values of velocity are taken as 186.3 m/s in y direction. Temperature is taken as 1583K Residual monitoring is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity - 0.001
X Velocity - 0.001
Y Velocity - 0.001
Energy - 0.001

The number of iterations is set up and iterations starts. The iteration continues till the convergence is reached and convergence history as shown in figure.

ANALYSIS OF NOZZLE AND TURBINE ROTOR BLADES AS A CASE OF PARTIAL ADMISSION:

The analysis is carried in fluent software by importing the meshed file saved in ANSYS FLUENT 14.5. The steps that are followed are given below which all the conditions and the boundaries value for the problem statement are included, for varied axial gaps of nozzle and turbine rotor blades as a case of partial admission.

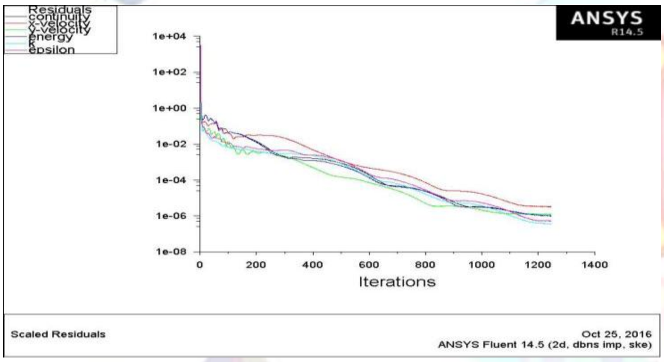

Figure: Convergence history of C-D Nozzle.

Checking of Mesh and Scaling:

The fluent solver is opened where 2DDP is selected and then the importing of the meshed file is done. The meshed file then undergoes a checking where number of grids are found. After this grid check is done followed by smoothing and swapping of grid is done.

Following this step, scaling is done. Scale is scaled to mm. Grid created then converted from m to mm. After this step, defining of various parameters is done.

Solver: Material Selection and operating condition defining:

The solver is defined first. Solver is taken as Segregated based and formulation as implicit, space as 2D and time as steady.

Velocity formulation as absolute and gradient options as Green Gauss Cell based are taken. Energy equation is taken into consideration. The viscous medium is also taken. Then analysis is carried out using K-epsilon turbulence model. The selection of material is done. Material selected as gas.

The properties of gas taken as follows:
Density of ideal gas is considered
Cp (Specific heat capacity) = 2034.6J/Kg.K
Thermal Conductivity = 0.0706 W/m-K
Viscosity = 6.07 e-5 (Kg/m-s)
Molecular weight = 23.05 (Kg/Kg-Mol)

The analysis is carried out under operating condition of Zero Pascal. Gravity is not taken into consideration.

BOUNDARY CONDITIONS:

Nozzle Inlet:

Pressure inlet is taken as inlet for nozzle. The value of pressure is 8101325 Pascal. Initial gauge pressure is taken as 7898681Pascal. Temperature is taken as 1583K.

Outlet blades of rotor:

The outlet is set as pressure outlet with a value of 101325 Pascal

Controls set up:

The solution controls are set as listed below: The under relaxation factor was set as given.
Pressure - 0.3
Density - 1
Body forces - 1
Momentum - 0.7
Pressure velocity coupling was taken as SIMPLE.

Discretization Equation is selected as given:
Pressure - standard
Density - 1st order upwind
Momentum - 1st order upwind
Turbulence Kinetic Energy (1st order upwind)
Turbulence dissipations rate (1st order upwind)
Energy - 1st order upwind

Initialization:

Solution initialization is done. Initial values of velocity are taken as 186.3 m/s in y direction. Temperature is taken as 1583K Residual monitorization is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity - 0.001
X Velocity - 0.001
Y Velocity - 0.001
Energy - 0.001

The number of iterations is set up and iterations starts.

