NUMERICAL INVESTIGATION ON THE CORRELATION BETWEEN SHOCK STRUCTURE AND THRUST PERFORMANCE IN AN OVEREXPANDED NOZZLE

Hyun Ah Choi*, Ho Dong Kam, Jeong Soo Kim***
*** Department of Mechanical Engineering, Pukyong National University, Busan, Republic of Korea**
**** R&D center, Hanwha Corp., Daejeon, Republic of Korea**

Keywords: *Overexpanded Nozzle, Shock Structure, Thrust Performance, Gas Dynamics*

Abstract

As a result of overexpansion of gas, nozzle exit pressure falls below the atmospheric pressure and irreversible phenomena like shock and flow separation occur inside the nozzle. The overexpanded nozzle is considered unsuitable to use as it causes a severe performance loss. However, the overexpanded nozzle is valuable to study effects of the shock structure on thrust performance when the shock exists inside the nozzle. Hence, a numerical simulation is made to the overexpanded nozzle flow to discover the interrelation of shock structure and thrust performance. Reynolds-averaged Navier-Stokes equation with k-ω SST turbulence model is employed to compute the nozzle flow at various NPR(Nozzle Pressure Ratio). In case the NPR is raised, the thrust performance monotonically increases under overexpanded conditions.

List of symbols

δ_{ij}	Kronecker delta
D	diffusion coefficient
E	total enegy
h	static enthalpy
$\vec{J}$	diffusion flux
k	thermal conductivity
M	Mach number
μ	molecular viscosity
p	static pressure
ρ	density
T	static temperature
τ	shear stress
u	mean velocity
x	Cartesian coordinate
Y	dissipation rate

Subscripts

eff	effective
m	mass
T	thermal
k	turbulence kinetic energy
ω	turbulent dissipation rate

1 Introduction

An overexpansion usually happens because nozzle exit pressure is below the atmospheric pressure when the nozzle has too large area-expansion ratio or low NPR (Nozzle Pressure Ratio) condition[1]. In the case of high overexpansion, the exhaust jet of supersonic nozzle separates from nozzle wall because of the large adverse pressure gradient. Correspondingly, to match the pressure of the separated flow region, an oblique shock is generated which evolves through the supersonic jet starting around the separation point[2]. And the rise of gas pressure at the nozzle exit to atmospheric pressure also causes shock waves and a contraction of plume diameter resulting in a thrust performance loss. In rocket design community, shock-induced separation is considered undesirable because an asymmetry in the flow can yield dangerous lateral forces, so-called side-loads which may damage the nozzle[3].

As the overexpanded nozzle has many critical disadvantages, such configuration is not preferred in the nozzle design process. Nevertheless, the overexpansion inevitably happens when the thruster is exposed to the firing test under ground condition. Furthermore large internal aerodynamic side forces can arise inside the nozzle-exit area undergoing an adverse pressure gradient. when the flow does separate asymmetrically, the forces are sometimes so large that the nozzle hardware can be broken down[4]. Therefore, a large amount of investigations about overexpanded nozzle are focused to study the side-loads[3,5,6]. Also the overexpanded nozzle can be very useful in scrutinizing a correlation between shock structure and thrust performance because the nozzle contains complex shock structures. For that purpose, this numerical study is focused on the overexpanded nozzle and shock structures are compared in various NPR conditions.

2 Computational Methodology

In order to simulate the overexpand nozzle flow, finite-volume CFD (Computational Fluid Dynamics) solver, Fluent[7], is employed and RANS equation and pressure-based coupled algorithm are selected. The algorithm solves a coupled system of equations comprised of the momentum equations and continuity equation. This procedure eliminates the pressure correction step that is the most expensive component of SIMPLE-like algorithms[7]. Since the equations are solved in a closely coupled manner, the rate of solution convergence significantly improves when compared to the segregated algorithm. Recently, Chen et al.[8] reported that the coupled method has been tested on several CFD benchmark cases with excellent results showing the numerical convergence which is improved and the significant reduction in computational time. To capture shocks and other irreversible phenomena, upwind scheme is used for the discretization.

For the effective analysis under the limited computer resource and time, following assumptions are applied: the flow is chemically frozen with the assumption of ideal-gas, and body forces are negligible. The time-independent transport equations of continuity (1), momentum (2), species (3), and energy (4) can be described in the below.

