Joint European Effort Towards Advanced Rocket Thrust Chamber Technology

Manuel FREY[1], Axel PREUSS[2], Gerald HAGEMANN[3], Stève GIRARD[4], Thierry ALZIARY DE ROQUEFORT[5], Philippe REIJASSE[6], Ralf STARK[7], Klaus HANNEMANN[8], Richard SCHWANE[9], David PERIGO[10], Luca BOCCALETTO[11], Hadrien LAMBARÉ[12]

[1,2,3]EADS SPACE Transportation, D-81663 München, Germany
manuel.frey@space.eads.net, axel.preuss@space.eads.net, gerald.hagemann@space.eads.net

[4,5]CEAT/LEA, 43 Route de l'Aérodrome F-86036 Poitiers Cedex, France
steve.girard@lea.univ-poitiers.fr, thierry.alziary@lea.univ-poitiers.fr

[6]ONERA DAFE, 8 rue des Vertugadins, F92190 Meudon, France
philippe.reijasse@onera.fr

[7]DLR Lampoldshausen, Langer Grund, D-74239 Lampoldshausen, Germany
ralf.stark@dlr.de

[8]DLR Göttingen, Bunsenstr. 10, D-37073 Göttingen, Germany
klaus.hannemann@dlr.de

[9,10]ESTEC, Keplerlaan 1, P.O. Box 299, NL-2200 AG Noordwijk ZH, The Netherlands
richard.schwane@esa.int, david.perigo@esa.int

[11,12]CNES, Direction des Lanceurs, Rond-point de l'Espace, F-91023 Evry Courcouronnes, France
luca.boccaletto@cnes.fr, hadrien.lambare@cnes.fr

Abstract – The achievements of the FSCD group during the last three years are described. For the case of nozzle flows in quiet ambience, new observations of the plume behaviour for truncated ideal nozzles are reported. Furthermore, some results of the hot Calo tests as well as of cold-gas sub-scale experiments with film injection are shown, and the transient start-up of the Vulcain 2 nozzle with TEG and dump film is numerically investigated. When operating on a launcher, the nozzle is exposed to a fluctuating ambience due to the launcher wake flow, which is simulated by advanced CFD-techniques. The coupling of a separated nozzle flow with these fluctuations is investigated numerically as well as by cold sub-scale experiments. Finally, the altitude-adaptive dual-bell nozzle concept is analysed. The transition between sea-level and altitude mode is studied in detail, and dedicated tests deliver important information about the behaviour of such nozzles during the launcher ascent.

1 Introduction

Since 1998, the European FSCD (Flow Separation Control Device) group deals with problems occurring in over-expanded rocket nozzles. The group consists of the industrial partners Snecma, EADS-ST Ottobrunn, Volvo Aero and of the research institutes DLR, ONERA, LEA Poitiers as well as ESTEC and CNES. The main focus of the group is a better understanding of flow separation and side-loads both in classical bell nozzles and altitude-adaptive nozzle concepts such as the dual-bell nozzle. The achievements of the group between 1998 and 2002 are summarised in Ref. [1], giving an overview about the experimental, numerical and analytical efforts of the group members.

In late 2002, the failure of the Ariane 5 ECA maiden flight L517 drew off the attention from side-load related topics, since the main focus was to get the Vulcain 2 engine back to flight, causing a very high work load for the involved industrial partners. In parallel, questions were raised concerning the future of any altitude-adaptive nozzle, namely how to qualify by test or calculation a new nozzle configuration for the launcher ascent. Nevertheless, work on flow separation and side-loads continued also after the A517 failure at most partners and got an additional boost by the successful return to flight of the Ariane 5 ECA in February 2005. Compared to earlier years, the main focus of the group's work has changed from the investigation of basic phenomena towards the more complex topics of separation in film-cooled nozzles, interaction with ambience and a quantitative analysis of the dual-bell transition.

As a continuation of Ref. [1], this paper gives an overview on the work done by the FSCD partners during the last three years and gives an outlook on planned activities.

2 Nozzle operation in still ambience

2.1 Plume investigations

During the early years of the European FSCD group, it was shown that two different plume patterns could exist in over-expanded rocket nozzle flows – the classical, well-known Mach disk and the cap shock pattern [1]. One major difference between the two is the shape of the strong shock: The Mach disk is curved concave viewed from upstream, slightly deflecting the flow towards the axis, whereas the small strong shock in the cap shock pattern is convex, deflecting the flow away from the axis and supporting the existence of a trapped vortex. While in truncated ideal nozzles (TIC), where the flow is uniform downstream of the design characteristic, only the Mach disk was observed, both patterns could be seen in thrust-optimised (TOC) or parabolic nozzles with their typical internal shock.

Later, it was shown from cold-gas tests that for low nozzle pressure ratios (NPR, defined as ratio of chamber pressure p_c to ambient pressure p_a) below p_c/p_a=15, the separated jet can locally reattach to one side of the nozzle [2]. This asymmetric behaviour occurs in both truncated ideal and thrust-optimised nozzles in an unsteady manner, causing a side-load peak, but vanishes as soon as the pressure ratio is increased. Furthermore, numerical simulations of a TIC nozzle showed a Mach disk with convex curvature and a trapped vortex [2].

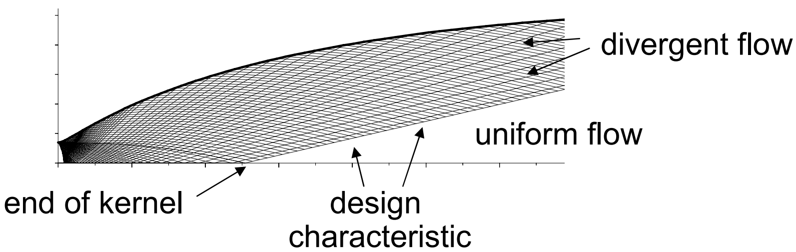


Fig. 1: Mesh of characteristics in a typical truncated ideal nozzle

In truncated ideal nozzles, the flow downstream of the design characteristic is uniform, i. e. has constant velocity and is parallel to the nozzle axis, see Fig. 1. For the usual design of a truncated ideal nozzle, the end of the kernel is far inside the nozzle. Thus, the flow in the exit plane is always uniform near the axis, whereas it is divergent with moderate angles and at a lower Mach number close to the nozzle wall. Hence, if a Mach disk appears outside these nozzles, its centre is always located in the region of uniform flow – the triple line representing the outer limit of the Mach disk might however be located in the region of divergent flow. It must be asked how the plume of a truncated ideal nozzle looks for decreasing pressure ratio as the Mach disk approaches or exceeds the end of the kernel, a question up to now unanswered mainly due to the poor observability inside a nozzle.

