

PROPULSIVE JET SIMULATION IN A HYPERSONIC LUDWIEG TUNNEL

S. Stephan and R. Radespiel.

Institute of Fluid Mechanics, Technische Universität Braunschweig,
Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany

R. Müller-Eigner

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. As a new approach to propulsive jet simulation a new jet facility was integrated into a hypersonic wind tunnel. This jet simulation facility resembles a Vulcain 2 rocket motor integrated into an Ariane 5 launcher. Details about the design and first experimental results are reported. This includes measurements of pressure, temperature and Mach number distribution.

1. INTRODUCTION

The interest in jet simulations is motivated by the need for experimental data in the area of afterbody flow phenomena. At atmospheric rocket flights there are a multitude of interactions between free stream and jet. These interactions influence the flight characteristics of rockets. While a number of investigations are reported in the literature on missiles, the literature on launchers with jet simulation is sparse. For example Peters [1], [2] researched the effect of the boattail drag for different jet simulation parameters by variation of the boattail, the nozzle exit to throat ratio, the jet temperatures and the gas composition. Kumar [3] presented investigations in boattail separated flows relevant to launch vehicle configurations. This included mean and fluctuation pressure measurements. The investigations were made at transonic speeds ranging from $Ma = 0.7 - 1.2$ and various boattail angles and diameter ratios. Investigations of the Ariane 5 European launcher afterbody at a scale of 0.01 have been conducted by Reijasse [4]. The first stage including the center engine and boosters was studied in a blow down wind tunnel at $Ma = 4$. The jet was simulated with cold high pressure air, which was expanded to resemble flight at an altitude of 30 km. As a result the complex flow field consisting of supersonic, subsonic and back flows was determined.

In this paper a new jet simulation facility for cost-effective research on turbulent afterbody flows is presented. The characteristic properties such as velocity ratio can be adjusted to match real flight conditions. The jet simulation facility in its current stage can be run in stagnant air at different counter pressures and also in hypersonic flow. The jet simulation facility is of the Ludwieg tube blow down type. Ludwieg tubes work with a fast acting valve with the benefit that a good flow quality can be obtained at low operational cost. Detailed information about Ludwieg tubes with fast acting valves are presented by Koppenwallner [5]. Usually Ludwieg tubes are cold blow-down tunnels. 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 adjust characteristic jet properties. The new jet simulation facility is integrated into

a large Ludwieg tube that generates the hypersonic flow field around the afterbody. The overall wind tunnel configuration is rather simple and it allows for cost-efficient but high-quality research typically performed by universities.

2. DESIGN APPROACH OF JET SIMULATION

For simulating rocket afterbody flows in wind tunnel facilities it is important to reproduce the major rocket plume flow parameters. A review of various techniques for simulation of jet exhaust in ground testing facilities is given by Pindzola [6]. The scaling of the rocket plume for the jet simulation facility used in the present work is based on discussions with industry in rocket propulsion [7]. The launcher to be scaled is the Ariane 5 with a Vulcain 2 rocket motor at an altitude of 50 km. The free stream conditions at flight and in the wind tunnel are shown in Tab. 1.

The easiest approach would be a geometric scaling, but for wind tunnel simulations that is very difficult approach. Rocket motors used in launchers have hot rocket plumes with temperatures up to 3500 K. This high temperature could destroy the wind tunnel models and sensors and the infrastructure cost are extremely high. Hence jet simulations in wind tunnels usually employ cold plumes. But for physics based ground simulation two major afterbody flow mechanisms are important and should be considered. The first mechanism is flow displacement by plume shape. The plume shape affects the positions of the shear layer

	HLB (air)	Ariane 5 trajectory (air)
p_{∞} [bar]	$1.3 \cdot 10^{-2}$	$7.6 \cdot 10^{-4}$
T_{∞} [K]	58	271
Ma_{∞}	5.9	5.3

Table 1. Free-stream conditions for the hypersonic wind tunnel HLB and the Ariane 5 trajectory.

	working gas air	working gas He	Vulcain 2
$p_{t,reservoir}$ [bar]	20.17	19.34	116
$T_{t,e}$ [K]	838.8	802.5	3630
p_e / p_∞	93.4	91.7	180
p_t / p_∞	1596	1530	152750
u_e / u_∞	1.07	2.64	2.4
$(u_{max} - u_\infty) / u_{max}$	0.306	0.689	0.662
T_e / T_∞	6.4	4.5	4.4
ρ_e / ρ_∞	14.5	2.64	19.3
$\rho_e u_e / \rho_\infty u_\infty$	16	7	47
$\rho_e u_{max} / \rho_\infty u_\infty$	21	9	57

Table 2. The jet simulation parameters for the facility used with air and helium and for the rocket motor Vulcain 2.

