한국기계가공학회지, 제19권, 제2호, pp.18～23(2020.02) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 19, No. 2, pp.18~23(2020.02) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2020.19.02.018

Flow Analysis near Shell Warhead

Kyekwang Choi*, Jaeung Cho**, #

*Department of Metal Mold Design Engineering, Kongju National UNIV.,
**Division of Mechanical and Automotive Engineering, Kongju National UNIV.

포탄의 탄두 주위에서의 유동해석

최계광*, 조재웅**,#

*공주대학교 금형설계공학과, **공주대학교 기계자동차공학부

(Received 3 November 2019; received in revised form 14 November 2019; accepted 24 November 2019)

ABSTRACT

The maximum speed and pressure distribution close to a warhead are altered based on the warhead shape, thereby resulting in changes to the flight distance and the destructive power. In this study, flow analysis was carried out based on the warhead shell shape. The maximum flow rate was detected at the side of shell, with a lower flow rate being found at the rear of the shell. In addition, the maximum pressure was detected at the warhead. It was also found that the reduction in the flow rate between the rear and the side of the shell in model A was smaller than that in model B. The obtained results are expected to be useful in the future design of shell warhead shape.

Key Words : Shell(포탄), Warhead(탄두), Flow Analysis(유동 해석), Flow Rate(유동속도), Pressure(압력)

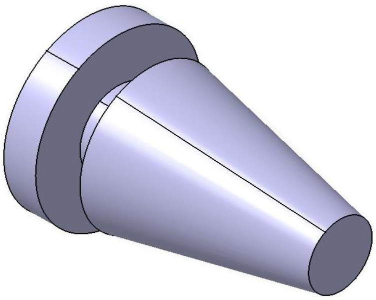
1. Introduction

On the 21'st century, there are many weapons in war. Of these weapons, many studies have been conducted on artillery. Most of the studies on artillery have been done with a lot of researches on the amount of explosive and the accurate calculation of the flight distance based on the amount of chemicals or the gun barrel. But there are few studies on the flow analysis due to the shape of the warhead of shell. The maximum speed and the pressure distribution near the warhead are changed by depending on the warhead shape. The flight distance and the destructive power can be changed due to the shape. For this reason, in this study, the maximum flow rate and the pressure distribution[1-6] near the warhead by shape of shell were investigated by carrying out the flow analysis[7-10]. The results of this study are thought to be useful at designing the warhead shape of shell.

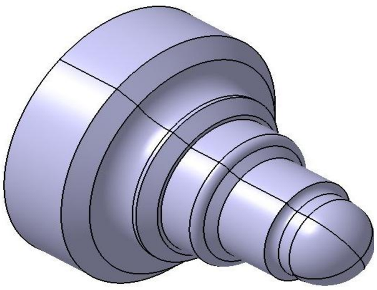
2. Configuration of Study Model

In this study, the warhead shapes of the shell in were designed by using CATIA program as shown

Corresponding Author : jucho@kongju.ac.kr
Tel: +82-41-521-9271, Fax:+82-41-555-9123



(a) Model A



(b) Model B

Fig. 1 Configuration of warhead model at shell

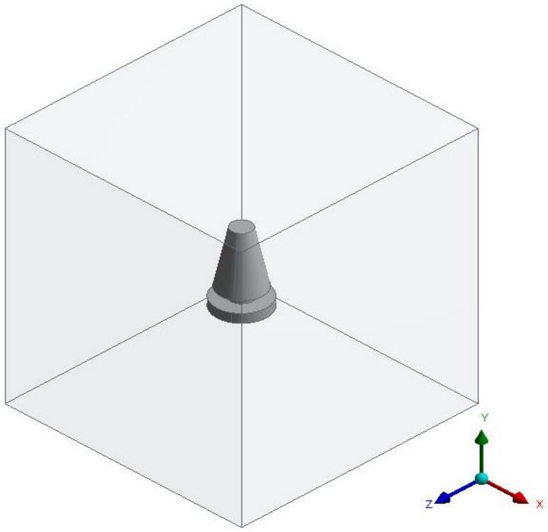
Table 1 Numbers of nodes and elements of warhead model at shell