Recent cold-flow tests performed at FOI (Swedish Defence Research Agency) [3] with a TIC nozzle truncated far beyond the original length foreseen by design revealed a convex shape of the Mach disk near the centreline, which is in contrast to all hitherto reported experimental observations, but resembles earlier numerical findings [2]. A numerical recalculation of the FOI experiment provided by ESTEC [4], [5] is in good agreement to Schlieren pictures as far as the shock locations are concerned, see Fig. 2. Furthermore, it shows a trapped vortex downstream of the convex curvature of the Mach disk as up to now only observed in thrust-optimised nozzles. This deviation from the expected behaviour could be attributed to the strong truncation of the nozzle and the resulting divergent flow even close to the centreline. However, before jumping to conclusions, it has to be carefully checked if the applied TIC nozzle really delivers a shock-free flow at the given operating point as indicated by the numerical simulations and if the ambient pressure fluctuations present in the test could be a reason for the Mach disk curvature.

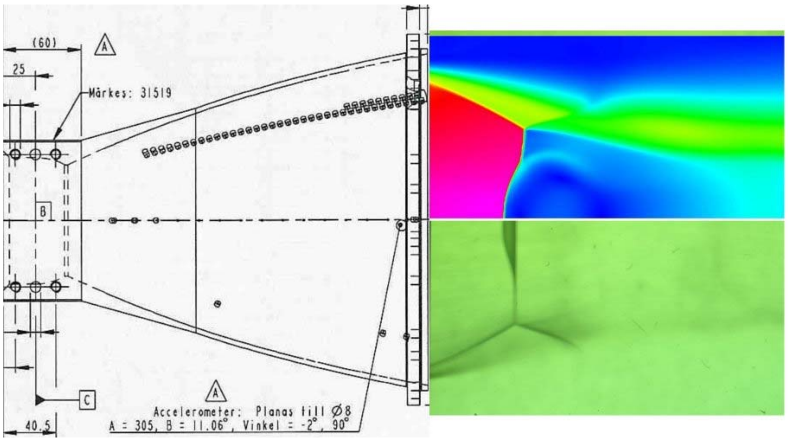


Fig. 2: : Flow in a truncated ideal nozzle with high divergence at $p_c/p_a \approx 30$ – Schlieren photograph from cold-gas test at FOI (bottom) and numerical simulation by ESTEC (top)

For the further examination of this phenomenon, additional tests with PIV (particle image velocimetry) flow visualisations are planned at FOI. In addition, DLR's Institute of Space Propulsion at Lampoldshausen plans a test series with a TIC nozzle to be successively shortened after each test.

2.2 Free shock separation

2.2.1 Calo tests

In the framework of the FSCD group, numerous cold gas tests were performed at different test benches [1]. In addition to these more basic experiments, EADS-ST Ottobrunn, DLR Lampoldshausen and Volvo Aero carried out hot runs with a 40 kN H_2/O_2 combustor at the P8 test facility at Lampoldshausen in 2003 [6], [7], [8]. Three different nozzle configurations – actively cooled without film (Calo A), actively and film cooled (Calo B) and only film-cooled without additional active cooling (Calo C) – with hardware manufactured by EADS-ST and Volvo Aero were tested at a wide variety of load points.

Numerous measurements were applied to ensure a thorough qualitative and quantitative description of the observed phenomena, e. g. calorimetric heat flux measurements, fast wall pressure transducers, hot gas side wall temperature sensors, infrared measurements as well as different video recordings. In total, 31 tests were run with the three different nozzle configurations resulting in approx. 2300 s test time. The database covers

- incremental change of the combustion chamber pressure between p_c = 30 bar to 120 bar
- transient change in combustion chamber pressure with different ramping speeds
- change in combustion chamber mixture ratio between O/F = 5 to 7.6
- change in secondary film mass flow rate between 45% to 120% of the design value

Fig. 3 left illustrates the measured wall pressure evolution for two selected load points together with calculated values, proving a good agreement between theory and test. Attached flow condition is achieved for the higher load point with p_c > 90 bar, while uncontrolled but steady flow separation is achieved at the reduced chamber pressure of p_c < 90 bar. An example of the wall pressure data for the three different thrust chamber configurations at similar load points is given in Fig. 3 right. The clean configuration Calo A reveals the lowest separation pressure. For Calo B and Calo C, the near-wall film alters the separation margin, a well known phenomenon caused by the lower momentum of the film.

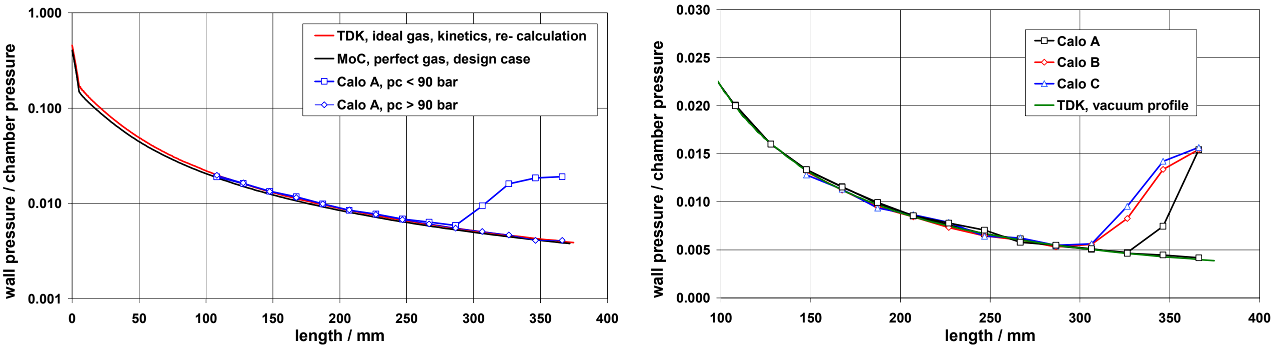


Fig. 3: Calo wall pressure distribution – left: numerical and experimental values for Calo A; right: measurements for Calo A, B and C at similar load point

The three-dimensional effects of film injection, i. e. variation in film efficiency over the circumference, on separation behaviour as known from Vulcain 2 full scale testing could fully be reproduced (see also Fig. 4 left). First data analysis shows a good correlation between full scale and sub scale separation effects with respect to film efficiency.

