N95- 23304

Optimized Design of A Hypersonic Nozzle

by

Ramesh Krishnamurthy
Department of Mechanical Engineering
Old Dominion University
Norfolk, VA 23529

Conventional procedures for designing nozzles involve the design of an inviscid contour (using the method of characteristics) that is corrected with a displacement thickness calculated from boundary-layer theory. However, nozzles designed using this classical procedure have been shown to exhibit poor flow quality at Mach numbers characteristic of hypersonic applications. The nozzle to be designed will be a part of the NASA HYPULSE facility which is being used for hypervelocity flight research. Thus, the flow quality of the nozzle is a critical question that needs to be addressed.

Design of nozzles for hypersonic applications requires a proper assessment of the effects of the thick boundary layer on the inviscid flowfield. Since the flow field is largely supersonic, the parabolized form of the Navier-Stokes (PNS) equations can be used. The requirement of a uniform flow at the exit plane of the nozzle can be used to define an objective function as part of an optimization procedure. The design procedure used in this study involves the coupling of a nonlinear (Least-Squares) optimization algorithm with an efficient, explicit PNS solver.

The thick boundary layers growing on the walls of the nozzle limit the extent of the usable core region (region with uniform flow) for testing models (especially rectangular). In order to maximize the region of uniform flow, it was decided to have the exit plane of this nozzle to be (nearly) rectangular. Thus, an additional constraint on the nozzle shape resulted, namely the nozzle will have a shape transitioning from a circular one at the inlet to that of a rectangle at the exit. In order to provide for a smooth shape transition, the cross sectional contour of the nozzle is defined by a superellipse. The nozzle is taken to be a meter in length. The axial variations of the major and minor radii of the superellipse are governed by cubic splines. The design parameters are the coefficients of the splines associated with the local nozzle wall slopes.

Extensive calculations have been made (with a three-dimensional Euler Code) to understand the effects of various parameters such as, location of the knot points of the spline function, different ways of characterizing the uniformity of the flow in the exit plane as well as the effect of constraining the area of the nozzle to be invariant. Turbulent flow (measurements indicate that the flow at the nozzle inlet is turbulent) calculations are now being performed (with the inviscidly designed nozzle contours) to assess the flow quality. It is planned to extend these computations by coupling the optimization program with a viscous (turbulent) PNS code. This would be undertaken after modifying the existing PNS code by incorporating a wall-function algorithm to reduce the computational costs.