[Khaleel, 3(9): September, 2014]

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Design and CFD Analysis of Conical Nozzle for 2.75 Inch Rocket System

MD. Khaleel*, N. Prabhu Kishore, Dr. A. Siva Kumar
*PG. Scholar, Assistant Professor, Professor
MLR Institute of Technology, Rangareddy, Telengana, India
khaleelpasha.fts@gmail.com

Abstracts

The 2.75 Inch Rocket System is a multi-service system consisting of launchers, warheads, and rocket motors. The air vehicle is a warhead and rocket motor combination that flies a free flight ballistic trajectory to the target. While the 2.75-Inch Rocket System has been modified over time to improve the performance, the system still experiences problems hitting a point target resulting in a high number of rockets needed to perform a mission. This problem is due to both low precision (wide impact distribution) and low system accuracy. Precision is affected by thrust misalignment, downwash and crosswinds whereas accuracy is affected by the launch system (aircraft and launcher) and the launch environment (air density). Both precision and accuracy need to be improved to increase the ability of the 2.75-Inch Rocket System to hit targets. The Naval Surface Weapons Center, Indian Head (NSWC-IHD) and General Dynamics Armament Systems (GDAS) are jointly developing a high-torque nozzle to improve the rocket precision. The current nozzle produces 3 ft-lbf of torque. The high-torque nozzle is designed to provide more than 10 ft-lbf of torque and significantly improves rocket precision by averaging out the thrust misalignment. The nozzle torque will shut-off at rocket exit from the launcher to limit the rocket spin rate below its natural bending frequency. A separate effort will modify the large pop-out fins to reduce the downwash effect from the helicopter. The program is currently in the design phase.

Keywords: Rocket Nozzle; Unigraphics; Fluid Dynamics; Pressure ratio; Control Volume

Introduction

Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object.

A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A nozzle is often a pipe or tube of varying cross sectional area and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them.

A nozzle is a specially shaped simple device used for the gases to escape. There are different types of nozzles used in rockets. In rockets a convergent divergent nozzles or CD nozzles as they are called are used. There are number of advanced designs in rocket design. Plug nozzle and aerospike nozzle comes under it. A plug nozzle consists of plug around where the working fluid flows and plug nozzle is used to adjust the flow rate of the fluids. Aero spike engine is a different type of nozzle which is a rectangular plug nozzle and has multiple combustion chambers. These nozzles are not yet used in commercial rockets as they are in still in the experimental phase.

Simply put, the nozzle is the component of a rocket or air-breathing engine that produces thrust. This is accomplished by converting the thermal energy of the hot chamber gases into kinetic energy and directing that energy along the nozzle's axis, as illustrated below.

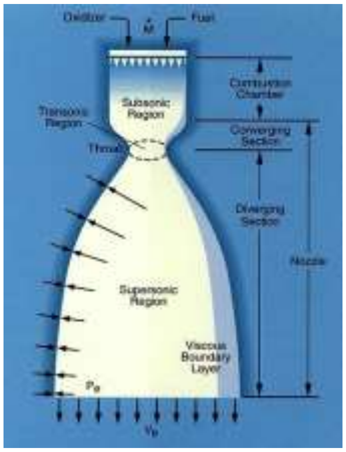

Fig.1 Simple representation of a Rocket Nozzle

Although simplified, this Fig illustrates how a rocket nozzle works. The propellant is composed of a fuel, typically liquid hydrogen (H $_2$), and an oxidizer, typically liquid oxygen. The propellant is pumped into a combustion chamber at some rate where the fuel and oxidizer are mixed and burned. The exhaust gases from this process are pushed into the throat region of the nozzle. Since the throat is of less cross-sectional area than the rest of the engine, the gases are compressed to a high pressure. The nozzle itself gradually increases in cross-sectional area allowing the gases to expand. As the gases do so, they push against the walls of the nozzle creating thrust.

Conical nozzle

The conical nozzle was used often in early rocket applications because of its simplicity and ease of construction. The cone gets its name from the fact that the walls diverge at a constant angle. A small angle produces greater thrust, because it maximizes the axial component of exit velocity and produces a high specific impulse (a measure of rocket efficiency). The penalty, however, is a longer and heavier nozzle that is more complex to build. At the other extreme, size and weight are minimized by a large nozzle wall angle. Unfortunately, large angles reduce performance at low altitude because the high ambient pressure causes overexpansion and flow separation.