and the plume shock. It mainly depends on the ratio of nozzle exit pressure to static pressure in the free-stream. The second mechanism is flow entrainment into the plume. The entrainment describes the effect of the shear layer to entrain gas from the base flow. Entrainment results from turbulent mixing and this is associated with the large turbulent structures in the afterbody flow. Simulation of turbulent mixing is therefore needed to represent buffet flow phenomena at the rocket afterbody. The differences between wind tunnel and rocket operation plume conditions affect the similarity parameters for entrainment such as the velocity ratio. The low total temperature used in wind tunnel facilities reduces the exhaust velocity. Varying the specific heat ratio of the plume fluid affects the exit Mach number and the flow expansion. The exhaust velocity is also affected by different gas molar masses. Finally, a higher density of the jet flow will affect the shear layer growth. In conclusion, entrainment and growth of mixing layers are governed by two important parameters. The first is the shear layer driving velocity to plume velocity ratio, $(u_p - u_\infty) / u_p$. This parameter is expected to govern turbulence production in the shear layer and unsteadiness of the afterbody flow. The second parameter is the momentum ratio $(\rho_p u_p) / (\rho_\infty u_\infty)$. This parameter will affect the growth of the shear layer and its position relative to the nozzle axis. Note that the plume velocity is not equal to the nozzle exit velocity. Rather, the plume velocity is the velocity in the region between the shear layer and the barrel shock of the underexpanded jet. The velocity ratio and the momentum ratio should be varied in significant amounts by using the facility to make experiments representative. For assessing the potential of the wind tunnel set-up a simple estimate is to observe the

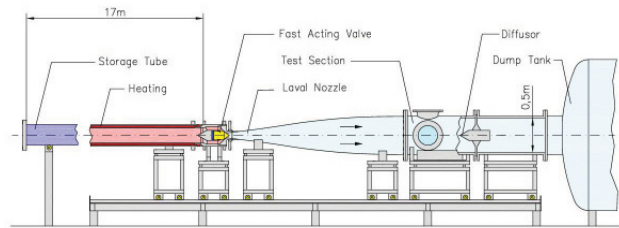


Figure 1. Sketch of the Hypersonic Ludwig Tube Braunschweig (HLB).

plume velocity for expansions to $p = 0$ ($Ma \rightarrow \infty$). In this case the plume velocity u_p is replaced by

$$u_{max} = \sqrt{\frac{2 \cdot \kappa \cdot \mathfrak{R} \cdot T_t}{\kappa - 1} M_{Mol}}$$

maximum velocity depends on the molar mass, the specific heat ratio and the total temperature. Therefore by changing the gas composition the velocity can be modified. In Tab. 2 the important ratios for jet simulation for the Vulcain 2 and for the jet simulation facility with different working gases are shown. The first, nozzle to free-stream pressure ratio p_e / p_∞ , shows the degree of underexpansion. The Vulcain 2 expansion is twice as large as in the simulation facility. The second ratio is the total nozzle to free stream pressure ratio p_t / p_∞ , this ratio is necessarily much smaller in wind tunnel facilities without high-pressure combustion. The nozzle exit velocity to free stream velocity ratio u_e / u_∞ is much too small compared to Vulcain 2 if air is used as a jet fluid. When helium is used as working gas the velocity ratios are quite similar. Also full simulation capability of the $(u_{max} - u_\infty) / u_{max}$ ratio and the nozzle exit to free stream temperature ratio is achieved. The density and the momentum ratios exhibit larger differences for both working gases as mentioned in the table. Especially for helium there are large differences, because of much lower helium molar mass. This should move the high-speed boundary of the mixing layer somewhat closer to the nozzle axis, as compared to Vulcain 2. However, the velocity ratios show good similarity and so the production of turbulence energy and level of turbulent shear stresses should be properly reproduced.

3. EXPERIMENTAL SETUP

3.1. Windtunnel HLB

Figure 1 shows the Hypersonic Ludwig Tube Braunschweig (HLB). The HLB is a cold Ludwig tube blow down tunnel with a fast acting valve. This valve separates the high pressure and the low pressure parts of the facility. The high pressure part consists of the 17 m long storage tube with a 3 m heated part. The maximum pressure in the storage tube is 30 bar. The low pressure part consist of the $Ma = 5.9$ Laval nozzle, the 0.5 m circular test section, the diffusor and the dump tank. Prior to running the facility the low pressure section is evacuated to a few mbar. The measuring time is 80 ms during which almost constant flow conditions are obtained. The unit Reynolds number range of the facility is $(3 \text{ to } 20) \cdot 10^6 \text{ 1/m}$. More

detailed information on the construction and operation of the HLB are given by Estorf [8].

3.2. Design of jet simulation facility and wind tunnel model

Figure 2 shows a sketch of the jet simulation facility. The working principle is similar to the HLB wind tunnel as this is a Ludwieg tube blow down tunnel with a fast acting valve as well. Outside of the HLB test section is the 32 m long heated storage tube. The diameter of the storage tube is 18.88 mm and it can be pressurized up to 140 bar and heated up to 900 K. The rocket model is placed in the centerline of the HLB test section. A tandem nozzle, consisting of the first nozzle, the settling chamber and the second nozzle, is integrated into the rocket model. The second nozzle is an axisymmetric Truncated Ideal Nozzle (TIC) with a mean exit Mach number, $Ma_e = 2.5$, and a $d_e = 43$ mm nozzle exit diameter designed by Saile [9]. The diameter of the settling chamber is $d_{SC} = 39$ mm. A system of three porous plates is integrated in the settling chamber to improve uniformity of the flow upstream of the second nozzle.

