Cfd Analysis of Convergent- Divergent Supersonic Nozzle

Ms. B.Krishna Prafulla[1], Dr. V. Chitti Babu [2] and Sri P. Govinda Rao [3]

[1]*Assistant Professor, Department of Mechanical Engineering, GMRIT, RAJAM, Srikakulam, AP*

[2, 3] *Associate Professor, Department of Mechanical Engineering, GMRIT, RAJAM, Srikakulam, AP*

ABSTRACT:

In the present work, CFD analysis of flow within, Convergent–Divergent rectangular supersonic nozzle and supersonic impulse turbine with partial admission have been performed. The analysis has been performed according to shape of a supersonic nozzle and length of axial clearance, with an objective of investigating 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 with in nozzle and the extent of flow expansion. Therefore nozzle-rotor interaction losses can be decreased by selecting appropriate length of axial clearance.

The present work has been carried out in two stages: Part 1 consists of modeling and analysis of flow for rectangular convergent divergent supersonic nozzle. Part 2 of the work is on prediction of optimal axial gap between the Nozzle and rotor blades by allowing the above nozzle flow. In the present study, flow through the convergent divergent nozzle study is carried out by using a finite volume commercial code, FLUENT 6.2. The nozzle geometry modeling and grid generation has been done using GAMBIT 2.3 Software. Computational results are in good agreement with the experimental results taken from the literature.

Keywords— *CFD, Modelling, Fluent, Gambit*

I. INTRODUCTION AND LITERATURE REVIEW

Many commercial software packages are being used in the design as well as analysis processes which not only save lead time and costs of new designs, but also are used to study systems where controlled experiments are difficult or impossible to perform. In the area of fluid dynamics, there are many commercial computational fluid dynamics (CFD) packages available for modeling flow in or around objects. Combined with the use of wind tunnel test data, CFD can be used in the design process to drive geometry change instead of being used mainly as a design validation tool. One of the most critical requirements for any CFD tool used for thermal applications is the ability to simulate flows along nozzles, turbines. Such challenging features as pressure gradients, shocks, velocity distribution, eddy location, stream line curvature, and stream wise vortices pose a challenge for computation. The small margins of improvement that are usually targeted in nozzle and turbines design today require precise tools capable of discerning small differences between alternative designs. Custom modelling tools that are based as simplified numerical methods and assumptions cannot provide the accuracy that can be obtained with CFD, which offers mainly inherent advantages for e.g.: it offers quick and cheap solutions in comparison to experimental solutions and more accurate in comparison to empirical methods used in design. Accurate simulation of flows through the nozzle is important for prediction of velocity pattern and pressure patterns.

Shin et *al.* [1] analyzed flow within supersonic impulse turbine with partial admission and concluded that turbine performance is depending upon the shape of nozzle, shape of rotor blades, and exile gap. Khan and Shenbharkar [2] in their paper opinions that one-dimensional in viscid theory does not reveal the complex flow features in a choked CD nozzle accurately. The code fluent has been used to compute RANS flow in a 2-D CD nozzle for nozzle pressure ratios (NPR) corresponding to presence of shock inside the diverging part of the nozzle. The computed solutions differ from the simple theory so for as shock location, shock structure and after – shocks are concerned. Majumdar and Rajani [3] evaluated grid generation for arbitrary 3-D configuration using a differential algebraic hybrid method. Layton and Sahin [4] developed problem solving approach using large Eddy simulation solver (LES) for flow over a back step.

Philip et *al.* [5] conducted a Quasi three-dimensional rotor/stator analysis for blade – to – blade flows in turbo machinery. The Bald winb lomox eddy-viscosity model is used for turbulent flows. B.Jodoin[6], conducted an extensive analytical study of the nozzle geometry, based on the one dimensional flow model to determine the optimal nozzle shape given the conditions, powder properties, and nozzle length. It was shown that the spray particle velocity is relatively insensitive to the nozzle shape and therefore that a single nozzle can be used for variety of operating conditions, as long as no shock waves are present in the nozzle.

The current study aims analysis of flow through the nozzle and prediction of optimal axial clearance. Solutions of flow along the nozzle involve only one phase of gas. Results are verified with the experimental data. In the present work nozzle study has been carried out and by using same nozzle, axial gap (clearance determination) has also been analysed.

METHODOLOGY AND MODELLING

To resolve a turbulent flow by **direct numerical simulation** requires that all relevant length scales be resolved from the smallest eddies to scales on order of the physical dimensions of problem domain. To get statistical quantities of interest, which cannot be measured experimentally, can be evaluated from the simulations (**dalea Anderson**). DNS is used for developing an understanding of the physics of the flow and deployed in developing turbulence models for simple flows. However, from an engineering point of view, It provides for more information than an engineer needs, and it is simply too expensive to be employed on a regular basis.Most engineering flows of interest are in the turbulence regime, which contain a wide range of length and times scales. In **large eddy simulation** the large eddies are computed and the smallest eddies are modeled, as the large –scale motions are more energetic than the small scale and are responsible for most of the important.

