

# Design Of Solid Propellant Rocket Nozzle Using ANSYS Software (De-Laval Type)

Dr N. Achara

Dr A. S. Imam (Major)

G. C. Japheth

Department of Mechanical Engineering Nigerian Defence Academy

**Abstract:** A Nozzle is an important component of a rocket propulsion system and depending on the mission of the system comes in a variety of shapes and sizes. By proper geometrical design of the nozzle, the exhaust of the combustion gasses is regulated in such a way that maximum effective rocket velocity can be reached. The design and simulation of two-dimensional solid propellant rocket nozzle using ANSYS Software has been carried out. In the study, parameters of interest considered include, Mach number, gas velocity and pressure. Computational results have been obtained for the three cases considered. The cases were based on area ratios of 4, 2.67 and 2. It has been found that of the three cases studied, case with area ratio of 4 gives the optimum performance result characterized by the highest thrust recorded at the exit of the nozzle

**Keywords:** de Laval nozzle, expansion ratio, Mach number, propellant, rocket motor, supersonic

## I. INTRODUCTION

Rocket engines may either work with solid or liquid propellant or a combination of both known as hybrid propulsion system. These rockets are energy conversion systems with heat release in the combustion chamber and subsequent expansion of the resulting combustion gases to produce thrust for propelling the system to a desired destination [1]. Most of the early design of rocket propulsion systems was dominated by a system in which gas flowed through a pin hole at the base of the rocket without the throat and diverging section of a normal nozzle to expand the exhaust gasses properly and make effective use of the fuel. It was not until around 19<sup>th</sup> century that workers including de-Laval started to work on nozzle shapes. The most common nozzle design in this group is the converging – diverging (C-D) nozzle which provides what is regarded today as best efficient nozzle for a solid propellant rocket motor [2]. As a result of the rising cost of rocket fuel, there have been further studies to improve the performance of de-lava nozzle without significant additional cost. One area where it is widely known

that such an increase in efficiency is possible, but has not yet been fully achieved is the configuration of the rocket nozzle [3]. At entry to the converging section of the nozzle, the speed of the gas is subsonic. As the nozzle contracts down the gas is forced to accelerate until at the throat where the cross-sectional area is smallest, the linear velocity becomes sonic. After the throat the cross-sectional area increases, the gas expands and the linear velocity becomes progressively more supersonic [4]. In the study to optimise the divergent angle of a rocket engine nozzle using computational fluid dynamics Kuttan and Sajesh found that the Mach number increases at the exit section of the nozzle with increasing divergent angle reaching Mach 2.20 for 4° and highest Mach 4.82 at 15°. Similar behaviours of increasing Mach number with divergent angle were recorded also at the throat. For the length of the rocket studied, the work further observed that oblique shocks are formed for flows through the nozzle, when the divergent angle is 4°, the first shock occurred at 1m from the inlet and the wave reflected from the walls of the nozzle caused another shock at 2m and it was also observed that any increase in the divergent angle displaces the shock further downstream