RESULTS AND DISCUSSION

Theoretical calculation of c-d nozzle:

Properties of Gases:
Gamma of gases □ □ □= 1.27
Thermal conductivity [K] = 0.0706 w/m-k

Molecular Weight = 23.05 Kg/Kg-Mol
Specific heat of gases [CP] = 2034.6 KJ/Kg-K
Density treated as Ideal gas
Gas constant (R) = 432
Viscosity (Kg/m-s) = 6.07 e -5

Boundary (Inlet/Exit) Conditions
NPR (Nozzle Pressure Ratio) : 6.16
Given Inlet pressure : 80 bar (8x106 pa)
Exit pressure : 12.987 bar
Inlet temperature : 1583K
Geometrical Parameters
Given Throat area : 2.85xe-5 m2
Given inlet Mach number : 0.2
Mach number definition : M = V/C
C = acoustic velocity

$$C = \sqrt{\gamma RT}$$

$$= \sqrt{(1.27)\,(432)\,(1583)}$$

$$= 931.931\text{m/s}$$

Inlet Velocity is

$$0.2 = \frac{V}{931.931}$$

$$V = 186.3 \quad \text{m/sec}$$

Throat pressure $\frac{P_T}{Pi} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}$

$$P_T = 44.096 \text{ bar}$$

Velocity at throat

$$P_T = sRT$$

$$80\text{x}10^5 = s\,(432)\,1583$$

$$s = 11.698 \text{ kg/m}^3$$

$$V_T = \sqrt{\frac{2\gamma}{\gamma-1}\,(P_1/s)\left(1-\left(\frac{P_T}{P_i}\right)^{\frac{\gamma-1}{\gamma}}\right)}$$

$$= \sqrt{\frac{2(1.27)}{0.27}\;\frac{80\text{x}10^5}{11.698}\;1-\left(\frac{44.096}{80}\right)^{\frac{0.27}{1.27}}}$$

$$= \quad 874 \text{ m/sec}$$

Velocity at Exit

$$Ve = \sqrt{\frac{2\gamma}{\gamma-1}\,P_i/s\left\{1-\left(\frac{Pe}{P_i}\right)^{\frac{\gamma-1}{\gamma}}\right\}}$$

$$= \sqrt{\frac{2(1.27)}{0.27}\;\frac{(8\text{x}10^5)}{11.698}\left\{1-\left(\frac{1}{6.16}\right)^{\frac{0.27}{1.27}}\right\}}$$

$$= 1436 \text{ m/sec}$$

So by the above results Exit Mach number is

$$M = \frac{Ve}{C}$$

$$= \frac{1436}{935}$$

$$= 1.540$$

Known Relation, Mach number related with other parameters

$$\frac{A_E}{A_T} = \frac{1}{M_{Exit}}\left(\frac{1+\frac{\gamma-1}{2}\,M_{ex}^2}{(\gamma+1)}\right)^{\gamma+1/2(\gamma-1)}$$

$$\frac{A_E}{A_T} = \frac{1}{1.540}\left(1+\left(\frac{\frac{(0.27)}{2}\;(1.54)^2}{2.27/2}\right)\right)^{2.27/2(0.27)}$$

$$\frac{A_E}{A_T} = 1.25568$$

So n $A_{EXIT} = 0.3495\text{X}10^{-4}\,\text{m}^2$
$A_{THROAT} = 0.285\text{X}10^{-4}\,\text{m}^2$

RESULTS FOR NOZZLE:

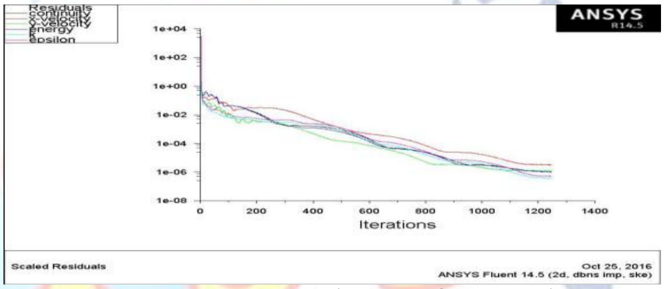

Figure: Convergence history of C-D nozzle

It had been observed that convergence history for C-D nozzle has good argument and flow is maintained smooth as shown in figure.

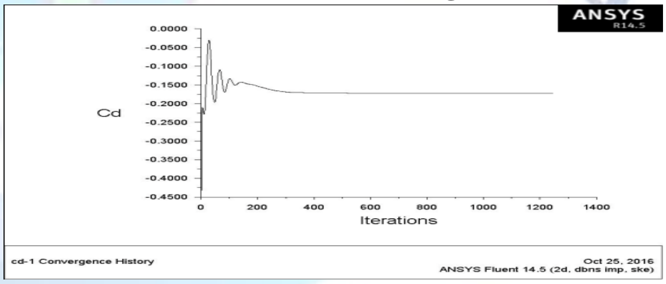

Figure: Co-efficient of drag for C-D nozzle

It had been observed that the co-efficient of drag. Initially it fluctuates at inlet and later on its remains stable while the flow continues, the co-efficient of drag value maintains the limitations and the results are good in agreement as shown is figure.