For scaling the Ariane 5 launcher the nozzle to body diameter ratio is used. For the Ariane 5 the nozzle diameter is $d_{Ariane} = 2.094$ m, and the body diameter is $D_{Ariane} = 5.4$ m and hence, the ratio is $d_{Ariane}/D_{Ariane} = 0.388$. The Ariane nozzle lip thickness is 2.5 mm and therefore negligible. The diameter of the cylindrical body is $D = 108$ mm while the model nozzle lip thickness is 0.5 mm. Therefore the ratio has been calculated for the inner and outer nozzle diameter. With the inner nozzle diameter $d_{inner} = 42$ mm the ratio is $d/D = 0.389$ and with the outer nozzle diameter $d_{outer} = 43$ mm the ratio is $d/D = 0.398$. Note that the nozzle length to body ratio $L/D = 1.2$ represents the Ariane 5 value as well.

After the start of the facility the flow detaches in the first nozzle and a shock system is generated. Because of this shock system the flow is decelerated to subsonic speed, at the entry of the settling chamber. In the settling chamber the flow uniformity is improved with porous plates

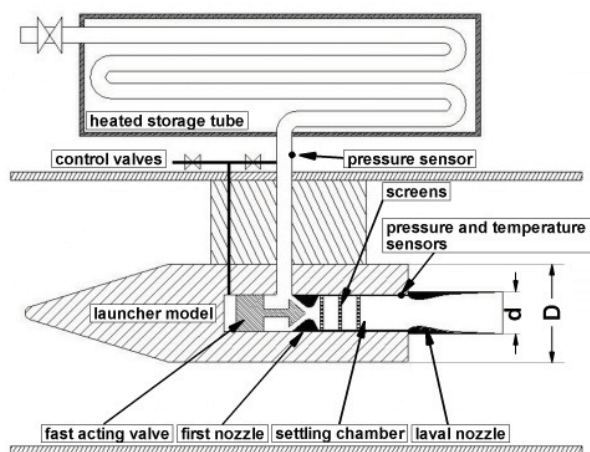


Figure 2. Sketch of the jet simulation facility (TSA). The diameters are $d = 43$ mm of the nozzle and $D = 108$ mm of the cylindrical model.

designed to reduce total pressure and to work as flow straighteners, see Fig. 2. In the second nozzle the flow is accelerated to $Ma_e = 2.5$ at the nozzle exit. In Fig. 3 the afterbody and the nozzle of the jet simulation facility mounted in the HLB are shown.

3.3. Instrumentation

The locations of the storage tube pressure sensor and the sensors in the settling chamber are shown in Fig. 2. A Gefran (Type: TKDA-N-1-Z-B16D-M-V) transducer is used for measuring the storage tube static pressure. The pressure range is between 0 bar and 160 bar and the response time for this transducer is less than 1 ms. For measuring the static pressure in the settling chamber three Kulites (XCEL-152) are used. The pressure ranges of these transducers are between 0 bar and 34.5 bar. The response time is less than 1 ms. The transducers are placed at the circumference with a 120 degree distance beginning from the top. Also in the settling chamber three temperature sensors are placed at the circumference with a 120 degree distance beginning from the bottom. K-Type thermocouples (TJC100-CASS-IM025E-65 Sensor from Omega) are presently used for measuring the settling chamber temperature. The response time of the thermocouple made with exposed 0.04 mm diameter wire is less than 20 ms. The total pressure at the nozzle exit is measured with a pitot rake consisting of 13 sensors. One sensor is located in the center of the nozzle exit. The other sensors are evenly distributed in two orthogonal sections. The distance between the sensors is 9 mm. Small and low cost transducers from Honeywell (True Stability Silicon Pressure Sensors Series Standard Accuracy), with a pressure range from 0 bar to 10 bar and 1 ms response time, are used in the pitot rake.

4. EXPERIMENTAL RESULTS

For qualifying the jet simulation facility a series of measurements with varying operation parameters was con-

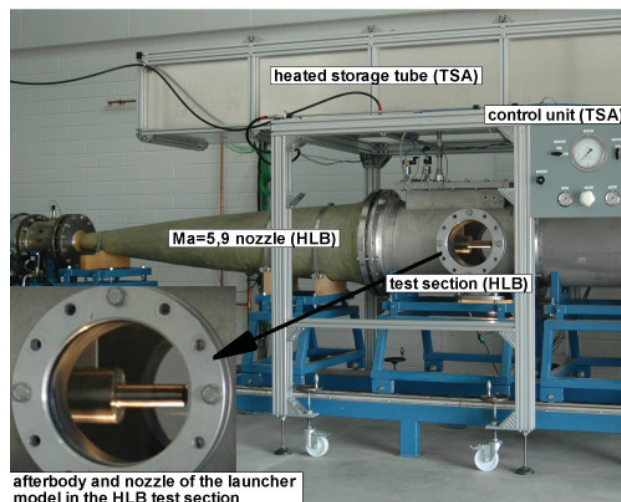


Figure 3. Jet simulation facility attached in the HLB test section (shown: nozzle and afterbody).