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\frac{\partial}{\partial x_i}(\rho u_i Y_j) = -\frac{\partial \vec{J}_i}{\partial x_i} \tag{3}$$

$$\begin{aligned}&\frac{\partial}{\partial x_i}\big(u_i(\rho E + p)\big)\\ &\quad= \frac{\partial}{\partial x_i}\left(k_{eff}\frac{\partial T}{\partial x_i} - \sum_j h_j \vec{J}_j + \left(\tau_{eff}\cdot \mu_j\right)\right)\end{aligned} \tag{4}$$

Diffusion flux (5) and shear stress (6) are represented as follows;

$$\vec{J}_i = -\rho D_{i,m}\nabla Y_i - D_{T,i}\frac{\nabla T}{T} \tag{5}$$

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij} \tag{6}$$

3 Verification of the Methodology

The nozzle flow contains rich combinations of flow interactions and flow physics. Turbulence phenomenon involving temperature fluctuation and compressibility effects is necessarily included in these combinations. As the irreversible phenomena like shocks and/or flow separation may occur inside the nozzle, the supersonic jet flow is featured with very complex structure. Therefore, it is very important to exactly predict the shock structure and flow characteristics in supersonic nozzle. Hence, assessment and validation of RANS turbulence models are conducted for the optimal analysis of supersonic converging-diverging nozzle through the comparison of computational results with experimental data. Several equations for turbulence closure such as Spalart-Allmaras, RNG k-ε, and k-ω SST are

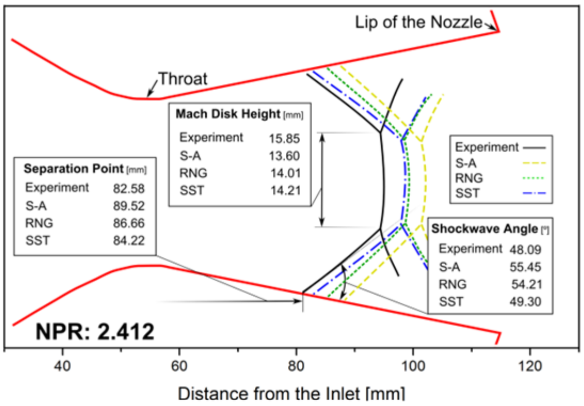

Fig. 1 Comparison of computational shock structure with experimental results at NPR=2.412[9]

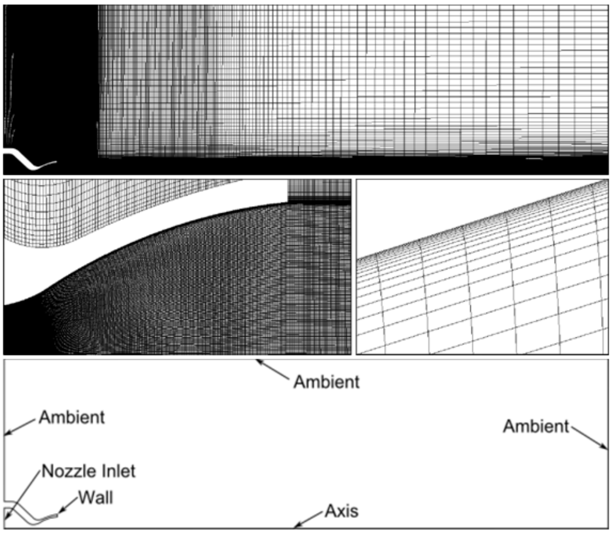

Fig. 2 Computational domain and grid configuration

employed to simulate the two- dimensional nozzle flow.

Computational results using the aforementioned turbulence models fairly well predict the shock structure as shown in Fig. 1. This figure contains detailed information including location of separation point, Mach-disc height, and shock wave angle, etc. It informs us that two-equation turbulence models predict more reasonable plume structure than the one-equation model. Especially k-ω SST turbulence model shows the best agreement to experimental results[9]. It is, therefore, expected that an analogous results to experiments is obtainable if any simulation is performed to the overexpanded supersonic nozzle with the prior-mentioned analysis method and k-ω SST turbulence model.