Model	Nodes	Elements
Model A	8120	43222
Model B	34804	189906

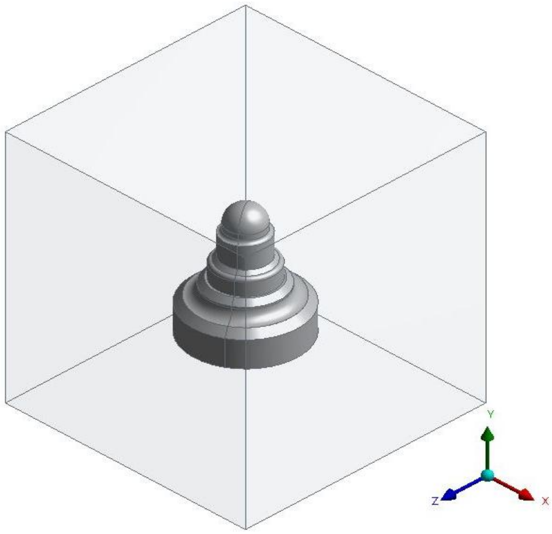
Fig. 1. Table 1 shows the numbers of nodes and elements of the shell by shape.

3. Condition of Flow Analysis

In this study, the analysis program of ANSYS was utilized in order to carry out the flow analysis due to the warhead shape of artillery shell. The flow field models of models 1 and 2 were made for the flow analysis as shown by Fig. 2. Fig. 3 shows the analysis conditions used in this study. The flow speed of 500 km/h was applied to the inlet. The outlet is applied as the pressure of 1 Pa.



(a) Model A



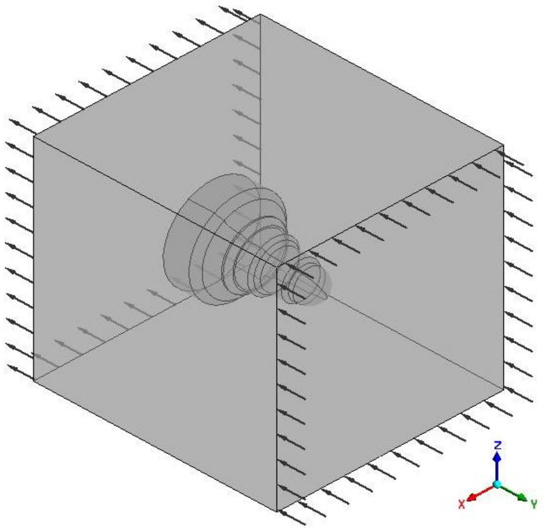
(b) Model B

Fig. 2 Flow field model due to warhead model

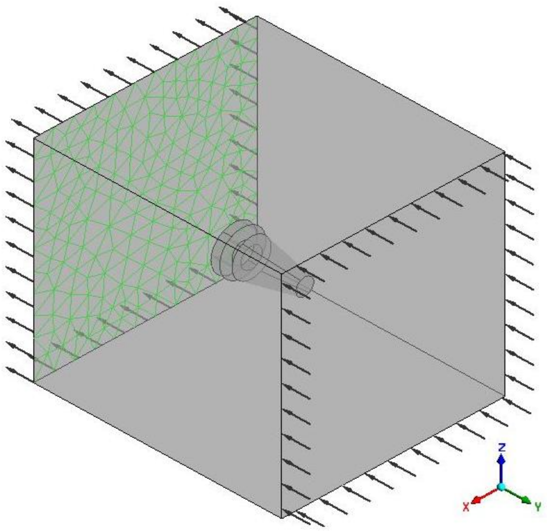
No slip condition was assigned to all other parts. In addition, the atmosphere used in the analysis had the property at 25°C as the general air.

