DESIGN & COMPUTATIONAL FLUID DYNAMICS ANALYSES OF AN AXISYMMETRIC NOZZLE AT TRANSONIC FREE STREAM CONDITIONS

S Wasim Akram[1], S. Rajesh[2]

[1] M.Tech Student, Department of Mechanical Engineering, Krishna Chitanya Institute of Technology & Sciences, Markapur.
[2] Sr. Assistant Professor, Department of Mechanical Engineering, Krishna Chitanya Institute of Technology & Sciences, Markapur.

Abstract - *A numerical investigation of boat tail nozzle has been carried out through computational software (GAMBIT 2.3.16, FLUENT 6.3). Effect of friction in adiabatic flow process has studied and the friction factor is satisfactory level around 0.001452. A variation of cross sectional area with respect to the corresponding Mach number relation output is achieved through programming language (C++). The isentropic process of variable area nozzle and its flow properties such as mass flow rate, total pressure, and free stream pressure are calculated. Mach number = 0.8 (Transonic regime) analysis have carried over modified cross sectional area of a throat section in geometry which it is referred (Teryn 2003). Throughout the investigation, validation work has done over coefficient of pressure chart with the effective utilization of K-epsilon turbulence model and its appropriate boundary conditions. By the application wise, reduction of pressure loss and improvement in design will suits the commercial jet planes.*

Key Words: *boat tail nozzle, computational software (GAMBIT 2.3.16, FLUENT 6.3)*

1. INTRODUCTION

Computational fluid dynamics analyses of an axisymmetric nozzle at transonic free stream conditions have been completed to determine the capabilities of Reynolds averaged navier stokes calculation to predict the details of flow conditions in the propelleling jet which comes from nozzle over external and internally. Pressure distribution of the nozzle over internal and external is to be compared with the standard data obtained in base reference [1].Before proceeds about the system; fundamentals of nozzle and its behavior are to be discussed in this chapter.

1.1 NOZZLE THEORY

Nozzle is a duct of varying cross sectional area used for increasing the velocity of steadily flowing stream of fluid. The fluid enters the nozzle with a high pressure and relatively small velocity. During the flow process, its pressure falls and velocity increases continuously from entrance to exit of the nozzle. The different expressions for the velocity of flows are discussed in the following chapters. Types of nozzle are converging or subsonic nozzle; the cross section of flow region decreases continuously from entry to exit and the acceleration of fluid is in the subsonic region, divergent nozzle or supersonic nozzle; the flow passage diverges from entry to exit and acceleration of fluid is in the supersonic velocity range, convergent- divergent or delavel nozzle is the cross section of the flow passage converges down from entry area to minimum area (throat area), and then diverges from throat to exit. The flow velocity at the throat equals the sonic velocity. But this boat tail nozzle is variable type.

The study of convergent- divergent type boat tail nozzle has been investigated by reference [1]. Boat tail nozzle over external and internal flow aerodynamic characteristics has carried out in under grant NCC3–922 and that was sponsored by the Propulsion Research and Technology Project of NASA's Next Generation Launch Technology Program.

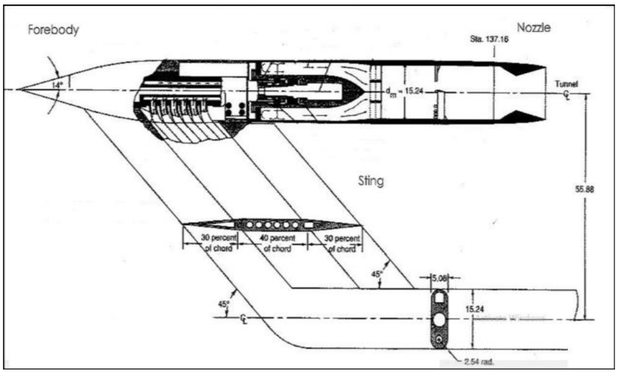


Figure 1.1 schematic of NASA Langley experimental rig (forebody/nozzle/support).

