A Computational Study on the Unsteady Lateral Loads in a Rocket Nozzle

Suryakant Nagdewe* · Heuydong Kim**

ABSTRACT

A numerical study of the unsteady flow in an over-expanded thrust optimized contour and compressed truncated perfect rocket nozzle is carried out in present paper. These rocket nozzles are subject to flow separation in transient phase at engine start-up and/or engine shut-down. The separation flow structures at different pressure ratios are observed. The start-up process exhibits two different shock structures such as FSS (Free Shock Separation) and RSS (Restricted Shock Separation). For a range of pressure ratios, hysteresis phenomenon occurs between these two separation patterns. A three-dimension compressible Navier-Stokes solver is used for the present study. One equation Spalart-Allmaras turbulence model is selected. The computed nozzle wall pressures show a good agreement with the experimental measurements. Present results have shown that present code can be used for the analysis of the transient flows in nozzle.

Key Words: Free Shock Separation, Restricted Shock Separation, Unsteady, AUSMDV

1. Introduction

Research on the development of supersonic nozzles are going on to increase the specific impulse and thrust. Nozzles such as Thrust Optimized (TO) are used in Space Shuttle main engine (SSME) and Vulcain whereas Compressed Truncated Perfect (CTP) used in Japanese LE-7A. These nozzles shows two problems during development stage of engine. One is excessive side load and other is damage to engine equipment [1]. It is reported in literature that these problems occur because of peculiar type of flow separation during start-up and shut-down of the engine. The flow separation pattern observed in nozzles are classified into two types as shown in Fig 1 and Fig. 2. The first flow pattern is called Free Shock Separation (FSS). FSS is a normal type of separation pattern in which separated jet does not reattach to nozzle wall as shown in Fig. 1. This FSS pattern is observed in many different geometries of nozzles, and reported in many publication dating from 50s and 60s [4]. This FSS pattern appear in thrust optimized contour nozzles for low pressure ratios. The other type is called Restricted Shock Separation (RSS). RSS is observed only in TO and CTP nozzles. In this case separated

* Researcher, School of Mechanical Engineering, ANU
** Professor, School of Mechanical Engineering, ANU
E-mail : kimhd@andong.ac.kr

jet reattaches to wall just downstream of separation point, forming a closed recirculation bubble. Moreover, the separation pattern evolve from a free shock separation to a restricted shock separation when the pressure ratio increases. Transition between these two kinds of separation pattern presents hysterisis phenomena. RSS is observed in a certain range of Nozzle Pressure Ratio (NPR) in TO and CTP nozzles during start-up and shut-down transient. It has been known that the occurence of RSS cause the large side load and damage to the regenerative tubes. The large side load is caused by the transition from FSS into RSS and its inverse. The transition FSS => RSS occurs for pressure ratio value that is higher than the one observed for the RSS/FSS transition. For example, FSS occur at low pressure during startup transient. As NPR increases the separation pattern changes into RSS. During this transition, the flow field becomes highly asymmetric because of the pressure distribution on the nozzle wall occurs flow disturbance. Therefore an imbalance of observed in many TOC nozzles.

Many researches had been conducted to

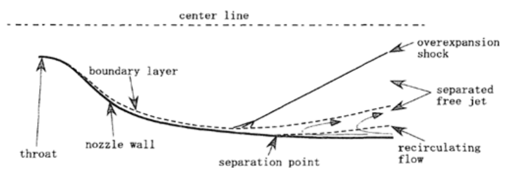


Fig. 1 Flow patternof FSS (Free Shock Separation)

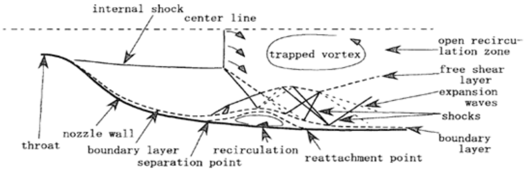


Fig. 2 Flow pattern of RSS (Restricted Shock Separation)

and results in the large side load. This phenomena, still imperfectly understood, is reveal the mechanism of the generation of the significant lateral force. However, the detailed flow structure and flow mechanism have not been understood sufficiently to enhance the reliability of rocket engines. Present study numerically Investigate the unsteady lateral side forces on the nozzle wall.