The small scale turbulence serves mainly to drain energy from the large scales through the cascade process, and is more universal land nearly isotropic, which makes it more suitable to be modeled the computational effort required for les is less than that of DNS by approximately a factor of 10 using present methods.The main thrust of present day research in computational fluid dynamics in turbulent flows is through the time-averaged Navier Stokes equations. These equations are referred to as the Reynolds equations of motion or the Reynolds averaged Navier Stokes (RANS) equations. Time averaging the equations of motion give rise to new terms, which can be interpreted as' apparent" stress gradients associated with the turbulent motion. These new quantities must be related to the mean flow variables through turbulence models.

The Reynolds equations are derived by decomposing the dependent variables in the conservation equations into time mean (obtained over an appropriate time interval) and fluctuating components and then time averaging the entire equation.The model development is carried out on gambit. It is the pre processing process where the model development is done and meshing of model is followed for further analysis. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing and assigning zone types to a model simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications.

The various components of the turbine are designed using GAMBIT. The various components include for present analysis is C-D nozzle and series of three blades of rotor. The component nozzle is modeled individually, the combined assembly of nozzle and blade modeling have been carried separately. The profiles are generated with the help of coordinates available which have been generated. All these components are modeled and meshing is done separately.

1.1 Modeling of the Super Sonic Nozzle:

The coordinates are provided in Table [1.1],[1.2],[1.3]for development of the 3D model of the supersonic nozzle. The profile of the nozzle has been shown in Fig. 1.1. For design purpose, the nozzle can be seen as an assembly of three separate sections operating in series a converging section, a throat, and finally the diverging section. In present analysis rectangular nozzle which stacks up 2-D supersonic nozzle profile in normal direction as shown in figure [5.4].Different models of nozzle can be observed in figures[5.2]&[5.3]

Sl.No.	X (mm)	Y (mm)
1	4.000	0.000
2	5.000	1.000
3	5.000	2.854
4	6.810	9.191
5	14.639	17.129
6	25.915	23.387
7	37.497	27.252
8	54.685	32.100
9	78.997	32.100
10	63.282	28.688
11	38.482	23.303
12	29.197	19.638
13	23.628	15.216
14	21.553	11.526
15	20.525	5.000
16	20.525	1.000
17	21.525	1.000

TABLE 1.1: Coordinates For End Points Of Nozzle Profile

CENTERS	X (mm)	Y(mm)	RADIUS (mm)
C1	4.000	1.000	1.000
C2	17.000	2.854	12.000
C3	31.531	-7.360	29.750
C4	48.045	-29.780	57.589
C5	56.123	-47.842	77.370
C6	-47.118	-12.171	36.510
C7	39.334	1.158	21.079
C8	30.695	8.812	9.536
C9	50.985	2.284	38.556
C10	21.525	1.000	1.000

TABLE 1.2: Coordinates For Arc Centers

SL.No.	X (mm)	Y (mm)	Z (mm)
1	0	0	0
2	0	32.1	0
3	90	32.1	0
4	90	0	0
5	0	0	22
6	0	32.1	22
7	90	32.1	22
8	90	0	22

TABLE 1.3: Coordinates for block on which nozzle profile is cut for 7.1 mm depth

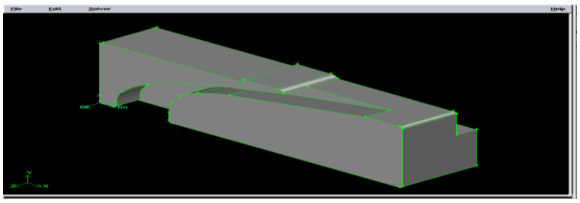

Fig. 1.1 Nozzle Profile

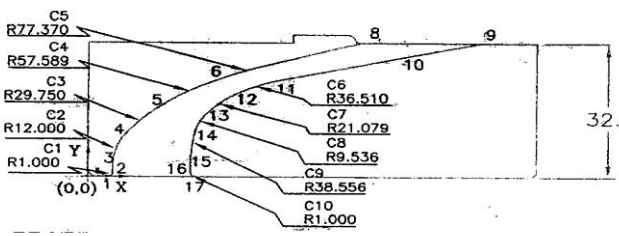

Fig. 1.2: Solid Model of Nozzle Profile Made on a Block.