2. LITERATURE SURVEY

Teryn et.al. [1] Reported the Boat tail nozzle over internal and external pressure distributions computed with the Wind code have been compared with experimental data obtained in the NASA Langley 16-Foot Transonic Tunnel. Using a range of turbulence models, including the Explicit Algebraic Stress model (EASM), the experimental data pressure profiles on two nozzle geometries has been predicted reasonably well.

The results show the greater pressure differences between the jet and free stream flow appeared to improve the EASM results. Values of free stream pressure, total pressure and total temperature has been chosen by this journal.

Oosthuizenet.al. [2] Formulated the formula for Mass flow rate of convergent nozzle, divergent nozzle, convergent divergent nozzle and variation of area nozzle i.e., isentropic variable area flow nozzle.

The adiabatic process of fluid flow with friction and without friction is discussed and reasonably possible conditions are stated for our problem. Programming code of adiabatic problem had taken by his book fundamental of compressible flow.

Shames [3] discussed the fluid behavior in compressible flows externally and internally also he stated about the effects of friction at real situations around the transonic Mach number variations in the throat section of the nozzle.

3. OPERATING CHARACTERISTICS OF NOZZLE

3.1 CHARACTERISTICS OF NOZZLE

The effect of changes in upstream and downstream pressures on the nature of the flow in and on the mass flow rate through a nozzle .i.e., through a variable area passage designed to accelerate a gas flow [2].The upstream stagnation conditions, are kept constant while the conditions in the downstream chamber into which the nozzle discharges are varied. Therefore the pressure in the downstream chamber is termed the back pressure. As the back pressure is decreased, flow commences, this flow initially being subsonic throughout the nozzle. Under these circumstances, the pressure on the exit plane of the nozzle, remains equal to the back pressure and mach number on the exit plane is less than 1.in this region of operation, a reduction in back pressure produces an increase in the mass flow rate..

When the back pressure has been decreased to this value, the mach number on the exit plane must be equal to 1 .further reduction in back pressure have no effect on the flow in the nozzle. i.e., exit pressure is equal to critical pressure, the mass flow rate remains constant and the mach number in the exit plane remains equal to 1.

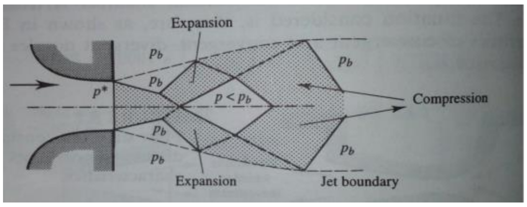


Figure 3.1 Effect of back pressure in convergent nozzle (Courtesy: Patrick H.Oosthuizen

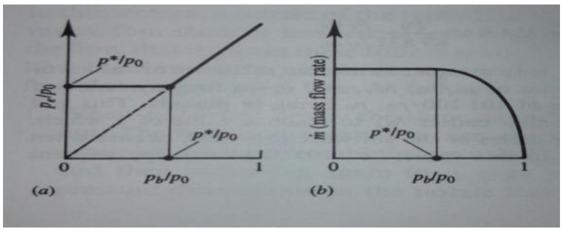


Figure 3.2 Effect of back pressure on a) exit plane pressure and b) mass flow rate through convergent nozzle.

3.2 NOZZLE PERFORMANCE CALCULATIONS

To find the stagnation condition of free stream surface and nozzle jet:

$$\frac{P_o}{P} = P\left[1+\frac{\gamma-1}{2}M^2\right]^{\frac{\gamma}{\gamma-1}} \quad \text{-- (Eqn. 1)}$$

$$\frac{\rho_o}{\rho} = P\left[1+\frac{\gamma-1}{2}M^2\right]^{\frac{1}{\gamma-1}} \quad \text{-- (Eqn. 2)}$$

$$\frac{T_o}{T} = P\left[1+\frac{\gamma-1}{2}M^2\right]^{\frac{1}{\gamma-1}} \quad \text{-- (Eqn. 3)}$$

$$m = p_t A_t \sqrt{\frac{\gamma}{RT_t}\left(\frac{2}{\gamma+1}\right)^{\left(\frac{\gamma+1}{\gamma-1}\right)}} \quad \text{-- (Eqn. 4)}$$

$$V_e = \sqrt{2c_p T_O\left(1-\frac{P}{P_O}\right)^{\left(\frac{\gamma-1}{\gamma}\right)}} \quad \text{-- (Eqn. 5)}$$

$$m\sqrt{\frac{2\gamma}{\gamma-1}RT_t\left[1-\left(\frac{P_o}{P_t}\right)\right]^{\left(\frac{\gamma-1}{\gamma}\right)}} \quad \text{-- (Eqn. 6)}$$