2. Numerical Simulations

Present solver is a three dimensional finite volume code written for structured multi-block meshes. Reynolds Average Navier-Stokes (RANS) equations with finite volume approach are used for the present study. Convective fluxes at the cell interfaces are calculated using AUSMDV [6] numerical scheme. Second order accuracy is achieved through MUSCL interpolation. Viscous terms are calculated using second order central difference scheme. Dual time step approach is used for marching the flow with time. The time step used in present study is 1x10^-5. Spalart-Allmaras one equation turbulence model is selected for the present investigation.

Fig. 3 shows the nozzle contour and location of pressure measurement. Wall pressure is measured at axial location A to C as shown in Fig. 3.

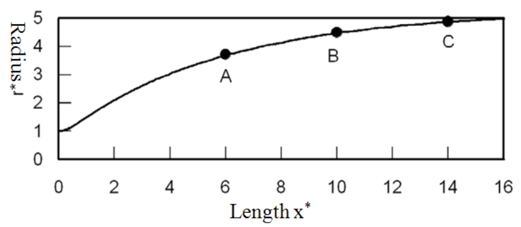
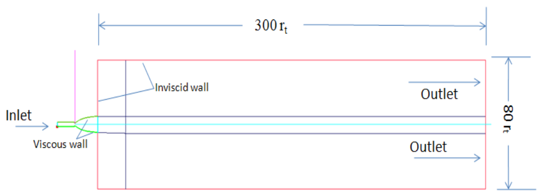


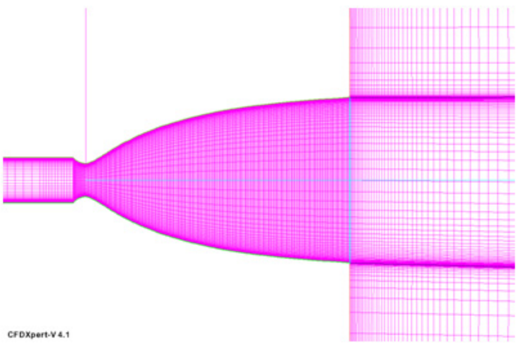
Fig. 3 Nozzle contour

Fig. 4 (a) shows schematic of the computational domain. Fig 4 (b) shows a computational grid around nozzle. The simulations are performed over a grid involving 360x100 cells inside the nozzle and 100x160 cells outside the nozzle. The two numbers correspond respectively to the stream-wise and to the normal directions.

At inlet boundary total pressure and total temperature boundary condition is specified. Nozzle walls are treated as viscous wall and other wall boundary as inviscid wall. Inlet total pressure and total temperature considered as 100kPa and 290K respectively. At the exit boundary, static pressure 1kPa was specified as a back pressure. Computational domain is initialized to a pressure of 1kPa, temperature of 290K and a velocity of zero.



(a) Geometry of computational domain



(b) Computational grid

Fig. 4 Computational domain

3. Results and Discussion

Computational domain was set to a pressure of 1 atm, a temperature of 290K and a velocity of zero as a initial condition. At inlet, 100 times the atmospheric pressure was set. Static pressure of 1 atm was given as back pressure and varied to 5 times to atm pressure.