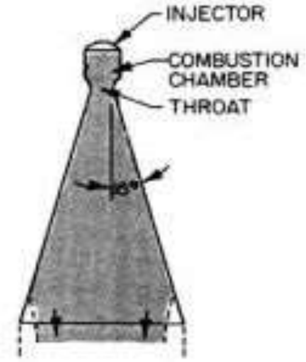

Fig.2 Conical Nozzle

Design methodology of nozzle

The conical nozzle is simply a cone shape described by the cone's half angle as viewed from the side. For instance if the nozzle is described as a 12 degree conical nozzle this means that from the centerline of the nozzle to the inside wall there is a 12 degree angle. This type of nozzle is seen historically and is typical for solid and hybrid rocket engines. Some of the advantages of a conical nozzle are its simple cone shape for design, and it contains no inflection as the propellants are expelled from the combustion chamber. This lack of inflection means that the nozzle is a straight line coming out of the throat to the exit. This lack of inflection is critical for solid and hybrid engines because these types of engines usually have some pieces of solid propellant expelled all the way out of the nozzle. Therefore, a conical nozzle is desired for solid and hybrid propellant types due to the lack of inflection.

The exit angle of a conical nozzle is the same as the cone angle, the flow exits at not parallel but rather at the cone angle. When there is an angle at the exit the flow experiences a divergence loss which causes energy loss and in turn a loss of nozzle efficiency.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

Fig.3 Extruded nozzle body

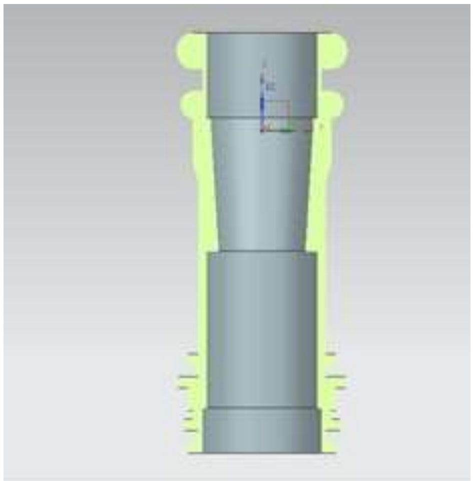

Fig.4 Nozzle cross-sectional view

With a conical nozzle the exit angle is large and therefore the divergence loss is maximized. Another disadvantage is a conical nozzle contains more material and therefore mass than a Bell nozzle of the same design. In general for a design, the more massive the launch vehicle is the more expensive it will be. These factors are all some of the downfalls when using a conical nozzle design. The Fig.3 & Fig .4 shows the CAD model of Extruded nozzle body and Nozzle cross sectional view

The conical nozzle CAD model was designed by first making a profile of the cone half angle. From the MAT outputs the exit diameter and expansion ratio were known values. Using this expansion ratio (ε) and the exit area (A_e) based on the exit diameter, the throat area (A_t) and thus throat diameter could be determined using below equation.

$$\varepsilon = \frac{A_e}{A_t}$$

The profile view was then rotated along the centerline in order to make a solid nozzle shape. The nozzle shape was then shelled out in order to make the nozzle. The finished CAD model can be seen in Fig.5 & Fig.6

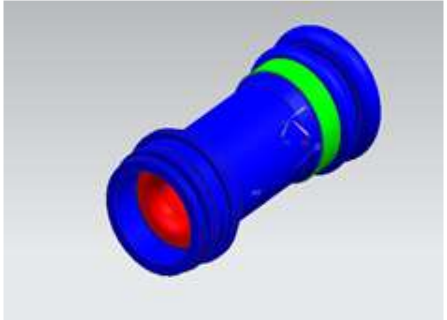

Fig.5 Assembly model

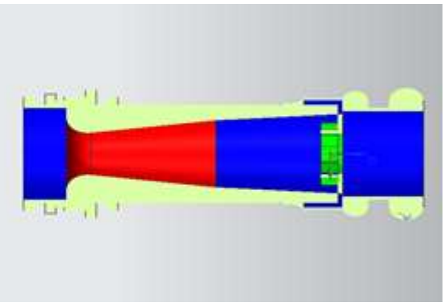

Fig.6 Cross-sectional view of the Assembly model

Computational fluid dynamic analysis of the nozzle

Computational fluid dynamics or CFD is the analysis of systems involving heat transfer, fluid flow and associated phenomenon like chemical reaction by means of computer based simulation. CFD is a new branch of design engineering which integrates the discipline of Fluid mechanics/Dynamics with mathematics and also with computer science as shown in fig.7