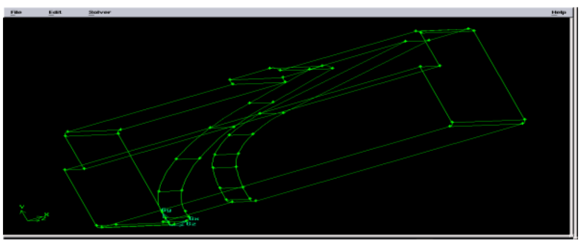

Fig. 1.3: Wire Frame Model of Nozzle Profile Made on a Block.

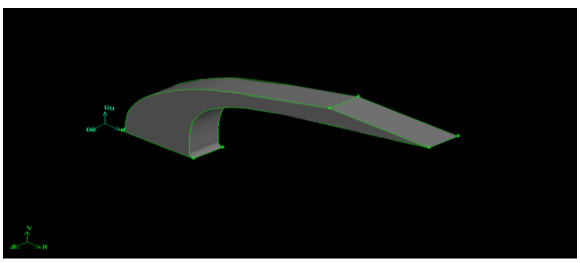

Fig. 1.4: Solid Model of nozzle Profile.

1.2 Modeling of the Blade Profile

The 2D model of blade profile is as shown in figure [1.5].Modeling of blade 2D is carried out by the coordinates provided table [1.4],[1.5]. A series of three blades is shown in figure[1.6] on which nozzle flow will impinges.

S.No.	*X (mm)*	*Z (mm)*
1	2.285	1.440
2	4.657	3.419
3	6.064	5.373
4	6.248	5.309
5	4.699	1.385
6	3.427	-1.235
7	2.960	-2.023
8	2.491	-2.636
9	2.017	-3.059
10	1.063	-3.604
11	-0.383	-3.730
12	-1.356	-3.373
13	-2.337	-2.373
14	-3.331	-1.249
15	-4.343	0.944
16	-5.744	5.119
17	-5.582	5.212
18	-5.021	4.516
19	-3.538	2.844
20	-0.362	1.039

TABLE5.4: Arc End Co-Ordinates

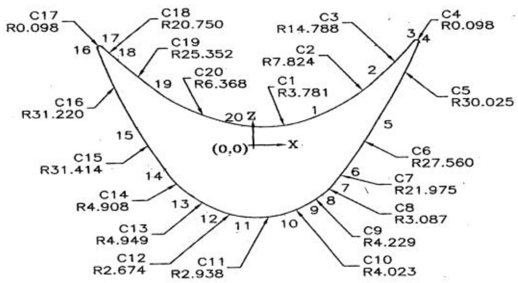

Fig. 1.5: Blade Profile.

CENTERS	X(mm)	Z (mm)	RADIUS (mm)
C1	0.432	4.736	3.781
C2	-1.443	8.319	7.824
C3	-6.601	13.009	14.788
C4	6.153	5.331	0.098
C5	-22.385	14.346	30.025
C6	-20.693	12.099	27.560
C7	-15.716	9.556	21.975
C8	0.295	-0.467	3.087
C9	-0.437	0.129	4.229
C10	-0.437	0.129	4.023
C11	0.093	-0.831	2.938
C12	0.032	-1.088	2.674
C13	1.119	0.912	4.949
C14	1.089	0.884	4.908
C15	24.666	12.998	31.414
C16	24.481	12.941	31.220
C17	-5.651	5.144	0.098
C18	-21.447	-8.162	20.750
C19	14.666	20.489	25.352
C20	1.065	7.245	6.368

TABLE 1.5: Arc Center Coordinates

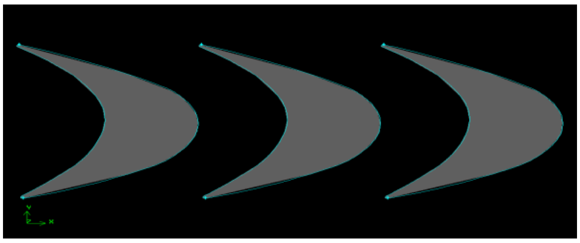

Fig. 1.6: Series of Rotor Blade.

1.3 Modeling of Nozzle and Turbine Blades as Partial Admission Case:

2D modeling of combined nozzle and series of 3 blades in a partial admission turbine rotor is carried out by using the nozzle and blade profile generated data provided in Tables [1.1],[1.2],[1.3],[1.4]and [1.5]. 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 Figs. [1.7],[1.8] and [1.9] for 3mm,4mm & 5mm axial gaps respectively.

