Acoustics Study of a Supersonic C-D Nozzle

Arun Raj S[1], Jaison Joseph[2], Jeffrey John Varghese[2], Joseph Tom Joseph[2], Subnas L C[2]

[1]Assistant Professor, Christ University Faculty of Engineering, Bangalore, India.
[2]UG student, Christ University Faculty of Engineering, Bangalore, India

***Abstract*— Supersonic flows associated with aircraft and rocket nozzles are very often steady. The objective of the present work is to simulate and understand the acoustic characteristics of single jet supersonic flows using Broadband noise model for 3 pressure ratios (1.27, 2.3 and 5.5). Single jet flows, generated by axisymmetric and asymmetric nozzles, have been numerically investigated by FLUENT software. Acoustic characteristics of the 4 different Convergent Divergent (C-D) nozzle configurations are revealed studying the main flow parameters (pressure, velocity, kinetic energy) in order to be able to discover the most efficient type of nozzle regarding the acoustic power level emitted.**

Keywords*— acoustic noise, CFD, turbulence, domain, mesh, boundary condition.

Nomenclature

Symbols

U	Free stream velocity
k	Turbulent kinetic energy
I	Turbulent intensity
P(y)	Acoustic power emitted
M_c	Mach number
ρ_o	Density
l	Eddy viscosity length
u’	Eddy viscosity

I. INTRODUCTION

Numerical analysis of 2-D steady air-flow over 2 different types of nozzle is carried out using the commercial CFD software Ansys FLUENT 14. Ansys Workbench is used to create the pre-defined grid required for simulation. The flow results are expressed by velocity, pressure and turbulent kinetic energy whereas acoustic characteristics by acoustic power level. Acoustic analysis strongly depends on the accuracy of the numerical methods used. It is known that the high order schemes like QUICK scheme will give higher accuracy. But for the present analysis second-order upwind method has been chosen in-order to choose an optimised solution between computational time and accuracy. The results have been compared to reveal the most efficient nozzle regarding noise.

Wleizen, R.W. and Kibens, v., [1988] et al. [1] investigated non-axis-symmetric nozzles having constant diameter and various exit cut-out shapes. He performed analysis on 3 types of nozzle configurations for 4 different pressure ratios (2.2, 2.8, 3.4 and 4.0) and divergent angles (up to 30°). It was concluded from the research that a small variation in the geometry can cause a huge variation in thrust, mixing enhancement and noise.

Samanta et al [2] showed that all acoustic analogies lead to the same numerical sound integral if the full DNS solution is used for acoustic source modelling.

II. GEOMETRY MODEL AND FLOW DOMAIN DISCRETISATION

For the present analysis, 4 different nozzle configurations are considered. All 4 nozzle configurations have a convergent length of 150 mm and divergent length of 117 mm. The exit to throat area ratio for nozzle 1, 2, 3, 4 are 1.5, 1.66, 1.14 and 1.21 respectively. The details of the nozzle configurations are given below.

Nozzle 1: Symmetric type: Base model, where the divergent angle from 2.801 degrees.

Nozzle 2: Symmetric type: Increasing the divergent angle from 2.801 degrees to 3.89 degrees

Nozzle 3: Asymmetric type 1: Introducing contraction angles at the bottom wall at distances of 68.13 mm (1st contraction angle) and 8.22 mm (2nd contraction angle) from the location of the 1st contraction angle along the wall of the divergent section.

Nozzle 4: Asymmetric type 2: Introducing contraction angles at the top and bottom walls at distances of 67.6 mm (1st contraction angle) and 72.0 mm (2nd contraction angle) of the divergent section.

The two asymmetric models, Model 2 and Model 3 have a divergent angle of 2.801 degrees at the nozzle throat [3]. The computational domain, namely convergent – divergent section, jet plume section is modelled as 2 zones using the program Ansys Design Modeler. Zone 1 is the convergent-divergent nozzle section and Zone 2 is the jet plume section as shown in the computational domain (Figure 2.1). These two zones share a common region (nozzle/plume interface boundary) where the fluid at the exit of the nozzle mixes with the fluid at the jet plume region.

Figure 2.2 shows the grid adopted for the analysis of nozzle 1. The mesh is generated by using the program Ansys Workbench. In-order to capture the boundary layer effect inflation layers have been created in the grid. Also the skewness for the grid is maintained less than 0.7 as recommended by Workbench manual. The governing equations of flow and acoustics are then solved over the grid developed using the program Ansys FLUENT 14.