Table -1:

PROPERTIES/PARAMETERS	VALUES
Mach number	0.8
Total pressure	193Kpa
Total temperature	297K
Free stream pressure	66.6Kpa
Area of throat section	0.0762m
Mass flow rate	3.4444Kg/sec
Force/thrust	367.748KN

Table 3.1 Physical properties of nozzle condition.

3.3 REAL NOZZLE FLOW AT DESIGN CONDITION OVER ADIABATIC FLOW WITH FRICTION

If M < 1, i.e., the flow is subsonic so decreasing the area which increases the velocity and vice versa. If M > 1, i.e., the flow is supersonic so, decreasing the area which decreases the velocity and vice versa. If M = 1 dA/dV=0 and A reaches the extremum, on that case A must be minimum.

But, only way to check out the Mach number variation by dA is,

$$\frac{dA}{A} = \frac{M^2 - 1}{1 + \frac{(\gamma - 1)M^2}{2}} \frac{dM}{M} \qquad \text{-- (Eqn.7)}$$

From this eqn.7, it follows that
When M < 1, A increases and M decreases.
When M > 1, A increases and M increases.
When M = 1, dA = 0 i.e., A is minimum.
dA can also be 0 when dM is 0.

Therefore, a minimum in the flow area can also be associated with a maximum or minimum in the mach number. Hence, the effects of area change on the Mach number.

Table -2:

PARAMETERS	VALUES
Wall shear	0
Viscosity of air	0.00001907
Frictional factor	0.001452

Table 3.2 variation of parameter value due to friction

4. COMPUTATIONAL INVESTIGATION:

Investigation were conducted with CFD codes (Gambit2.3.16 and Fluent 6.3), a general purpose tool which solves the pressure based , implicit with absolute velocity formulation and green gauss cell based node centered finite difference approach. It is used for steady state calculations and utilize second order upwind scheme. The solver was configured to run with the following technical specifications: steady state time stepping, two equation models K-epsilon and K-omega with the necessity condition of perfect gas.

4.1 GEOMETRY DEFINITIONS AND BOUNDARY CONDITIONS:

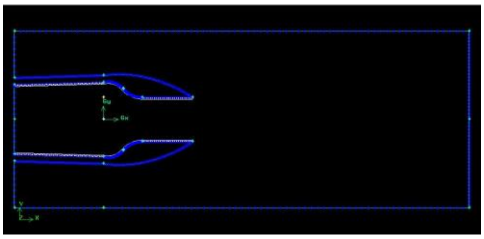
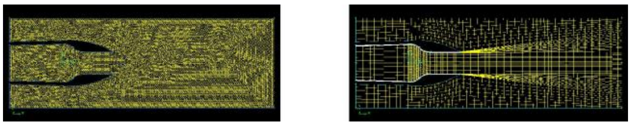


Fig -4.1: Geometrical configuration of nozzle.

2-Dimensional, structured, geometry and computational grids were generated for all cases using Gambit 2.3.16 software. In all cases, the nozzle is attached to a rear body of engine, whose geometry is included in the computational models due to the well developed boundary layers along the length of the body. The entire control domain has a coordinates of (-15,-15) (45,-15) (45, 15) (-15, 15). The entire length assembly is 60cm; upstream nozzle diameter is 7.0cm; exit nozzle diameter is 3.5cm; and the variable throat diameter is 5.25cm; is 30cm located from the mid of global co-ordinate system is shown in Figure 4.1.



a) Non-structured grids b) Structured grids

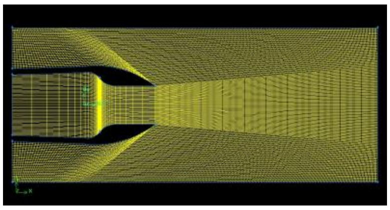


Figure.4.2 a) Structured and non-structured grids Grid generation (structured and non-structured)

A grid dependence study was carried both structurally and non-structurally and it is shown in fig 4.2. a, b and c. The unstructured grid contained 30% more grid points than structured mesh. Because of ram region in inside the nozzle i.e. throat section of the nozzle needs more grids than other regions both externally and internally. For the non- structured grids are more suitable for external flows and in complex fluid problems but here the geometry is in simple construction so we can avoid structured mesh and considered only combined structured and unstructured mesh for ease of solutions attaining. Here, we made 57,570 nodal points for reducing CPU memory consumption. And it is sufficient to capture pressure distributions.

