"HENRI COANDA"
AIR FORCE ACADEMY
ROMANIA

"GENERAL M.R. STEFANIK"
ARMED FORCES ACADEMY
SLOVAK REPUBLIC

INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER
AFASES 2015
Brasov, 28-30 May 2015

ANALYSIS OF FLOW IN CONVERGENT-DIVERGENT ROCKET ENGINE NOZZLE USING COMPUTATIONAL FLUID DYNAMICS

Bogdan-Alexandru Belega*, Trung Duc Nguyen**

*Military Technical Academy, Bucharest, Romania, **Paul Sabatier University, Toulouse, France

Abstract: *Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that exhaust from them. Nozzles come in a variety of shapes and sizes depending on the mission of the rocket, this is very important for the understanding of the performance characteristics of rocket. By the proper geometrical design of the nozzle, the exhaust of the propellant gases will be regulated in such a way that maximum effective rocket velocity can be reached. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. After getting heated the propellant first converges at the throat of the nozzle and then expands under constant temperature in the divergent part. In the present paper, flow through the convergent divergent nozzle study is carried out by using a finite volume rewarding code, FLUENT 6.3. The nozzle geometry modeling and mesh generation has been done using GAMBIT 2.4 Software. Computational results are in good acceptance with the experimental results taken from the literature.*

Keywords: *CFD, fluent, nozzle, Gambit, combustion chamber.*

1. INTRODUCTION

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 temperature.

The general range of exhaust velocity is 2 to 4.5 kilometer per second. The convergent and divergent (also known as convergent-divergent nozzle – figure 1) type of nozzle is known as DE-LAVAL nozzle. [2,3]

The inlet Mach number is less than one, Convergent section accelerates it to sonic velocity at the throat and further accelerated to supersonic velocities by the diverging section.

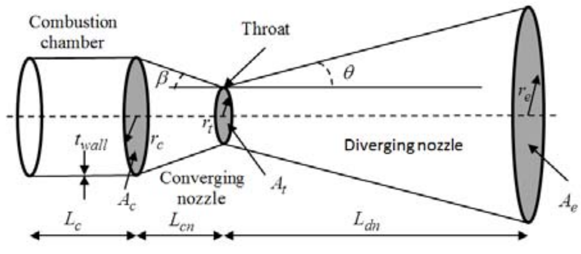

Figure 1 – Convergent-divergent nozzle

In this project the designing and analysis of CD nozzle geometry is done in the CFD (Computational Fluid Dynamics software). Firstly the design of nozzle is made in Gambit software and then the nozzle geometry is further analyzed in fluent software in order to analyze the flow inside the CD nozzle and to get the view of the behavior of fluid inside the convergent-divergent section of nozzle. [1,2]

2. NOZZLE GEOMETRY ANALYSIS

2.1 Rocket nozzle equations. The function of the nozzle is to accelerate gases produced by the propellant to maximum velocity in order to obtain maximum thrust. The amount of thrust produced by the engine depends on the mass flow rate through the engine, the exit velocity of the flow, and the pressure at the exit of the engine. The value of these three flow variables are all determined by the rocket nozzle design.

For steadily operating rocket propulsion system moving through a homogeneous atmosphere total thrust and specific impulse are:

$$F = \dot{m} \cdot v_e + (p_e - p_0) \cdot A_e \tag{1}$$

$$I_{sp} = \frac{F}{\dot{m} \cdot g_0} \tag{2}$$

The first term is the momentum thrust and the second term represents the pressure thrust. The rocket nozzle is usually so designed that the exhaust pressure is equal or slightly higher than the ambient fluid pressure. Because changes in ambient pressure affect the pressure thrust, there is a variation of the rocket thrust with altitude (between 10% and 30%).