4. Analysis Result

As shown by Fig. 4, the contours of flow rates

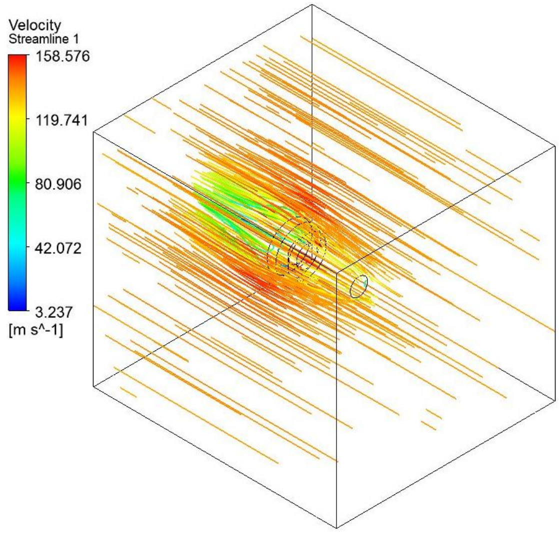


(a) Inlet condition - 500 Km/h

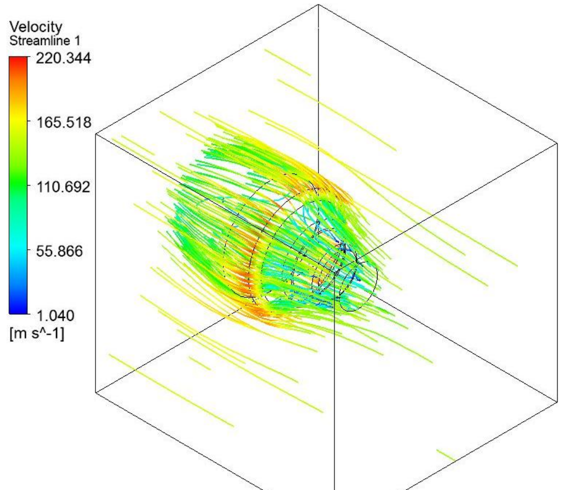


(b) Outlet condition - 1 Pa

Fig. 3 Condition of flow analysis



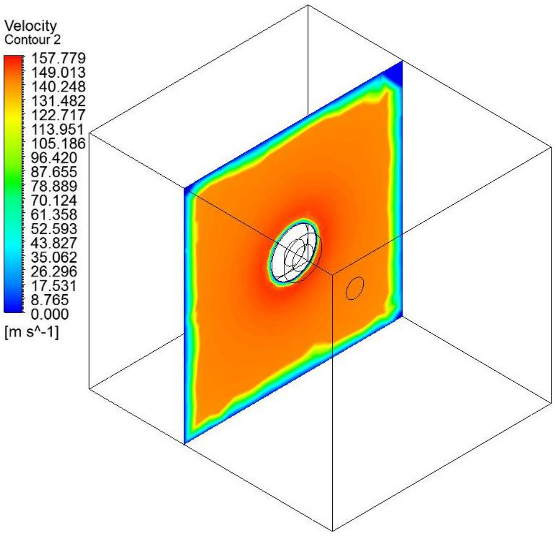
(a) Model A



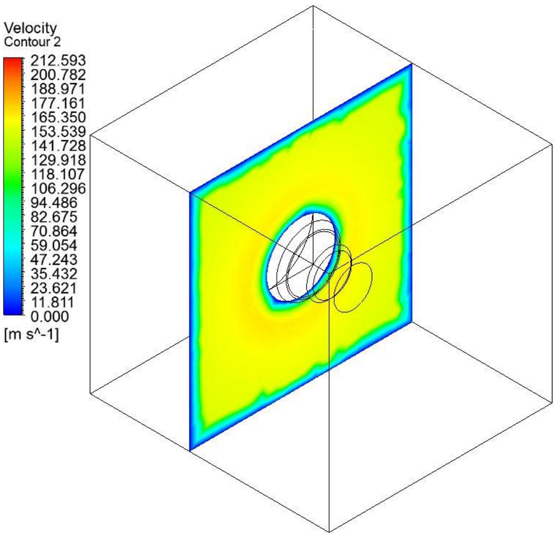
(b) Model B

Fig. 4 Contours of flow rates around the shell

by model around the shell were compared according to the warhead shape of shell. It was shown that the maximum flow rates at models A and B were shown to be 158.576 m/s and 220.344 m/s, respectively. Fig. 5 shows the contours of flow rates near the tail of shell according to shell shape. It was shown that the maximum flow rates near the tail of shell at models A and B were shown to be 157.779 m/s and 212.593 m/s, respectively. In case of two models A and B, the maximum flow rates were shown near the side of the shell. So, the flow rate near the rear of shell was decreased than the side. The reduction of flow rate between the vicinities of rear and side of shell at model A was