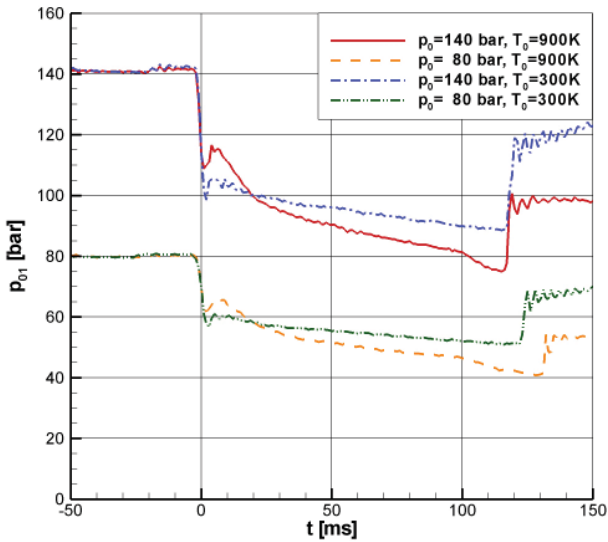


Figure 4. Static storage tube pressure for different initial storage tube pressures and different storage tube temperatures.

ducted. At least five tunnel runs were done and several time segments during each tunnel run were evaluated. The pressure and the temperature in the settling chamber and the pitot pressure at the nozzle exit were measured for two different storage tube pressures ($p_0 = 80$ bar and $p_0 = 140$ bar) and for different storage tube temperatures ($T_0 = 300$ K and $T_0 = 900$ K). Also the pitot pressure distributions in the jet plume flow with surrounding flow for two axial positions were measured. Here, the settling chamber pressure for the TSA was $p_{t,SC} = 16.13$ bar and the temperature was $T_{SC} = 470$ K and $T_{SC} = 620$ K. The flow properties of the surrounding flow are described by $p_{t,\infty} = 16.79$ bar, $T_{t,\infty} = 470$ K, $Ma_\infty = 5.9$. The working gas for the HLB and the jet simulation facility was air during these initial tests and the surrounding initial pressure in

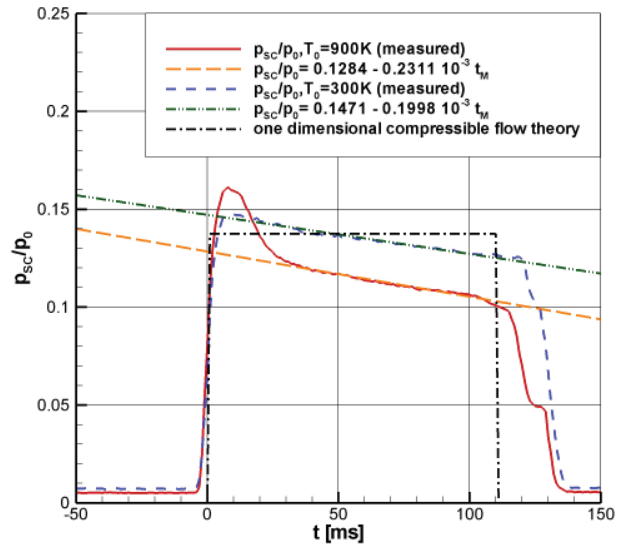


Figure 6. Settling chamber pressure to initial storage tube pressure ratio (measured, calculated and linear function over measuring time).

the HLB test section was always less than 3 mbar.

4.1. Flow Measurements in Storage Tube and Settling chamber

Figure 4 shows the static storage tube pressure for different initial storage tube pressures and different temperatures. Right after the opening of the fast acting valve ($t = 0$ ms) there is a pressure loss of 24 % for low temperatures and of 26 % for high temperatures because of the unsteady starting process. During the measuring time of 100 ms the pressure drops further by 15 %. This pressure loss is described by Koppenwallner [10]. Koppenwallner investigated Ludwig tubes with Laval nozzles of

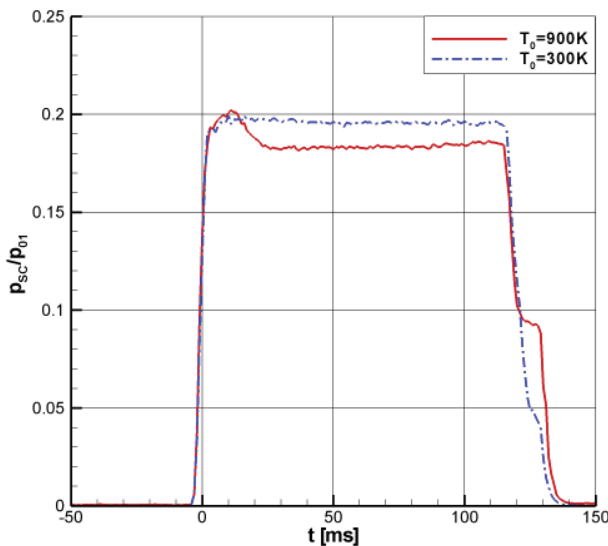


Figure 5. Settling chamber pressure to storage tube pressure during runtime for different temperatures.

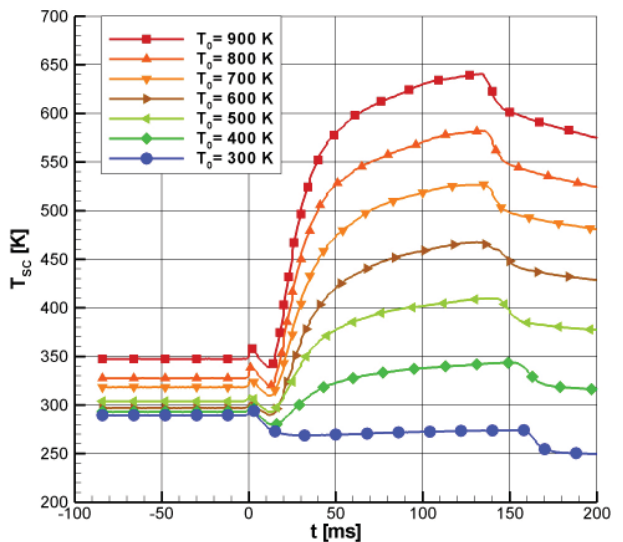


Figure 7. Settling chamber temperatures at varied storage tube temperatures ($p_0 = 140$ bar).

