Drag optimization for axisymmetric afterbodies with jet plume using CFD

*Masoud Mirzaei and Sina Arabi

KNT University of Technology, Tehran, Iran

As Supersonic aircraft and missiles reach higher velocities and required greater performance, the procedures used for their design must be evaluated. New performance requirements often dictate new designs, and previous method of analyzing aerodynamics may need to be improved. A major constraint on the performance of an aerodynamic body is drag. A supersonic body will have major drag contribution from pressure drag, skin friction drag, and base drag (in some conditions that the vehicle is not in powered flight phase). Obtaining valid predictions for these drag components, and thus having valid tools for design purposes is difficult at best. The recent advances in computational capabilities, namely increased computer memory size and processing speed, as well as improved numerical methods, have enabled attempts at solving the Navier-Stokes equations for drag prediction.

A logical next step is to begin using these computations as a practical preliminary design tool, especially as a method for optimizing geometries for minimizing drag. This paper considers the numerical solution of the Navier-Stokes equations for compressible, turbulent flow as applied to an axisymmetric body with a boat tailed afterbody with a jet plume.

The solver is a finite volume code working in unstructured grid based on kinetic flux vector splitting method for inviscid flux approximation and using a two equation $K - \epsilon$ model for turbulence modelling. A four step Runge-Kutta method has been adopted for time integration.

The boat tail angle is then optimized to determine the geometry for minimum total drag. The jet plume is considered to have a constant mass flux for various boat tail angles. When the angle is increased, the exit area of the jet is decreased. This causes a smaller pressure on the exit area and consequently lower pressure force on the area. On the other hand, increasing the boat tail angle causes an increase in the tail surface and a decrease in the pressure, and may lead to an increase in the pressure force on the boat tail section. So there is an optimized boat tail angle at which the drag force (contribution of pressure force at exit area and boat tail section) is a minimum. In this paper the main goal is to find this optimized angle using computational fluid dynamics and an optimization method by the name of golden section method. The method requires a prior knowledge that the objective function is unimodal (has a single extremum) within the constrained section of the design space. The biggest advantage of the method is that it does not require the use of gradient information which in the present case would require several evaluations of the objective function (drag) for each search step.