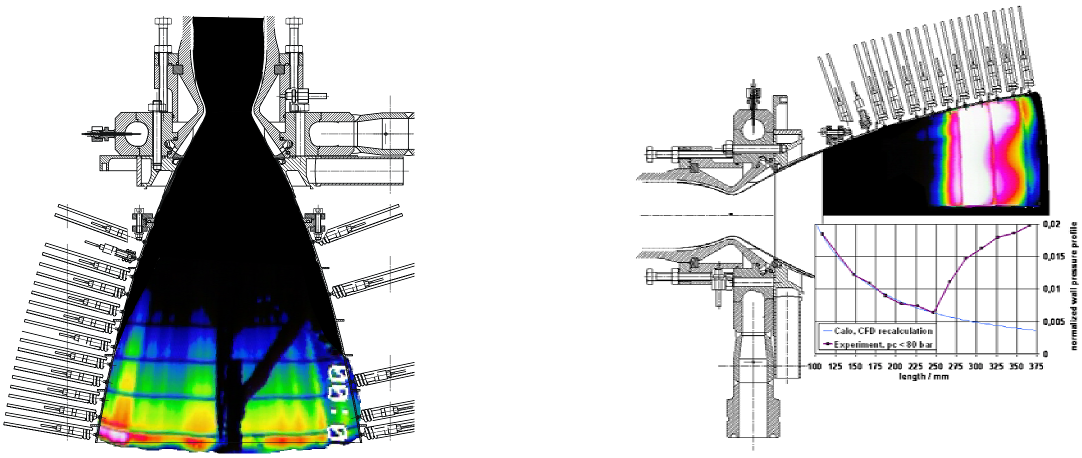


Fig. 4: Calo C infrared images – left: nozzle during high load point; right: infrared imaging compared to pressure measurements for a lower load point

The established database enables a clear distinction of the influence of chamber pressure, mixture ratio, wall temperature evolution, and secondary film mass flow on free shock separation. Furthermore, the fast wall pressure measurements enable the detailed analysis of the shock-boundary layer interaction within the incipient separation region.

2.2.2 Flow separation with cooling film

Cold-gas nozzle side-load tests at stabilised nozzle pressure ratios (NPR) have been performed in the ONERA R2Ch wind tunnel [9], [10] on a TOC nozzle able to run with or without a wall film. The characteristics of the film were chosen in similarity with Vulcain 2 concerning the ratios of static pressure, Mach number and mass flow rate. The side-load torque as well as the side force were measured independently by different strain-gauge mountings, and additional thrust measurements were performed.

For the case of flow separation upstream of the film injector, i. e. NPR $\leq$ 45, the measured side-load torque shows comparable values for the cases with and without film (see Fig. 5 left). However, the measured torques differ for flow separation downstream of the film injector(NPR > 45); a drop in side-load torque is observed for the case without film as the separation point leaves the film injection lip. For even higher NPRs resulting in flow separation near the nozzle exit (NPR $\approx$ 64), this difference vanishes again. However, when considering the measured side forces and normalising them by the thrust measured at each NPR, the film influence is hardly visible for all NPRs, see Fig. 5 right. This is mainly due to the separation location, which is significantly further downstream in the case without film, hence resulting in higher overexpansion and lower thrust.

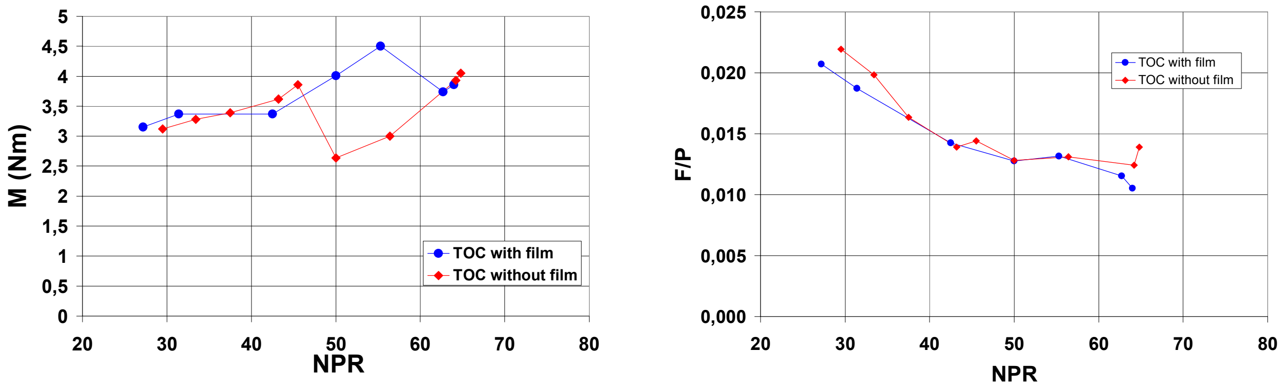


Fig. 5: Influence of a near-wall film on the side-load activity in a TOC nozzle at stabilised NPR – left: measured side-load torque; right: measured side force normalised by measured thrust

During the on-going test campaign further side-load measurements will be performed. In the near future, the influence of some control devices equipping the reference TOC nozzle profile (i.e. without the film capacity) on the side-loads will be experimentally investigated.

2.3 Transient phenomena

One characteristic of the Vulcain 2 engine is the injection of turbine exhaust gases (TEG) into the main nozzle. The purpose of this system is to cool the metallic wall of the lower part of the nozzle while optimising engine performance. The TEG gases' mixture ratio and temperature are typical of gas generators, thus fairly low in comparison to the main jet. The upper portion of the nozzle extension is dump-cooled, and the exhausted hydrogen is injected into the nozzle just above the TEG system.

During engine start-up, the three mass flow rates inside the nozzle (main jet, TEG, dump) experience different time evolutions; their interaction may therefore be different from the one during steady-state operation. In order to better understand the physical mechanisms which could play an important role in side-loads generation during transient phases, numerical simulations of such a configuration have been carried out and new experimental devices have been set up.

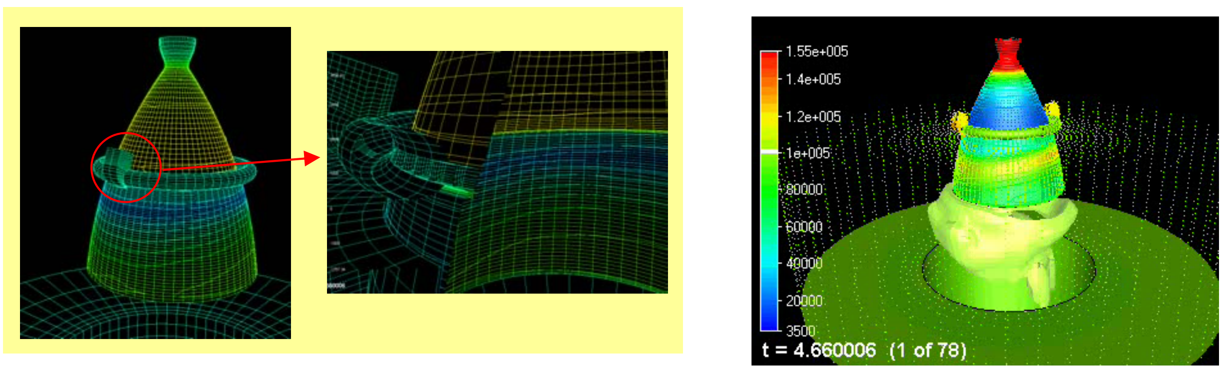


Fig. 6: 3D simulations of Vulcain 2 start-up – left: details of the computational domain; right: instantaneous pressure field