Velocity of sound and Mach number:

$$a = \sqrt{\gamma \cdot R \cdot T} \tag{3}$$

$$M = \frac{v}{a} \tag{4}$$

The stagnation properties of a flow are those properties which would result if the flow is isentropic. Stagnation properties are constant in an isentropic flow. Thus, properties along the nozzle are best referenced against the stagnation properties. With these assumptions of ideal gas and isentropic flow, ratios of pressure, density and temperature can be related to the stagnation pressure, density and temperature at a given Mach number.

$$\frac{T_0}{T} = \left[1 + \frac{(\gamma - 1)}{2} \cdot M^2\right] \tag{5}$$

$$\frac{p_0}{p} = \left[1 + \frac{(\gamma - 1)}{2} \cdot M^2\right]^{\frac{\gamma}{\gamma-1}} \tag{6}$$

$$\frac{\rho_0}{\rho} = \left[1 + \frac{(\gamma - 1)}{2} \cdot M^2\right]^{\frac{1}{\gamma-1}} \tag{7}$$

Additionally, the ratio of the local area to the throat area can be specified by the Mach number:

$$\frac{A}{A_t} = \frac{1}{M}\left[\frac{2}{\gamma + 1}\left(1 + \frac{\gamma - 1}{2} M^2\right)\right]^{\frac{\gamma+1}{2\cdot(\gamma-1)}} \tag{8}$$

In a converging-diverging nozzle a large fraction of the thermal energy of the gases in the chamber is converted into kinetic energy. The flow velocity can be obtained from the conservation of total enthalpy h_0:

$$v_e = \sqrt{2 \cdot (h_0 - h_e)} \tag{9}$$

From the isentropic relations the equation becomes:

$$v_e = \sqrt{\frac{2\gamma}{\gamma - 1} \cdot \frac{RT_0}{M} \cdot \left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{10}$$

An increase of the ratio $\frac{T_0}{M}$ will increase the performance of the rocket. The influences of the pressure ratio $\frac{p_0}{p_e}$ and of the specific heat ratio γ are less pronounced.

The nozzle area expansion ratio (ε) is an important nozzle design parameter:

$$\varepsilon = \frac{A_e}{A_t} \tag{11}$$

The maximum gas flow per unit area occurs at the throat (critical values):

$$\frac{p_t}{p_0} = \left[\frac{2}{(\gamma + 1)}\right]^{\frac{\gamma}{\gamma-1}} \quad (0.53 - 0.57) \tag{12}$$

$$\frac{T_t}{T_0} = \left[\frac{2}{(\gamma + 1)}\right] \quad (0.83 - 0.91) \tag{13}$$

$$\frac{\rho_t}{\rho_0} = \left[\frac{2}{(\gamma + 1)}\right]^{\frac{1}{\gamma-1}} \quad (0.62 - 0.63) \tag{14}$$

Throat velocity v_t is:

$$v_t = \sqrt{\frac{2\gamma}{\gamma + 1} RT_0} = \sqrt{\gamma RT_t} \tag{15}$$

To attain sonic/supersonic flow:

$$\frac{p_0}{p_e} \geq \left[\frac{(\gamma + 1)}{2}\right]^{\frac{\gamma}{\gamma-1}} \quad (1.75 - 1.89) \tag{16}$$

The mass flow rate as a function of nozzle geometry and fluid properties can be found from basic continuity where v the average velocity is, A is the nozzle area, ρ is the density and $\dot{m}$ is flowrate:

$$\dot{m} = \rho v A = constant \quad (17)$$

After substitutions we have De Saint Venant's Equation:

$$\frac{\dot{m}}{A} = p_0 \sqrt{\frac{2\gamma}{\gamma - 1} \frac{1}{RT_0} \left(\frac{p}{p_0}\right)^{\frac{2}{\gamma}}} \cdot \sqrt{\left[1 - \left(\frac{p}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} \quad (18)$$

When sonic velocity is reached at the throat, it is not possible to increase the throat velocity or the flow rate in the nozzle by further lowering the exit pressure (choking the flow).

Choking is a compressible flow effect that obstructs the flow, setting a limit to fluid velocity because the flow becomes supersonic and perturbations cannot move upstream; in gas flow, choking takes place when a subsonic flow reaches $M = 1$.

