Research on fairings with different aspect ratio under subsonic flow Based On FLUENT

Keqing Hu*

Department of aerospace engineering, University Of Bristol, Bristol, BS32, UK

*Corresponding author: Dante190213@hotmail.com

Abstract. When rocket flying at high speed, large air resistance will generate at the front and around the rocket fairing. Results in significant reduction of the rocket's range and speed. The research shows that the air resistance produced by the fairing accounts for more than half of the total resistance of the rocket. As the shape of fairing is axisymmetric, a 2D model with triangle mesh was used for simulation to analyze the pressure around fairing. Eight group of fairing with aspect ratio from 0.25 to 2 were studied under the compressible non viscous airflow with 200 m/s incoming speed. The air resistance decreases with the increase of the aspect ratio, and the high pressure area around the fairing also reduces from more than a half to one third of the total fairing length. In these eight groups of fairings, the fairing with aspect ratio 2 performs best, receives the minimum air resistance and with pressure mainly distributed at the front end of the fairing.

Keywords: rocket; air resistance; axisymmetric.

1. Introduction

As an important high-tech equipment developed by a country, its technological development can often represent the industrial level and the highest technological development of the entire country. Especially in the new century, all countries are paying close attention to the development of aerospace [1-2]. The maturity of rocket technology plays an important role in the development of the aerospace industry. At present, the components of each part of the rocket must be highly coordinated and coordinated during the lift-off process to ensure that the rocket can successfully complete its mission [3]. In the early days, the air resistance was often enormous due to the speed at which the rocket was lifted into the air. Therefore, the outer fairing of the rocket often faces huge mass pressure, otherwise it will have an impact on the overall quality and working performance of the rocket [4]. When the rockets of various countries fly at high speeds, a large air resistance will be generated in front of and around the rocket fairing [5]. This results in a significant reduction in the range and speed of the rocket. Studies have shown that the air resistance created by the fairing accounts for more than half of the rocket's total drag. Therefore, optimizing the design of the fairing has direct significance for reducing air resistance and improving rocket performance [6].

This paper will analyze the external flow field and pressure distribution of the rocket fairing by means of numerical simulation, and obtain the flow field parameters outside the fairing through numerical simulation, which can provide important reference and reference for the design of the rocket fairing in the future.

2. Method

As the shape of fairing is axisymmetric, a 2D model with triangle mesh was used for better calculation efficiency as the calculation accuracy was also ensured [7-8]. The diameter of the fairing are all 20 cm, and 20 cm length was set as a standard L. From 0.25 L to 2 L, use 0.25 L as a width step, eight groups of fairings were studied. For meshing, unstructured mesh was used as it can support any shape of cells with unlimited number of cell edges, makes it can simulate natural geometric boundaries well [9]. The grid division results of the external flow field are shown in Figure 1. The length of the flow field area is 150 cm and the height is 70 cm.

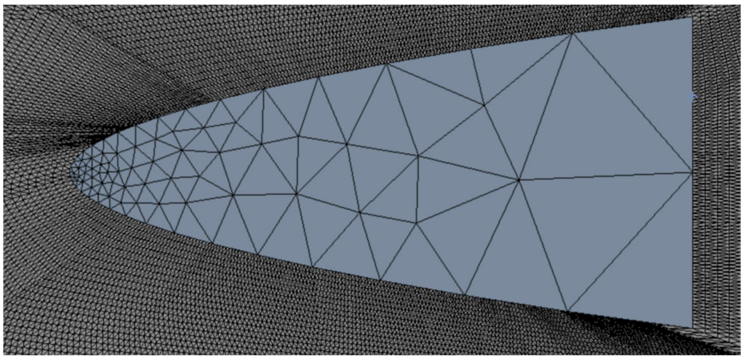
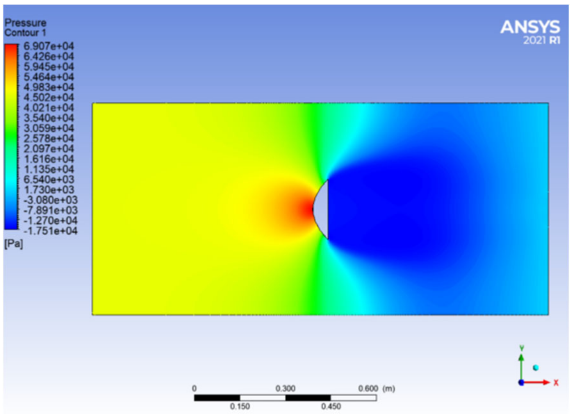