Fig. 2 shows computational domain, grid configuration, and boundary conditions for the

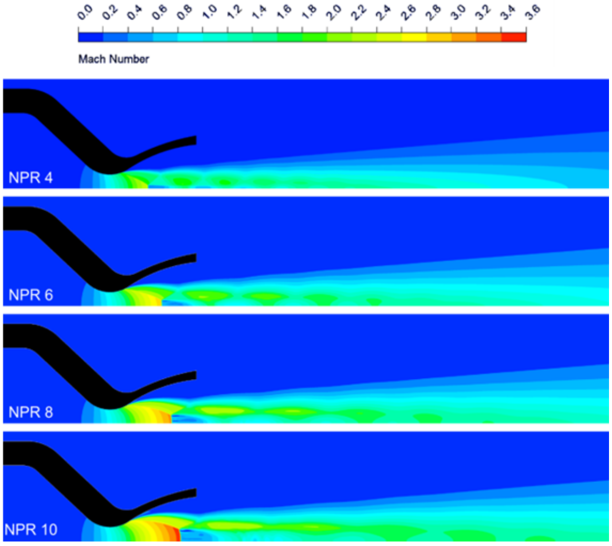

Fig. 3 Mach contours of the nozzle at various NPR's

simulation. The nozzle employed for this simulation was designed for ground-firing test of hydrazine thruster[10]. Various NPR is endowed with the nozzle inlet condition where experimental data[11] of hydrazine thruster are used to specify temperature and chemical composition. All solid walls of the nozzle were treated as no-slip adiabatic surface.

The structured grid configuration shown in the figure was made by ANSYS-ICEM CFD[12]. It consists of about 200,000 cells. In an attempt to capture complicated physics of the interaction process on shock-boundary layer, the nozzle-inside area is densely gridded with cells. The cells around the throat are also concentrated to ensure proper wall boundary-layer development and flow separation.

4 Analysis Results

Computational simulations are conducted with the NPR of 3 to 10. Figure 3 illustrates Mach contours of the nozzle flow with various NPR. Sonic flow occurs at the nozzle throat and the flow expands continuously until it faces the oblique shock in diverging sections where the flow is fully detached from the wall for all the NPR's. Region of the plume becomes enlarged as the NPR rises. Mach discs are found in the core of nozzle, which are pushed to outside by the NPR being raised.