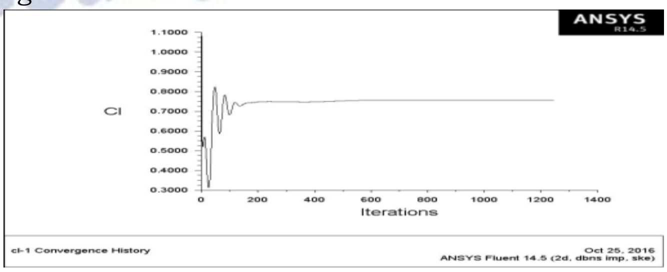

Figure: Co-efficient of lift for C-D nozzle

It had been observed that the co-efficient of drag. Initially it fluctuates at inlet and later on its remains stable while the flow continues, the co-efficient of drag value maintains the limitations and the results are good in agreement as shown is figure. As extent of results in normal direction is nothing but 3D flow through the nozzle for given input condition with velocity as 183 m/s and maximum output was observed as a 1423 m/s, such that Mach Number is increased from 0.2 to 1.54 in which nozzle is acting as supersonic nozzle and contours of mach number as shown in figure. The velocity contours of nozzle is plotted in figure, the pressure contours of nozzle is plotted in figure and temperature contours of nozzle is plotted in figure. The velocity, temperature, Mach number, pressure variation along the nozzle is compared with theoretical calculation and with experimental too. These three results are good in agreement with each other.

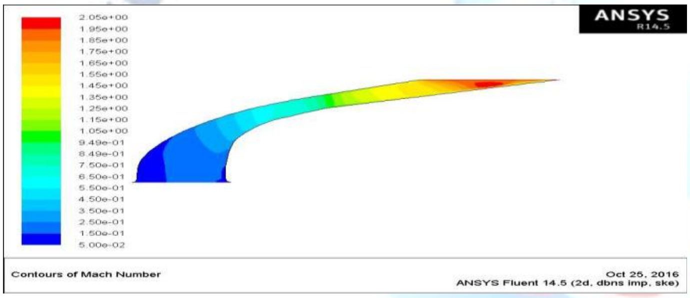

Figure: Mach number contours of nozzle.

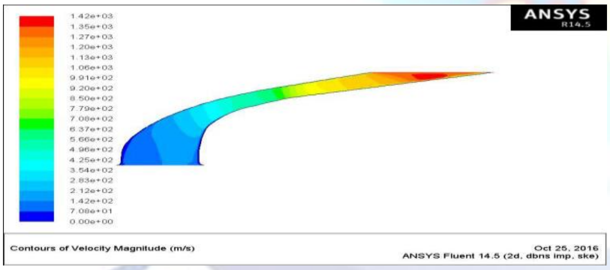

Figure: Velocity contours of nozzle.

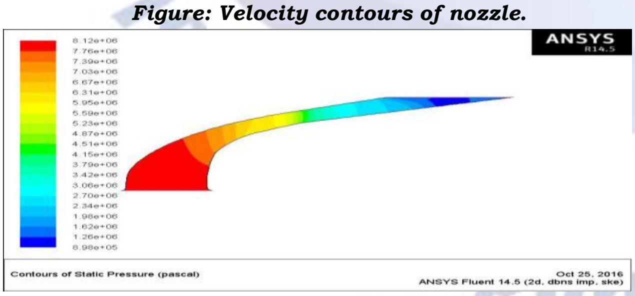

Figure: Pressure contours of nozzle

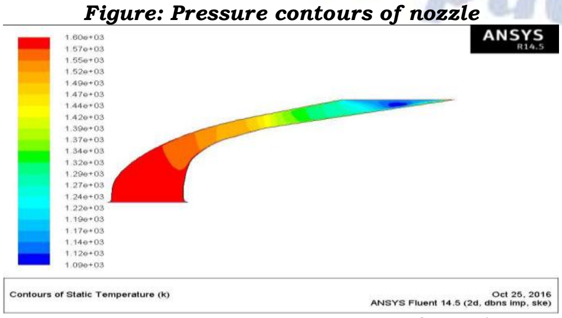

Figure: Temperature contours of nozzle.

