Jet Simulation Facility of the Ludwieg Tube Blow Down Type

S. Stephan, R. Radespiel* and R. Müller-Eigner***
** Technische Universität Braunschweig, Institute of Fluid Mechanics*
Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany
*** HST GmbH, Max-Planck-Str. 19, 37191 Katlenburg-Lindau, Germany*

Abstract

Afterbody flow phenomena represent a major source of uncertainties in the design of a launcher. Therefore there is a demand for measuring such flows in a wind tunnel. The interest in jet simulation is motivated by the need for experimental data in the area of jet-afterbody interactions. During atmospheric rocket flight there are several ways of aerodynamic and aero-structural interactions where the propulsive jet is involved.

As an advanced approach to propulsive jet simulation a new jet facility was integrated into the hypersonic Ludwieg tube Braunschweig (HLB). The jet simulation facility resembles a single large scale rocket motor integrated into a heavy space launcher. In this paper the new jet simulation facility for cost-effective flow research is presented. The design approach is that characteristic properties such as the velocity ratio can be adjusted to match real flight conditions. The jet simulation facility is again of the Ludwieg tube blow down type, as sketched in figure 1. The jet simulation Ludwieg tube is designed in a tandem-nozzle set up so that good flow quality can be obtained at low operational cost and with a relatively small storage tube diameter. In the present case the storage tube used for jet simulation can be heated up to 900 K and gas of low molar mass can be employed to achieve high jet velocities. With this approach it is possible to systematically vary the major rocket plume flow properties, e.g. the flow displacement by plume shape and the entrainment by turbulent mixing. The paper will discuss the potentials and the limits of this approach.

For qualifying the jet simulation facility a series of measurements with varying operation parameters was conducted. The pressure in the settling chamber and the pitot pressure at the nozzle exit were measured for different storage tube pressures and temperatures. In figure 2 the Mach number distribution calculated from measured pitot data with the Rayleigh pitot formula is shown as an example. Also the Mach number obtained from CFD computations of the same nozzle is included. The

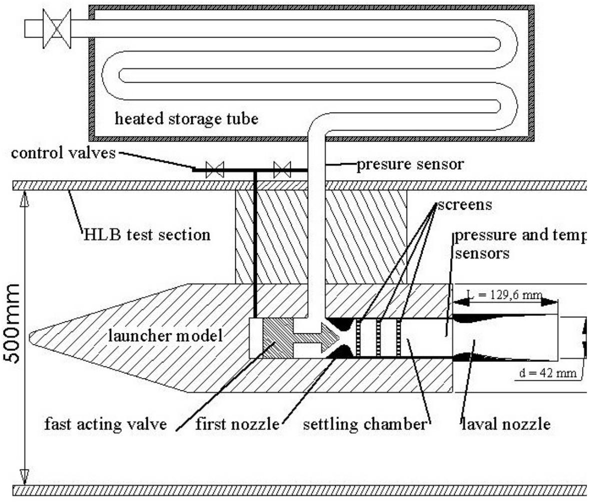

Figure 1 Sketch of the jet simulation facility

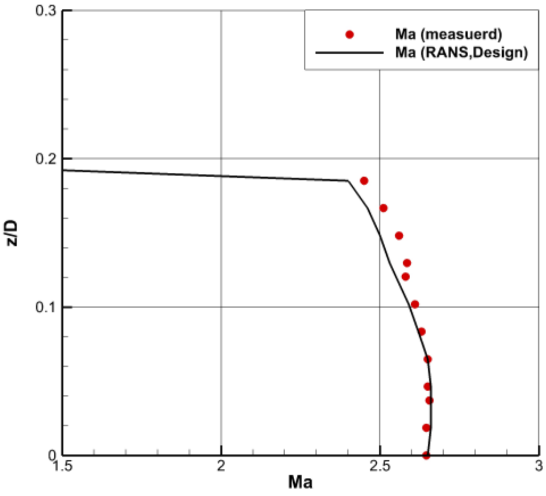

Figure 2 Mach number distribution along the nozzle exit

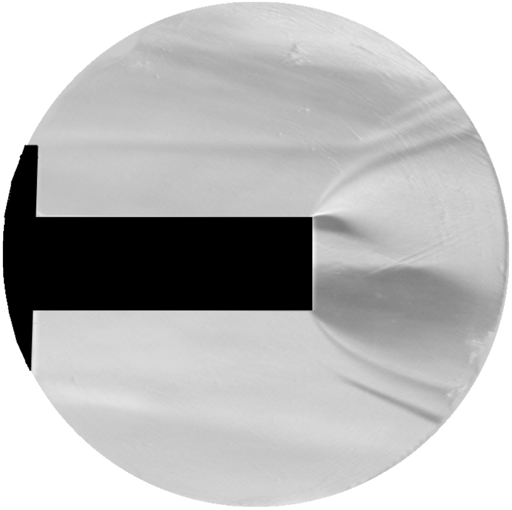

Figure 3 Averaged Schlieren picture with jet ($p_{t,jet}$ = 16.13 bar, T_{jet} = 470 K, $Ma_{e,jet}$ = 2.5) and surrounding flow ($p_{t,\infty}$ = 16.79 bar, T_{∞} = 470 K, Ma_{∞} = 5.9)

figure shows a good agreement between the computed and the measured Mach number distribution. Measurements of the pitot pressure distribution of the plume flow structure at various axial positions are also conducted. Figure 3 shows a Schlieren picture obtained by averaging 30 individual images. It shows a displaced recompressions shock caused by the underexpanded jet. Also the expansion fan, the plume barrel shock and the jet mixing region are displayed.

The full paper will present and discuss the results of these measurements, where nozzle exit pressure and plume total temperature are systematically varied. Moreover, effect of plume flow parameters on the unsteady base pressure and the unsteady pressure on the surface of the nozzle measured with Kulites (XCS-093/ LE-062) sensors will be presented.