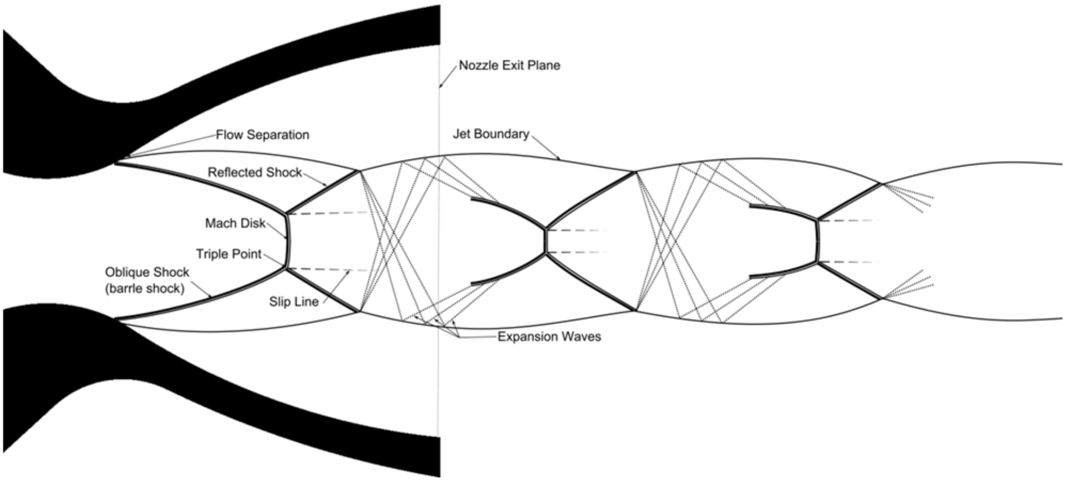

Fig. 4 Shock structure of the thruster nozzle at NPR=6

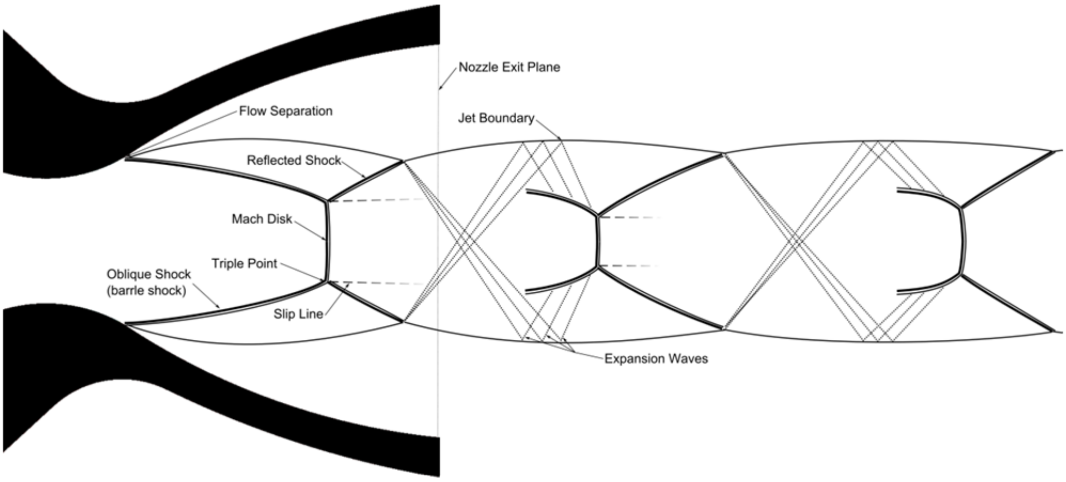

Fig. 5 Shock structure of the thruster nozzle at NPR=8

To get out the specific information regarding the nozzle plume, detailed shock structure is sketched in Fig. 4 and Fig. 5. The figures show a typical shock structure in overexpanded nozzles. Pressure of the nozzle flow decreases with its acceleration. When the flow pressure becomes approximately 40% of the ambient pressure, flow separation and oblique shock caused by an adverse pressure gradient are formed. The shock is necessary to permit decelerating supersonic flow and increasing its pressure up to the ambient condition. The oblique shock is reflected by lens-shaped Mach disc centered on the flow axis. In the case of chemical-rocket engine, the flow passing through normal shock wave is considerably heated so that any excess fuel present in the exhaust is reignited and burnt. The afterburning makes the Mach disc be bright and visible generating a ring pattern known as shock-diamonds. After the shocks, the flow is compressed too much so that its pressure exceeds the ambient pressure. As a result, the flow begins turning outward and the exhaust expands again to be equalized with the external conditions. This expansion is accomplished through a series of expansion waves being reflected off the free jet boundary. Jet pressure is decreased by the expansion fans but it becomes again too lower than the ambient pressure. As the expansion waves reach the contact discontinuity, they are reflected back again inward to form compression waves and a compression fan. These compression waves force the flow to turn back inward and increase the pressure. The compression waves merging into an oblique shock wave form a new Mach disc similar to that of the first appearance. This process repeats over and over again generating a series of Mach discs until the plume pressure is

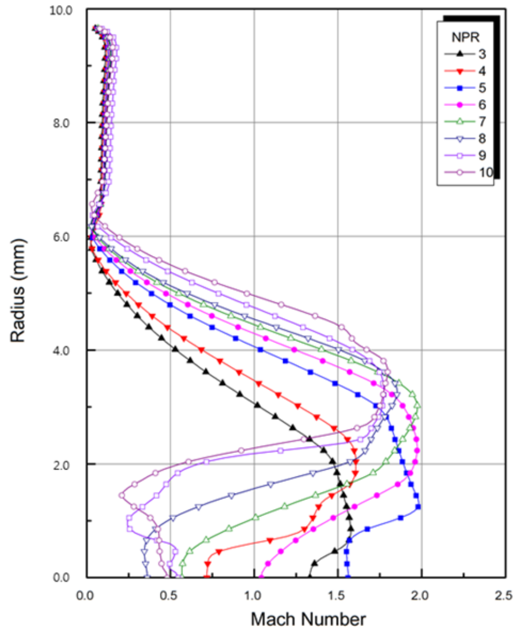

Fig. 6 Mach number distribution on the exit plane of the overexpanded nozzle at various NPR's

equalized with that of ambient. It is observed in the figures that the overall shock structure including separation point, Mach disc, oblique shock and expansion/compression waves moves downward the nozzle exit when the NPR rises from 6 to 8.

Mach number distribution on the exit plane at various NPR is shown in Fig. 6. There are insignificant flows over the outer region of jet boundary inasmuch as the main streams are fully detached at each separation point. There exist supersonic flows over the zone ranging in 2-4 mm of radius because of the flow passing through oblique shocks with its tangential momentum being maintained. In the core region up to 2 mm of radius from symmetry axis, the flows are generally at subsonic conditions for higher NPR's and at supersonic ones for low NPR's. The dependency of flow velocity on the NPR is resulted from the shock structure involving the location of Mach discs, reflected shocks, and expansion waves. For example, when the expansion waves are formed in front of the nozzle exit plane, the flow just on the exit plane is supersonic. It is also found that plume diameter on the nozzle exit is gradually recovered as NPR rises.

