International Journal of Research in Science and Technology http://www.ijrst.com

(IJRST) 2017, Vol. No. 7, Issue No. III, Jul-Sep **e-ISSN: 2249-0604, p-ISSN: 2454-180X**

AERODYNAMIC DESIGN OF MACH 5 NOZZLE IN LARGE HYPERSONIC WIND TUNNEL

***Vandana Vidyadhari, **S.Rajakumar**
**PG Scholar, Aeronautical Engineering*
*** Assistant Professor, Mechanical Engineering*
Anna University, Tirunelveli

ABSTRACT

The paper involves a design of Mach 5 Nozzle from an existing Mach 6 Nozzle of a large Hypersonic Wind Tunnel in Vikram Sarabhai Space Centre, Thiruvananthapuram in order to obtain the simulation of Mach 5. The work deals with design of nozzle contour using specific boundary conditions, where by changing minimum numbers of existing nozzle components. A new contour design can be arrived without sacrificing exit flow quality of the nozzle. Mach 6 nozzle is converted into Mach 5 nozzle by changing 3 nos. of segments viz. convergent, throat and 1st divergent segment in order to reduce cost of design, installation and manufacturing. Axisymmetric CFD flow analysis of Mach 5 contour is done in Fluent to confirm the flow uniformity at the nozzle exit.Different contours are generated by varying the throat location and radius of curvature of throat. The flow simulation carried out using CFD for the different contours in order to obtain the best flow quality at the nozzle exit.

Keywords: axisymmetric nozzle; contour; flow uniformity; flow quality; grid sensitivity analysis

INTRODUCTION

ANozzle is a conduit with a variable cross-sectional area through which a fluid accelerates to a high-velocity stream. The purpose of wind tunnel nozzle is to expand high pressure air to desired Mach number and to generate uniform flow at the test section to carry out aerodynamic simulation of scale down model of aerospace vehicles. A convergent-divergent nozzle is used to obtain supersonic flow with $M > 1$ (when choked at throat). The VSSC Hypersonic Wind Tunnel is an Intermittent Blow-down type wind tunnel with flow simulation capability of Mach number range 6 to 12 with an enclosed free jet type. The compressed air is allowed to enter the settling chamber where the flow is settled by reducing turbulent fluctuations. A model incidence mechanism is used to provide desired attitude to the model and to inject the model after the flow is established.Mach 6 nozzle of the wind tunnel has one convergent, throat and six divergent segments. It is planned to replace convergent, throat and divergent segment 1 to convert Mach 5 nozzle.The intended flow qualities of the Mach 5 nozzle contour are flow angularity of <0.2 deg and Mach no. uniformity of < 1.5% at the nozzle exit.

NOZZLE DESIGN METHODOLOGY

The nozzle design procedure makes use of determination of throat radius deduced from Area-Mach number relationship as Mach number is a function of Area ratio where nozzle exit area (A) can be calculated with the known exit diameter (d).

$$\left(\frac{A}{A_t}\right)^2 = \frac{1}{\mathrm{M}^2}\left[\frac{2}{\gamma+1}\left(1+\frac{\gamma-1}{2}\mathrm{M}^2\right)\right]^{\frac{\gamma+1}{\gamma-1}} \qquad (1)$$

The radius of throat is $0.1d$ for the contour generation of axisymmetric nozzle in theoretical calculation. Boundary layer correction [1] of $0.097d$ is applied on the throat radius using CFD result. The Mach 5 contour is defined using a 5th order polynomial curve fit using the boundary conditions at the starting of convergent segment, throat and end of 1st divergent segment. The boundary conditions are used are: (i) slope and radius at the starting of convergent segment; (ii) radius and radius of curvature of throat and (iii) radius, slope and 2nd derivative at the end of 1st divergent segment [5,9]. Assuming a curve equation fitting the above boundary conditions as,

$$y = \sum_{n=0}^{n} A_n x^n \, , n = [0,5] \qquad (2)$$

Applying the boundary conditions, simultaneous equations are solved using matrix inversion method to obtain the coefficients of the equation. The contours derived by applied above mentioned boundary conditions are shown in the Figure I. Various curve fits using different functions such as Piecewise cubic Hermite Interpolant, Bezier curve, Splines are also studied in order to match the existing Mach 6 contour. It is observed that best fit is achieved using 5th order polynomial.