towards the exit of the nozzle. When the divergent angle was increased to  $7^\circ$ , the second shock was found to be eliminated from within the nozzle. Finally at  $15^\circ$  the shock was completely eliminated from the nozzle and this could be considered as a good design for the nozzle. [2]. Venkatesh and Jaya in their work on the modeling and simulation of supersonic nozzle using computational fluid dynamics concluded that contour nozzle gives a greater expansion ratio compared with a conical nozzle. The study recommended that conical nozzle be used at sea-level and that the contour nozzle be used at higher altitudes since for a given nozzle length, greater expansion ratio is required at higher altitudes [5]. Since the performance of rocket motors is highly dependent on the aerodynamic design of the expansion section, the main design parameters being the shape, divergent half angle and the area ratio/expansion ratio, many studies in this area have focused interest in the optimization of these parameters. It is recognized that the area of the throat section and operating conditions of the combustion chamber determine the mass flow rate through a rocket nozzle. Therefore, design changes in the configuration of the convergent portion of the nozzle would influence the mass flow of the exhaust gases and also, to some extent, the combustion efficiency achieved in the chamber [6]. These are engineering problems that require solution and there are various methods for approaching the solution. Some of methods include the analytical method as well as the experimental methods with the associated prototyping. Except for very simple systems, the analytical method is complicated and difficult. The experimental method on its part is very costly and time consuming. If any errors in the design were detected during prototype testing, another prototype has to be produced in which all the errors have been rectified and again tested. This process can continue for a considerable length of time. The introduction of Computational Fluid Dynamics has not only helped to overcome these difficulties but has as well revolutionized the field of engineering. In the CFD approach problem is simulated using an appropriate software package. This is achieved by setting up the transport equations associated with the problem and solving with computer assistance to obtain values for relevant parameters. [7]. Therefore the purpose of this work is to use computational fluid dynamics software (ANSYS software) to study the behavior of a convergent-divergent nozzle of varying area ratio.

## II. METHOD AND MATERIALS

In this work the design and flow analysis of a two dimensional convergent -divergent nozzle using ANSYS software package has been considered in order to fully understand the internal flow behaviour of fluid through the C-D section of the nozzle. The parameters of interest studied include Mach number, pressure and velocity along the nozzle length. The work has mainly looked at the behaviour of the relevant parameters with given expansion ratio. In order to make the analysis of gas flow through the de Laval nozzle manageable, a number of simplifying assumptions are necessary and these include:

- ✓ the assumption of ideal gas,

- ✓ isentropic gas flow with constant entropy,
- ✓ steady state gas flow in the combustion chamber,
- ✓ the gas flow is symmetric along the nozzle's axis,
- ✓ the gas fluid is compressible since the flow is at very high velocities.
- ✓ Inlet pressure to the nozzle is 5bar.
- ✓ expansion ratios computed are 4, 2.67 and 2 respectively.

ANSYS software uses the characteristics/mathematical flow model equations {RANS- Reynolds Average Navier Stoke equations}, which are differential in nature and govern the flow behaviour with, associated boundary flow conditions and the physical system. The model equation for the turbulent kinetic energy  $K$  is given as:

$$\frac{DK}{Dt} = \frac{\partial K}{\partial t} + u_j \frac{\partial K}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \frac{V_t}{\sigma_k} \frac{\partial K}{\partial x_j} \right] + P - \varepsilon = \text{Rate of increase}$$

of  $K$  + Convective transport= diffusive transport + Rate of production-Rate of destruction [8].

And

The model equation for the turbulent dissipation  $\varepsilon$  is given as:  $\frac{D\varepsilon}{Dt} = \frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \frac{V_t}{\sigma_k} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon_1} \frac{P\varepsilon}{K} - C_{\varepsilon_2} \frac{\varepsilon^2}{K} = \text{Rate of increase of } \varepsilon + \text{Convective transport} = \text{diffusive transport} + \text{Rate of production-Rate of destruction}$

The major advantages of this model are that it is relatively simple to implement, it leads to stable calculations, and it is a widely validated turbulence model. Also by the conservation of momentum the net force on the gas generated by the combustion of the solid propellant fuel must be equal to the flux out of the combustion chamber, therefore.

$$\int_{A_c}^0 P_c dA - A_e P_e = \frac{m}{g} V_e \quad (3)$$

The integral describes the thrust produced by the solid propellant rocket

$$F_R - A_e P_e = \frac{m}{g} V_e \quad (4)$$

$F_R$  = rocket thrust force,

$V_e$  = gas velocity at exit of the nozzle,

$g$  = gravitational constant =  $9.81 m/s^2$ ,

Assuming that the rocket nozzle is operated at an ambient pressure  $P_a=0$  (high altitude condition). Thus equation (4),

When operated at a finite ambient pressure  $P_a>0$  (low altitude condition), therefore

$$F_R = \frac{m}{g} V_e + A_e (P_e - P_a) \quad (5)$$

Where  $\frac{m}{g} V_e$  = momentum thrust and  $A_e (P_e - P_a)$  = pressure thrust. [9]

When  $P_e > P_a$ , a condition known as under expansion of gases expelling from the nozzle occurs. Also when  $P_e < P_a$ , a condition known as overexpansion of gas expelling from the nozzle.