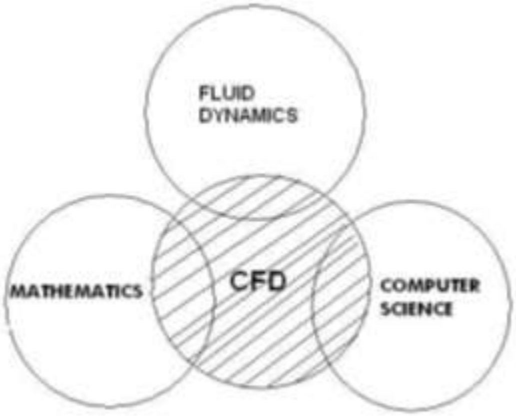

Fig.7 Computational Fluid Dynamics interrelation

CFD is especially dedicated to the study of fluid in motion and how the fluid flow behavior affects processes that may include heat transfer. The ultimate and unique aim of developments in the CFD fields is to provide a capability comparable to other CAE (computer aided engineering) tools such as stress analysis codes etc. The CFD simulation technique is very powerful and it covers a wide range of industrial and non industrial application areas.

There are several unique advantages of CFD over experiment based approaches to fluid system design. This method is used to Substantial reduction in lead times and costs of few designs and CFD enables us to study systems where controlled experiments are difficult or impossible to perform (Large systems). CFD Provides the environment to study under hazardous conditions and beyond their normal performance limits.

CFD consists the mathematical basis for a general purpose practical model of fluid flow and heat transfer from the basic principles of physics. These principles are conservation of mass, conservation of momentum and conservation of energy. This leads to governing equations of flows and a discussion of necessary secondary conditions. CFD package consist of three main parts. These are Pre-processor, Solver, Post processor. Pre-Processor defines the Definition of geometry (computation domain), Grid generation (formation of sub-domain), Selection of physical or chemical phenomenon, Fluid property definition and Specification of boundary conditions.

There are three different streams of numerical solution technique. Finite difference, finite element and spectral methods. Basically solver contains numerical methods that perform three steps, there are Approximation of unknown flow variables by means of simple functions, Discretization and Solution of algebraic equations. Post processor includes the geometry of domain of problem and display, the vector plots, 2D-3D surface plots, colored postscript output and animation for dynamic result display.

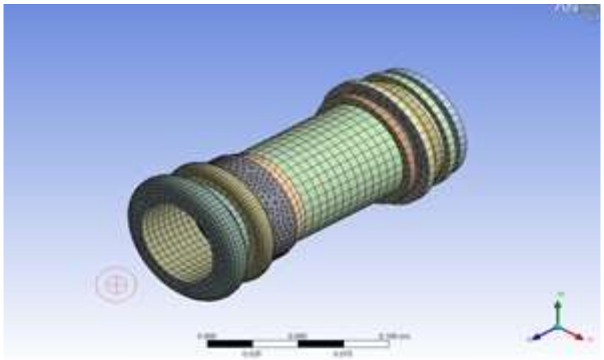

Fig.8 Generating mesh Isometric view

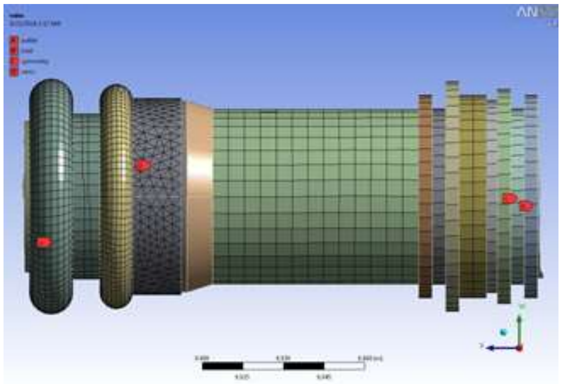

Fig.9 All named sections on nozzle

Results

There is symmetric flow as observed from the above figure. There is a constant increase in the velocity magnitude of the fluid as we move from the inlet of the convergent section (=1.74e+02 m/s) to the throat (=3.92e+02 m/s) and to the exit of the divergent section (=8.70e+02 m/s). The flow is turbulent, so near the wall flow separation takes place causing the velocity to decreases to nearly 8.70e+01 m/s. The maximum velocity is reached at the exit.