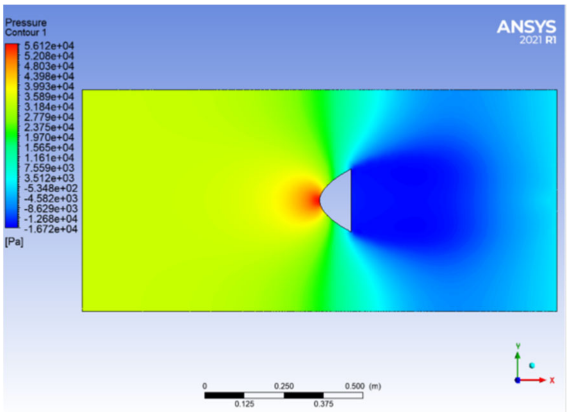
Fig. 1 Example of mesh of fairing with length 2L

Pressure based solver and coupled method were used as the flow field is subsonic. The left boundary of fluid area was named as inlet and the right boundary was named as outlet. The inflow speed from inlet was set as 200 m/s. residual error value was set to 1*10-5, K-epsilon model was used [10-11].

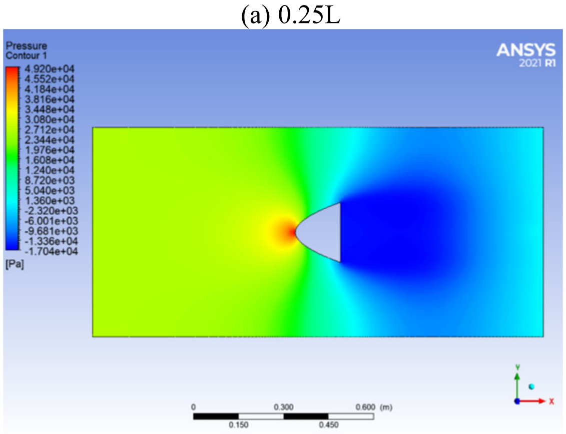
3. Results and Discussion



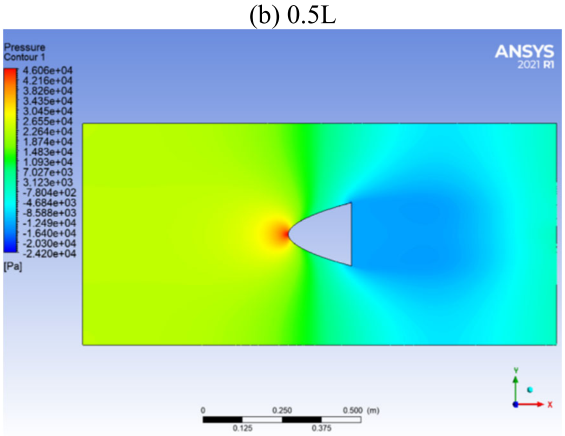
(a) 0.25L



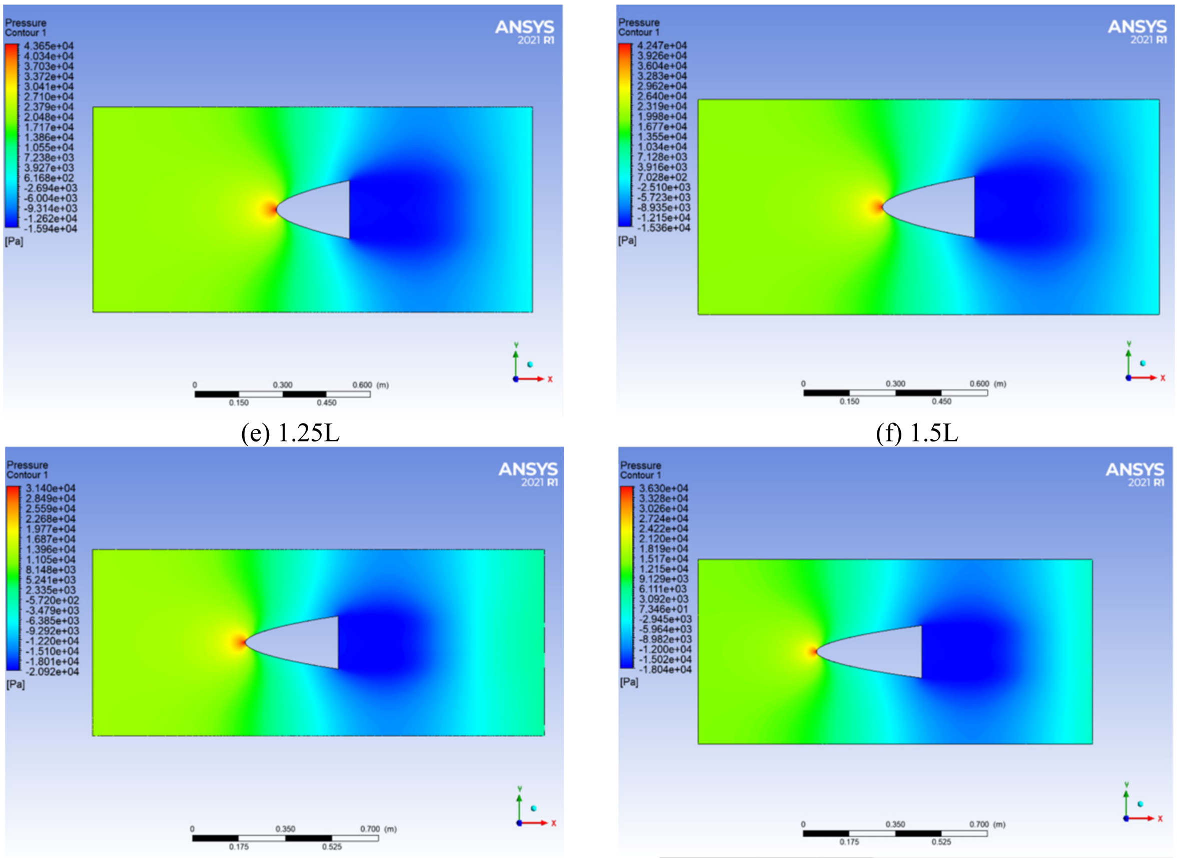
(b) 0.5L



(c) 0.75L



(d) L



(e) 1.25L (f) 1.5L

(g) 1.75L (h) 2L

Fig. 2 Pressure contour of fairing

For the eight groups of fairing studied, facing the same incoming air flow of 200 m/s speed. The air resistance decreases with the increase of the aspect ratio, from the greatest 0.25 L case, at about 69 kpa as shown in Figure 2(a), to the smallest 2 L case as shown in Figure 2(h), around 36.3 kpa. In terms of pressure, say that 16 kpa is a high pressure region for body, then for aspect ratio less than 1, the high pressure region covers half or more than a half of the fairing body, as shown in Figure 2(b). For aspect ratio between one and two however, it only affects the front one third region of the fairing as can be inspected clearly from Figure 2 (h), and for the middle part of fairings with aspect ratio more than 1, the average pressure is around 70 pa. As a result, for the eight group of fairings studied, under the premise of ideal non viscous air flow of 200 m/s, the 2 L fairing performs best, receives the minimum air resistance and with pressure mainly distributed at the front end of the fairing [12-14].

The overall results are quite accurate compares with Olivares simulation [1], this model has much more meshes and better calculation model [15]. But since this simulation only simulates the relationship between compressible air flow and pressure, without considered the effects of viscosity, heat, angle of attack and other factors, small differences are expected to occur when facing the actual flight data in reality [16].