Maximum thrust is achieved when  $P_e = P_a$ , but since this condition could not hold in real or practical operating condition of the nozzle, the nozzle designer should assume an optimum design point that would give an exit gas pressure slightly below or above  $P_a$  (the ambient pressure) since too much under expansion and over expansion are undesirable. [10].

After the ANSYS workbench has been launched, a two dimensional C-D nozzle geometry is drawn and a curve fitted in the workbench, meshed in the in the mesh window and then

exported to the fluent solver where the model, material, operating condition, boundary condition are defined and solution initialized. The material is selected as air and used density as ideal gas to make the solution simpler, also energy equation chosen which includes viscous model and finally the flow simulation. Results were obtained after a minimum of 500<sup>th</sup> iterative simulations and displayed in form of figures and graphs for various parameters.

### III. RESULTS

4 is for velocity. The variations of relevant parameters with given location of the nozzle have been computed and plotted in the figures 2 to 4. Figure 1 is a representative sample of the type of meshing used in the simulation process. Figures 2 and 2.1 relate the the variation of Mach number. Typical examples of pressure plots are given in figures 3 and 3.1. The final plot in figure variation along the nozzle.

ANSYS  
14.0

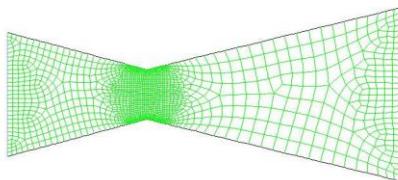


Figure 1: C-D Rocket Nozzle Geometry/ Meshed

#### CASE A: EXPANSION RATIO = 4

##### MACH NUMBER

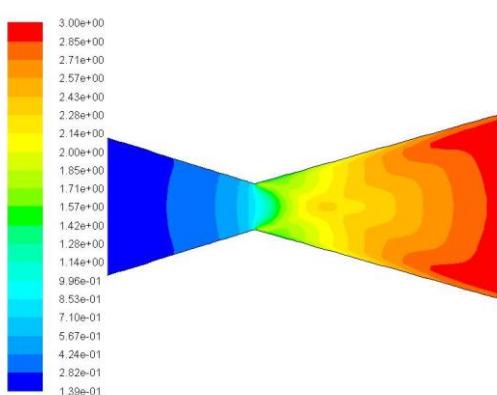
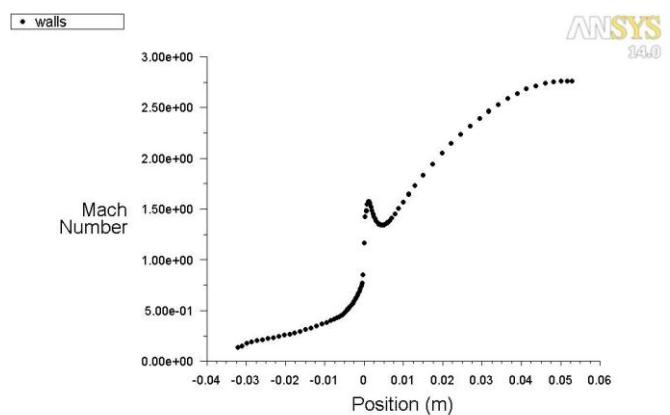