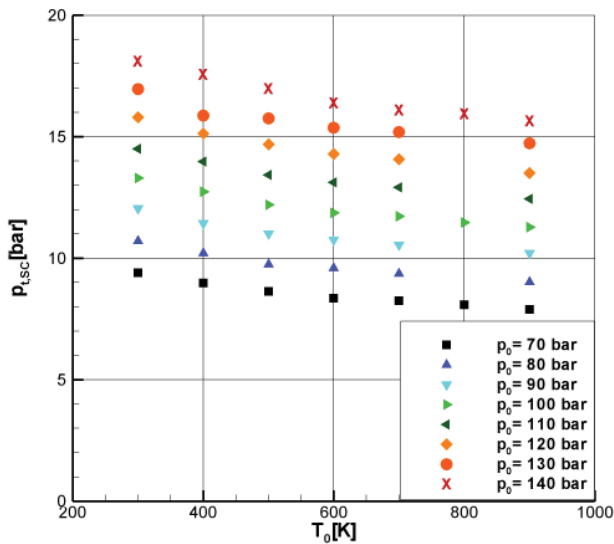


Figure 8. Settling chamber pressure for different storage tube temperatures at 80 ms.

different area ratios. He detected that the pressure loss over the measuring time depends on the storage tube Mach number. With rising Mach number the pressure loss increases. The pressure drop of the present facility with a storage tube Mach number of 0.2 is in good agreement with the results of reference [10].

In Figure 4 there is a further decrease of the pressure seen after about 100 ms. This pressure drop indicates the maximum operating time of the jet simulation facility that depends on the storage tube temperature. It is found that after 100 ms the expansion wave from the starting process appears back in the valve section. Note that the measuring time depends on the speed of sound and hence on the storage tube temperature.

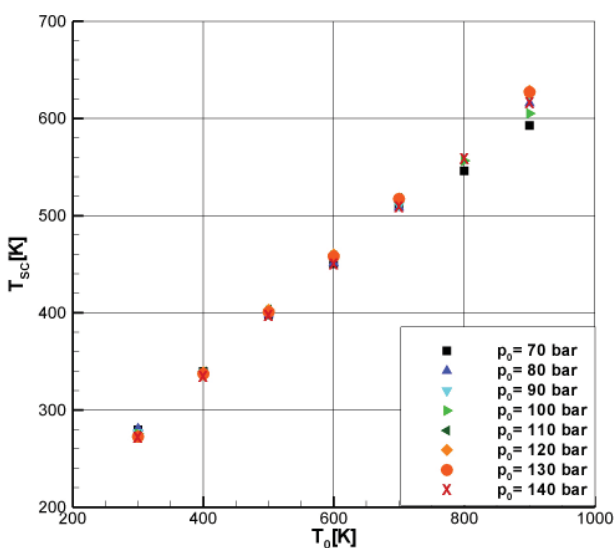


Figure 9. Settling chamber temperatures for different storage tube temperatures at 80 ms.

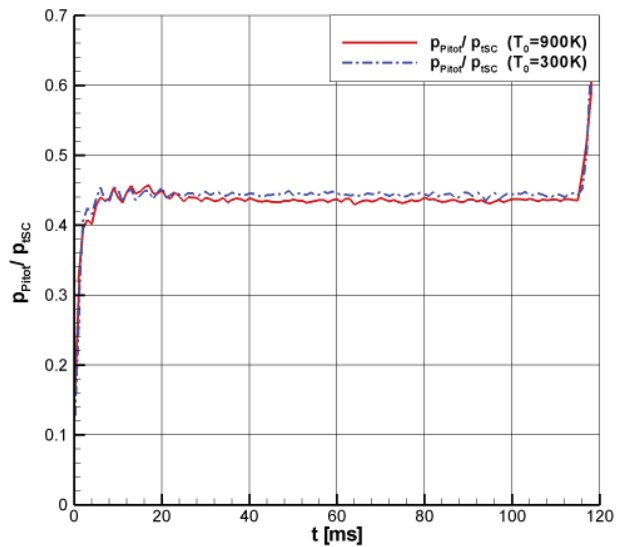


Figure 10. Pitot pressure in the center of the rocket nozzle exit.

The ratio of settling chamber pressure to storage tube pressure during runtime is shown in Fig. 5. The ratio is constant over time, but at higher temperature its value is lower. This ratio illustrates the strong pressure loss between the storage tube and the end of the settling chamber. The initial pressure loss agrees well with preliminary estimates based on one-dimensional flow theory. Figure 6 shows the ratio of settling chamber pressure to initial storage tube pressure ratio for different storage tube temperatures. The dash-dot curve shows the pressure ratio calculated with one-dimensional compressible flow theory. The long-dash and the solid curves show the measured ratios for low and high storage tube temperatures. The experimentally obtained settling chamber pressure can be well described with a linear function in dependence of the initial storage tube pressure and the measuring time.