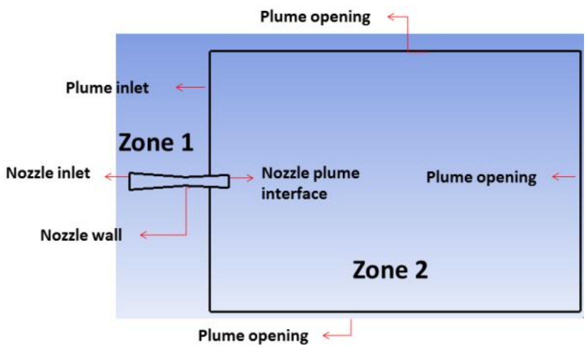

Figure 2.1: Computational domain for Nozzle configuration 1

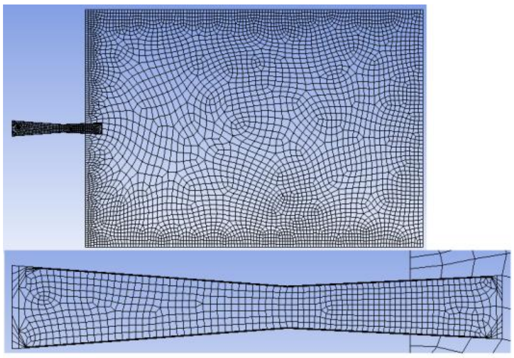

Figure 2.2: Grid for Nozzle configuration 1

III. SOLVER SETTINGS AND BOUNDARY CONDITIONS

To start with the simulation part, it is required to define the operating condition model, material, and boundary condition. Density based implicit solver has been chosen for the present 2-D simulation. This is because density based solver is designed for high speed flows. The material is selected as air and the density as ideal gas to make the problem easier. The standard k-ε model is chosen as the turbulence model because of its reasonable accuracy and robustness. One of the major advantages is that this model doesn't require a large time for doing the simulation. Broadband Noise Source model has been enabled to calculate the acoustic field. This model uses Proudman's equation, Lilley's equation and Goldstein's theory for calculating the acoustic field. All these approaches are based on Lighthill's equation for turbulent flows, which states that the acoustic power emitted by a source is:

$$P(y) = \frac{K{u'}^4 l^3 \rho_O (1+{M_C}^2)}{4\pi a_O^5 \tau_\varepsilon^4 (1-{M_C}^2)^4}. \ [4]$$

The initial values are computed from nozzle inlet in zone 1. It is required to select the residual value under monitors and check in plot to visualize the convergence. The boundary conditions required for the simulation is illustrated in Table 3.1.

Table 3.1: Boundary settings

Parameters	Value
Total pressure at nozzle	Case 1, 2, 3: 1.27, 2.3, 5.5
Inlet velocity Plume inlet	250 m/s
Plume opening	Conditions at sea level

IV. RESULTS AND DISCUSSION

Jet noise is becoming a major issue in aerospace industry. The present research implements CAA models to investigate jet noise. The sources of jet noise can be monopole (where there is fluctuation in mass flow), dipole (on surfaces where the flow causes fluctuating pressure) and quadrupole (turbulent wakes). For flow in a supersonic jet, noise generated may be due to several reasons such as i) flow separation ii) incident turbulence iii) turbulent boundary layer iv) vortices in the wake region and so on. The preferred Broadband noise source model has many times successfully used in different acoustic applications. The jet flow has been considered as potential source of the acoustic noise in Convergent Divergent (C-D) nozzle. Broadband noise source models allow acoustic sources to be estimated based on the results of steady-state simulations [8].

A. *Flow field study for noise source*

Figure 4.1 shows the Mach number contour for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5).

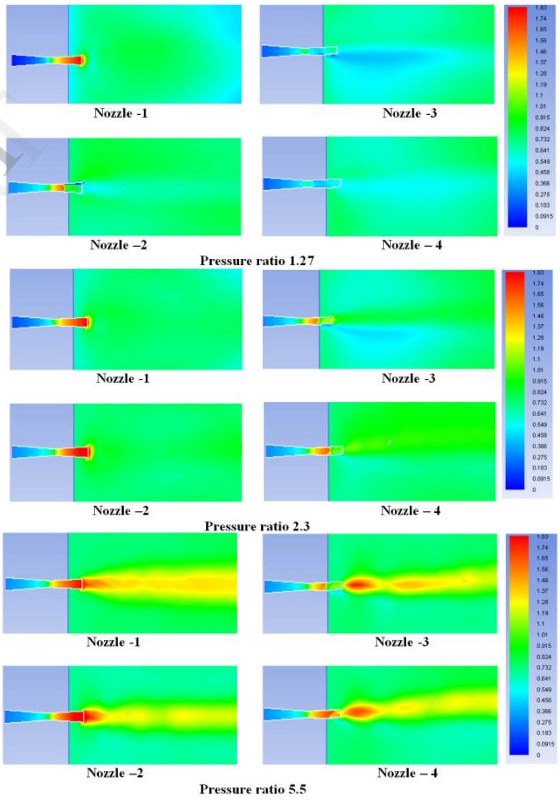

Figure 4.1: Mach number for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5)