It has been presented that Mach number distribution in the overexpanded nozzle is changed by the NPR. The Mach number on nozzle exit plane significantly affects the nozzle performance. Fig. 7 illustrates thrust performance of the nozzle according to the variation of NPR. The thrust monotonically increases according to the growth of NPR, as expected.

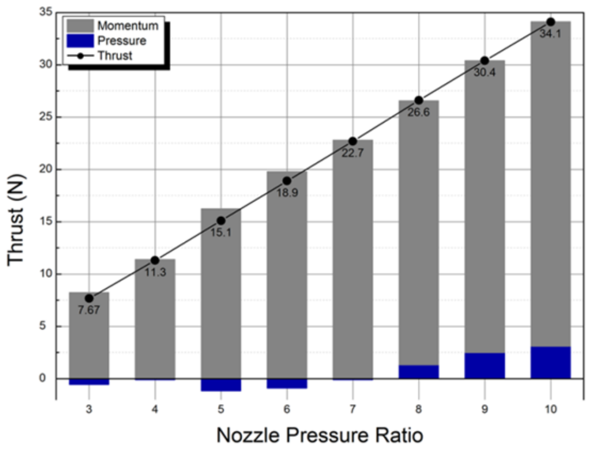

Fig. 7 Thrust performance of the nozzle according to NPR

5 Concluding Remarks

Computations for an overexpanded nozzle were conducted for various NPR's in order to investigate effects of the shock structure on thrust performance. RANS equation with k-ω SST turbulence model was selected to simulate the nozzle flow. To verify the computational methodology and turbulence model on supersonic CD nozzle, computational results of the nozzle flow were compared with the experimental data. The simulation carried out with the selected analysis method and turbulence model successfully provided analogous results to the experiment. Based upon the validation, analysis of the overexpanded nozzle that was designed for ground-firing test of hydrazine thruster has been performed. As the simulation results, detailed shock structure and shock-diamonds patterns were depicted. Finally, correlation between shock structure and thrust performance on the overexpanded nozzle were discussed thoroughly. As the NPR is raised, the thrust monotonically increases, and shock structure is pushed toward nozzle exit plane.

6 Acknowledgments

This work was supported by Advanced Research Center Program (No. 2013073861) through the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) contracted through Next Generation Space Propulsion Research Center at Seoul National University.

References

[1] Sutton G P. *History of liquid propellant rocket engines*. AIAA, 2006
[2] Nasuti F and Onofri M. Shock structure in separated nozzle flows. *Shock Waves*, Vol. 19, No. 3, pp 229-237, 2009.
[3] Nave L H and Coffey G A. Sea-level side loads in high-area ratio rocket engines. *AIAA paper* 73-1284, 1973
[4] Sutton G P and Biblarz O. *Rocket propulsion elements*. Wiley, 2010.
[5] Want T S and Guidos M. Transient three-dimensional side load analysis of a film-cooled nozzle. *Propulsion and power*, Vol. 25, No. 6, pp 1272-1280, 2009.
[6] Ruf J, Mcdaniels D M and Brown A M. *Nozzle side load testing and analysis at marshall space flight center*. AIAA paper 2009-4856, 2009.
[7] ANSYS Inc. ANSYS-Fluent User's Guide 14.5, 2012.
[8] Chen Z J and Prezekwas A J. A Coupled pressure-based computational method for incompressible/compressible flows. *Computational physics*. Vol. 299, pp 9150-9165, 2010.
[9] Kam H D and Kim J S. Assessment and validation of turbulence models for the optimal computation of supersonic nozzle flow. *Korean society of propulsion engineers*. Vol. 17, No. 1, pp 18-25, 2013.
[10] Kam H D, Kim J S, Lee J W and Kim I T. Performance analysis for the design optimization of a thruster nozzle used for ground firing test. *Asian Joint Conference on Propulsion and Power,* Xi'an, China, 2012-143, 2012.
[11] Lee S N, Baek S W, Kim S K and Yu M J. Analysis of combustor and nozzle for monopropellant satellite thruster. *Korean Society of combustion*. Vol. 15, No. 2, pp 12-18, 2010.
[12] SAS IP Inc. ANSYS-ICEM CFD 14.5 User Manual, 2012.

7 Contact Author Email Address

jeongkim@pknu.ac.kr

Copyright Statement