Figure 7 shows the measured temperature in the settling chamber over run time for a storage tube pressure $p_0 = 140$ bar and for different storage tube temperatures. For a storage tube temperature of $T_0 = 900$ K the total temperature in the settling chamber reaches $T_{SC} = 620$ K after 80 ms. We find that for high storage tube temperatures the settling chamber does not reach a constant level. This could be caused by heat transfer in the unheated support tube that connects the model with the heated storage tube. For future improved measurements this part should be isolated and heated as well. Figure 8 shows the settling chamber total pressure $p_{t,SC}$ as a function of storage tube temperature T_0 and storage tube pressure taken at 80 ms after the facility start. With higher storage tube temperatures the settling chamber pressure decreases. Figure 9 displays the settling chamber temperature as a function of the storage tube temperature for storage tube pressures from 70 bar to 140 bar (80 ms after the facility starts). Up to $T_0 = 700$ K the variance of the settling chamber temperature is ± 10 K. For higher storage tube temperatures the temperature variances increase.

4.2. Flow Measurements of Propulsive Jet

Figure 10 shows the ratio of nozzle exit pitot pressure to settling chamber total pressure ratio. The pitot pressures are measured in the center of the nozzle exit as an average over 5 tunnel runs. The values are almost constant over the measuring time. We find

$$p_{Pitot,C} = (0.4440 \pm 0.0033) \cdot p_{t,SC} \quad (T_0 = 300 \text{ K})$$

$$p_{Pitot,C} = (0.4357 \pm 0.0044) \cdot p_{t,SC} \quad (T_0 = 900 \text{ K}).$$

Figure 11 shows the vertical pitot pressure distribution across the nozzle exit. The distribution indicates a symmetrical jet flow. Also it is shown, that the pressure increases in radial direction. In Figure 12 the Mach number distribution calculated with the Rayleigh pitot formula is shown. Also the Mach number distribution obtained from CFD computations of the same nozzle by Saile [9] is included. The Figure shows a good agreement between the computed and the measured Mach number distribution. In Figs. 13 and 14 the pitot pressure distributions of the plume flow at the axial positions $x/d = 2$ and $x/d = 3$ are shown. The total pressure in the settling chamber was $p_{t,SC} = 16.13 \text{ bar}$ and the settling chamber temperatures were varied between 470 K and 620 K. The flow properties of the surrounding flow were $p_{t,\infty} = 16.79 \text{ bar}$, $T_\infty = 470 \text{ K}$ and $Ma_\infty = 5.9$. Both facilities were synchronized to overlap their measuring windows. The values used for the evaluation are the mean values between 60 ms and 80 ms after the start of the TSA. Figure 13 shows the pitot pressure distribution of the plume flow at the axial position $x/d = 2$. Note that the differences between the pitot pressure distributions at different temperatures may have also been affected by locally large variations from run to run in the mixing region at $z/D = \pm 0.4$ that divides the jet flow from the surrounding flow. In Figure 14 the pitot pressure distribution for the axial position $x/d = 3$ is shown. These measurements were only conducted for the lower part of the jet plume.

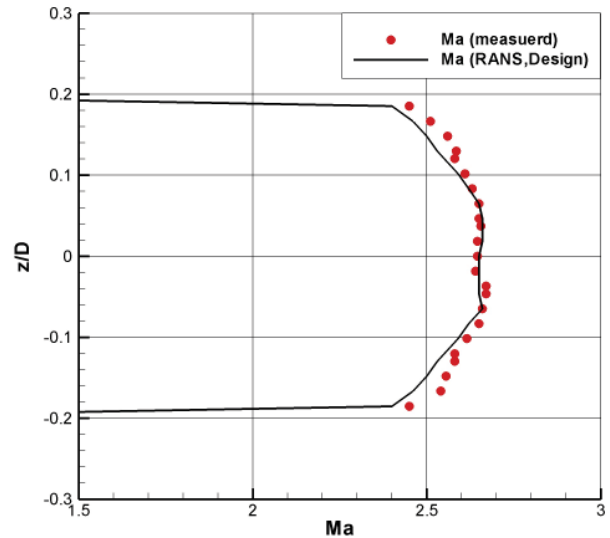


Figure 12. Pitot pressure distribution in the vertical plane of symmetry at the nozzle exit.

The flow field of the rocket afterbody was also characterized using Schlieren images. Shown are an instantaneous single image and an average of 30 images. Figure 15 shows the rocket afterbody without jet flow. In the lower part of the image the recompression shock of the external flow is visible. This recompression shock is significantly displaced by the underexpanded jet as shown in Figure 16. The images show the jet expansion fan, the plume barrel shock and the jet mixing region along with the outer compression shock. The oblique waves in the upper part of the flow are most likely caused by the model support.

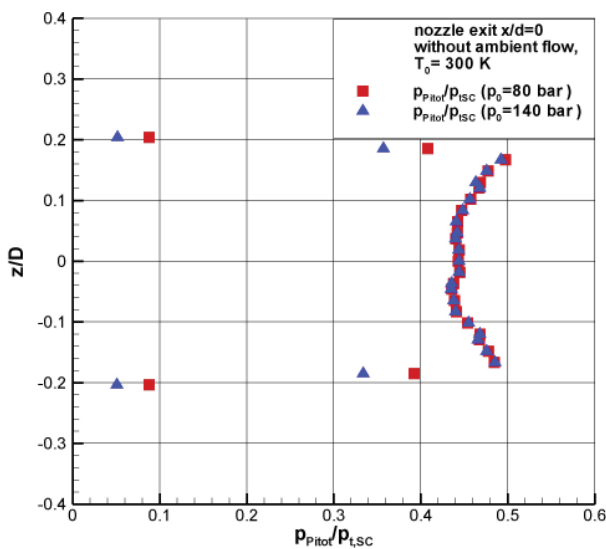


Figure 11. Pitot pressure distribution along the vertical nozzle exit.