The CPS code has been used for three-dimensional transient computations [11], see Fig. 6 as an example. The main purpose of this work was to obtain a first order – but realistic – representation of physical phenomena inside the TEG and the nozzle. These computations showed that:

- the TEG mass flow distribution is heterogeneous in circumferential direction
- the distribution of this heterogeneity evolves during the start-up transient, depending on the engine operating point

Thus, the numerical results confirm the experimental observations. Moreover, a strong interaction exists between the three flows inside the nozzle, and this interaction is sensitive also to the rate of change of the film mass flow. Consequently, steady state simulations for this kind of phenomena cannot be completely relevant.

Experimental activities have been initiated in order to consolidate trends highlighted by numerical results: the experimental device already used for steady state tests in the ONERA R2Ch supersonic wind tunnel has been modified in order to well control the film injection during transient phases. The circumferential homogeneity of the film can be controlled as well as the local and global mass flow rate injected into the nozzle. A large set of transducers has been implemented in order to measure the wall pressure, film injection conditions as well as axial and lateral forces generated during the transient phase.

Fig. 7: Experimental test device for flow separation experiments with inhomogeneous film

A first test campaign was carried out in June 2005 in order to characterise the test device and to optimise the test sequence. During this preliminary test phase the device operated as expected and the facility is now ready to start experimental activities. Fig. 7 shows the experimental device mounted inside the test facility. The four lines for the feeding of the film are well visible. Each of them can operate independently.

3 Nozzle operation in fluctuating ambience

During the launcher ascent, the nozzle is located in the wake of the rocket body and is exposed to considerable ambient pressure fluctuations typical of wake flows. A maximum activity of these fluctuations is reached as the launcher velocity approaches the transonic regime. This behaviour differs from most cold gas tests and full-scale engine tests at ground, where the ambient pressure is constant.

3.1 Pressure pulsations in the nozzle ambience

Accurate numerical simulation of the unsteady and fluctuating flow field in the wake of launch vehicles is a very challenging task and requires the application of advanced turbulence models such as DES or LES methods. Therefore, the experimental investigation of such flows with launcher models in wind-tunnels represents an essential part of the design process. According to these tests, pressure fluctuations associated with large turbulence scales can reach 12 % of the dynamic pressure at the end of the nozzle. The energy spectrum is concentrated between Strouhal numbers of 0.22 and 0.6 (~ 10...30 Hz at launcher scale).

Two different numerical approaches by ESTEC and DLR Braunschweig/Göttingen, respectively, have been used to simulate the transient flow field and pressure fluctuations along the Vulcain 1 engine and its thermal protection (PTM) located at the base of the Ariane 5G wind tunnel model. ESTEC applied an unsteady-laminar approach with the NS code EURANUS (no dedicated modelling of sub-grid scale dissipation). DLR, Institute of Aerodynamics and Flow Technology used Detached Eddy Simulation with a Spalart-Allmaras based sub-grid dissipation model. Identical computational grids consisting of five million points have been employed for both computations. The resulting time-averaged streamlines are comparable for both approaches; as example, the results from the DLR computations are presented in Fig. 8 centre.

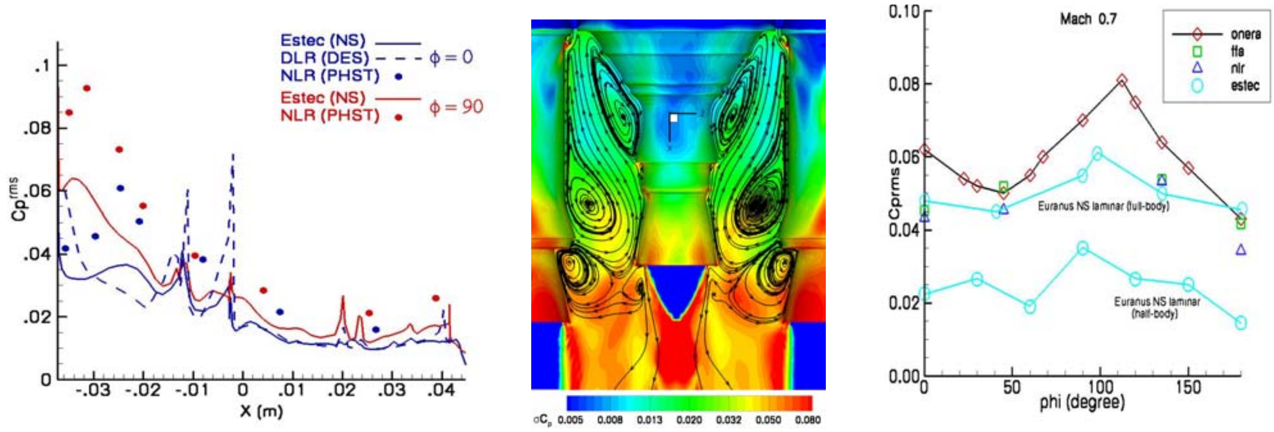


Fig. 8: 3D simulations of the Ariane 5G model aftbody – left: distribution C_p^{rms} along the nozzle and PTM; centre: C_p^{rms} distribution and time-averaged streamlines in the wake flow; right: C_p^{rms} at the nozzle exit for azimuthal angels in the range of 0°...180°

Fig. 8 left shows the RMS-value of the pressure coefficient C_p^{rms} along the Vulcain 1 engine with PTM. Two different C_p^{rms} distributions for azimuthal angles of 0° and 90°, respectively, are shown for the ESTEC computations. The distribution along 0° is compared with the DES result from DLR. In general, the DES approach shows larger fluctuations than those resulting from the unsteady-laminar solution. The experimental data from the NLR PHST facility [12] for 0° is included in the comparison. Both numerical solutions show the right trend in the C_p^{rms} distribution despite a relatively lower C_p^{rms} magnitude compared to test. The helium tank that was present in the experiment was not included in the CFD simulations, which could be a reason for this deviation. Fig. 8 right compares C_p^{rms} distributions around the nozzle exit from unsteady-laminar computations performed by ESTEC to a number of experimental data from ONERA, FOI and NLR, resulting in a satisfactory agreement for the full-body calculations, but revealing less adequate agreement if advantage is taken of the symmetry conditions by only calculating a half-body problem.