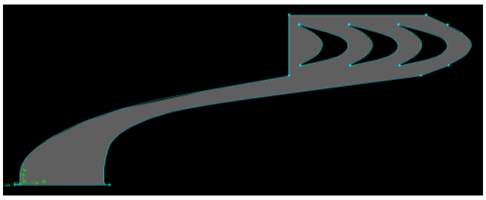

Fig. 1.7: Solid Model as a Partial Admission Case of Nozzle and Rotor Blades with 3mm axial gap.

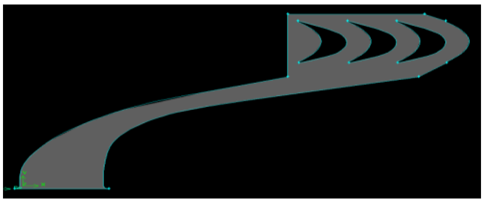

Fig 1.8: Solid Model as A Partial Admission Case of Nozzle and Rotor blades with 4mm axial gap.

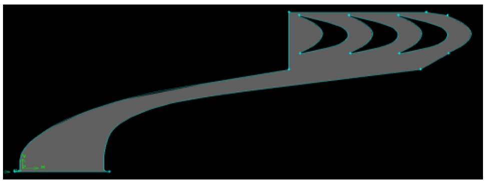

Fig. 1.9: Solid Model as a Partial Admission Case of Nozzle and Rotor Blades with 5mm axial gap.

II. GRID GENERATION AND BOUNDARY CONDITIONS

The first step of the actual work is the geometry creation and then considers a certain domain of the fluid in which the body is present under analysis. Then the volume occupied by the body as well as fluid is divided into discrete cells called meshing Physical modeling is then defined i.e., equations of motion, no. of phases in the flow, turbulence model etc. boundary conditions are defined now. This involves specifying the fluid behavior and properties at the boundaries of the problem. Then the simulation is started and the equations are solved iteratively as steady state or transient. Finally a post processor is used to view the results. These steps are followed for both C-D nozzle and for combined nozzle and rotor blades.

2.1 Mesh Generation for 3D C-D Nozzle

The flow domain is required to be discretized to convert the partial differential equations in to series of algebraic equations. This process is called grid generation. A 3D Model is created in GAMBIT as shown in Fig.1.4. To generate the structured grid as shown in Fig.2.1 with hexahedral cells gambit is used. This mesh are in good agreement with turbulence model gave good results, which otherwise didn't match, with the experimental ones.

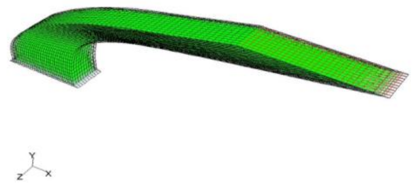

Fig. 2.1 Meshed View of 3 D Nozzle Profile

2.2 Mesh Generation for 2D C-D nozzle

A 2D nozzle profile is modeled in GAMBIT. To generate the structured grid as shown in figure 2.2 with quadrilateral elements Gambit is used. This mesh are in good agreement with turbulence model gave good results, which otherwise didn't match with the experimental ones. Nozzle inlet as pressure inlet and outlet as pressure outlet and remaining as walls the nozzle path are defined as fluid. File is saved for analysis in fluent.

A 2D model of the rotor blades and nozzle for different axial gaps of nozzle exit and turbine rotor blades are modeled in GAMBIT as shown in Figs. 1.7,1.8 and 1.9. To generate the structured grid with triangular elements Gambit is used. This mesh are in good agreement with the different model adopted gave good results, which other wise didn't match with the experimental ones the boundary conditions are defined asgiven. Inlet as pressure inlet and outlet as pressure outlet remaining as walls. Meshed models of a partial admission Case of Nozzle and rotor blades with axial gaps of 3mm, 4mm and 5mm have also been shown in Figs.2.3, 2.4 and 2.5.

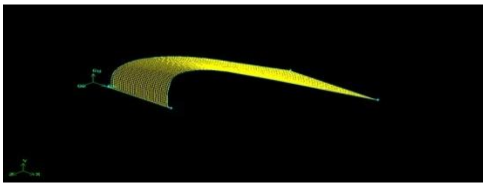

Fig. 2.2 Meshed View of 2 D Nozzle Profile

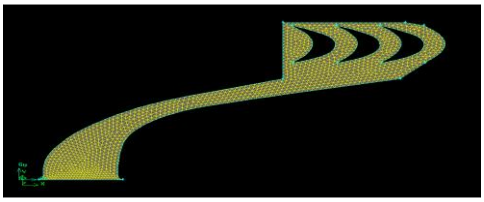

Fig. 2.3: Meshed View as a Partial Admission Case of Nozzle and Rotor Blades With 3mm Axial Gap.