RESULTS FOR INTERACTION OF NOZZLE AND ROTOR BLADES:

Flow passage from exit of nozzle and entry of turbine rotor blades is allowed under static condition of rotor (blades). As a case of partial admission of axial impulse turbine the flow will suddenly impact to the blades during the of nozzle flow at rotor blades so many factors can be considered to improve the performance of turbine. In this study the tangential velocity is selected as a parameter for better performance of turbine speed. Velocity distribution is shown in figures for 3mm, 4mm and 5mm axial gap respectively. Velocity vector contours are shown in figures 3mm, 4mm and 5mm axial gap respectively. By observing the tangential velocity contours for 3mm, 4mm, 5mm gap axial clearances of nozzle and turbine rotor blades as shown in figures. The maximum average tangential velocity will act a 3mm axial clearence, 3mm gap of axial clearance will be the better one and also assumed that co-efficient of drag and lift of nozzle-rotor interaction, all are in good agreement as shown in figures.

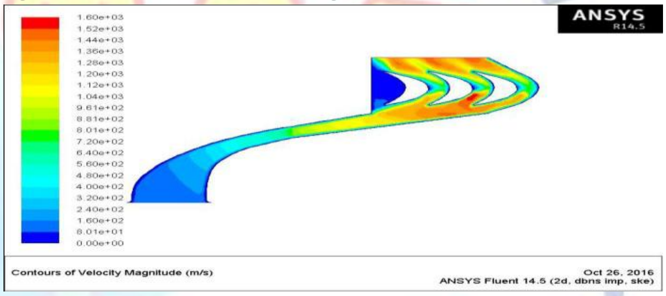

Figure: Velocity contours with 3mm axial gap between Nozzle and Rotor blades.

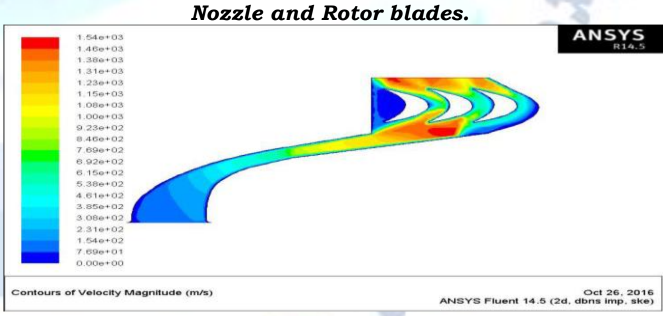

Figure: Velocity contours with 4mm axial gap between Nozzle and Rotor blades.

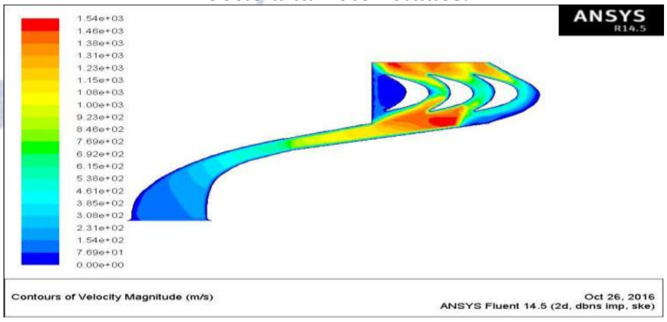

Figure: Velocity contours with 5mm axial gap between Nozzle and Rotor blades.

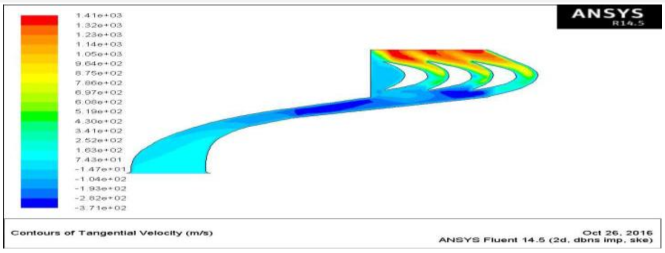

Figure: Tangential velocity contours with 3mm axial gap between Nozzle and Rotor blades.