5. RESULTS AND DISCUSSION

Results are presented as contours of pressure distribution on the internal and external nozzle surfaces, vector plot, and coefficient of pressure plots over the flow field especially with respect to distance (x/c). A variation of area is found to decrease monotonically with Mach number variation as discussed through programming code (C++). But in reference [6], mass flow rate is found to decrease monotonically with Mach number. Variation of static pressure increases with Mach number less than 0.8 found to be identical from inlet throat to exit for Mach number greater than 0.8. Solutions are evaluated at specified nozzle pressure ratio corresponds to mass flow rate, maximum velocity, maximum pressure, and maximum thrust force.

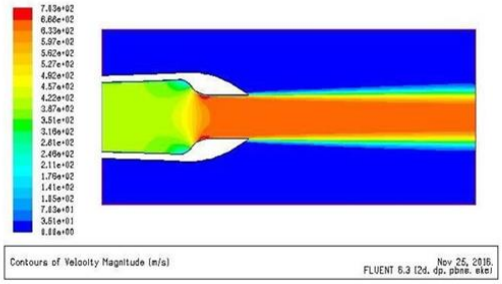


Figure.5.1 Velocity magnitude through streamlines.

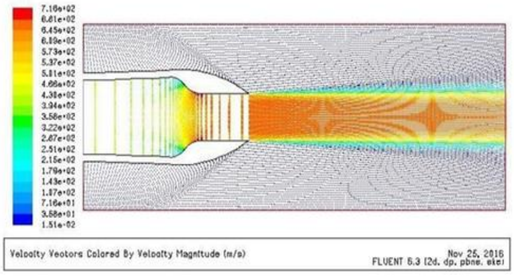


Figure 5.2 Velocity vectors of the flow nozzle

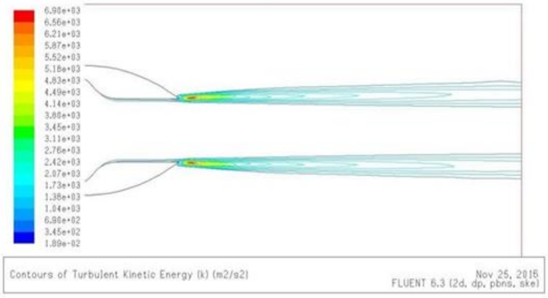


Figure 5.3 Turbulent kinetic energy contour of nozzle (tail region).

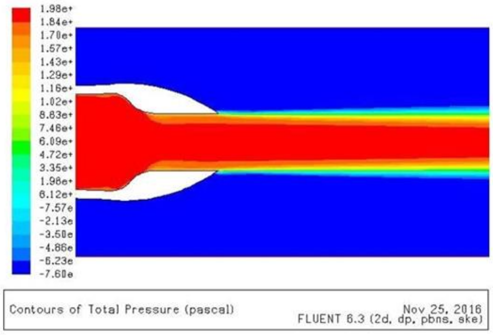


Figure 5.4 Contour plot of total pressure.

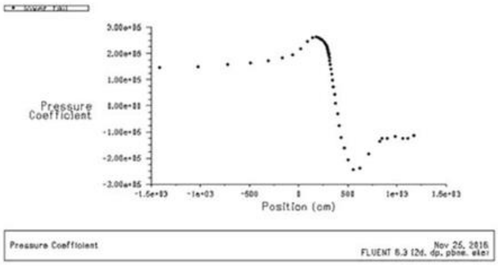


Figure 5.6 Plot between coefficient of pressure and position of nozzle.