For unguided rockets, reducing flight resistance and being able to eject at an appropriate time are the two main objectives of the fairing. For long-distance high-performance rockets, reducing resistance is even more important. Therefore, in order to meet the special requirements of rocket design, a more accurate simulation with specific environmental condition will be redone in the future.

4. Conclusion

Eight group of fairing with aspect ratio from 0.25 to 2 were studied under the compressible non viscous airflow with 200 m/s incoming speed. The air resistance decreases with the increase of the aspect ratio, and the high pressure area around the fairing also reduces from more than a half to one third of the total fairing length. In these eight groups of fairings, the 2 L fairing performs best, receives the minimum air resistance and with pressure mainly distributed at the front end of the fairing. A more accurate simulation with specific environmental condition will be redone in the future.

References

[1] Ernesto F. Subsonic Compressible Fluid Flow Analysis by the Finite Element Method. Academia Journal of Scientific Research, 2021, 5(10): 460-467.

[2] Chen Z., Guo F. Numerical simulation research on flow distribution of tube system of furnace based on Fluent-Impact of Different manifold bland structures on flow distribution. Petroleum Refinery Engineering, 2015, 67: 78-83.

[3] Hilton F. Tests of a Fairing to Reduce the Drag of a Supersonic Swept-Wing Root. Physics of Fluids, 2012, 56(3): 120-128.

[4] Zhou Y., Yang S., Luo B. Research on the Pressure-flow Performance of a Large-diameter Spool Valve Based on Fluent. Machine Tool & Hydraulics, 2011, 78(8): 79-83.

[5] Bai X., Tang K., Hui L. I. Research on Three-Dimensional Flow Field Numerical Simulation and Its Experiment Based on FLUENT. Mechanical Engineer, 2012, 67: 230-239.

[6] Peng X. I., Xiong Y., Ji-Jun P. U. Mesh comparison on E799A cavitation performance based on Fluent. Ship Science and Technology, 2019, 89(8): 119-124.

[7] Vishnu B., Dileep M., Prahlad K. A Study On Recent Trends In High Subsonic Flow Over Delta Wings. Journal of Engineering Research and Applications, 2014, 4(1): 122-127.

[8] Sven S., Christian J. Investigation of the Base Flow of a Generic Space Launcher with Dual-Bell Nozzle. CEAS Space Journal, 2021, 13(1): 197-216.

[9] Zhang R., Liu L., Zhu Y. Research on the Flow-pressure Performance of a Metal-rubber Micro-flow Valve Based on Fluent. Machine Tool & Hydraulics, 2013, 78: 115-119.

[10] Sergevich A., Prasad P. The new era of robotics: Devices and systems. International engineering journal for research & development, 2018, 3(2): 78-82.

[11] Yang G., Li Z. Simulation and Research on Internal Flow Fields of Ball Valve Based on Fluent Software. Mechanical Science and Technology for Aerospace Engineering, 2014, 78: 118-123.

[12] Zhao Y. Research on Flow Field Features of Regulator Tank in Flow Standard Device Based on Fluent. Electronic Science and Technology, 2018, 56(8): 304-319.

[13] Wei J., Jin F. W., Huang C. S. Simulation Research on a Coal-fired Burner Based on Fluent Software. Journal of Sanming University, 2017, 67: 102-108.

[14] Yan-Ping H. U., Deng Z. Research on the Pressure-flow Performance of Solenoid Valve Based on Fluent[J]. Hydraulics Pneumatics & Seals, 2016, 56(7): 341-348.

[15] Yu H. Research on Static Characteristics of Radial Aerostatic Bearings Based on FLUENT[J]. Lubrication Engineering, 2009.

[16] Liu H. F., Zhang H. S., Ye Y. Research on the Nozzle of the Licorice Cleaning Equipment Based on FLUENT[J]. Hydraulics Pneumatics & Seals, 2016, 45(7): 78-82.