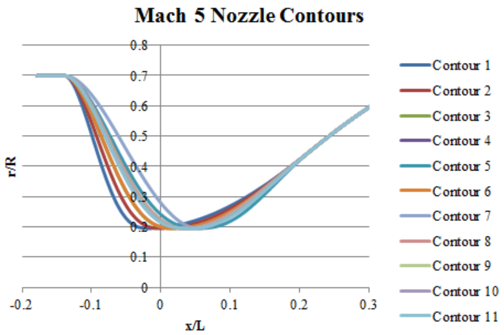

Figure I. Contours of Mach 5 nozzle

The contour generated using the above polynomial equation is modeled using ANSYS Design Modeler. A structured mesh is generated with clustering towards wall as shown in Figure II. Quality of mesh plays a significant role in accuracy and stability of numerical computation. Mesh sizing control over the edges is done are appropriately defined to achieve a wall $y^+ \sim 0.1$.The first cell height from the wall can be calculated over the edge by geometric progression with the help of Bias factor. Bias Factor is defined as the ratio of the largest edge to the smallest edge.

$$\text{Bias factor} = \frac{\text{Largest edge}}{\text{Smallest edge}} = \frac{a_n}{a_1} \qquad (3)$$

Where, n = Number of divisions

Figure II. Meshed model of an axisymmetric nozzle

Basic axisymmetric mesh of the nozzle region consisted of nodes in the axial and vertical directions which were clustered towards nozzle wall and the throat section. This mesh was further improved with a resolution grid of 400x100 in order to obtain y+ at the wall around 0.1 to capture the turbulent boundary layer correctly.

The fluid flow analysis is carried out using the boundary conditions with an inlet pressure and temperature. Density-based implicit solver is used to solve the governing equations of continuity, momentum and energy simultaneously as a set of equations. It is important to consider the energy equation since the problem is to be dealt with compressible flow and temperature effects. Spalart Allmaras is a one-equation turbulence model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. 3-coefficient method of Sutherland law for viscosity is used. In compressible flow, density is not constant and specified as ideal gas. Air, which is treated as a perfect gas with ratio of specific heat (γ=1.4), no-slip viscous condition of constant temperature is introduced at the wall.

The Full Multigrid (FMG) initialization, which can provide a better solution, is used at the start of the calculation, providing initial and approximate solution at a minimum cost to the overall computational expense where flow convergence can be accelerated. The default CFL for the density-based implicit formulation is 5.0. It is observed that a lower CFL is required during startup, but it can be increased as the solution progresses until it obtains convergence. Solution steering in the density-based implicit solver provides navigation of the flow solution from a starting initial guess to a converged solution with minimum user interaction. The type of flow that best characterizes the solution domain and the maximum desired accuracy should be selected, and then allow the solver to take the solution to convergence. Calculations are performed for the case of Mach 5 nozzle with different geometries to compare the calculated flow characteristics from various fluid flow analysis.

RESULTS

Grid sensitivity analysis has been studied and an optimized grid of 400x100 quadrilateral elements is adopted in the numerical analysis. CFD computation is validated using the Mach 6 experimental result as shown in FigureIII. Computed result matches with 5% of the experimental results.