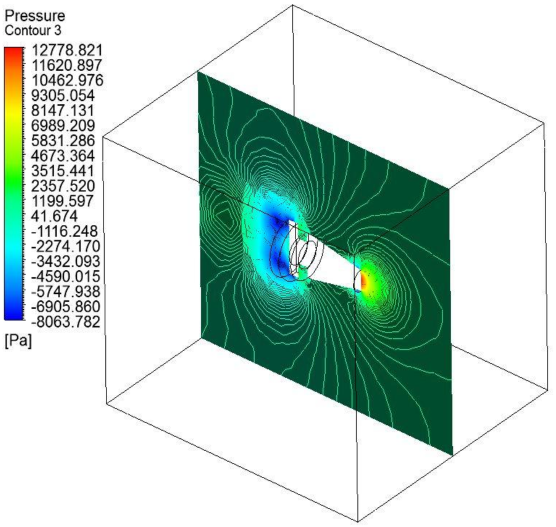


(a) Model A

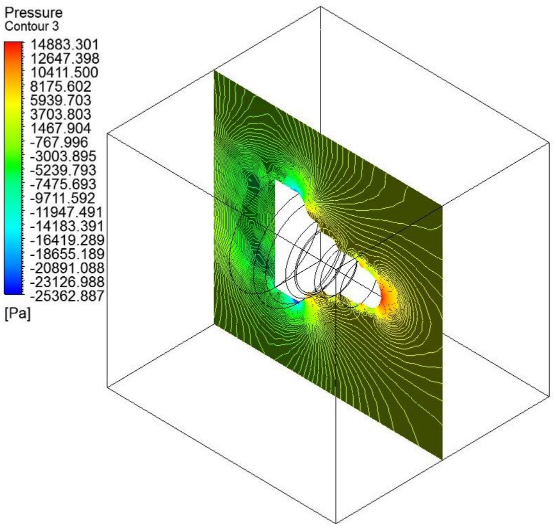


(b) Model B

Fig. 5 Contours of flow rates near the tail of shell



(a) Model A



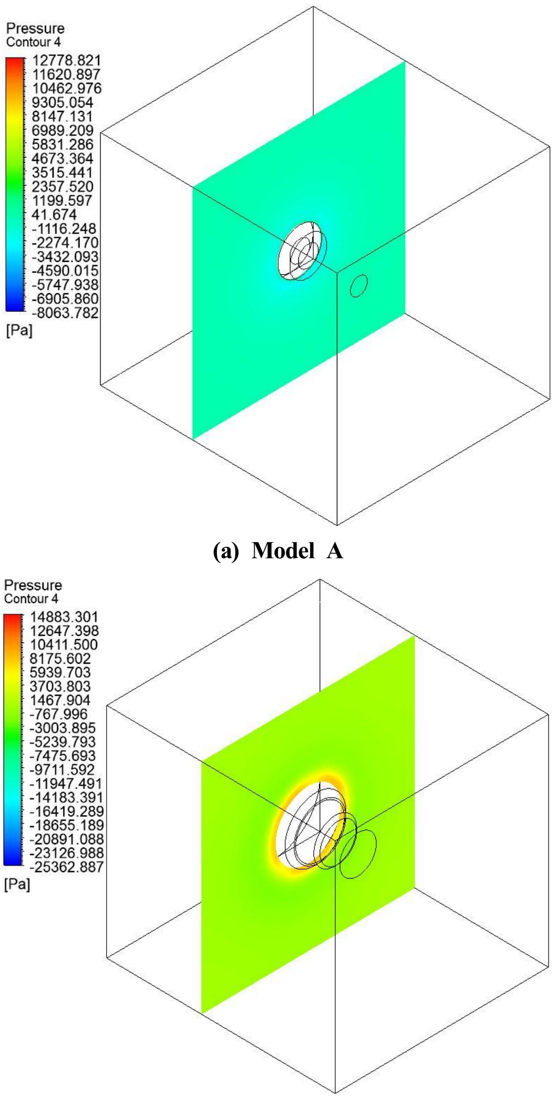
(b) Model B

Fig. 6 Contours of pressure distributions around the shell

smaller than model B.

