Aerothermodynamics issues of the DLR hypersonic flight experiment SHEFEX-I

J.M.A. Longo[1], T. Barth, Th. Eggers
German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, 38108 Braunschweig, Germany

Here is presented an overview of the hypersonic experiment on sharp edge concepts, SHEFEX-I. The project, being performed under responsibility of the German Aerospace Center (DLR), is aimed to investigate the behavior and the possibilities of an improved shape for aerospace vehicles considering sharp edges and facetted surfaces. It is a basic in-flight experimentation research on hypersonic technologies for future launcher vehicles but not a re-entry experiment. Additionally, the SHEFEX-I project is the starting point for a series of experiments which enable the acquisition of important knowledge in hypersonic free flight experimentation and which are an excellent test bed for new technological concepts. The experiment which successfully flew on top of a two-stage solid propellant sounding rocket on October 27th, 2005 from Andøya Rocket Range in northern Norway, enabled time accurate investigation of the flow effects and their structural answer during a hypersonic flight 5.5<Mach<6.5 from 90 km down to an altitude of 20 km. The present paper gives an overview about the aerothermodynamic philosophy and introduces some main outcomes of the post-flight analysis.

I. Introduction

After almost fifty years flying hypersonic vehicles, the fatal flight of the US-Orbiter Columbia in 2003 has shown the severity of the hypersonic environment. Indeed, hypersonic systems are complex, difficult to design and expensive to build due to a lack of physical understanding on the involved flow-regimes (**Fig. 1**) and a lack of available data for design. Although there is considerable energy spend designing new concepts and developing the technology base required to support them, today access-to-space vehicles do not provide routine, low-cost operations. Several expendable and partially or fully reusable concepts discussed in the past require essential improvements over current vehicles in order to ensure economic viability and to fulfill mission and safety constraints. In particular, many the technological developments required to reduce cost and to improve the reliability of accessing space are strongly associated to the progress in knowledge achieved on aero(thermo)dynamics. Indeed, computational fluid dynamics (CFD) has now matured to the point that it is widely accepted as a key tool for aerospace design. While algorithms have been the subject of intensive development for the past three decades, for high-speed flows the effective use of CFD to more complex applications, require more sophisticated algorithms since one of the key problems here is the treatment of multiple space and time scales [01]. Also the strong need to validate the computer codes by comparison with experiments is for flows related to high speed vehicles not an easy task due to the wide spectrum of physical conditions to be reproduced in the experiments. In spite of these issues, numerical computations have the potential to become the principal tool of aerospace design processes because of the flexibility it provides for the rapid and comparatively inexpensive evaluation of alternative designs, and because it can be integrated in a numerical design environment for both multidisciplinary analysis and multidisciplinary optimization. The demands for new space transportation systems is calling for a very strong disciplinary coupling of the major technology fields of high speed vehicles such as aerodynamics, thermal and electromagnetic environments, propulsion, materials and structures and guidance and control. Any shortcoming in a particular field can affect strongly the other fields. Interdisciplinary applications coupling CFD with the computational analysis of other disciplines is playing an increasingly important role.

[1]Head Spacecraft Branch, Lilienthalplatz 7, 38108 Braunschweig, Jose.Longo@dlr.de, AIAA Senior Member

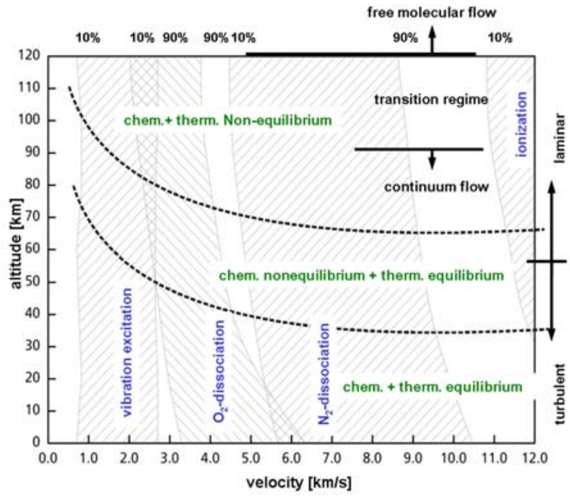

Fig. 1: Basic physical and chemical features of high speed flow.