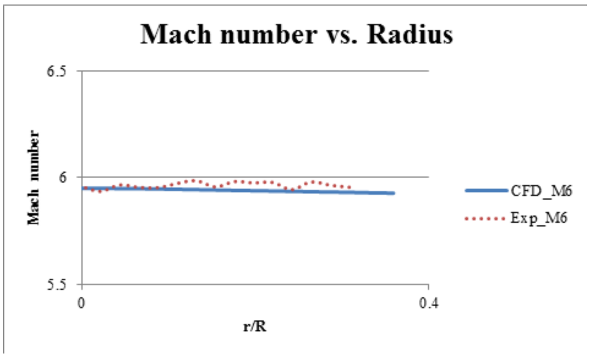

Figure III. CFD vs. Experimental M6 Nozzle results

11 contours are generated and studied which are tabulated in Table I. In the study, throat locations are varied from -0.002 to 0.006, throat curvature ratio (radius of curvature of throat / throat radius) from 10 to 60.

Based on the validation, the numerical simulations were applied to investigate flow field on these contours. It is observed from the computation that flow angle distributions are greatly influenced by the nozzle throat shift. In the core region, the variation is indicated with a slight change in Mach number. The flow non-uniformity is presented as percent deviation w.r.t average exit Mach number. Flow uniformity in terms of Mach no. variation, static pressure and flow angularity are tabulated in Table I w.r.t different test cases.

Table I. Flow uniformity w.r.t to different throat location and radius of curvature ration of throat

Contour No	*Throat Location (x_T/L)*	*Curvature Ratio $\left(\frac{R_c}{R_t}\right)$*	*Divergence angle ($\emptyset_{max}$) in degrees*	*M average*	*ΔM (in %)*	*ΔP (in %)*	*(θ_{max}) in degrees*
1	-0.002	50	9.66°	4.9	0.85	5.64	0.07°
2	0		9.73°	4.9	0.74	4.8	0.08°
3	0.002		9.99°	4.89	0.55	3.56	0.13°
4	0.004		10.71°	4.88	0.24	1.81	0.24°
5	0.006		11.89°	4.86	0.27	1.63	0.44°
6	0.002	40	9.89°	4.89	0.6	3.88	0.37°
7	0.006	10	9.76°	4.89	0.58	3.75	0.04°
8	0.004	20	9.93°	4.89	0.56	3.63	0.1°
9		30	10.35°	4.89	0.39	2.61	0.18°
10		40	10.57°	4.88	0.3	2.11	0.22°
11		60	10.81°	4.88	0.2	1.61	0.26°

The effects of the results from parameter analysis are as follows:

- Exceeds limiting divergence angle (<11 deg) when throat location is shifted towards divergent side keeping constant throat curvature ratio.
- Large pressure variation is observed in the core region when throat is shifted towards convergent side.
- Non-uniform Centerline Mach number distribution when throat location shifted towards convergent side.
- Mach number uniformity increases with decrease in throat curvature ratio
- Increase in flow angularity with respect to increase in throat location shift.

The best design of nozzle can be summarized by Mach flow quality, Mach number and pressure variation. These requirements are accomplished by a nozzle location shifted towards divergent side with a minimum radius curvature ratio. The average pressure, Mach-number deviation is ±3.63%, ±0.56% and the minimal flow angle deviation is ±0.1° are obtained for the nozzle contour of x_T=0.004 with $\left(\frac{R_c}{R_t}\right)$=20.

An average Mach number of 4.89 is achieved at outlet of the nozzle as shown in contour developed in Figure IV. The typical pressure and temperature distribution through the nozzle developed from the 8th contour are shown as contour plots in Figure V, Figure VIand are compared with the values obtained from isentropic relations expressed in the form of:

$$\frac{P_0}{P} = \left(\frac{1}{1+\frac{\gamma-1}{2}M^2}\right)^{\gamma/(\gamma-1)} \qquad (4)$$

$$\frac{T_0}{T} = \left(\frac{1}{1+\frac{\gamma-1}{2}M^2}\right)^{\gamma/(\gamma-1)} \qquad (5)$$

Where,M = Mach number

γ = 1.4 (for air)

P_o, T_o=Inlet pressure and temperature (in Pascal and K).