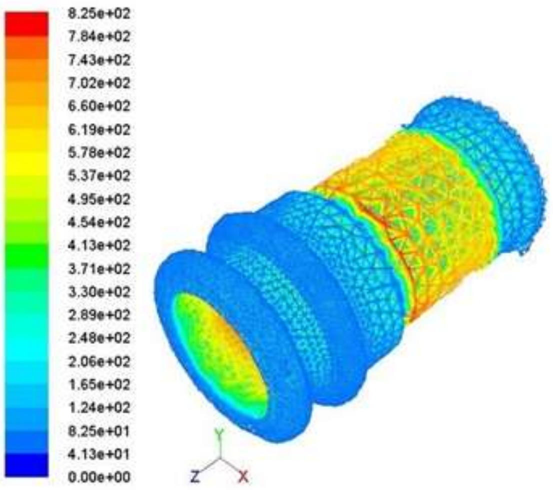

Fig.10 Contours of velocity magnitude

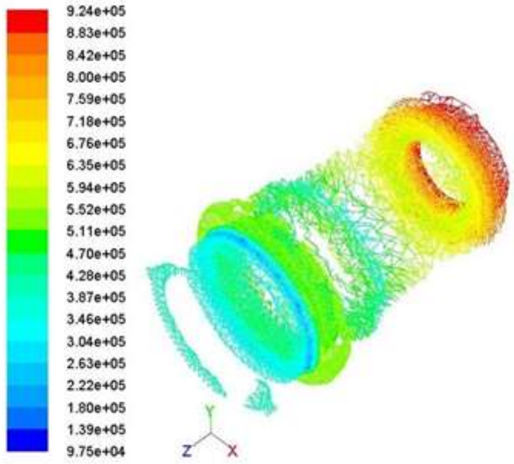

Fig.11 Contours of static pressure

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that the gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3.23e+05 Pa and as we move towards the throat there is a decrease and the value at the throat is found out to be 2.30e+05 Pa. After the throat, there is sudden expansion and the static pressure falls in a more rapid manner towards the exit of the nozzle. At the exit it is found to be 1.13e+04

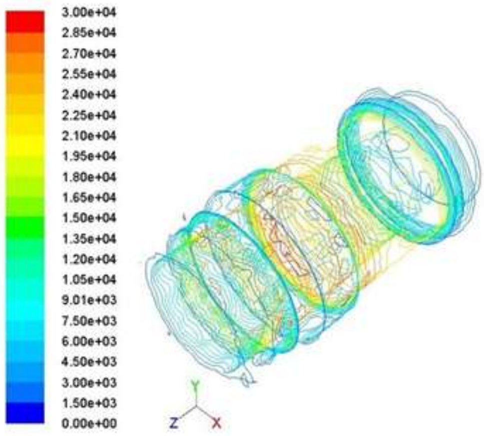

Fig.12 Contours of dynamic pressure

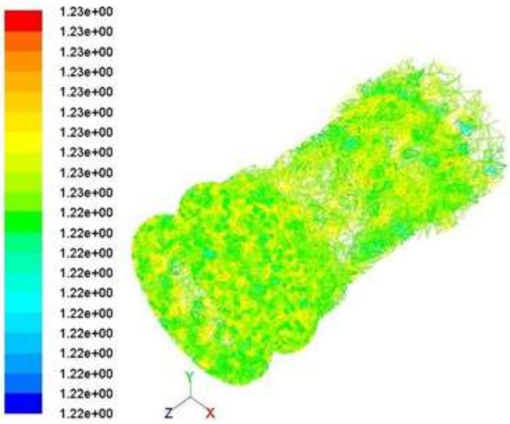

Fig.13 Contours of density

From the above figure, you can easily visualize in the above figure that, there is decrease in stagnation pressure near the nozzle walls. This is due to the viscous effects, whereas the stagnation pressure remains almost constant in the centre. The value uniformly in the centre is found to be 3.64e+05 Pa except near the walls where it decreases to 5.41e+04 Pa. There is a slight increase in the total pressure in the centre as well as the walls when compared to the default angle version of the nozzle.