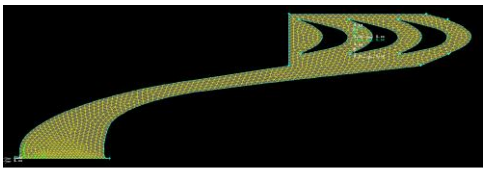

Fig. 2.4 Meshed View as a Partial Admission Case of Nozzle and Rotor Blades With 4mm Axial Gap.

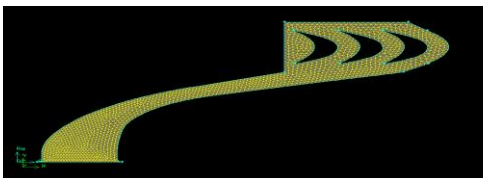

Fig. 2.5 Meshed View as a Partial Admission Case of Nozzle and Rotor Blades With 5mm Axial Gap.

III. ANALYSIS

The analysis is carried in fluent software by importing the meshed file saved in Gambit. The steps that are followed are given below which include all the conditions and the boundaries value for the problem statement.

3.1 Analysis of C-D Rectangular Nozzle

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. Grid check is followed by smoothing and swapping of grid. Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done. 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 in to consideration. The viscous medium is also taken. They analysis is carried using K-epsilon turbulence model. Material selected is gas. The properties of gas taken as follows: 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 in to consideration.

Pressure inlet was taken as inlet for nozzle the value of pressure is 8101325 Pascal. Initial gauge pressure was taken as 7898681 Pascal. Temperature was taken as 1583K. The nozzle outlet is set as pressure outlet with a value of $13e^{5}$. The solution controls are set as listed below: The under relaxation factor was set as given. Turbulence Kinetic Energy as 0.8, Turbulence Dissipation rate 0.8 and Turbulence Viscocity as 1. Discretization Equation is selected as Flow (Second order up wind), Turbulence Kinetic Energy (1^{st} order upwind), Turbulence dissipations rate (1^{st} order upwind). Solution initialization has also been done. Initial values of velocity are taken as 186.3 m/s for 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 Fig. 3.1.

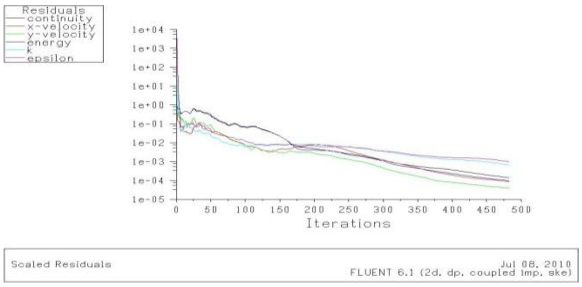

Fig. 3.1 Convergence History for C-D Nozzle.

3.2 Analysis of Nozzle and Turbine Rotor Blades as a Case of Partial Admission

The analysis is carried in fluent by importing the meshed file saved in Gambit. The steps that are followed are given below which include all the conditions and the boundaries value for the problem statement, for varied axial gaps of nozzle and turbine rotor blades as a case of partial admission. 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 following which smoothing and swapping of grid is done.

Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done. 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 in to consideration. The viscous medium is also taken. They analysis is carried using K-epsilon turbulence model. The selection of material is done. Material selected is gas. The properties of gas taken as follows: 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 in to consideration. Pressure inlet was taken as inlet for nozzle the value of pressure is 8101325 Pascal. Initial gauge pressure was taken as 7898681 Pascal. Temperature was taken as 1583K. The outlet is set as pressure outlet with a value of 101325 Pascal. 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 Pressure-standard, Density -1^{st} order upwind, Momentum-1^{st} order upwind, Turbulence Kinetic Energy (1^{st} order upwind), Turbulence dissipations rate (1^{st} order upwind), Energy -1^{st} order upwind.

Solution initialization has also been done. Initial values of velocity are taken as 186.3 m/s for 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 iteration continues till the convergence is reached and convergence history has been shown in Figs. 3.2, 3.3 and 3.4 respectively for 3mm,4mm & 5mm axial gaps of nozzle and turbine rotor blades as a case of partial admission.

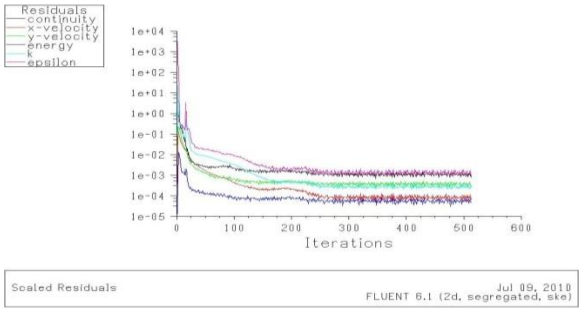

Fig 3.2 Convergence history as a Partial admission case of nozzle and rotor blades with 3mm axial gap.