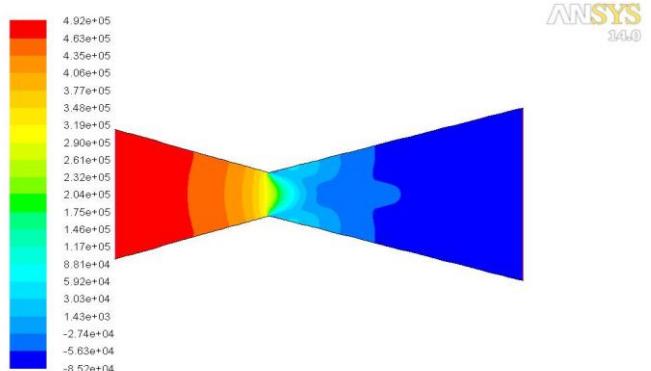
Figure 2: Contours of Mach number in the Nozzle



Mach Number  
ANSYS FLUENT 14.0 (2d, dbns imp, lam)  
Feb 23, 2016

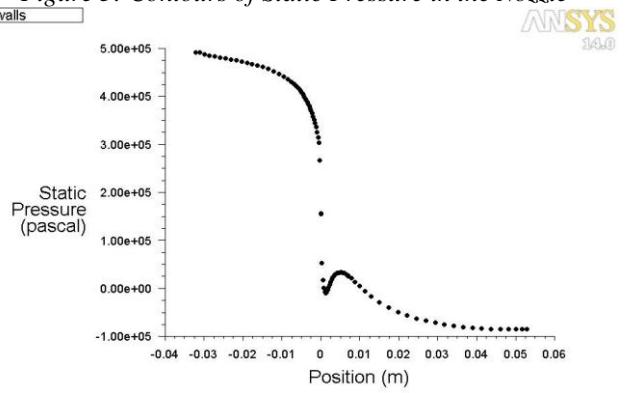
Figure 2.1: Plot of Mach number with its position in the Nozzle

##### STATIC PRESSURE



Contours of Static Pressure (pascal)  
ANSYS FLUENT 14.0 (2d, dbns imp, lam)  
Feb 23, 2016