A first validation example considers inlet pressure value of 10.2x10^5 Pa and

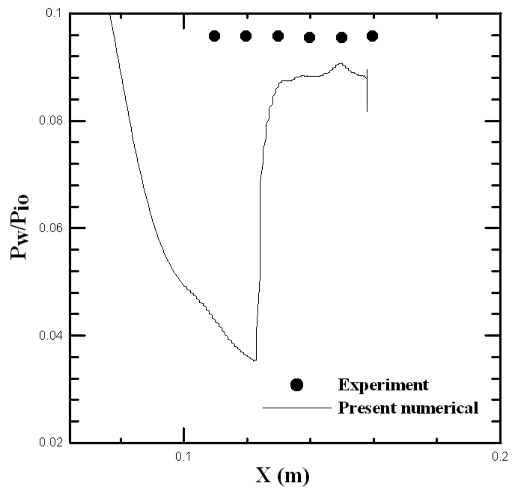


Fig. 5 Nozzle wall pressure

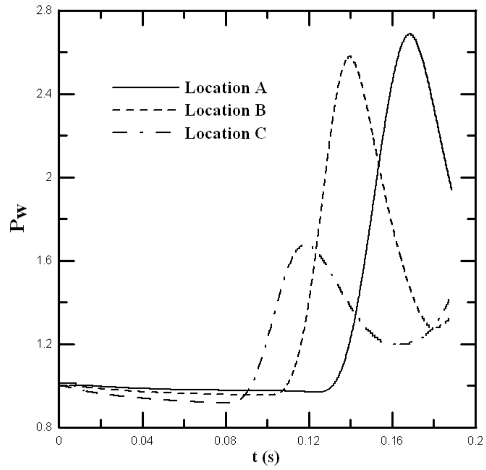


Fig 6 Pressure variation at location A, B and C

temperature 270 K. The computational domain is initialized with pressure value of 10^5 Pa, temperature 288 K and velocities set to zero. Analysis of wall pressure profiles as shown in Fig. 5 shows that wall pressure measurements and computational results are in reasonable agreement.

Second validation considers static pressure variation of 1kPa to 20kPa at outlet boundary condition in 1.0 second. Fig. 6 shows wall pressure versus NPR at the three points shown in Fig. 6. In both experimental and numerical results, peak is observed. This peak is caused by the shock waves. At point A, no pressure peaks are found in both experiment and numerical results. The pressure increases at NPR of 20 in experiment, while at NPR 18 in the numerical simulation. This shows that our simulation code predicts the location of the separation point downstream of the actual location.

4. Conclusion

Rocket engines are designed to produce axial thrust to power the vehicle. Several engine failures were attributed to non-axial static and/or dynamic forces (side load). Present work investigate the unsteady flow in an over-expanded rocket nozzle during a shutdown. The aim of present study is to investigate the lateral side load during the nozzle shutdown or starting condition using computation methodology. The present numerical computations has been carried out with density based solver on multi-block structured grid. Present study considers two nozzle pressure ratio i.e. 100 and 20. At 100 NPR, restricted shock separation (RSS) pattern is observed while, 20 NPR shows free shock separation (FSS) pattern. Side load is observed during the transition of separation pattern at different NPR.

References

1. J. Ostlund, Flow processes in rocket engine nozzles with focus on flow separation and side loads, PhD Thesis, 2002.
2. M. Frey and G. Hagemann, Restricted shock separation in rocket engines, Journal of Propulsion and Power, Vol. 16, No. 4, pp. 478-484, 2000.
3. M. Onofri and F. Nasuti, The Physical Origins of Side Loads in Rocket Nozzles, AIAA 99-2587, 1999.
4. K. Yonezawa, T. Morimoto, Y. Tsujimoto, Y. Watanabe and K. Yokota, A study of an asymmetric flow in an overexpanded rocket nozzle, Journal of Fluid Science and Technology. Vol. 2, No. 2, pp. 400-409, 2007.
5. A. Nebbache and C. Pilinski, Pulsatory phenomenon in a thrust optimized contour nozzle, Aerospace Science and Technology, Vol. 10, pp. 295-308, 2006.
6. Y. Wada and M. -S. Liou, An accurate and robust flux splitting scheme for shock and contact discontinuities, SIAM Journal on Scientific Computing, Vol. 18, No. 3, pp. 633-657, 1997.