Flow Separation in Out-of-Round Nozzles, a Numerical and Experimental Study

Chloé Génin and Ralf Stark.
German Aerospace Center, DLR, Institute for Space Propulsion
Lampoldshausen, Germany, D-74239

Sebastian Jack
German Aerospace Center, DLR, Institute of Aerodynamics and Flow Technology
Braunschweig, Germany, D-38108

and

Sebastian Karl
German Aerospace Center, DLR, Institute of Aerodynamics and Flow Technology
Göttingen, Germany, D-37083

Abstract
The start-up and shut-down transient phases in supersonic rocket engine nozzles generate high side loads. The Eigen modes of the nozzle structure can excited internal and external flow fluctuations. The flow separation in out-of-round nozzle intensifies the initial structural deformation, leading in worst case, to the collapse of the nozzle. In the framework of the DLR internal cooperation program ProTAU, the ovalization of nozzles is being investigated from both the numerical and the experimental point of view. Three nozzle geometries have been designed and deformed applying various ovalization methods. The flow behavior in the initial axisymmetric and the obtained out-of-round contours has been numerically investigated. The most promising contours will be manufactured as stiff out-of-round nozzle models and tested under cold flow conditions.

The separated flow in ovalized supersonic nozzles has been the focus of various numerical investigations. The results of these studies are partly contradictorily concerning the flow behavior and the structural response. The flow-structure-interaction is very difficult to measure experimentally. The focus of the present study is to understand and validate the simulation of the flow behavior in an out-of-round nozzle. In this first step, the structural response is not taken into account.

Various deformation approaches have been applied to three truncated ideal contours featuring different design Mach numbers. The amplitudes of deformation were set between 5% and 15% related to the nozzle exit diameter, which corresponds to an exaggeration of the expected actual deformation but enable extensive experimental measurements. Begin of the deformation can be set at different points of the contour: either at the nozzle throat or at the interface to the thrust chamber, corresponding to an area ratio of 5. The function defining the deformation evolution in the axial direction was also investigated.
The flow in the initial and the out-of-round contours were both investigated with TAU, the DLR in-house Navier-Stockes solver. The TAU code is a second order finite-volume flow solver for the Euler and Navier-Stokes equations in integral form. The position of the separation point as well as the shape of the shock system were of particular interest.

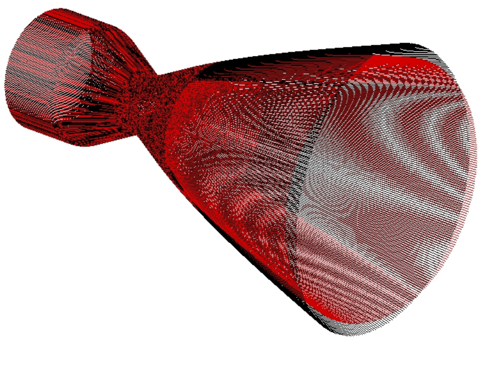

Fig.1. An example of contour ovalization in a TIC nozzle

The initial nozzles were manufactures and tested under cold flow conditions at DLR P6.2 test bench. Wall pressure measurements and schlieren imaging of the nozzle jet deliver information for comparison with the simulation. The validated numerical method is applied to study seven out-of-round nozzles. The most promising deformation approaches will be applied for the manufacturing and testing of three stiff deformed nozzle models.