It is shown that the axi-symmetric nozzle (Nozzle 1 and 2) gives higher exit Mach number than a-symmetric nozzle (Nozzle 3 and 4) for all pressure ratios. This is because for a supersonic flow the exit Mach number is directly dependent on the area ratio (A_e/A_t) of the nozzle. Nozzle 1 and 2 has higher exit Mach number comparing Nozzle 3 and 4. This is because the exit to throat area ratio (A_e /A_t) is higher for Nozzle 1 and 2 comparing Nozzle 3 and 4. [5]

B. Acoustic field study for noise source

Figure 4.2 shows turbulent intensity (I) for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5). Turbulence intensity (I) which is defined as: $I = \frac{\sqrt{\frac{2}{3}k}}{U}$ with ‘U’ mean velocity and ‘k’ turbulent kinetic energy, which is a product of Reynolds stress tensor, obtained from simulation.

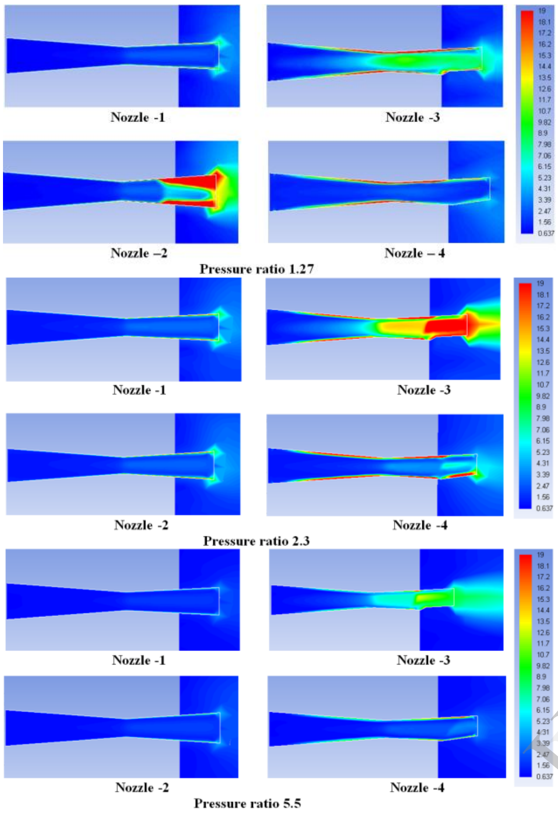

Figure 4.2: Turbulent Intensity for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5)

It is clear from the Mach number contour that the flow near the exit section of the nozzle experiences asymmetric separation due to boundary layer effect which in turn creates a shock wave. This shock wave increases the turbulent intensity value in that region. [7].

Figure 4.3 shows Acoustic power for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5). Acoustic power is measure of quadrupole noise source. The turbulent intensity is influential for broad-band noise level and is presented in Figure 4.2. It is clearly shown from the result that the axi-symmetric nozzle (Nozzle 1 and 2) emits less acoustic power than a-symmetric nozzle (Nozzle 3 and 4) for all pressure ratios. This can be explained by analysing the turbulent intensity contour.