Figure 3: Contours of Static Pressure in the Nozzle



Static Pressure  
ANSYS FLUENT 14.0 (2d, dbns imp, lam)  
Feb 23, 2016

Figure 3.1: Plot of Static Pressure with its position in the Nozzle

## VELOCITY

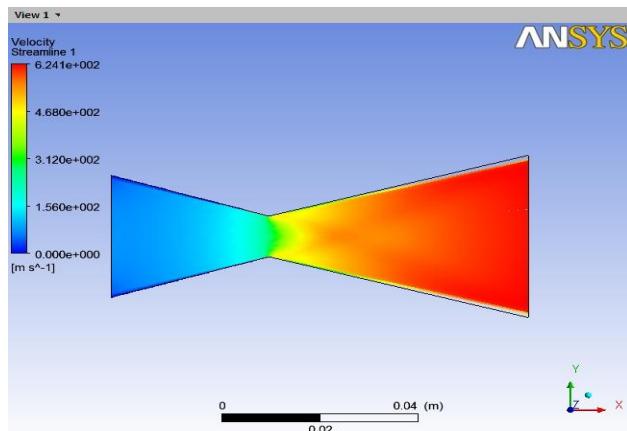


Figure 4: CFD Velocity Streamline Flow of the Nozzle

## IV. DISCUSSION

- ✓ In figure 2, the inlet section as showings blue in colour indicating low Mach number, a value of 0.139 At the throat the Mach number has risen to a value of 0.996. The Mach is found to be increasing as it passes through the divergent section. At the exit section shown in red in colour, it is found to have reached a value of 3.00 Fig 2.1 is a linear variation of the Mach number and of interest is the peak and trough which thought to be due to the formation and reflection of shock waves around the throat area.
- ✓ The peak and trough notwithstanding, fig 2.1 is a curve of increasing mach number from inlet to the exit of the nozzle.
- ✓ The static pressure contour fig3 shows a reduction in the static pressure throughout the nozzle. At the inlet the pressure shown in red is high with a reading of  $4.92 \times 10^5$  Pa. At the throat it has reduced to  $2.9 \times 10^5$  Pa. This value again reduces to  $-8.52 \times 10^4$  Pa and remains constant till the exit section shown in blue colour.
- ✓ Fig 3.1 is a curve of decreasing static pressure from inlet to the exit of the nozzle, as pressure decreases along the nozzle, the gas passing through the nozzle gains more velocity .
- ✓ The inlet section of fig.4 shown in blue colour has a zero velocity and at the throat the velocity has increased to a value of 312.0 m/s. The velocity is found to be increasing as it passes through the divergent section. At the exit section shown in red colour, the velocity is found to be 624.1m/s

## V. CONCLUSION

A two dimensional convergent-divergent nozzle geometry has been designed, meshed and simulated using ANSYS software. The results predicted have shown the variations of relevant parameters along the length of the nozzle. In the

particular case of area ratio, the investigation has found that of the three cases, case A will give the highest thrust for the rocket at the exit. Step by step procedures and necessary analysis on how to carry out the operations with ANSYS software are found in the main work.

## REFERENCES

- [1] Oskar J. (2008) "Advanced Rocket Engines" Institution of Space Propulsion, German Aerospace Center (dlr) 74239 lampoldshausen Germany. Page 1- 6.
- [2] Biju P, andSajesh. M. (2013)"Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" Department of Mechanical Engineering, NSS College of Engineering, the International Journal of Engineering and Science (Ijes) ||Volume||2 ||Issue|| 2 ||Pages|| 196-207 ||2013|| Issn: 2319 – 1813 Isbn: 2319 – 1805 www.theajes.com The IJES Page 196 .
- [3] Taylor. N, Streetant. J. and Bond.R. (2011)"Experimental Comparison of Dual Bell and Expansion Deflection Nozzles" 47TH AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibition 31 July - 03 August 2011. San diego, California Page.2 and 13.
- [4] Boyanapalli. R, Vanukuri. R. S. R, Gogineni. P, Nookala. J, Yarlagadda. G. K and Gada. V. B (2013) "Analysis Of Composite De- Laval Nozzle Suitable For Rocket Application" International Journal Of Innovation Technology And Exploring Engineering (IJITEE) ISSN: 22783075 Volume 2, Issue 5 April 2013. Pg 336 to 341.
- [5] Venkatesh. V, and Jaya pal Reddy. C, (2015) "Modeling and Simulation of Supersonic NozzleUsing Computational Fluid Dynamics" International Journal of Novel Research in Interdisciplinary Studies. Vol.2, Issue 6, November-December 2015 @www.noveltyjournals.com. Page 16-27.
- [6] Rao. G.V.R "Recent Developments in Rocket Nozzle Configurations" National Engineering Science Co. Pasadena, Calif, Download by Aerospace Cooperation on August 5, 2014 Http://Arc.Aiaa.Org/ DOI: 10251418537, Ars Journal Pg 1488
- [7] Pandey. K. M andYadav. S. K "Cfd Analysis of a Rocket Nozzle with Two. Inlets at Mach 2.1" Journal of Environmental Research and Development Vol5, No2; October-December 2010. Pg 320
- [8] ComsolMultiphysics Cyclopedia Publication on "Navier Stokes Equation' 100 District Avenue Burlington MA01803 USA COMSOL.Inc, www.wikipedia.com.
- [9] Leroy J. Krzycki "How to Design, Build and Test Small Liquid-Fuel Rocket Engines" Printed in the United States of America First printing: March 1967 Second printing: March 1971 ISBN 9600-1980-4 PDF version created by Tim Patterson, http://www.rocketry.org/~tim/.8115 June 1975. Pg 23-103.
- [10] Dieter. K. Huzel and Dard. H. Haung "Design of Liquid Propellant Rocket Engines" Published by National Aeronautics and Space Administration. July 19/1967.