Graduate Research in Engineering and Technology (GRET)

Volume 1
Issue 7 *Rockets and Missiles Technologies.*

Article 15

June 2022

CFD Analysis of Nozzle Flow with Sudden Expansion In Aerospace Engineering

Yogesh Bhawarker Mr.
Department of Aerospace Engineering, Sandip University, SOET, NASHIK, MH, yogeshbhawarker@gmail.com

Ritik Patidar
Sandip University, Nashik, patidarritiksailana@gmail.com

Prakash Katdare
Sagar Institute of Research and Technology Bhopal India, prakshkatdare21@gmail.com

Recommended Citation
Bhawarker, Yogesh Mr.; Patidar, Ritik; and Katdare, Prakash (2022) "CFD Analysis of Nozzle Flow with Sudden Expansion In Aerospace Engineering," *Graduate Research in Engineering and Technology (GRET)*: Vol. 1: Iss. 7, Article 15.
DOI: 10.47893/GRET.2022.1119
Available at: https://www.interscience.in/gret/vol1/iss7/15

CFD Analysis of Nozzle Flow with Sudden Expansion In Aerospace Engineering

Yogesh Bhawaker[1], Ritik Patidar[1], Prakash Katdare[2]
[1,2]Department of Aerospace Engineering, Sandip University, Nashik India.
[3]Department of Mechanical Engineering, Sagar Institute of Research and Technology, Bhopal India
Corresponding Author email: yogesh.bhawarker@gmail.com

***Abstract*— As the need for missiles and rockets has increased exponentially, the challenges associated with the gas dynamics of these vehicles continues to be a concern. The pressure in the downstream is sub atmospheric whenever there is a sudden expansion. This low pressure in the recirculation zone causes significant drag, accounting for almost two-thirds of the net drag of the aerospace vehicles. Hence, the purpose of this paper is to deliver a Computational Fluid Dynamic (CFD) analysis of the impact geometry and flow parameters have on the thrust force generated by the flow from convergent divergent nozzles to a suddenly expanded circular duct with a wider cross-sectional area. By observing all of the results, it is possible to conclude that the flow field in an enlarged duct is greatly influenced by the area ratio.**

Keywords-Area Ratio, CFD, Mach number, Nozzle Pressure Ratio, Velocity, Thrust Force.

I. INTRODUCTION

Unexpected flow expansion is a major difficulty in many fluid-flow applications in the field of high-speed flows such as subsonic and supersonic flow conditions. In a high-speed application, the micro jets effect is critical in the case of rapid expansion flows. It is observed that for jet and rocket engine test cells, the systems are used to imitate high-speed and high altitude situations; a jet creates a violent ejection of pressure, which is known as sub atmospheric pressure. Because of its widespread application, numerous researchers are investigating the behavior of fluid in a suddenly expanded conduit. The study of base pressure in drastically expanded flows is an important subject of research that has applications in a variety of fields. The rocket nozzle base Pressure field is one of the many applications. At general, the pressure in the base region is lower than the atmospheric pressure. It is critical to consider the aforesaid difficulty when designing the expanded duct and selecting appropriate geometrical parameters. Different situations are examined in this work by adjusting geometrical and flow characteristics in order to determine the optimal area ratio, i.e. the ratio of increased duct area to nozzle exit area, for a given nozzle pressure ratio, i.e. the ratio of stagnation pressure to back pressure. Flow control is a key topic of fluid dynamics that is rapidly evolving. The thrust force created by the flow from the nozzle is the most critical factor in any projectile such as a rocket or an aircraft. The difference between the thrust force provided by the flow from the nozzle and the drag forces developed in the base region of the larger duct is the resultant thrust force. Because the pressure in the base region of the larger duct is often less than air pressure, drag forces arise. The resultant thrust force decreases as the drag forces grow. It is critical to investigate the effect of flow and geometry factors on thrust force and drag force while examining the aforesaid problem. The larger duct must be created based on the CFD data so that anyone may select appropriate flow and shape parameters for a given Mach number or nozzle pressure ratio to achieve maximum thrust.