The pressure, temperature values from the analysis are compared against isentropic values for this contour. The Mach number distribution at centerline is plotted as a function of the axial distance of the nozzle is shown in Figure VII. As the flow moved downstream in the nozzle, it expands to higher Mach number with a trend:

- Of maximum expansion up to inflection point
- With compression waves beyond inflection point
- Flow is made uniform in further segments.

The Mach number distribution in the core region is found to be smooth and continuous along the profile. The exit Mach-number is plotted versus radius where the Mach number distribution of 8th contour with shift in throat location and minimum curvature ratio is shown in Figure VIII. In the essentially inviscid core region of 0 to 700mm in diameter, the percentage of average Mach number for this contour is about 0.56%. The pressure distribution for 8th contour from Table Iis shown in Figure IX. In the core region, the percentage of average pressure value for this contour is about 3.63%. For this nozzle contour, the flow angle is plotted with respect to radius as shown in Figure X. Flow angle is found to be minimal in the 8th contour with a value of 0.1° in contrast to the other contours shown in Table I.

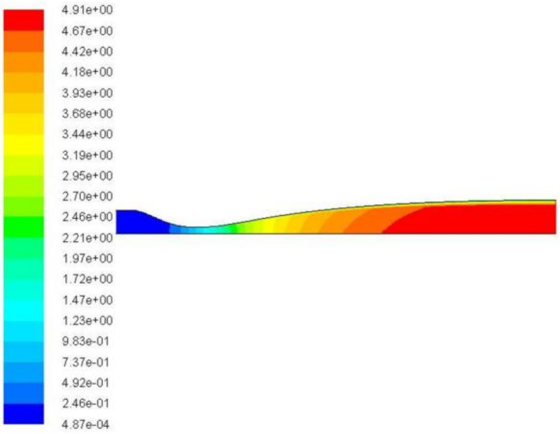

Figure IV. Mach number Contour

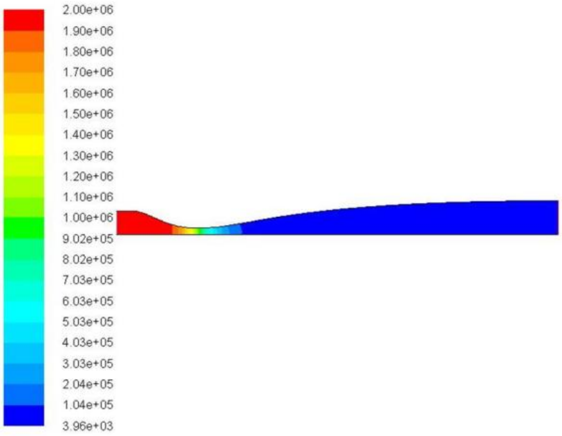

Figure V. Pressure Contour

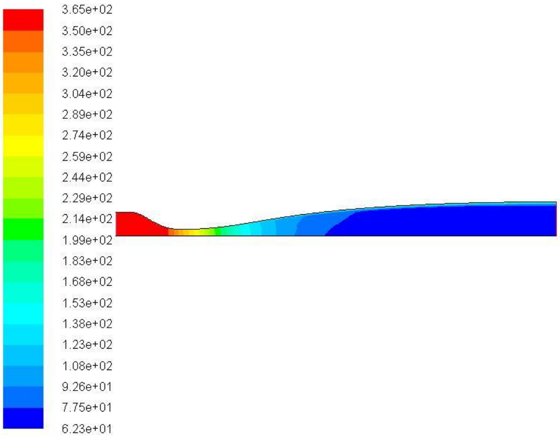

Figure VI. Temperature Contour

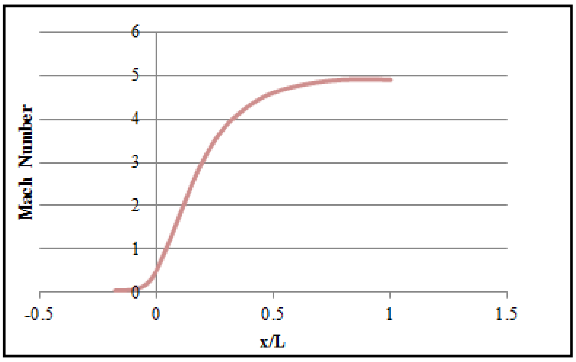

Figure VII.Mach number vs. X Coordinate @Centerline

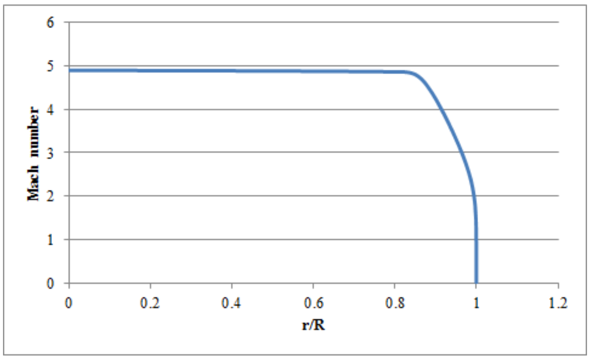

Figure VIII. Mach number vs. R Coordinate @Outlet

