Supersonic Rocket Thruster Flow Predicted by Numerical Simulation

Despite efforts in the search for alternative means of energy, combustion still remains the key source. Most propulsion systems primarily use combustion for their needed thrust. Associated with these propulsion systems are the high-velocity hot exhaust gases produced as the byproducts of combustion. These exhaust products often apply uneven high temperature and pressure over the surfaces of the appended structures exposed to them. If the applied pressure and temperature exceed the design criteria of the surfaces of these structures, they will not be able to protect the underlying structures, resulting in the failure of the vehicle mission.

An understanding of the flow field associated with hot exhaust jets and the interactions of these jets with the structures in their path is critical not only from the design point of view but for the validation of the materials and manufacturing processes involved in constructing the materials from which the structures in the path of these jets are made.

The hot exhaust gases often flow at supersonic speeds, and as a result, various incident and reflected shock features are present. These shock structures induce abrupt changes in the pressure and temperature distribution that need to be considered. In addition, the jet flow creates a gaseous plume that can easily be traced from large distances.

To study the flow field associated with the supersonic gases induced by a rocket engine, its interaction with the surrounding surfaces, and its effects on the strength and durability of the materials exposed to it, NASA Glenn Research Center's Combustion Branch teamed with the Ceramics Branch to provide testing and analytical support.

The experimental work included the full range of heat flux environments that the rocket engine can produce over a flat specimen. Chamber pressures were varied from 130 to 500 psia and oxidizer-to-fuel ratios (o/f) were varied from 1.3 to 7.5.
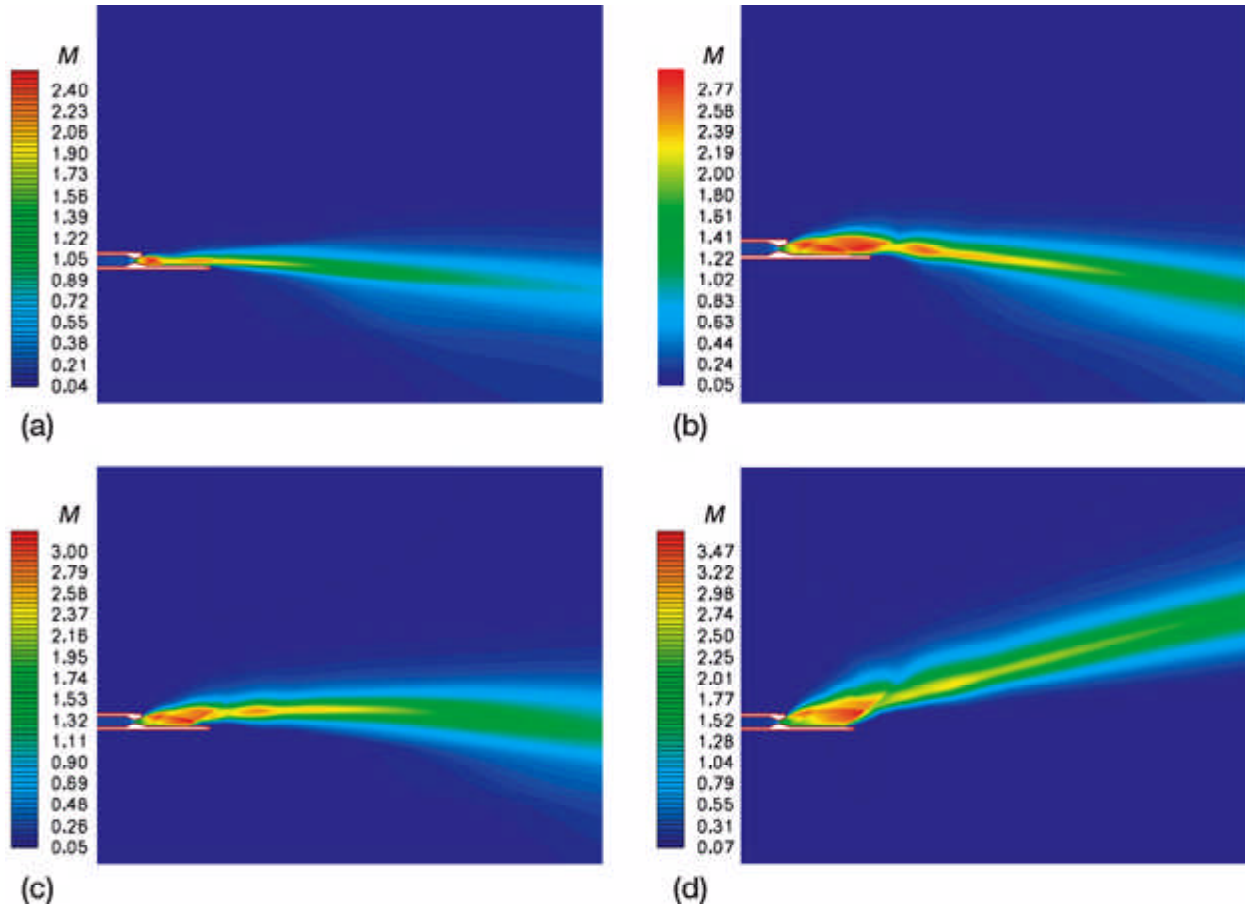
Plume prediction of a rocket thruster operating at four different combustion chamber pressures (M, Mach number). (a) 130 psia. (b) 250 psia. (c) 300 psia. (d) 500 psia.

