Paper ID: 43071
student

IAF SPACE PROPULSION SYMPOSIUM (C4)
Interactive Presentations - IAF SPACE PROPULSION SYMPOSIUM (IP)

Author: Mr. Jaecheong Lee
Chungnam National University, Korea, Republic of, 123wocjd@naver.com

Mr. Jongin Choi
Chungnam National University, Korea, Republic of, jic316@daum.net
Mr. Yuseok Lee
Chungnam National University, Korea, Republic of, aimatgo@naver.com
Prof. Hwanil Huh
Chungnam National University, Korea, Republic of, hwanil@cnu.ac.kr

EFFECT OF NOZZLE GEOMETRY ON COUNTERFLOW JETS FOR DRAG REDUCTION OF A HIGH SPEED VEHICLE

Abstract

Several researches are being conducted to improve the performance of a high speed vehicle. The Shock waves in the front part of the vehicle directly affect the drag force, and studies are also being conducted to reduce drag. The counterflow jets can reduce the drag by injecting the jets at the stagnation point of the nose cone to deform the structure of the shock waves. In the case of a blunt body, a bow shock wave moves to opposite direction of freestream by injecting the jets at the stagnation point. The position of the shock wave is changed by the jets, and a recirculation zone is formed between the shock wave and the body to reduce the pressure on the surface of the body. As the pressure distribution acting on the surface becomes smaller, the drag acting on the body is reduced. The shock waves originate on the nose cone of the vehicle, which is shaped differently depending on the shape of the nose cone and the condition of free stream. The pressure ratio of the free stream and jets is the main variable that distinguishes the structure of the flow field. When the jet pressure is higher than the specific jet pressure range, there is a critical pressure in which the flow changes from unsteady flow to steady flow. From the importance of nozzles among the components of counterflow jets, we are analyzing the characteristics according to the shape of the nozzle. The objective of this research is to select the exit area and Mach number of the injection nozzle as the main variables by means of the flow visualization and quantitative measurement. The shock stand-off distance is the main variable to determine the amount of drag reduction. The flow visualization can deduce the shock stand-off distance. Experimental study on counterflow jets using air in supersonic condition with compressed air was carried out. The fundamental experiments were performed both flow visualization to observe the change of the flow structure according to the jet injection pressure and measurement experiment to quantitatively observe the drag reduction effect. Future plans will include additional nozzles to analyze the nozzle exit diameter and Mach number. Through the above process, the characteristics of the supersonic nozzle and the sonic nozzle can be compared.