Validation and error estimation are critical challenges for CFD because uncertainties in predicting the vehicle performance increase the design margins, adding weight and lower the vehicle performance. Numerical code accuracy and the accuracy of the physical models used in the code are the two main sources of error. Under ideal circumstances, the numerical error estimation is based on fully grid-converged solutions. However, grid convergence studies are necessary but insufficient for establishing error estimates in the less-than-ideal circumstances that usually prevail in hypersonic applications. For geometrically and physically complex configurations, obtaining a grid converged solution is either not possible due to the nature of the flow or is precluded by the high demand in computer resources. Therefore, comparisons of numerical solutions with experimental data are necessary but unfortunately not enough to determine physical modeling errors, since numerical accuracy and physical modeling errors are coupled by the fact that physical models are functions of flow parameters and their gradients, which are in turn functions of the grid resolution. While CFD error estimation is predominantly based on code validation experience, confidence in CFD predictions depends ultimately on comprehensive comparisons with experimental data with well-defined error bounds [02-03]. Wind tunnels are the major source of flow data for CFD validation. They are very important because under ideal conditions they allow controlled building block experiments, i.e. experiments which satisfy predefined conditions such as the precise definition of the test geometry, the absence of uncontrolled parasitic effects and complete information on the uncertainty margins. However, for hypersonic applications it is recognized that the simulation of all flight parameters in one selected type of wind tunnel is not possible. As is shown in **Fig. 2**, the simultaneous simulation of flight Reynolds number, flight Mach number and binary scaling parameter which controls the dissociation reactions in shock layer flows is not possible in high enthalpy flows. Also, wind tunnels have limitations inherent to the type of facility, kind of operation and instrumentation used. Indeed, the complexity of hypersonic flows requires that experiments in ground based facilities are strongly linked with computational fluid dynamics investigations. These common activities range from the calibration process of the facility and the study of basic aerodynamic configurations, which are well suited to look at fundamental aspects of high enthalpy flows fields, to the investigation of complex configurations [4].

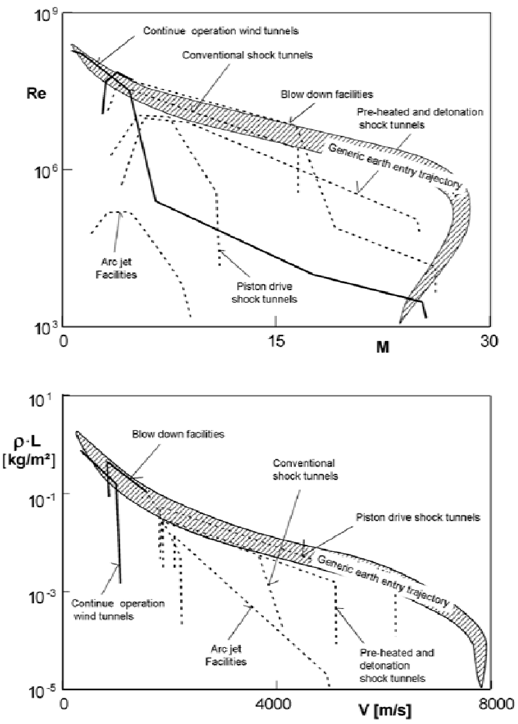

Fig. 2: Wind tunnel operating ranges. Top: Diagram based on Reynolds and Mach number. Bottom: Diagram based on the binary scaling parameter and velocity.

Finally, in-flight-measurements constitute the only way to obtain data for prediction tool validation and calibration under real conditions and therefore, they are irreplaceable for CFD validation. However, flight measurements are not unexpensive, they require considerable time for preparation and their complete repeatability is not always possible. Indeed, to repeat the same flight path under similar atmospheric conditions is one of the major sources of the difficulties. In addition, the data obtained for quantities which can only be measured indirectly, may contain important uncertainties [05]. Further, in the last 25 years many X-programs, such as Hermes, the X-30, Sänger, X-33, X-34 and the X-38 have failed, in some cases short before the flight test that would produce the data, so necessary to advance our understanding of the hypersonic environment and how to design vehicles that would survive in that environment. Those failures have been partially due to economical reasons and partially due to the technological challengers associated with the projects, being the first cause also close related to the second one. One of the big mistakes of this period of time has been the strong coupling of the progress in hypersonic physics with new operational systems developments instead of use less complex systems like sounding rockets as test bed for new technological concepts or for simple gathering flight data representative of real hypersonic environments. Sounding rockets may pay the way of access to a certain type of “sky based facility” (in analogy to the ground based facilities) but extremely less expensive and less risky than the so-called X-vehicles. Emerging examples of such strategy are the SOAREX experiment of NASA [06], the HyShot experiment of the University of Queensland [07] and the HiFIRE program of the US-AFRL [08]. At DLR R&D this approach has been adopted with the SHEFEX Program [09], while the project to be described in the following chapters, SHEFEX-I, has been the pathfinder experiment.