II. MODELLING AND MESHING

A. Modeling

ANSYS Workbench 14.5 is used for modelling and meshing. The problem is solved as a 2D axi-symmetrical problem since the geometry is symmetrical about the axis. The geometry is depicted in the diagram below [1].

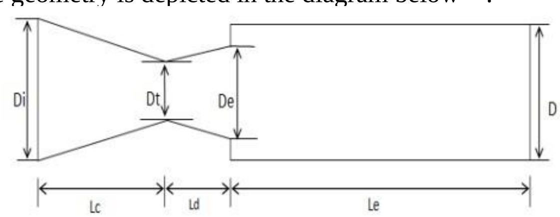

Fig: Detailed dimensions of Nozzle[4].

Where,
Di – inlet diameter of nozzle
ϴc – Half cone angle of convergent section
Lc – Convergent Length
Dt – Throat diameter
ϴd – half cone angle of divergent section
De – exit diameter of nozzle
Ld – Divergent Length
Le – Extended Length

The choice of a model in the ANSYS software is critical for CFD simulation. The solutions to the governing equations were obtained by numerically simulating the flow using (RANS) and the best and most effective turbulent models. For the simulation of viscous effects, the well-known k-standard model for modelling turbulent flow is used [5].

B. Meshing

The mesh should be structured to provide high-quality output. The model is broken down into several sections, each of which is meshed separately. As seen in picture, a fully structured grid is created[(1)].

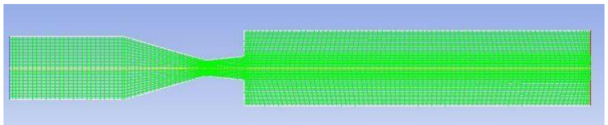

Fig : Model of a nozzle with a mesh and a larger duct(1).

III. Finite Element Modelling

ANSYS can be used to observe the fluid flow variation on the CD nozzle in finite element modelling.

IV. Two-Dimensional CD Nozzle

In this scenario, ANSYS workbench was used to model the geometry of the CD nozzle and a 2D planar body was evaluated. To begin the solution with perfect ideal flow, boundary conditions are defined by considering the plane's edges. Figure depicts a 2D planar finite element model. The ANSYS workbench was utilized for meshing, and structural mesh was produced to improve fine mesh[(4)].

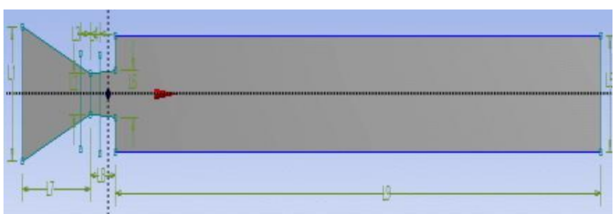

Fig: 2D Planar Fluid Body[(4)]

V. Boundary Conditions

The boundary conditions defined in Ansys fluent to evaluate all situations are "pressure intake at inlet" and "pressure outlet at outlet." Atmospheric pressure is used as the operational pressure. In all circumstances, the input pressure is computed using the Nozzle Pressure Ratio NPR. In all circumstances, the output pressure is assumed to be 1 atmospheric pressure. The analysis is done with Ansys Fluent. The analysis is carried out for all feasible flow and geometrical parameter combinations, and the results are compared using tables[(1)].

VI. CFD Results

Graphs are created using the Ansys software's postprocessor. The results are plotted on the nozzle and enlarged duct axes using different area ratios for the same nozzle pressure ratios (NPR) and different nozzle pressure ratios for the same area ratio[(3)].

VII. Analysis

Variations in area ratios and nozzle pressure ratios are used to conduct the analysis. The nozzle pressure ratios that were considered were 2, 4, 6, 8, and 10. The area ratios evaluated for each value of nozzle pressure ratio are 1, 2, 4, 6, 8, and 10. For each area ratio, the expanded duct diameter (D) is determined. All other dimensions are maintained. The analysis is carried out for all feasible combinations of flow and geometry parameters, and the results are graphed[(2)].

VIII. Result

The study is carried out by taking into account various area ratios and nozzle pressure ratios, and XY graphs are generated using the Ansys postprocessor. The findings are presented individually after considering different area ratios at the same nozzle pressure ratios (NPR).