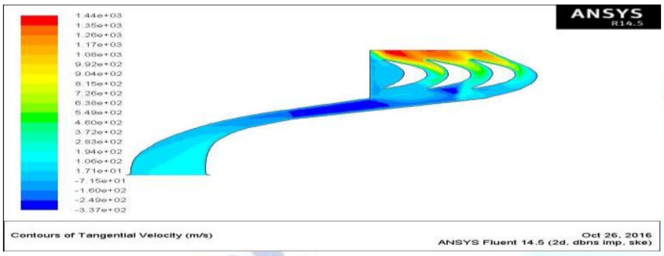

Figure: Tangential velocity contours with 4mm axial gap between Nozzle and Rotor blades.

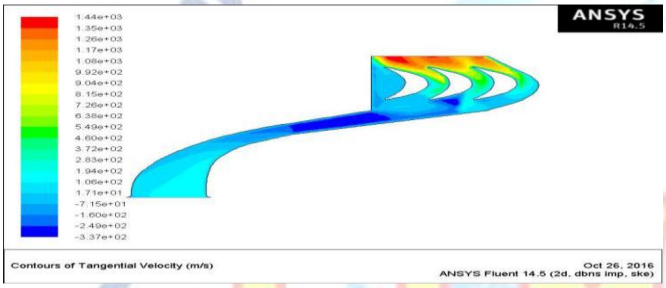

Figure:: Tangential velocity contours with 5mm axial gap between Nozzle and Rotor blades.

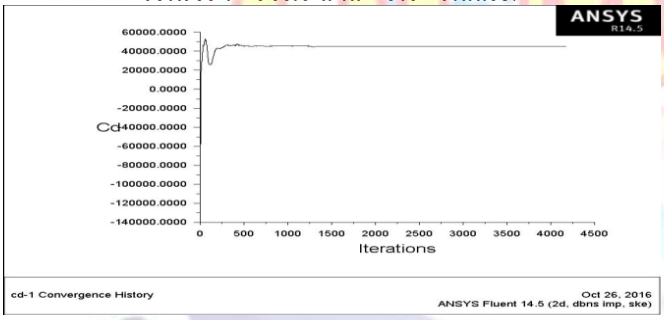

Figure: Co-efficient of drag with 3mm axial gap between Nozzle and Rotor blades.

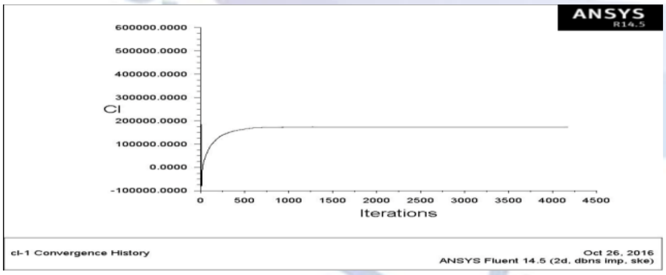

Figure: Co-efficient of lift with 3mm axial gap between Nozzle and Rotor blades.

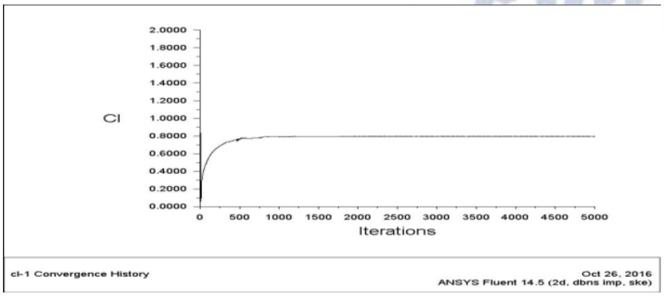

Figure: Co-efficient of lift with 4mm axial gap between Nozzle and Rotor blades

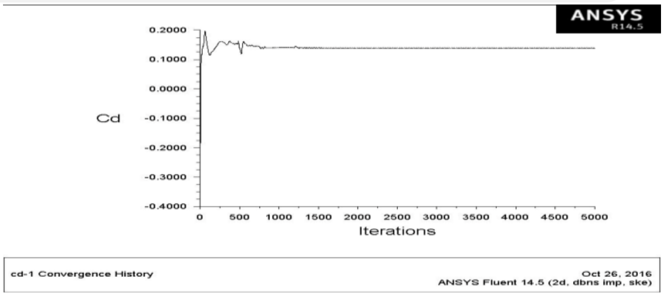

Figure: Co-efficient of drag with 4mm axial gap between Nozzle and Rotor blades

IV. LIMITATIONS AND SCOPE

- CD Nozzle flow is carried out for given pressure ratio and input output conditions as specified.
- The above nozzle used for this purpose is a special type manufactured by N.S.T.L for their special applications.
- During the study of nozzle and turbine the flow of gases is assumed to be at its full length and turbine blades are assumed under static condition.