Mass flow rate:

$$\dot{m} = \rho_t v_t A_t = \frac{p_0 A_t \sqrt{\gamma}}{\sqrt{RT_0}} \cdot \left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma+1}{2(\gamma-1)}}$$

$$\Rightarrow \quad \dot{m} = \Gamma \cdot \frac{p_0 A_t}{\sqrt{RT_0}} \quad (19)$$

where $\Gamma = \sqrt{\gamma} \cdot \left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma+1}{2(\gamma-1)}}$

The area ratio is:

$$\frac{A_e}{A_t} = \frac{\Gamma}{\sqrt{\frac{2\gamma}{\gamma - 1} \left(\frac{p_e}{p_0}\right)^{\frac{2}{\gamma}} \left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]}} \quad (20)$$

The velocity ratio is:

$$\frac{v_e}{v_t} = \sqrt{\frac{\gamma + 1}{\gamma - 1} \left[1 - \left(\frac{p_e}{p_0}\right)^{\frac{\gamma-1}{\gamma}}\right]} \quad (21)$$

2.2 Calculated values for nozzle dimensions and geometry. The dimensions of the convergent-divergent nozzle geometry are obtained through the following equations which are used in every spacecraft available during the present day.

Mass flow in rocket is calculated by:

$$\dot{m} = \frac{F_{thrust}}{v_e} \quad (22)$$

where $F_{thrust} = 1.2MN$ and $v_e = 3500m/s$

Putting the given values in the equation we obtain:

$$\dot{m} = 342.86 kg/s$$

For high altitude (100 km or higher) expansion ratio in nozzle, given by (11), are between 40 and 200.

Area of the nozzle throat:

$$A_t = \frac{\dot{m}}{P_c \sqrt{\gamma \left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma+1}{\gamma-1}} \frac{M}{R_u T_c}}} \quad (23)$$

Given:

$M = 10$; $R_u = 287 \frac{J}{kg \cdot K}$; $T_c = 3000K$; $\gamma = 1.2$; $P_c = 12MPa$ and putting these values in (23) we get:

$$A_t = 0.0129m^2$$

Now by using equation (11) we can get the value of A_e for $\varepsilon = 74$:

$$A_e = 0.95m^2$$

Also the exit area of nozzle is given by:

$$A_e = \pi \cdot r_e^2 \quad (24)$$

obtaining: $r_e = 0,55m$

Convergence area in Nozzle is:

$$A_c = 3 \cdot A_t \quad (25)$$

$$A_c = 0.0387 m^2$$

Radius of throat:

$$r_t = \sqrt{\frac{A_t}{\pi}} \quad (26)$$

$$r_t = 0.064m$$

Combustion radius:

$$r_c = \sqrt{\frac{A_c}{\pi}} \quad (27)$$

$$r_c = 0.111m$$

Given $\theta = 15^0$ and $\beta = 60^0$ we can calculate diverging Nozzle length:

$$L_{dn} = \sqrt{\frac{A_e}{\pi}} \cdot \frac{1}{\tan\theta} \quad (28)$$

$$L_{dn} = 2.04m$$

Length of the converging nozzle:

$$L_{cn} = \sqrt{\frac{A_c}{\pi}} \cdot \frac{1}{\tan\beta} \quad (29)$$

$$L_{cn} = 0.081m$$

Length of the combustion chamber:

$$L_c = \frac{A_t \cdot L^*}{\pi \cdot r_c^2} \quad (30)$$

A parameter describing the chamber volume required for complete combustion is the characteristic chamber length, L^*, which is given by:

$$L^* = \frac{V_c}{A_t} \quad (31)$$

where V_c is the chamber volume (including the converging section of the nozzle) and A_t is the nozzle throat area. For gaseous oxygen/hydrocarbon fuels, an L^* of 1.27 to 2.54 meters is appropriate. For $L^* = 1.27m$, chamber length is:

$$L_c = 0.423m$$

3. METHODOLOGY AND IMPLEMENTATION

3.1 Modeling the nozzle. The geometry of the nozzle was created using the Geometry workbench of GAMBIT modeling package for developing the wire frame which resembles the cross-section of the rocket nozzle (figure 2). A tri-dimensional geometry of the nozzle was created by using the value of $A_e; r_e; A_t; r_t; L_{dn}; A_c; r_c; L_{cn}; L_c; t_{wall}$.