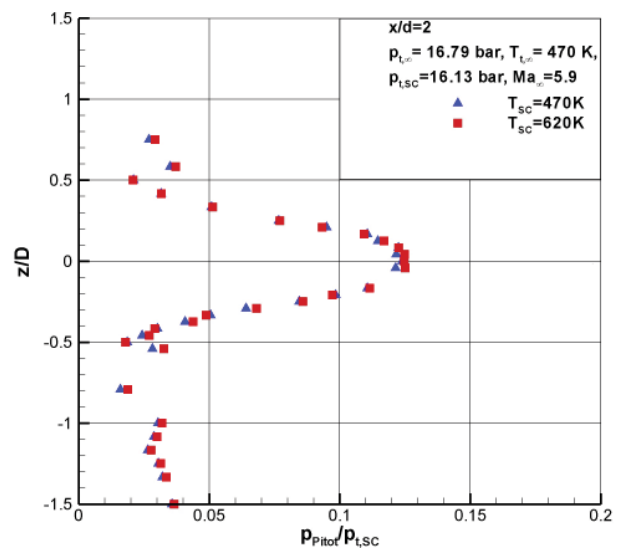


Figure 13. Pitot pressure distribution in jet plume at $x/d = 2$.

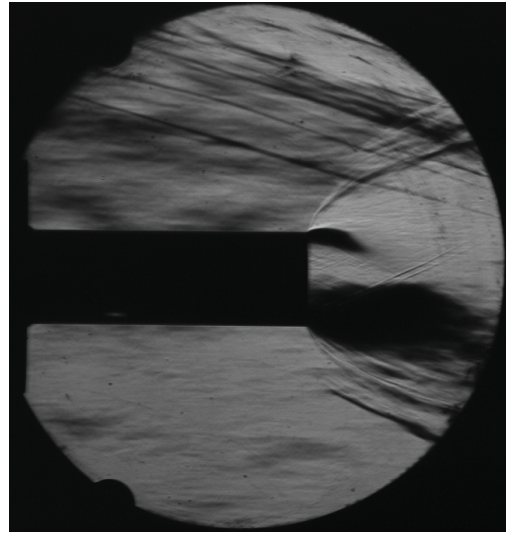
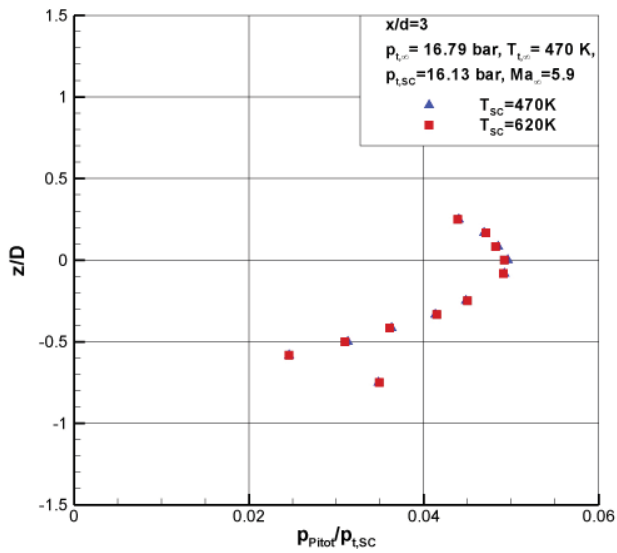


Figure 14. Pitot pressure distribution in jet plume $x/d = 3$.

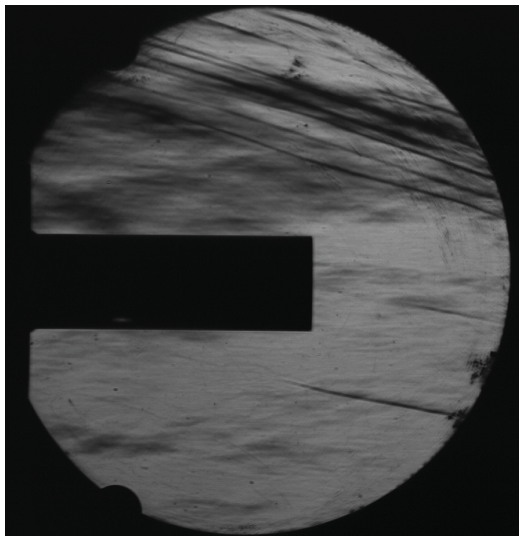


Figure 16. Schlieren image with jet and surrounding flow ($p_{t,SC} = 16.13$ bar, $T_{SC} = 470$ K and $Ma_e = 2.5$, $p_{t,\infty} = 16.79$ bar, $T_\infty = 470$ K and $Ma_\infty = 5.9$, top: single image bottom: averaged image).