V. CONCLUSION & FUTURE SCOPE

For the present study, the following conclusions are drawn:

- CFD results of convergent divergent nozzle were in good agreement with the experimental values and theoretical values and the nozzle is acting as a super sonic nozzle.
- CFD predictions for convergent – divergent nozzle and turbine rotor find good agreement with the experimental results of N.S.T.L

Future work should focus on employing a finer resolution grid than the one employed in present study different turbulence models can be tested in the simulations.

- Dynamic mesh can employ for further simulations.
- Nozzle can be studied for different N.P.R to meet the other range of knots.
- Further research can be done under dynamic condition of rotor.

REFERENCES

[1] Bong-Gunshin, Young-Jae Kwak, Kui-soon Kim and Jin-Han Kim, “A study on Nozzle – Rotor interaction losses for geometric conditions of a partial admission impulse turbine”. Proceedings of Asian joint conference on propulsion and power 2006 Beijing, China, April 20-23, 2006.

[2] A.A.Khan and T.R.Shem bharkar, "Viscous flow analysis in a Convergent – Divergent nozzle". Proceedings of the international conference on Aerospace science and Technology, Banglore, India, June 26-28, 2008.
[3] H.K.Versteeg and W.Malala Sekhara, "An introduction to Computational Fluid Dynamics", British library cataloguing pub, 4th edition, 1996.
[4] David C.Wil Cox, "Turbulence modeling for CFD" Second Edition 1998.
[5] S. Majumdar and B.N.Rajani, "Grid generation for Abritrary 3-D configuration using a Differential Algebraic Hybrid Method, CTFD Division, NAL, Bangalore, April 1995.
[6] Layton, W. Sahin and Volker.J, "A problem solving Approach using Les for a backward facing – step" 2002.
[7] M.M.Atha vale and H.Q. Yang, "coupled field thermal structural simulations in Micro Valves and Micro channels" CFD research corporation.
[8] Lars Davidson, "An introduction to turbulence Models", Department of thermo and fluid dynamics, Chalmers university of technology, Goteborg, Sweden, November, 2003.
[9] Mas Mohas Rai, "Three-Dimensional Navier – Stokes Simulations of turbine Rotor-Stator interaction, NASA Ames Research Center, Moffett Field, California Vol.5, No.3, May – June 1989.
[10] Kazuhiro Nakahashi, "Navier – Stokes Computations of two and three dimensional cascade flow fields", Vol.5, No.3, May – June 1989.
[11] Philip C.E. Jorojenson and Rodrick V.China, " Explicit Range–Kutta Method for unsteady Rotor Stator Interaction", Vol.6, No.6, June 1989.
[12] Md.Akhtaruzzaman Sarker and David G.Rhodes, "3D free surface model of laboratory channel with Rectangular Broad – crested weir", proceedings 28th IAHR Congress, Graz, Austria, 1999.
[13] Leonid Moroz, Yuri Gororuschenko, Petr Pagur, "Axial turbine stanges design: ID/2D/3D simulation, Experimal, Optimization", Proceedings of GT 2005 ASME Turbo Expo 2005: Nerada USA.
[14] Nicholas J. Georgia dis, Teryn W.Dal Bellow, chorles J.Tretry, and Albert L.Johns, "Aerodynamic Design and analysis of High performance nozzles for Mach 4 Accebrator Vehicles", 44th AIAA Aero Space Sciences meeting and exhibit, 9-12 January 2006, Nerada.
[15] Eric Gamble and Dan Haid, "Improving off Design nozzle performance using fluid injection" AIAA-2004 – 1206 USA.
[16] K.Sukati and Y.Shinizu, "Effect of the increase in the entrance convergent section length of the gun nozzle on the high velocity oxygen fuel and cold spray process", 24th October 2000 ASM international.
[17] E.V.Volpe, L.C. decastro sartos "Boundary and internal conditions for adjoins fluid – flow problems". 29 June 2007 Springer science Business Media B.V. 2008.
[18] B.Jodoin, "Cold Spray nozzle Mach number limitation', February 14 2001, ASM international.