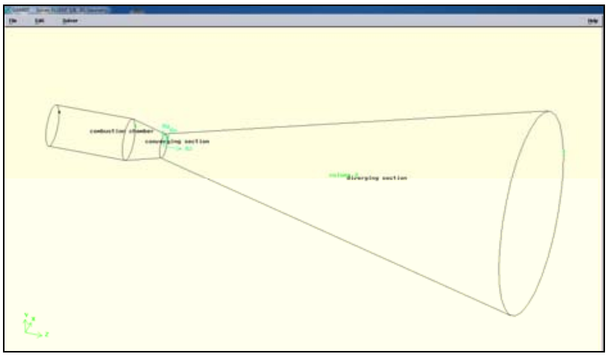

Figure 2 – Nozzle Geometry

3.2 Meshing in Gambit. The next task was to mesh the geometry created. In GAMBIT the mesh used was tetrahedral mesh elements and proper care is taken while meshing the regions near the walls of the nozzle so as to get more refinement in that particular regions. As any computational process requires a mesh to carry computation this step is a primary and most important to start the problem. The mesh created in GAMBIT is as shown below figure.3.

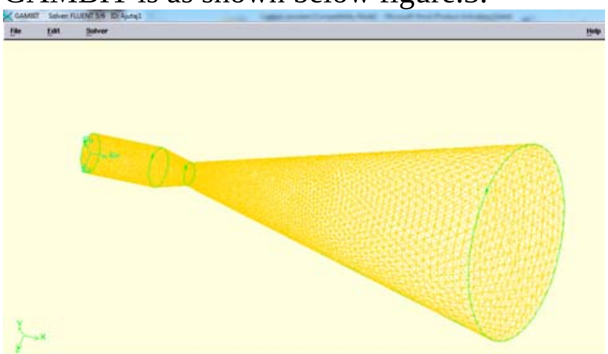

Figure 3 – Nozzle Geometry (Mesh)

We can see that the meshing near the boundary of the nozzle is more refined when compared to other regions of mesh. The mesh was refined to the third degree using the refinement option of the GAMBIT. After meshing, the inlet, the axis and the outlet boundaries were named.

3.3 Boundary Conditions used. Specification of the boundary zones has to be done in GAMBIT only, there is no possibility to specify the boundary zones in FLUENT. Accordingly the geometry of the nozzle is divided into zones and boundary conditions given to these are: the inside nozzle surfaces are given as WALL; the inlet is given as mass flow inlet; the outlet is given as pressure outlet. With all the zones defined properly the mesh is exported to the solver. The solver used in this paper is FLUENT. The exported mesh file is read in fluent for solving the problem.

The fluent conditions to the design nozzle geometry are energy equation included, viscous model- K epsilon, materials are Hydrogen and Titanium for fluid flow and wall respectively, mass flow rate (Inlet) $10kg/s$ and outlet gauge pressure and temperature are 0 and 273 k respectively.

3.4 Solving. After initiating the numerical analysis, convergence was obtained after 332th iteration in case of nozzle analyzed. The steady axisymmetric implicit formulation with coupled solver with K-e turbulence model is chosen. The mass flow of the burning solid propellant was modeled using a user-defined function. [4, 5]