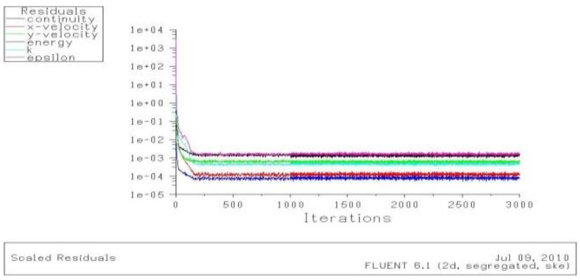

Fig. 3.3: Convergence history as a Partial admission case of Nozzle and rotor blades with 4mm axial gap.

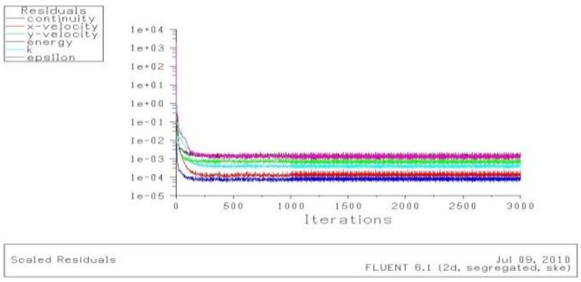

Fig. 3.4: Convergence history as a Partial admission case of Nozzle and rotor blades with 5mm axial gap.

IV. RESULTS AND DISCUSSION

Nozzle profile which is examined in 2D is as shown in Fig. 1.1. Results shown 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 1436 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 Fig. 4.1. The velocity contours of nozzle are plotted in Fig. 4.2, the pressure contours of nozzle is plotted in Fig.4.3, and temperature contours of nozzle are plotted in figure4.4. The velocity, temperature, Mach number, pressure variation along the nozzle is compared with theoretical calculation and with experimental results obtained from the literature.

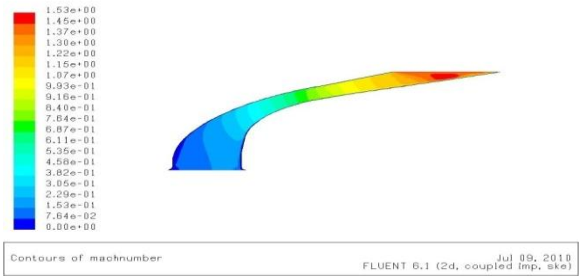

Fig. 4.1: Mach number contours of nozzle.

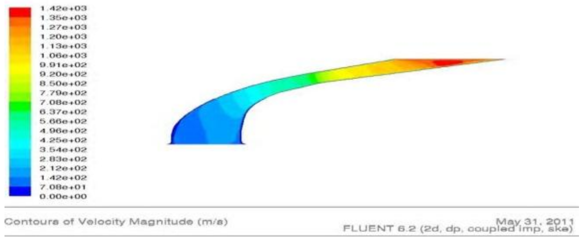

Fig. 4.2: Velocity Contours of Nozzle.

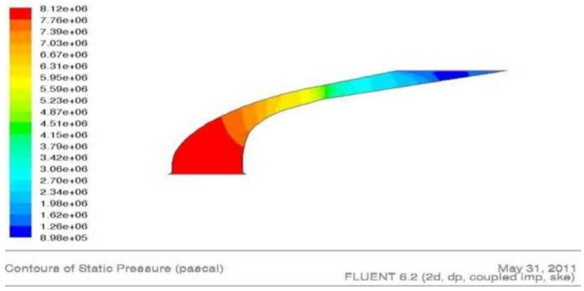

Fig. 4.3: Pressure Contours of Nozzle.

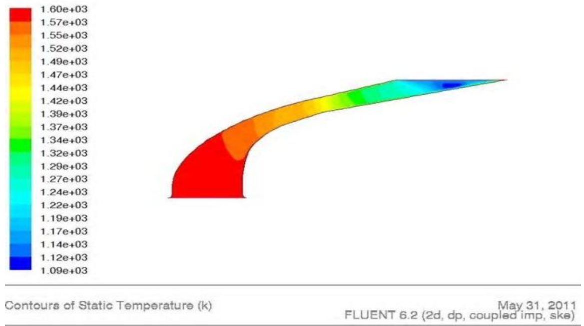

Fig. 4.4: Temperature Contours of Nozzle.