Fig. 6 shows the contours of pressure distributions around the shell by comparing with models A and B.

It was shown that the maximum pressures at models A and B were shown to be 12778.821 Pa and 14883.301 Pa, respectively. In case of two models A and B, the maximum pressures were equally shown at the warhead of shell. Fig. 7 shows the contours of pressure distributions near the tail of shell according to the shape of shell by comparing with models A and B. It was shown that the maximum pressures near the tail of shell at models



(a) Model A

(b) Model B

Fig. 7 Contours of pressure distributions near the tail of shell

A and B were shown to be 12778.821 Pa and 14883.301 Pa, respectively. It was shown that the maximum pressure near the tail of shell at model B became higher than model A. The warhead model A of the shell is expected to maintain the flow performance better than model B.

5. Conclusion

In this study, the flow analysis by the shape shell of models A and B was carried out. The study result is summarized as follows;

1. The contours of flow rates by model around the shell were compared according to the warhead shape of shell. The contours of flow rates near the tail of shell by the shape of shell were also compared with each other. In case of two models A and B, the maximum flow rates were shown near the side of the shell. So, the flow rate near the rear of shell was decreased than the side. The reduction of flow rate between the vicinities of rear and side of shell at model A was smaller than model B.
2. The contours of pressure distributions around the shell were compared with models A and B. In case of two models A and B, the maximum pressures were equally shown at the warhead of shell. The contours of pressure distributions near the tail of shell by the shape of shell were compared with models A and B. It was shown that the maximum pressure near the tail of shell at model B became higher than model A.
3. The warhead model A of the shell is expected to maintain the flow performance better than model B. The results of this study are thought to be useful at designing the warhead shape of shell.

References

1. Lee, C. R. and Kim, B. H., “Flow Analysis of Cylindrical Helical Water Turbine for Micro Hydro-power”, Journal of the Korean Society of Mechanical Technology, Vol. 20, No. 2, pp. 187-193, 2018.
2. Kim, G. H. and Lee, D. R., “A Study on The Flow Analysis in the Turbocharger Turbine with

Different Rotational Speeds", Journal of the Korean Society of Mechanical Technology, Vol. 18, No. 5, pp. 734-739, 2016.

3. Lee, J. B., Yang, D. H., Kim, S. Y. and Kim, K. S., "A Study on Flow Analysis of the Blast Valve for Anti-aircraft Shelter", Journal of the Korean Society of Mechanical Technology, Vol. 17, No. 5, pp. 1107-1115, 2015.
4. Suk, O. B. and Cho, J. U., "A Convergence Study through Flow Analysis due to the Configuration of Automotive Air Breather", Journal of the Korea Convergence Society, Vol. 9, No. 10, pp. 265-270, 2018.
5. Cho, J. U., "A Flow Analysis on Wing Shape of Cooling Fan at Automobile", Journal of the Korea Convergence Society, Vol. 5, No. 4, pp. 75-79, 2014.
6. Lee, H. C. and Cho, J. U., "A Study on Air Flow Analysis due to the Shape of Automotive Body", Journal of the Korea Convergence Society, Vol. 5, No. 2, pp. 19-23, 2014.
7. Lee, D. H., Park, S. S., Ko, T. J. and Shim, J. S., "Effect of the Texture Shape Aspect Ratio on Friction Reduction in a Hydrodynamic Lubrication Regime," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 2, pp. 63-68, 2017.
8. Gwak, C. Y., Shin, B. S., Go, J. S., Kim, M. J., Yoo, C. J. and Yun, D. H., "A Study on the Simulation Analysis of Nozzle Length and Inner Spiral Structure of a Waterjet," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 1, pp. 118-123, 2017.
9. Jun, C. W., Sohn, J. H. and Yang, M. S., "Comparison of Fluid Modeling Methods Based on SPH and ISPH for a Buoy Design for a Wave Energy Converter," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 3, pp. 94-99, 2017.
10. Kuk, Y. H. and Choi, H. J., "Analysis of Fluid-Structure Interaction of Cleaning System of Micro Drill Bits," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 15, No. 1, pp. 8-13, 2016.