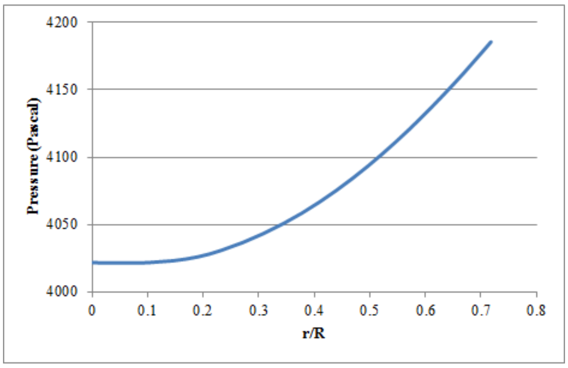

Figure IX. Pressure vs. R Coordinate @Outlet

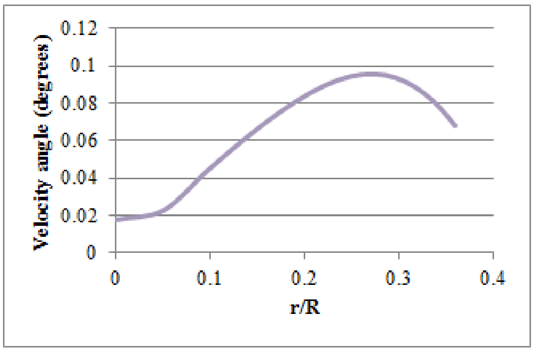

Figure X. Velocity angle vs. R Coordinate @Outlet

CONCLUSION

The Analytical design of the nozzle is studied and CFD analysis of an axisymmetric nozzle is proposed in this work. This involved use of fine mesh and relatively less computing time using Full Multigrid initialization. Care was taken to ensure that the CFD solutions are converged and accurate. The nozzle design and its exit characteristics such as centerline, exit Mach number, pressure and flow angularity variation are analyzed using ANSYS FLUENT. The accuracy of the approximation method was tested by comparing the results with the exact theoretical ones; these resulted within the accuracy of construction nearly complete agreement. The flow uniformity survey measures the uniformity of the total pressure, flow angularity, and temperature as well as boundary layer characteristics for different nozzle configurations and the desired contour is obtained. The fully turbulent CFD simulations were conducted to analyze the flow quality of the nozzles.Computational validation which includes the confirmation of Mach flow at the exit of the nozzle, the prediction of boundary layer thickness and an assessment of the nozzle exit flow quality is done.Mach number of 4.89 is achieved and flow uniformity of 0.56% and flow angularity of 0.1° is observed in the contour where throat location is shifted by 0.004 downstream. The optimum contour with required exit flow properties can be incorporated in the design of Mach 5 nozzle for the large hypersonic wind tunnel.