Conclusions

This Nozzle design and modeling demonstrated the potential for shorter duration increased torque generation from an eroding torque ring. Two design approaches were investigated; a plastic erodible vane installed within the motor nozzle and a metal tabbed vane extending into the plume immediately aft of the nozzle that rotated out of the plume after launcher exit, plastic erodible vanes are preferable. A tube launched rocket comprising a torque vane system having a plurality of substantially low erodible vanes, the torque vane system attachable to the rocket, wherein the plurality of low erodible vanes are positionally fixed aft of the rocket propelling thrust effective to impart rapid spin to the rocket and one or more erodible retainers attachable to the torque vane system, wherein the one or more erodible retainers retain the torque vane system in a manner that secures the low erodible vanes in a fixed angular orientation within the rocket propelling thrust effective to provide torque to the rocket until the one or more erodible retainers erode wherein the torque vane system remains attached to the rocket and ceases to provide torque to the rocket and launching the rocket from the launch tube In addition, the greater division of thrust for torque and the new nozzle configuration did not adversely affect the motor's ballistic or safety performance. .The nozzle is used to convert the chemical thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature.

References

1. Stark R., Kronmüller H., and Zerjeski D., "Advanced Flow Visualisation Techniques in Cold Gas Subscale Nozzles, a Comparison", AIAA Paper 2003-5180, 2003.
2. Kronmüller H., Schäfer K., Zimmermann H., Stark R., "Cold Gas Subscale Test Facility P6.2 at DLR Lampoldshausen", 6th Symposium on Propulsion for Space Transportation of the XXIth century, Versailles, 2002
3. Makatar Wae-hayee, Perapong Tekasakul, and Chayut Nuntadusit "Influence of nozzle arrangement on flow and heat transfer characteristics of arrays of circular impinging jets" Songklanakarin J. Sci. Technol. 35 (2), 203-212, Mar. - Apr. 2013
4. Balaji Krushna.P, P. SrinivasaRao, B. Balakrishna "Analysis Of Dual Bell Rocket Nozzle Using Computational Fluid Dynamics" Ijret, Eissn 2319-1163 | Pissn 2321-7308
5. Biju Kuttan P, ,M Sajesh " Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" IJES Volume2 Issue 2 ||2013|| Issn 2319 – 1813 Isbn 2319 – 1805
6. Hagemann G., Preuss A., Grauer F., Frey M., Kretschmer J., Rydén R., Jensen K., Stark R., and Zerjeski D., "Flow Separation and Heat Transfer in High Area Ratio Nozzles", AIAA Paper 2004-3684, 2004
7. Ralf H. Stark "Flow Separation in Rocket Nozzles, a Simple Criteria" German Aerospace Center, Lampoldshausen,D-74239, Germany .
8. Tom Farabaugh, and Mike Kessinger "Improved IM Response for Future 2.75" APKWS with Composite Case Technology" Alliant Techsystems Inc. Christina Davis AMRDEC, Redstone Arsenal, Alabama
9. Zill A, "Flow separation in rectangular over-expanded supersonic nozzles", Paper AIAA 2006-17, 2006 .
10. A. A. Khan and T. R. Shembharkar "Viscous Flow Analysis In A Convergent-Divergent Nozzle" Proceedings of the International Conference on Aerospace Science and Technology 26 - 28 June 2008, Bangalore, INCAS-2008-004
11. Dipak J. Choudhari, Uday V. Asolekar. " Efficiency Analysis of an Aerospike Nozzle" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 National Conference on Emerging Trends in Engineering & Technology (VNCET-30 Mar'12).
12. A. Bernstein, W. H. Heiser, C. Hevenor "Compound-Compressible Nozzle Flow" Transactions of the ASME September 1967
13. Raviteja Boyanapalli,Raja Sekhara Reddy, Vanukuri, Prudhvi Gogineni, Janakinandan Nookala, Goutham Kumar Yarlagadda,VinayBabu Gada " Analysis of Composite De-Laval Nozzle Suitable for Rocket Applications" International Journal of Innovative Technology and Exploring Engineering (IJITEE) ISSN: 2278-3075, Volume-2, Issue-5, April 2013.
14. Khaled S. Abdol-Hamid1, S. Paul Pao2, Craig A. Hunter3, Karen A. Deere , Steven J. Massey, Alaa Elmiligui "PAB3D: Its History in the Use of Turbulence Models in the Simulation of Jet and Nozzle Flows" NASA Langley Research Center, Hampton, VA 23681, Eagle Aeronautics, Hampton, VA 23666, AS&M, Inc., Hampton, VA 23666