3.2 Coupling of nozzle flow with pulsating external pressure field

Interaction of a nozzle plume with unsteady ambient flow is a major concern for side-load generation, especially in the sub- and transonic flow phase, in which the inherent unsteadiness of a rocket plume and its effect on the ambient flow can be transported upstream and feed back, e. g. on the separation location inside the nozzle. This generates a flow environment in which unsteady flow structures can be amplified until the related loads exceed the structurally acceptable values.

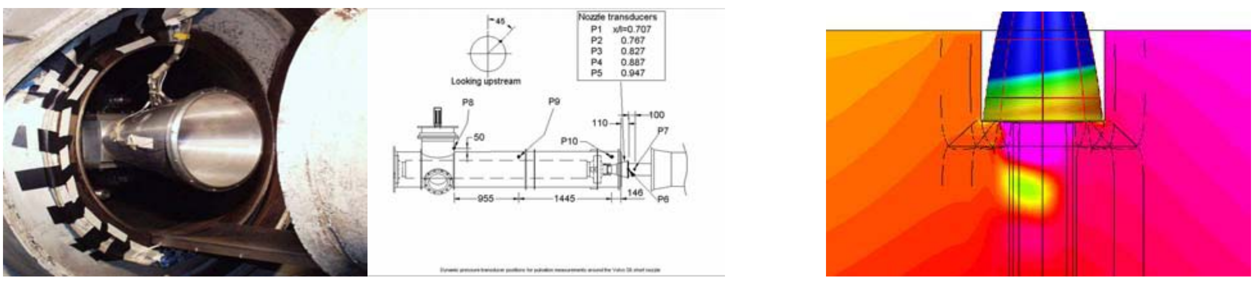


Fig. 9: Tests with fluctuating ambient pressure at FOI – left: photograph of nozzle; centre: sketch of modified tunnel set-up; right: result of numerical simulation of test under three-dimensional ambient excitation

To investigate this potentially critical load case, the HYP500 hypersonic wind tunnel facility at FOI has been modified [3], see Fig. 9 left and centre. The ambient flow entrained by the plume into the test chamber has to pass through a fast acting valve, which is used to generate a fluctuating ambient pressure of variable frequency. The resulting time dependent separation location in the nozzle can be observed by means of fast wall pressure measurements.

First results from this facility indicated a strong dependency of the amplitude of the separation location fluctuation due to ambient excitation for a certain frequency range, resembling the characteristics of weakly damped oscillators under external excitation. Due to limitations of the used instrumentation, however, this experiment was not fully conclusive and is therefore presently repeated with improved instrumentation to quantify the effect of coupling experimentally. Nevertheless, as a consequence of the test results, the flow has been rebuilt by CFD methods and by analytical modelling, both giving evidence of a possible resonance phenomenon [5], [13]. Fig. 9 right depicts a snapshot from the time dependent flow-field in which the separating plume is excited by ambient flow with 10% pressure fluctuation under 180 degrees phase angle between opposite sides of the nozzle. The deflection of the separation shock that generates large side-loads due to non-axisymmetric pressure distributions inside the nozzle is clearly visible.

Ultimately, the observation of resonance between ambient flow and separation location has led to the conclusion that the development of a nozzle with a large uncontrolled separation during sub- or transonic flow is not feasible. Research has consequently focussed on shapes with passive (dual-bell) or active (extendible nozzle) separation control.

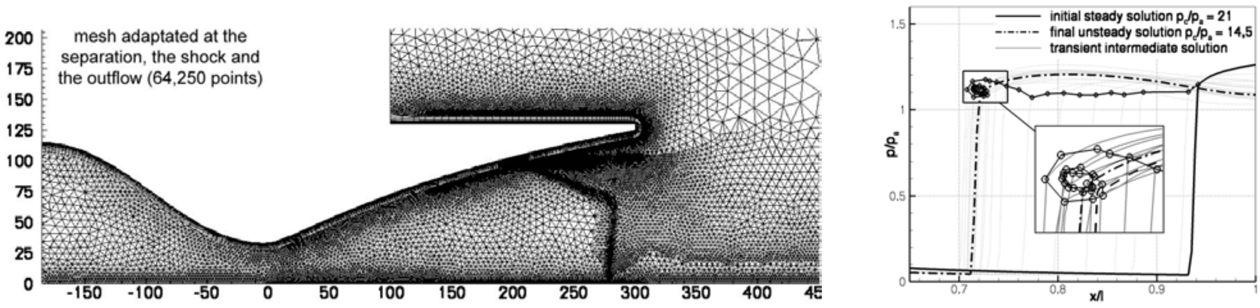
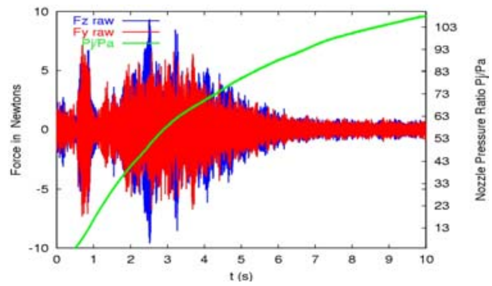


Fig. 10: Recalculation of FOI experiments with fluctuating ambient pressure – left: hybrid computational grid after adaptation; right: unsteady pressure distributions along the line of symmetry after sudden decrease of the nozzle pressure ratio from 21 to 14.5

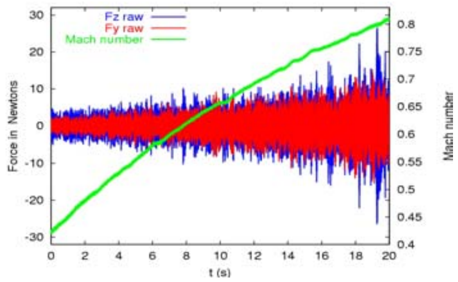
Numerical simulations of the interaction of separated flow in rocket nozzles with ambient pressure fluctuations were also carried out at DLR's Institute of Aerodynamics and Flow Technology [25]. The primary goal was to rebuild the afore-mentioned experiments performed by Torngren at FOI [3]: As the nozzle pressure ratio p_c/p_a is suddenly decreased, the flow field in the nozzle and the position of the shock system behaves like a damped oscillator as previously reported in Ref. [14]. A typical computational grid and the evolution of the pressure distribution along the axis of symmetry are shown in Fig. 10.