Flow passage form 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 as shown in Figs 4.5, 4.6 and 4.7 respectively for 3mm,4 mm and 5mm axial gap respectively. Velocity vector contours are shown in Figs. 4.8, 4.9 and 4.10 respectively for 3mm,4mm and 5mm axial gap respectively. Tangential velocity contours for 3mm, 4mm, 5mm gap axial clearances of nozzle and turbine rotor blades are shown in Figs. 4.11, 4.12, 4.13, 4.14, 4.15and 4.16. The maximum average tangential velocity will act at 3mm axial clearance and hence it can be deduced that 3mm gap of axial clearance will be the better one.

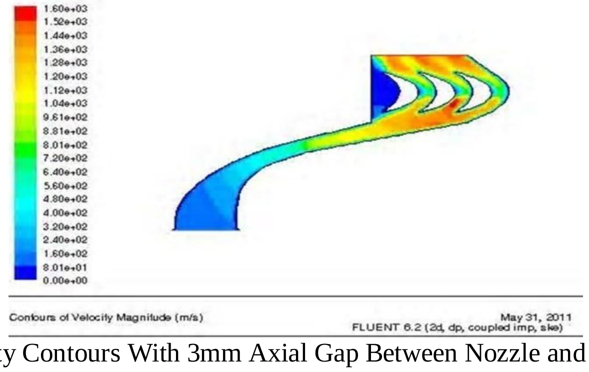

Fig. 4.5: Velocity Contours With 3mm Axial Gap Between Nozzle and Rotor blades.

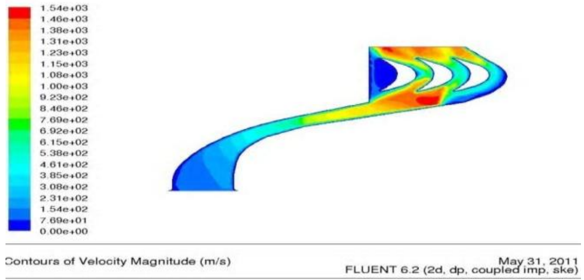

Fig. 4.6: Velocity Contours With 4mm Axial Gap Between Nozzle and rotor blades.

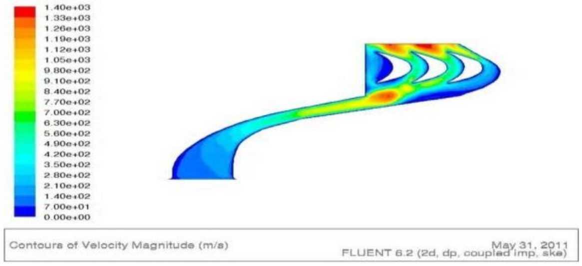

Fig. 4.7: Velocity contours with 5mm axial gap between nozzle and rotor blades.

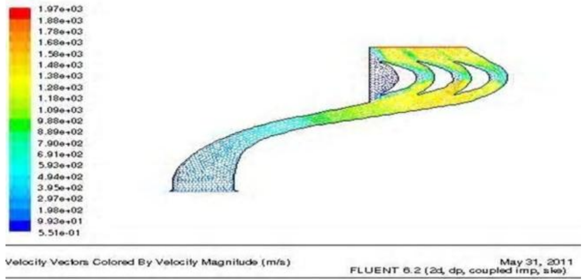

Fig 4.8 Velocity vectors with 3mm Axial gap between Nozzle and Rotor blades.

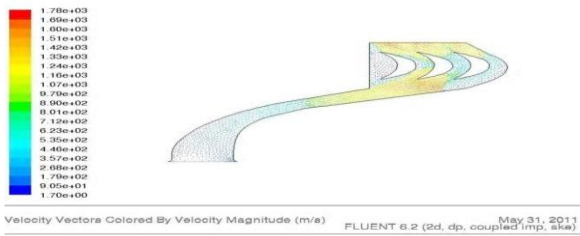

Fig. 4.9: Velocity vectors with 4mm Axial gap between Nozzle and Rotor blades.

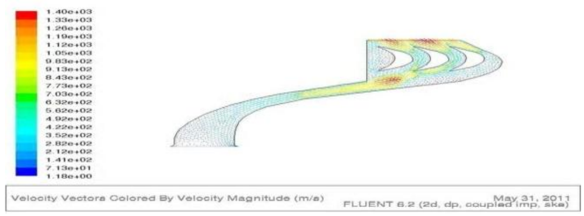

Fig. 4.10: Velocity vectors with 5mm Axial gap between Nozzle and Rotor blades.

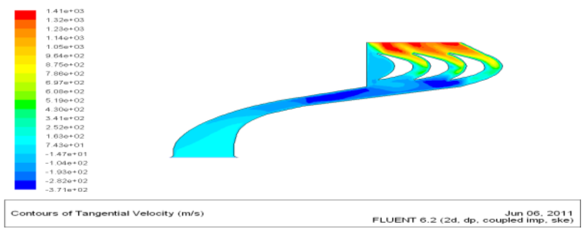

Fig. 4.11: Tangential velocity contours with 3mm axial gap between Nozzle and Rotor blades.