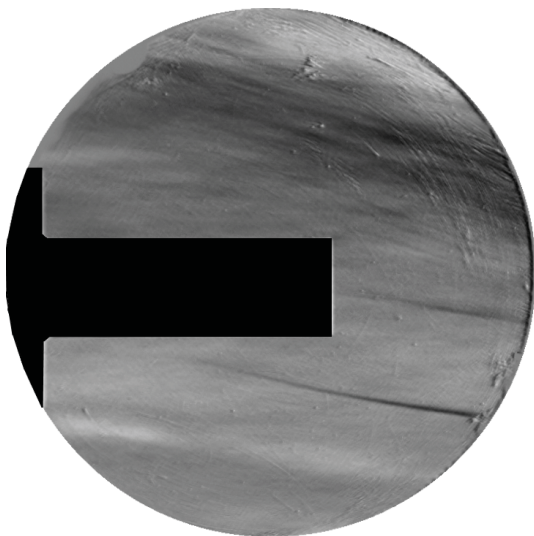


Figure 15. Schlieren image without jet flow ($p_{t,\infty} = 16.79$ bar, $T_\infty = 470$ K and $Ma_\infty = 5.9$, top: single image bottom: averaged image).

5. SUMMARY

The design approach for an efficient jet simulation facility and first measured results were presented. It was discussed that the jet simulation for rocket afterbodies depends on several similarity parameters. Important parameters are the velocity ratio to reproduce the turbulent stresses and associated mixing process and the jet flow momentum ratio to reproduce the mixing layer growth. It appears that using the Ludwig Tube operation principle for jet simulation opens the path to suited variations of these parameters in afterbody flow research, since the jet simulation can be efficiently performed with low molar mass gas Helium. As the diameter of the jet flow storage tube is limited for given model strut sizes our present experiments show pressure losses during runtime of the jet simulation facility that depend on the storage tube Mach number. The pitot pressure distributions of the plume flow at different axial positions are investigated and

discussed. The experiments confirm the high-quality nozzle flow and plume representation of the new facility. Future works will further examine the afterbody flow fields by using averaged and time resolved pressure measurements and PIV.

NOMENCLATURE

d	=	nozzle diameter, m
D	=	model body diameter, m
L	=	length, m
r	=	radius, m
p	=	static pressure, bar
p_t	=	total pressure, bar
T	=	static temperature, K
T_t	=	total temperature, K
ρ	=	static density, kg/m ³
ρ_t	=	total density, kg/m ³
κ	=	ratio of specific heat
Ma	=	Mach number
M_{Mol}	=	molar mass, kg/mol
\mathcal{R}	=	universal gas constant,
Re	=	Reynolds number
u	=	velocity
TSA	=	Jet simulation facility
HLB	=	Hypersonic Ludwig tube Braunschweig
x	=	axial direction
y	=	horizontal direction
z	=	vertical direction

Subscripts

c	=	nozzle center
e	=	nozzle exit
Pitot	=	Pitot pressure, total pressure in the nozzle exit
p	=	conditions at the nozzle plume
SC	=	settling chamber states
∞	=	free-stream conditions
0	=	storage tube states before running the facility
01	=	storage tube states along test

ACKNOWLEDGEMENT

The work was funded by the German Research Foundation (Deutsche Forschungsgemeinschaft, DFG) within the framework Sonderforschungsbereich Transregio 40 (Technological foundations for the design of thermally and mechanically highly loaded components of future space transporting systems).

REFERENCES

- [1] Peters, W. L., Kennedy, T. L., "Jet Simulation Techniques – Simulation of Aerodynamic Effects of Jet Temperature by Altering Gas Compositions," *AIAA-1979-327*, 1979.
- [2] Peters, W. L., Kennedy, T. L., "An Evaluation of Jet Simulation Parameters for Nozzle/Afterbody Testing at Transonic Mach Number," *AIAA-1977-106*, 1977.
- [3] Kumar, R., Viswanah, P. R., "Mean and Fluctuating pressure in Boat-Tail Separated Flows at Transonic Speeds," *Journal of Spacecraft and Rockets*, Vol. 39, No. 3, 2002.
- [4] Reijasse, P., Détery, J., "Experimental Analysis of the Flow past the Afterbody of the Ariane 5 European Launcher," *AIAA-91-2897*, 1991.
- [5] Koppenwallner, G., Müller-Eigner, R., Friehmelt H., "HHK Hochschul-Hyperschall-Kanal: Ein "Low-Cost" Windkanal für Forschung und Ausbildung," *DGLR Jahrbuch*, Band 2, 1993.
- [6] Pindzola, M., "Jet Simulation in Ground Test facilities," *AGARDograph 79*, 1963.
- [7] Frey, M. "personal communication", EADS Astrium.
- [8] Estorf, M. and Wolf, T. and Radespiel, R., "Experimental and Numerical Investigations on the Operation of the Hypersonic Ludwig Tube Braunschweig." *In Proceedings of the 5th European Symposium on Aerothermodynamics for Space Vehicles*, ESA SP-563, 579–586, 2005.
- [9] Saile, D., "Vorschlag eines Düsenprofils für SFB-TRR-40," DLR Institut für Aerodynamik und Strömungsmesstechnik, 2010, (unpublished).
- [10] Koppenwallner, G., Hefer, G., "Kurzzeitversuchsstand zur Treibstrahlsimulation bei hohen Drücken," *DFVLR*, IB 252-76 H 12, Göttingen, 1976, (unpublished).