A further cold-gas study of interaction between transonic base flow and flow separation in an over-expanded nozzle was carried out by LEA Poitiers [15], [16]. Direct measurements of side-loads were performed by means of a specially designed strain-gage balance which measures the two components of the side force on a sub-scale TIC nozzle with external flow at Mach numbers between 0.5 and 0.82. The experiment corresponds to a transonic free jet of diameter 200 mm around an axisymmetric body of diameter 70 mm whose base is equipped with a nozzle of exit diameter 30 mm.

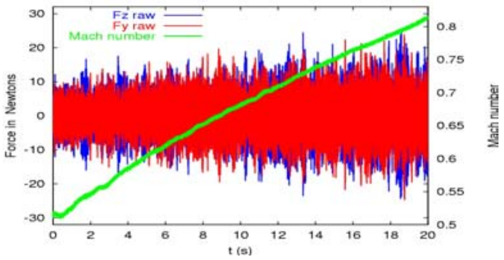
On Fig. 11 (a) the nozzle pressure ratio p_c/p_a is increasing slowly from 3 up to 110. High side-loads are first observed in the range NPR ≈ 4...8, corresponding to the asymmetric reattachment of the jet to one wall [2]. For higher pressure ratios, the force level increases with the pressure ratio up to a maximum at $p_c/p_a \approx 60$ and decreases to a very low value at $p_c/p_a \approx 100$, corresponding to the full-flowing regime. Fig. 11 (b) to (d) show the influence of the external Mach number Ma_∞ first without nozzle flow and then for NPR = 60 and NPR = 100. The comparison between Fig. 11 (b) and (c) highlights the increase of the side-loads level due to the nozzle flow. The contribution of the external flow to the side-loads can be evaluated by comparing Fig. 11 (a) at NPR = 60 (maximum level about ±5 N) and Fig. 11 (c) (maximum level about ±10...15 N for $0.5 < Ma_\infty < 0.82$), indicating an increase by a factor of 2 due to the external flow. Finally, the comparison of Fig. 11 (b) and (d) shows that, for high NPR (full flowing condition), the nozzle flow can decrease the side-loads level – most likely because entrainment by the nozzle jet moves the reattachment point of the external flow on the nozzle outside upstream.



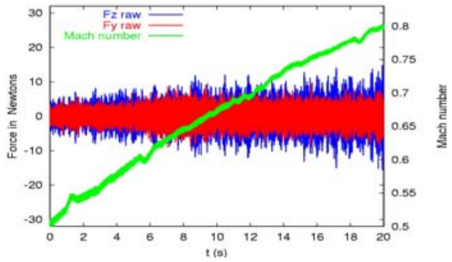
(a) Increase of p_c/p_a without external flow



(b) Increase of Ma_∞ without nozzle flow



(c) Increase of Ma_∞ at $p_c/p_a = 60$

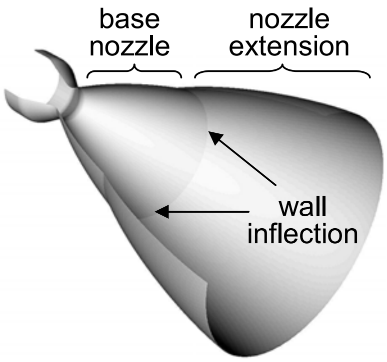


(d) Increase of Ma_∞ at $p_c/p_a = 100$

Fig. 11: Evolution of side-loads induced by increase of the nozzle pressure ratio p_c/p_a or by increase of the external flow Mach number Ma_∞ – raw results [16]

4 Advanced concepts – the dual-bell nozzle

A key issue for the realisation of the altitude-adaptive dual-bell nozzle is the transition between sea-level mode (controlled flow separation fixed at the wall inflection) and altitude mode (full-flowing nozzle). During this transition, the separation point jumps from the wall inflection (see **Fig. 12** left) to the nozzle exit. Thus, the nozzle experiences a short phase of uncontrolled flow separation, possibly resulting in an increased side-load activity.



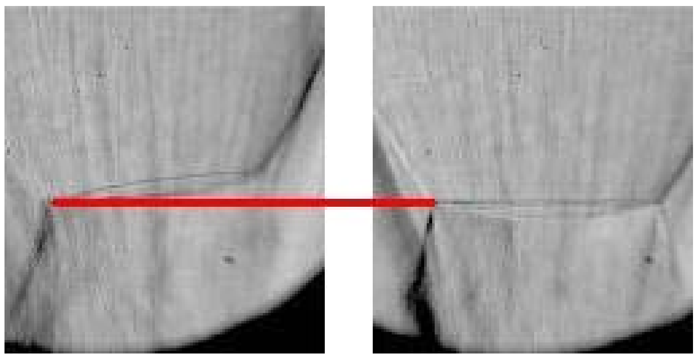


Fig. 12: Dual-bell nozzle – left: sketch of concept; right: asymmetric behaviour of transition compared to end of transition

DLR's Institute of Space Propulsion, Lampoldshausen, investigates among others experimentally the transition process and its influence on closed high altitude simulation chambers [17]. To visualise the behaviour during transition, black and white high speed Schlieren images were taken at a frame rate of 2 kHz. The typical duration of transition was ~2.5 ms, for a nozzle extension length of 0.1 m. An increasing Mach disk tilt angle indicated a skewed separation front of increasing asymmetry. If a wall-near flow velocity of 685 m/s is assumed, the separation front moved during transition with an averaged Mach number of Ma ≈ 0.28. The transition delay between two opposite nozzle sides can be determined from images to be lower than 0.5 ms. The maximum Mach disk tilt angle taken from images, e. g. Fig. 12 right, was approximately 8° and occurred right before the end of transition.

The Institute of Aerodynamics and Flow Technology in Braunschweig and Göttingen performs numerical simulations of flows in dual-bell nozzles [18]. The DLR TAU code [19] is used for 2D as well as 3D analyses of the flow field. Special focus is put on the prediction of transition time and pressure ratio. The goal of this investigation is to quantify the influence of different parameters such as turbulence and external pressure fluctuation on the transition process. Fig. 13 shows the Mach number distribution and streamlines obtained from a CFD simulation of a dual-bell nozzle in sea level mode with separation at the wall inflection (left). Furthermore, the transition time between sea level mode and altitude mode is displayed as a function of the temporal variation of the nozzle pressure ratio (right).

