Plume Interference Effects on the Missile with a Simplified Afterbody at Transonic/Supersonic Speeds

H. S. Kim, H. D. Kim
School of Mechanical Engineering, Andong National. University, Korea

Y. K. Lee
School of Aeronautical Engineering, Queen's University of Belfast, United Kingdom

The powered missiles with very high thrust level can make highly underexpanded jet plume downstream of the exhaust nozzle exit so that strong interactions between the exhaust plume and a free stream occur around the body at transonic or supersonic speeds. The interactions result in extremely complicated flow phenomena, which consist of plume-induced boundary layer separation, strong shear layers, various shock waves, and interactions among these. The flow characteristics are inherent nonlinear and severe unstable during the flight at its normal speed as well as taking-off and landing. Eventually, the induced boundary layer separation and pitching and yawing moments by the interactions cause undesirable effects on the static stability and control of a missile.

There have been many theoretical and experimental studies to date to understand these complex flow physics. However, there is still a lack of complete understanding of the physics on the plume-afterbody interaction control techniques to alleviate the problem associated with. Experiments in a wind tunnel to understand the problem are expensive and also suffer from support interference effects. In the present study, a CFD analysis was performed to obtain a quality understanding of the flow physics. The aims of the current research are fully to depict the flow characteristics around missile bodies and clearly to understand the plume-interference effects on them with CFD methods, and to capture minor phenomena hardly to be simulated by theoretical and experimental methods for the effective tactical missile design at transonic/supersonic speeds..

The computational model used in the current computation can be basically represented as an ogive forebody and straight afterbody without tail pins. A convergent-divergent nozzle with the design Mach number of 2.07 was used to initiate a supersonic plume expansion. Two-equation turbulence model, RNG k-ε, which is modified to take account for compressibility effect, is employed to close the mass-averaged Navier-Stokes equations. The second order upwind scheme was selected for making it feasible to capture the shock structure near the forebody and the wake flow downstream of the afterbody of the missile model.

In the computation, the plume pressure ratio and free stream Mach number were changed to provide various characteristics of plume-free stream interactions. Both factors were changed in the range of 20~400 and 0.9~3.0 respectively, as following the interests in moderately or highly underexpanded plumes imbedded in transonic and supersonic flows.

The results showed the severe influence of the free stream Mach number on the flow field around the missile model. The increase of free stream Mach number led to the reduced expansion of the plume and the stronger compression waves upstream of the plume. It also resulted in the downstream movement of the plume-induced shock in the supersonic flow regime, where the shock wave occurred. The shock-induced boundary layer separation on the missile afterbody was generated at high plume pressure ratios for Mach numbers > 1.2.

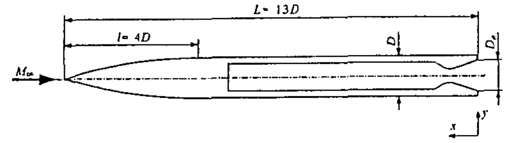
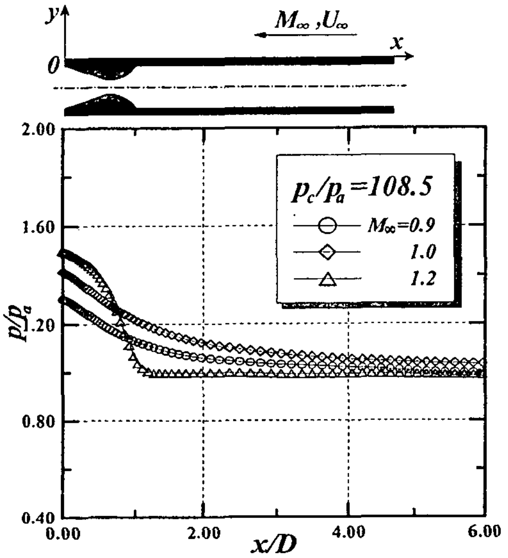
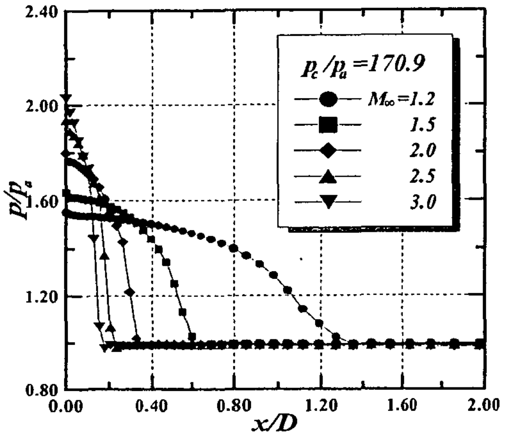


Fig.1 Simplified missile model



(a) Transonic speeds, p_c/p_a=108.5



(b) Supersonic speeds, p_c/p_a=170.9

Fig.2 Wall static pressure distributions along the afterbody surface

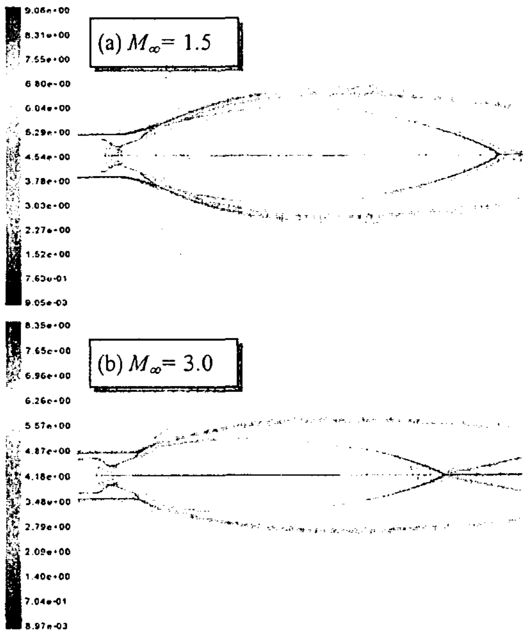


Fig 3. Mach number contours around the plume for p_c/p_a = 170.9 at supersonic speeds.

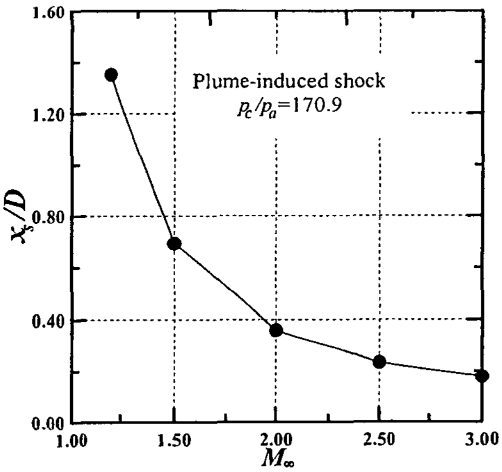


Fig.4 Movement of the plume-induced shock for various Mach numbers at p_c/p_a=170.9