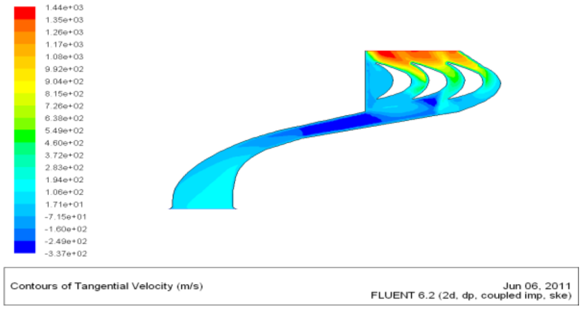

Fig. 4.12: Tangential velocity contours with 4mm axial gap between Nozzle and Rotor blades.

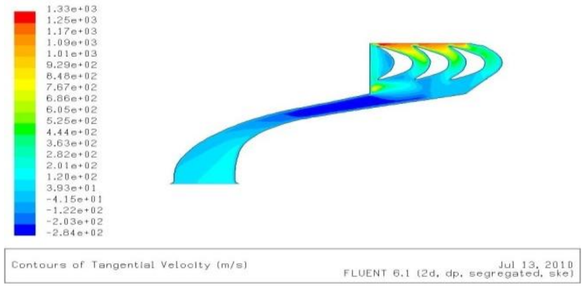

Fig. 4.13 Tangential velocity contours with 5mm axial gap between Nozzle and Rotor blades.

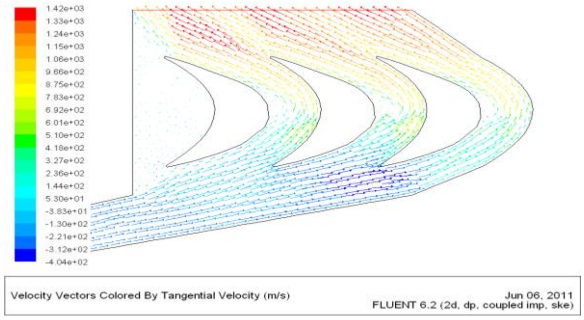

Fig. 4.14: Tangential velocity vectors with 3mm Axial gap between Nozzle and Rotor blades.

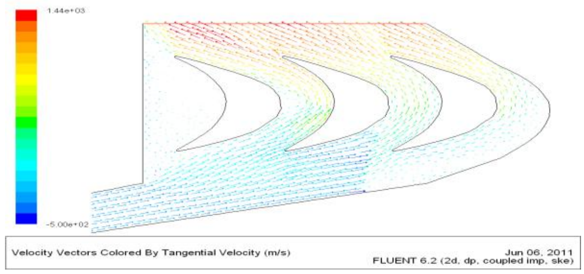

Fig. 4.15: Tangential velocity vectors with 4mm Axial gap between Nozzle and Rotor blades.

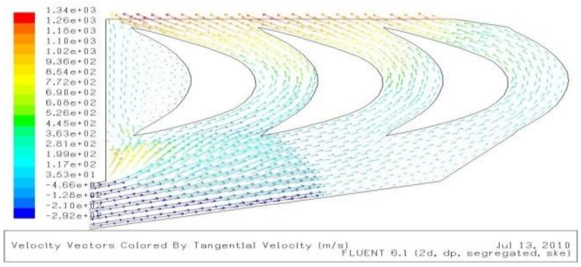

Fig. 4.16 Tangential velocity vectors with 5mm axial gap between Nozzle and Rotor blades.

V. LIMITATIONS

- CD Nozzle flow is carried out for given pressure ratios and input, output conditions as specified.
- The above nozzle used for this purpose is a special type manufactured by Naval Science and Technological Laboratory, Visakhapatnam, India for their special applications.
- During the study of nozzle and turbine the flow of gases is assumed at full length and turbine blades are assumed to be under static condition.

VI. CONCLUSIONS

- CFD results of convergent divergent nozzle were in good agreement with the experimental values and theoretical values and the nozzle is acting as a supersonic nozzle.
- CFD predictions for convergent – divergent nozzle and turbine rotor find good agreement with the experimental results of Naval Science and Technological Laboratory, Visakhapatnam.

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 Aero space science and Technology, Banglore, India, June 26-28, 2008.

[3] 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.

[4] 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.

[5] 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.

[6] B.Jodoin, "Cold Spray nozzle Mach number limitation', February 14 2001, ASM international. A. Husain, D. K. Sehgal, and R. K. Pandey, "An inverse finite element procedure for the