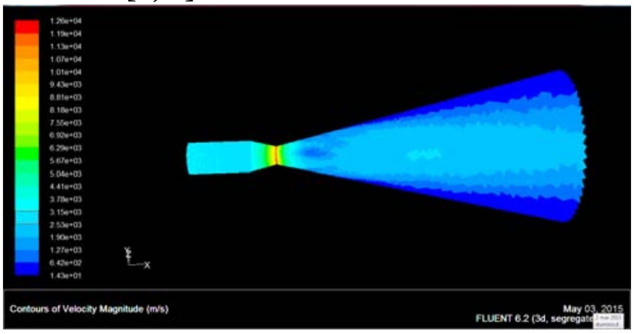

Figure 4 – Velocity distribution of nozzle geometry

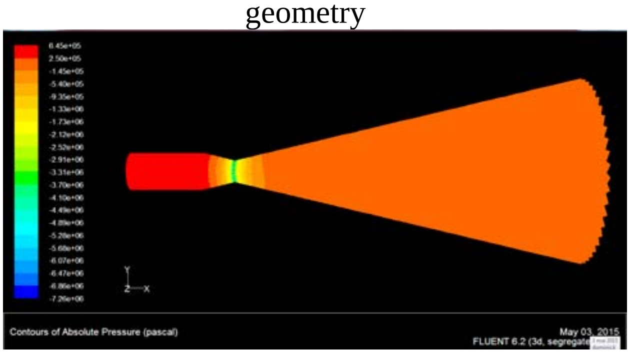

Figure 5 – Absolute pressure distribution of nozzle geometry

In figure 4 and figure 5 velocity distribution and absolute pressure of nozzle geometry is shown respectively.

It is clearly seen the velocity is increasing along with the length of the nozzle. Due to shocking in the nozzle, the velocity decreased for a while but later began to increase as the fluid expanded through the divergent portion.

Pressure gradually decreased along the length of the nozzle except a slight rise during the shocking. However, the rise was not significant comparing to the total fall in pressure. According to Bernoulli's equation, pressure decrease as velocity increases along the expansion zone.

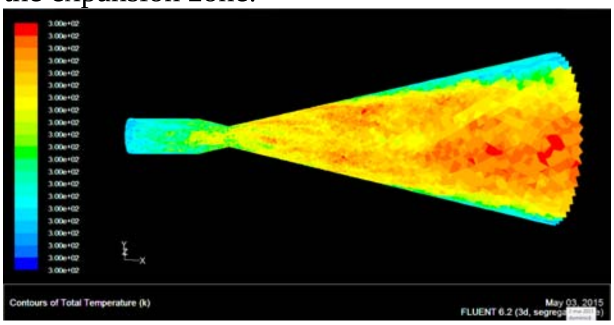

Figure 6 – Total temperature distribution of nozzle geometry

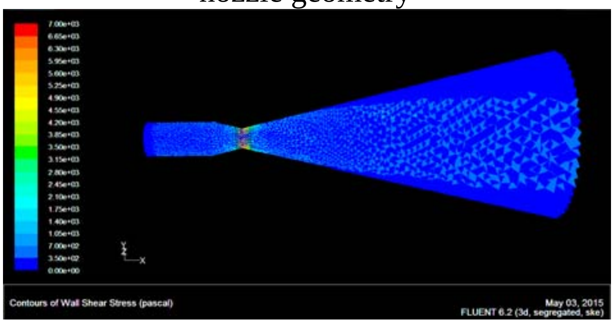

Figure 7 – Wall shear stress distribution of nozzle geometry

In figure 6 and figure 7 total temperature and wall shear stress of nozzle geometry is shown respectively.

It is seen that temperature decreased gradually except a slight increase. The slight increase occurs in the shock zone where rapid change of fluid properties takes place. But the rise in temperature was not significant with respect to the fall in temperature throughout the distance.

4. CONCLUSIONS & ACKNOWLEDGMENT

After successfully completing this simulation of a design created, the decisions were finally confined into the following points.

From the analysis, it is clearly observed that nozzle created based on exit parameters it is in accord with the scope.

REFERENCES

1. Mohan Kumar G., Dominic Xavier Fernando, R. Muthu Kumar, - *Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation* - Advances in Aerospace Science and Applications. ISSN 2277-3223 Volume 3, Number 2 (2013), pp. 119-124.
2. Mayur Chakravarti - *Design analysis of flow in convergent divergent rocket nozzle* - University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India, 2011.
3. Vipul Sharma, Ravi Shankar, Gaurav Sharma, - *CFD Analysis of Rocket Nozzle* - University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India, 2011.
4. Biju Kuttan P, M Sajesh - *Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics* - The International Journal Of Engineering And Science, volume 2, pages 196-207, 2013, ISSN: 2319 – 1813.
5. MD. Safayet Hossain, Muhammad Ferdous Raiyan, Nahed Hassan Jony - *Comparative study of supersonic nozzles* - International Journal of Research in Engineering and Technology, volume 3, pages 351-357, oct.2014 ISSN: 2319-1163.