The computational work used the National Combustion Code (NCC) to produce numerical results for several operating conditions. The accompanying figures show the computational results obtained with this code. The preceding figures show the plumes predicted by the Reynolds average Navier-Stokes numerical simulations for the supersonic flow field induced by a H_2-O_2 rocket thruster with an attached panel, under a variety of operating conditions. The plume direction was controlled by the shocks. It moved from a straight and slightly downward direction to an upward direction as the combustion chamber pressure was increased from 130 to 500 psia. The 500-psia case exhibits the highest plume angle, where the Mach number remains very high past the initial shock, aft of the inclined ramp, and over the flat panel. The following figures show the Mach number distributions, demonstrating the structure of the shocks beyond the nozzle exit. The geometry is asymmetric about the nozzle centerline; hence, the shock patterns are not symmetric. The degree of asymmetry in the shock pattern is a function of the velocity of the jet exiting the nozzle. In the 130-psia case shown in part (a), the exit velocity is not high enough to create excessive asymmetry about the centerline of the nozzle. The typical diamond-shaped shock pattern can still be recognized. As the chamber pressure and, consequently, the exit velocity increases, the asymmetry becomes more pronounced, as in part (b). Eventually, the asymmetry increases to the degree that the shock emanating from

the top edge of the nozzle exit is directed away from the panel, as in the 300-psia case shown in part (c). Finally, it is parallel to the flat panel, as in the 500-psia case shown in part (d).
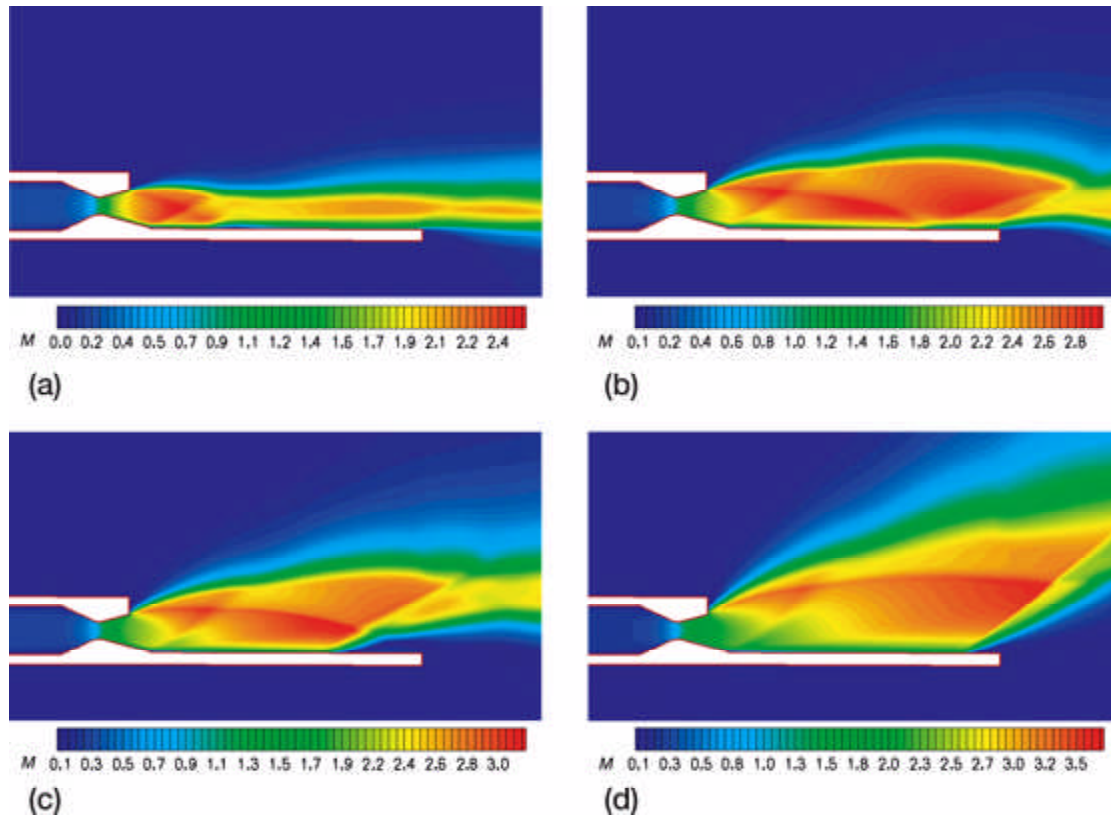
Numerical simulation of shock waves and their interaction in a supersonic rocket engine operating at different chamber pressures (M, Mach number). (a) 130 psia. (b) 250 psia. (c) 300 psia. (d) 500 psia.

The simulations have captured details of the supersonic flow field, such as the plume formations, the direction and expansion of the plumes, the formation of the shock waves, and the effects of the shock waves on the temperature and pressure distributions on the walls. A comparison showed that the results of the simulations are consistent with related measurements obtained from rig tests.

References

1. Davoudzadeh, Farhad; and Liu, Nan-Suey: Shock Waves and Plume Prediction of a Rocket Thruster With an Attached Panel. Proceedings of FEDSM'03 4th ASME-JSME Joint Fluids Engineering Conference, FEDSM2003-45077, 2003.
2. Davoudzadeh, Farhad; and Liu, Nan-Suey: Simulation of the Flow Field Associated With a Rocket Thruster Having an Attached Panel. NASA/TM--2003-212347, 2003. http://gltrs.grc.nasa.gov/cgi-bin/GLTRS/browse.pl?2003/TM-2003-212347.html
3. Liu, Nan-Suey: On the Comprehensive Modeling and Simulation of Combustion Systems. AIAA Paper 2001-0805, 2001.

University of Toledo contact: Dr. Farhad Davoudzadeh, 216-433-8876, Farhad.Davoudzadeh@grc.nasa.gov
Glenn contact: Dr. Nan-Suey Liu, 216-433-8722, Nan-Suey.Liu-1@nasa.gov
Author: Dr. Farhad Davoudzadeh
Headquarters program office: OAT
Programs/Projects: PR&T, RLV