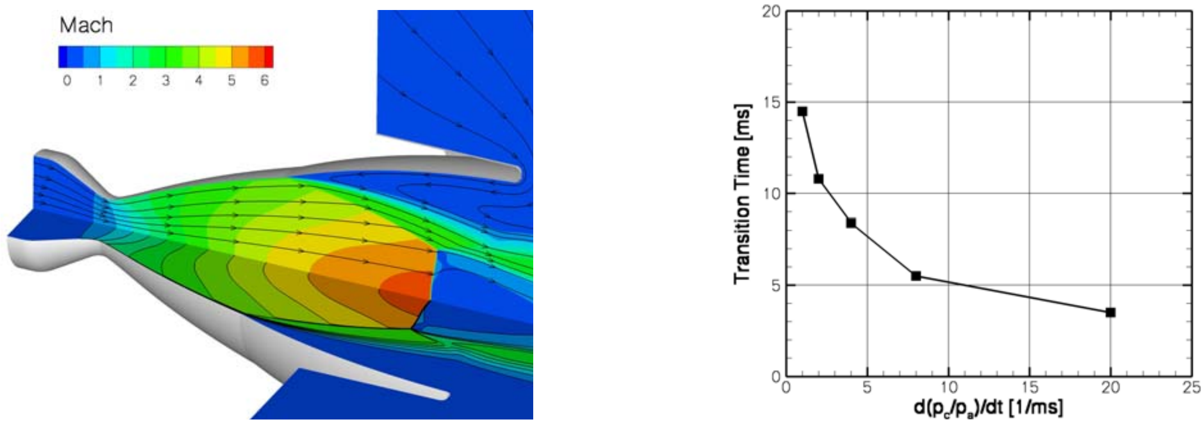


Fig. 13: Time-accurate simulations of a dual-bell nozzle flow – left: Mach number contours and streamlines during sea-level mode; right: dependence of transition time on pressure ratio change

Side-loads on a dual-bell nozzle were measured by LEA Poitiers [15], [16]. Their evolution without external flow with respect to the nozzle pressure ratio is illustrated on Fig. 14 left. During 20 seconds, the nozzle pressure ratio p_c/p_a is increased slowly from 3 up to 110 and then decreased. As soon as the flow approaches the inflection region at $p_c/p_a \approx 75$, increasing side-loads are observed up to a sudden decrease which indicates the transition from sea level to altitude mode. When the pressure ratio is decreased, the separation point moves back toward the contour inflection point. This reverse transition takes place at $p_c/p_a \approx 65$, thereby demonstrating a significant hysteresis. Figure Fig. 14 right shows that the same behaviour occurs with external flow at $Ma_\infty = 0.8$, the hysteresis occurs again between 75 and 65.

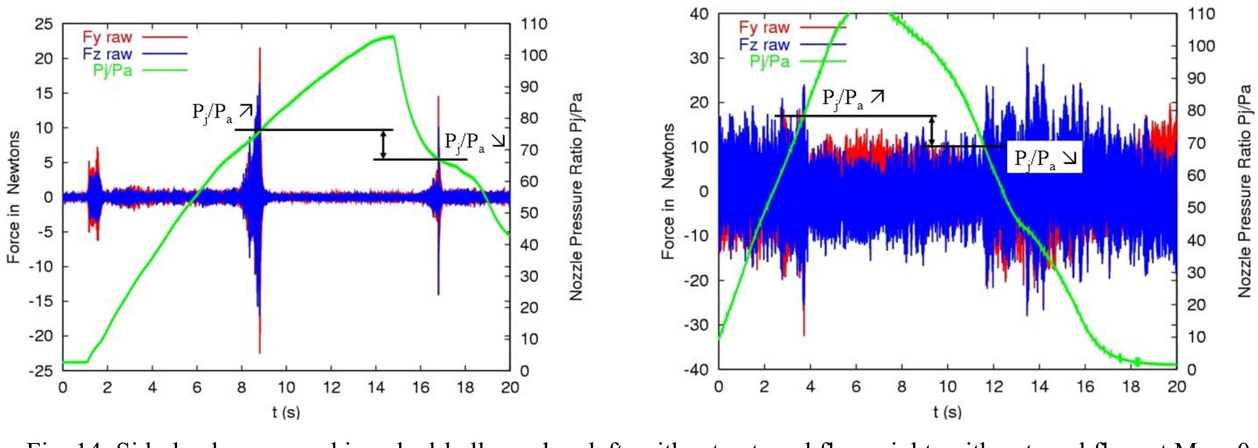


Fig. 14: Side-loads measured in a dual-bell nozzle – left: without external flow; right: with external flow at Ma_∞=0.8

One focus of future dual-bell nozzle tests and CFD investigations will be the influence of total and back pressure fluctuations during sea level mode and in particular during transition. Furthermore, EADS-ST plans to demonstrate the dual-bell concept on a 40 kN combustor with a fuel-cooled ceramic nozzle.

5 Conclusion

During the past three years, the European FSCD group made further progress on the field of flow separation and side-loads:

- The first experimental proof has been found for the existence of a convex Mach disk in a truncated ideal nozzle. The numerical recalculation is in good agreement to the experimental observation and additionally reveals the existence of a trapped vortex downstream
- Tests on the Calo H_2/O_2 subscale combustor with three differently cooled nozzle configurations have enhanced the knowledge on the influence of a coolant film and different wall temperatures on flow separation
- Cold sub-scale tests with film have been performed, and tests with controlled uneven film distribution around the circumference are planned
- Transient 3D simulations of the Vulcain 2 start-up have revealed details about the interaction between main flow, dump and TEG film
- LES and DES simulations have been applied to characterise the pressure fluctuations in the nozzle ambience for an Ariane5G wind tunnel model
- The coupling of separated nozzle flow with ambient pressure fluctuations caused by the launcher wake has been investigated by numerical simulations, indicating the possibility of resonance phenomena; furthermore dedicated cold sub-scale tests allow the quantification of side-loads for different ambient flow regimes
- The transition between sea-level and altitude modes for a dual-bell nozzle has been analysed with high-speed Schlieren imaging, revealing and quantifying the asymmetry of this process; numerical simulations show the dependence of the transition duration on the temporal change of the nozzle pressure ratio. Side-load measurements during dual-bell cold gas tests with and without ambient flow have given insight how such a nozzle could behave during the ascent of a launcher

A continuation of the experimental, computational as well as analytical activities is foreseen on all fields relevant for the implementation of an altitude-adaptive nozzle on a future European launcher.

6 Acknowledgements

The Calo tests were supported by the technology programme ASTRA, sponsored by the German Space Agency, DLR Bonn, under contract no. 50JR0015. The Swedish contribution to the Calo programme was sponsored by Swedish National Space Board (SNSB).
The new test campaign for transient phenomena at R2Ch is co-funded by CNES and ONERA and the work of LEA was funded by CNES, both in the framework of the ATAC program.