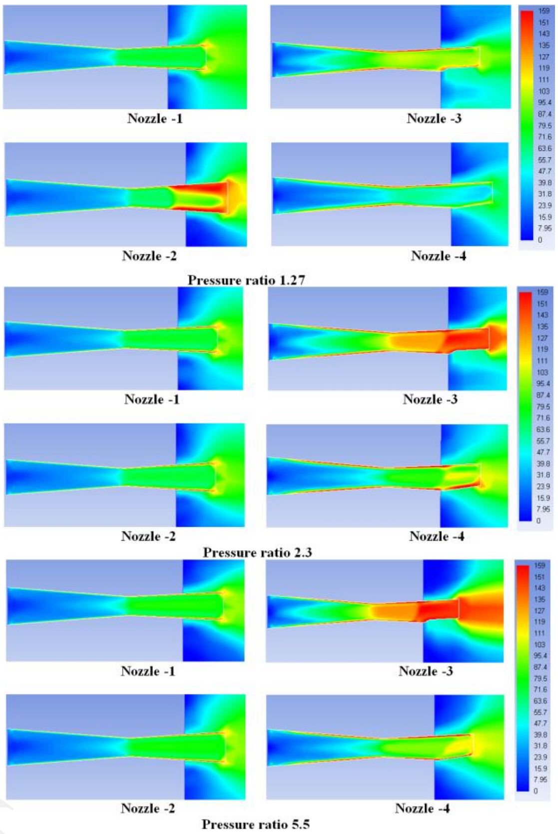

Figure 4.3: Acoustic Power for different nozzle configurations (Pressure ratio: 1.27, 2.3, and 5.5)

The variation of acoustic power (AP) and turbulent intensity (TI) looks almost similar. The reason is the turbulent wake which raises the turbulent intensity in that region. It acts as a quadrupole source which in turn increases the acoustic power in that region. [6]

Table 4.1: Acoustic Power (AP) for different nozzle configurations (Pressure ratio (PR): 1.27, 2.3, and 5.5)

PR	Cases	AP (dB)	TI (%)
1.27	Nozzle 1	116.19	2.36
	Nozzle 2	122.82	7.60
	Nozzle 3	121.81	6.63
	Nozzle 4	117.53	4.76
2.3	Nozzle 1	116.30	1.07
	Nozzle 2	117.71	2.3
	Nozzle 3	139.60	9.39
	Nozzle 4	140.61	10.2
5.5	Nozzle 1	113.59	0.92
	Nozzle 2	115.19	1.14
	Nozzle 3	137.29	4.11
	Nozzle 4	140.58	5.51

It can be shown from the Table 4.1 that the most efficient type of nozzle regarding the acoustic power level emitted for a pressure ratio of 1.27, 2.3 and 5.5 is Nozzle 1 (Symmetric nozzle with area ratio 1.5). The acoustic power level emitted by the Nozzle 1 for a pressure ratio of 1.27, 2.3 and 5.5 is about 116 dB, 116 dB and 113 dB respectively.

V. CONCLUSION

The noise generated by the steady flow in supersonic jet is analyses by means of Broadband noise source model. In the present study a 2-D simulation has been carried out for 4 different nozzle configurations at 3 different pressure ratios (1.27, 2.3, and 5.5). The grids employed are too coarse to resolve the turbulent eddies sufficiently and so there could be a small variation in the computation. Acoustic model namely Broadband noise source model has been tried to calculate the jet noise. The acoustic power (dB) predicted by the Broadband noise source model for 4 different nozzle configurations at 3 different pressure ratios (1.27, 2.3, 5.5) are analysed. Among the four nozzle configurations nozzle 1 is found as the most efficient for all 3 pressure ratios regarding the acoustic power level emitted. Also it appears that Broadband noise source model may be best suited for predicting the noise of supersonic jet.

REFERENCES

[1]. Wlezien R. W. and Kibens. V. Influence of nozzle asymmetry on supersonic jets. AIAA Journal., 26(1), January 1988.
[2]. Samanta, A., Freund, J.B., Wei, M. and Lele, S.K. Robustness of Acoustic Analogies for Predicting Mixing-Layer Noise, AIAA Journal, Vol. 44, No. 11, November 2006, pp. 2780-2786
[3]. E.M.S. Ekanayake, J.A. Gear, and Y. Ding. Flow simulation of a two dimensional rectangular supersonic convergent divergent nozzle. ANJIAM Journal 2009, pages C377 -C392.
[4]. Kateřina HORÁKOVÁ, Karel FRAŇA. Acoustics of Jet Flow. Journal of applied science in the thermodynamics and fluid mechanics. Vol. 1, No. 1, 2007.
[5]. J.D. Anderson. Modern Compressible Flow with Historical Perspective. McGraw -Hill, 3rd edition, 2003.
[6]. K. M. Pandey, Virendra Kumar, and A.P.Singh., Study on Supersonic Free Single Jet Flow: A Numerical Analysis with Fluent Software, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp.179-184.
[7]. E.M.S. Ekanayake, J.A. Gear, and Y. Ding. Numerical simulation of a supersonic convergent divergent nozzle with divergent angle variations for under expanded condition. 17th Australasian Fluid Mechanics Conference Proceedings, 51, 2010.
[8]. ANSYS FLUENT 12.0 - Theory Guide, ANSYS Inc.