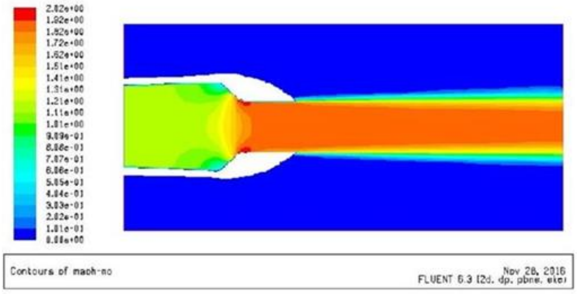


Figure 5.7 Contours of Mach number=0.8

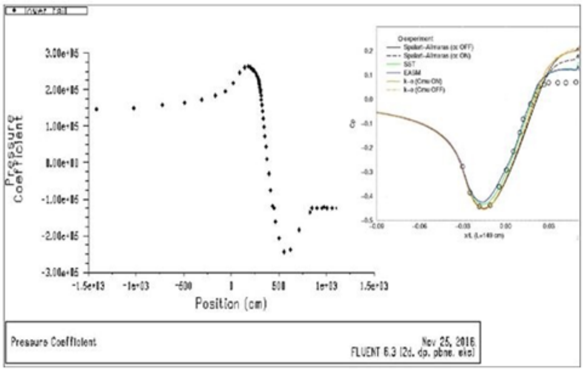


Figure 5.8 Validation chart over co-efficient of pressure at M = 0.8

6. CONCLUSION AND FUTURE ENHANCEMENTS

We conclude that, CFD analyses have been done through the codes (GAMBIT 2.3.16 and FLUENT 6.3) and boat tail nozzle at transonic Mach numbers have been performed to better understand the effects of high-speed nozzle geometries on the nozzle internal flow and the surrounding boat tail regions. Operation of this nozzle at lower-than-design nozzle pressure ratio provides a challenging flow field for studying the capabilities of K-epsilon turbulence models to accurately predict nozzle aerodynamics. In general, all turbulence models are under predicted the pressure distribution internally and externally of the nozzle but in advance k-epsilon plays important role with realizable mode. The frictional effect causes the flow in adiabatic consideration i.e., real applications but it is considerably neglected in isentropic flows. In isentropic flows, shocks are not induced rapidly but that are consumed to be negligible and the induction of shockwave may affect the flows entirely at any transonic regime. In future, discussion of shockwaves, separation of shockwaves and it's control at same geometry and transonic regime M=0.8 are to be analyzed with this same CFD code.

REFERENCES

[1] "Computations of internal and external axisymmetric nozzle aerodynamics at transonic speed" and "computational study of axisymmetric nozzle through off design performance" by Teryn Dalbello, Nicholas Georgiadis and Dennis Yoder, Theo Keith, NASA/TM-2003-212731.

[2] "Compressible fluid flow" by Patrick H.Oosthuizen and William E. Carscallen, McGraw-HILL INTERNATIONAL EDITIONS 1997.Page no: 191-194 and 250-257.

[3] "Mechanics of fluids" by Irving H. Shames, McGraw-HILL INTERNATIONAL EDITIONS 1992-498.

[4] "Fanning friction factor for pipe flow" by JIM MCGOVERN, DUBLIN INSTITUTE OF TECHNOLOGY.

[5] "Modeling of turbulent flows and boundary layer" by Dr.SrinivasaRao.P, C-Dac, India published in INTEC, Goatia, January- 2010 Pp.420.

[6] "Flow analysis of convergent- divergent nozzle using CFD" by Gutty Rajeswararao and et.al, IJRME publication, vol.1 october-december 2013, P p.136-144

BIOGRAPHIES

S Wasim Akram is currently pursuing M.Tech in CAD/CAM at Krishna Chaitanya Institute Of Technology & Sciences, Markapur. He received B.Tech degree in Mechanical Engineering from Rao and Naidu Engineering College Ongole in the year 2013.

.

S. Rajesh working as Assistant Professor in Krishna Chaitanya Institute of Technology & Sciences, Markapur.
He received M.Tech in Machine Design from Rajeev Gandhi Memorial college of Engineering and Technology, Nandyal in the year 2015.