7 References

References cited in this article in the order of appearance

[1] M. Frey, R. Rydén, Th. Alziary de Roquefort, G. Hagemann, Ph. James, Th. Kachler, Ph. Reijasse, R. Schwane, R. Stark: *European Cooperation on Flow Separation Control, 4th International Conference on Launcher technology*. "Space Launcher Liquid Propulsion", 3-6 December 2002 - Liege (Belgium)

[2] W. Kwan, R. Stark: *Flow separation phenomena in subscale rocket nozzles*, AIAA2002-4229

[3] L. Torngren: *Correlation between Outer Flow and Internal Nozzle Pressure Fluctuations*. Proceedings of the 4th European Symposium on Aerothermodynamics for Space Vehicles, Capua, Italy 15-18 October, 2001.

[4] D. Perigo, R. Schwane: *Simulation of unsteady three dimensional separation behaviour in truncated ideal contour nozzles*. AIAA 2004-4015

[5] R. Schwane, H. Wong, L. Torngren: *Validation of Unsteady Turbulent Flow Predictions for Over-Expanded Expansion Nozzles*. Proceedings of ICCFD2, Sidney, Australia July 15-19, 2002

[6] G. Hagemann, J. Alting, J. Kretschmer, M. Terhardt, M. Frey, R. Rydén, R. Stark: *The Calorimeter Nozzle Programme*. AIAA 2002-3998

[7] G. Hagemann, A. Preuß, F. Grauer, J. Kretschmer, M. Frey, R. Rydén, R. Stark, D. Zerjeski: *Technology Investigation for High Area Ratio Nozzle Concepts*. AIAA 2003-4912

[8] G. Hagemann, A. Preuß, F. Grauer, M. Frey, J. Kretschmer, R. Ryden, K. Jensen, R. Stark, D. Zerjeski: *Flow Separation and Heat Transfer in High Area Ratio Nozzles*. AIAA 2004-3684

[9] P. Reijasse, L. Morzenski, D. Blacodon, J. Birkemeyer: *Flow Separation Experimental Analysis in Overexpanded Subscale Rocket-Nozzles.* AIAA 2001-3556.

[10] F. Bouvier, P. Reijasse: *Mesures de charges latérales dans les tuyères surdétendues. Soufflerie R2Ch. Programme FSCD.* Onera Rapport Technique N° RT 81/07571DAFE, November 2002

[11] L. Boccaletto, L. Lequette: *CFD Computation for Rocket Engine Start-up Simulation*. AIAA 2005-4438

[12] J. J. Meier: *Analysis of Ariane 5 base flow measurements in the NLR-PHST and FFA T1500 wind tunnels*. NLR-CR 99449

[13] H. Wong: *A Theoretical Prediction of Resonance in Nozzle Flows*. AIAA Journal, 2004

[14] R. Schwane, D. Perigo, Y. Xia. *Unsteady simulation of the flow in a truncated ideal contour nozzle under free shock separation conditions*. AIAA 2003-3676

[15] T.H.H. Le, S. Girard, Th. Alziary de Roquefort: *Direct Measurement of Side Loads with Transonic Buffeting.* 5th European Symposium on Aerothermodynamics for Space Vehicles, 8-11 November 2004, Cologne, Germany, SP-563

[16] T.H.H. Le: *Etude expérimentale du couplage entre l'écoulement transsonique d'arrière-corps et les charges latérales dans les tuyères propulsives*. PhD Thesis, 27/06/2005, University of Poitiers, France

[17] R. Stark, Ch. Böhm, O. J. Haidn, H. Zimmermann: *Cold flow testing of dual-bell nozzles in altitude simulation chambers*. European conference for aerospace sciences (EUCASS 2005), July 4-7, 2005, Moscow, Russia

[18] S. Karl, K. Hannemann: *Numerical investigation of transient flow phenomena in dual-bell nozzles.* 6th International Symposium on Launcher Technologies, Munich, 2005

[19] A. Mack, V. Hannemann: *Validation of the unstructured DLR-TAU-Code for Hypersonic Flow*, AIAA 2002-3111

Further references of the FSCD group for information

[20] Th. Alziary de Roquefort: *Low frequency fluctuations in separated turbulent compressible flows.* AFI 2002 Symposium, 17/12/2002, Tohoku University, Japan, ISSN 1344-2236, 2002

[21] S. Deck, A.T. Nguyen: *Unsteady Side Loads in a Thrust-Optimized Contour Nozzle at Hysteresis Regime.* AIAA Jour. Vol. 42 No. 9, 1878-1888, 2004

[22] A. Gross, O. Haidn, R. Stark, W. Zeiß, C. Weber, C. Weiland: *Experimental and Numerical Investigation of Heat Loads in Separated Nozzle Flow*. AIAA 2001-3682

[23] G. Hagemann, Th. Nguyen, H. Immich, G. Dumnov: *Rocket Engine Nozzle Concepts*. AIAA, Progress in Astronautics and Aeronautics, Vol. 200 Chapter 12, pp.437-468, 2004

[24] H. Kronmüller, K. Schäfer, H. Zimmermann, R. Stark: *Kaltgas-Höhensimulations-Prüfstand P6.2 des DLR Lampoldshausen*, DGLR-2002-031

[25] H. Lüdeke, A. Filimon: *Investigation of Transient Flow Phenomena at the Ariane-5 Propulsion System During Ascent*. 5th European Symposium on Aerothermodynamics for Space Vehicles, 8.-11. November 2004, Cologne, Germany

[26] H. Lüdeke: *Investigation of the Ariane-5 Nozzle by DES*. European conference for aerospace sciences (EUCASS 2005), July 4-7, 2005, Moscow, Russia

[27] M. Neracher, B. Hemmerling, D. Kozlov, W. Kwan, R. Stark, D. Klimenko, W. Clauss, M. Oschwald: *Rocket nozzle cold-gas flow velocity measurements using laser-induced gratings*. Journal of Raman Spectroscopy based on presentation at ECW2002; 33: 912-918

[28] A.T. Nguyen, H. Deniau, S. Girard, Th. Alziary de Roquefort: *Unsteadiness of Flow Separation and End-effects Regime in a Thrust-Optimized Contour Rocket Nozzle.* Flow, Turbulence and Combustion, Vol. 71 No. 1-4, 161-181, 2003

[29] K. Schäfer, Ch. Böhm, H. Kronmüller, R. Stark, H. Zimmermann: *P6.2 Cold Gas Test facility for Simulation of Flight Conditions - Current Activities*. European conference for aerospace sciences (EUCASS 2005), July 4-7, 2005, Moscow, Russia

[30] R. Stark, H. Kronmüller, D. Zerjeski, B. Wagner: Advanced *Flow Visualisation Techniques in Cold Gas Subscale Nozzles, a Comparison*. AIAA 2003-5180

[31] R. Stark: *Flow Separation in Rocket Nozzles, a Simple Criteria*. AIAA 2005-3940