ACKNOWLEDGMENT

I sincerely thank Shri.Amitava Mandal, Ankur Nagpal for their valuable suggestions and their support in designing the nozzle. I would like to thank Mrs.Rani for her co-operation in this design.

REFERENCES

[1] Ying-Nien Yu, *NATO AGARDograph*, “A Summary of Design techniques for Axisymmetric Hypersonic Wind Tunnels”.

[2] James C. Sivells, *Journal of Spacecrafts and Rockets, Vol. 7, No. 11*, “Aerodynamic design of Axisymmetric Hypersonic Wind Tunnel Nozzles”.

[3] Richard L Gaffney Jr., Brian K. Stewart, Stephen F. Harwin, *25th AIAA Aerodynamic Measurement Technology and Ground testing Conference*, June 2006, “Design of a High Q Mach 5 Nozzle for the NASA Langley 8Ft HTT”.

[4] Tharen Rice, John Hopkins University, *Applied Physics Laboratory*, June, 2003,“2D and 3D method of characteristics tool for complex nozzle development”.

[5] D. F. Long and G. **J.** Hanust, *AIAA-0044*, “Aerodynamic design and Analysis of a Dual throat Hypersonic Nozzle”.

[6] R. Sauer, *NACA (National Advisory Committee for Aeronautics), Technical Memorandum No.1133*, “Method of Characteristics of 3D-Axially symmetric supersonic flows”.

[7] B.KrishnaPrafulla, V.ChittiBabu and P.GovindaRao, *International Journal of Computational Engineering Research, Vol 03, Issue 5*, May 2013,“CFD Analysis of Convergent-Divergent Supersonic Nozzle”.

[8] Cronvich.LL, *Journal of Aeronautical Sciences, Vol 15,No.3*, “A Numerical-Graphical Method of Characteristics for axially symmetric Isentropic flow”.

[9] Kevin L.Mikkelsen and D.F.Long, *43rdAIAA Aerospace Sciences Meeting and Exhibit*, Jan 2005, “Refurbishment,Calibration and Initial testing in the ASE Hypersonic Wind Tunnel”.

[10] Kenney JT, Webb LM, *AGARDograph3*, “A Summary of techniques of Variable Mach Number Supersonic Wind Tunnel Nozzle design”.

[11] H.Lewis*Flight Propulsion Laboratory*, Cleaveland, Ohio, “Internal performance characteristics of a short convergent-divergent exhaust nozzle designed by method of characteristics”.

[12] Pu Zhang, JingleiXu, ZhibinQuan, Jianwei Mo, *ActaAstronautica*, September 2016,“Effects of Nonuniform Mach-number Entrance on Scramjet Nozzle Flowfield and Performance”.

[13] Olivera P. Kostic, Zoran A. Stefanovic, Ivan A. Kostic, *FME transactions*, 2015, “CFD modeling of supersonic airflow generated by 2D nozzle with and without an obstacle at the exit section”

[14] GUO Shan-guang, WANG Zhen-guo, ZHAO Yu-xin, *Science and Technology on Scramjet Laboratory*, 2015, “Design of a continuously variable Mach-number nozzle”.

[15] S.M.Aulchenko, V.M.Galkin, V.I.Zvegintsev, A.N.Shiplyuk, *International Conference on Methods of Aerophysical Research, ICMAR* 2008, “Design of a multimode axisymmetric nozzle for a hypersonic wind tunnel by methods of numerical optimization”.

[16] H. Wei, W. Y.K. Chan, P. A. Jacobs, and R. G. Morgan, *19th Australasian Fluid Mechanics Conference*, December 2014 , “Computational Optimisation and Analysis of A Truncated Hypersonic Nozzle for X3 Expansion Tunnel”.