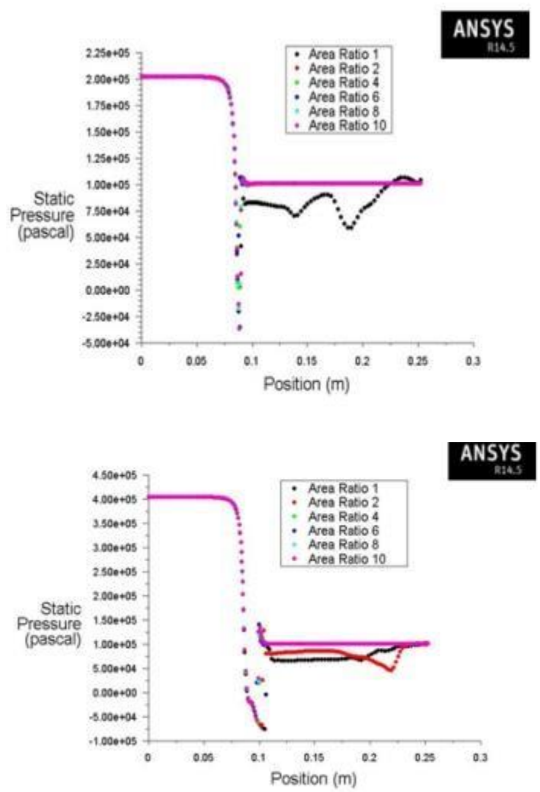

Fig: Pressure Distrubution[(2)]

Table 1 . Nozzle Pressure Ratio

S. No.	Area Ratio	NPR
1	1	1
2	2	2.2
3	4	2.90
4	6	3.368
5	8	3.677
6	10	3.923

IX. CONCLUSIONS

The outcomes for all of the instances are listed below. The flow field in an extended duct is greatly impacted by Nozzle Pressure, area ratio, Mach number, and L/D ratio, as shown by the preceding data. Due to the non-availability of the relaxation to the flow when the area ratio is low, the fluctuations are quite high. When the NPR is very high, the flow becomes severely under expanded, and the flow field is full of waves, which strike the duct wall and are reflected; this process continues until the under expansion is minimized and the pressure equals the back pressure. The reattachment length is a function of the area ratio; it is shortest for the smallest area ratio and gradually grows as the area ratio increases. When the flow is over expanded for a given Mach number, NPR, area ratio and L/D ratio, after exiting from the nozzle, the jet under goes compression in view of the presence of the oblique shock wave, and this process will continue till it attains the value of the ambient atmospheric pressure.

REFERENCES

[1] Khizar Ahmed Pathan, S.A. Khan, P. S. Dabeer ‘CFD Analysis of Effect of Flow and Geometry Parameters on Thrust Force Created by Flow from Nozzle’IEEE Conference Publication, pp 1121-1125, Dec 2017

[2] Pathan Khizar Ahmed , P. S. Dabeer, S. A. Khan ‘CFD Analysis of The Supersonic Nozzle Flow with Sudden Expansion’.. IOSR Journal of Mechanical and Civil Engineering, pp 05-07, Dec 2016

[3] Khizar Ahmed Pathan, S.A. Khan, P. S. Dabeer ‘CFD Analysis of Effect of Mach number, Area Ratio and Nozzle Pressure Ratio on Velocity for Suddenly Expanded Flows’. IEEE Conference Publication, pp 1104-1110, Dec 2017

[4] Sher Afghan Khan , Abdul Aabid , Maughal Ahemed Ali Baig ‘CFD analysis of cd nozzle and effect of nozzle pressure ratio on pressure and velocity for suddenly expanded flows’. International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) , Vol.2. PP- 1147-1152, Jun 2018

[5] Sher Afghan Khan , Abdul Aabid , Fharukh Ahmed Mehaboobali Ghasi , Abdulrahman Abdullah Al-Robaian , Ali Sulaiman Alsagri ‘Analysis of Area Ratio In a CD Nozzle with Suddenly Expanded Duct using CFD Method’ CFD Letters, Issue 5, PP- 61-71, 2019.