II. Definition of the SHEFEX-I Experiment

Aerodynamic sharp-edged vehicles have better aerodynamic performances than hypersonic blunt-nosed ones. Indeed, sharp edged configurations have minimal drag and offer more gradual kinetic energy conversion and flat heat flux time profile along the upper part of the re-entry trajectory avoiding high peaks in the energy dissipation and heat flux rate; controllability of the vehicle along the entire flight path; low pressure forces and decelerations; low landing speed; a very large landing footprint due to the long re-entry duration; low angles of attack for efficient control surfaces operations; require lower thrust during ascent. Cross-range improvement affects launch abort, the re-entry windows and the number of locations required for landing sites. Finally, sharp leading edges minimize the number of electrons that jam radio transmissions and that cause communications blackouts during the re-entry of blunt-body vehicles. Small radius leading edges and nose-tips were used in early hypervelocity vehicle concepts to minimize wave drag until further analysis demonstrated that extreme aerothermodynamic heating blunted the available thermal protection system materials. Supersonic aircrafts of the 1940s routinely featured needle noses and tapered wing tips that reduced drag until the beginning of the space-age in the 1950s, where it was recognized that materials scientists could not then provide thermal protection systems enabling the design of Earth entry vehicles with sharp leading edges [10]. The use of blunt-nosed shapes tends to alleviate the aerodynamic heating problem because the heat flux for blunt bodies scales inversely with the square root of the nose radius and in addition, the reduction in heating rate for a blunt body is accompanied by an increase in heat capacity, due to the increased volume. This concept leads the design of many vehicles in the past, like Mercury, Gemini, Apollo and also the Shuttle Orbiter. But shape optimization studies in the 1960s and 70s suggested that improved volumetric packaging efficiency and improved aerodynamics would be realized with vehicles using sharp leading edges. Hence, designing a hypersonic vehicle involved a trade off between making the vehicle sharp enough to obtain acceptable aerodynamic and propulsion efficiency and blunt enough to reduce the aerodynamic heating in stagnation regions.

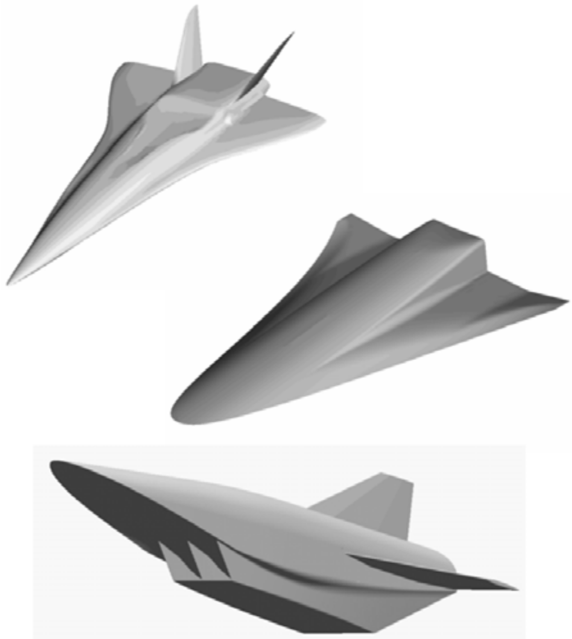

Fig. 3: Sharp edge concepts studied at DLR. From top to bottom: Sänger, F-8 and JAPHAR.

During the 1990s, the development of ceramic composite and ultra high temperature materials for TPS applications led to a renewed interest in sharp edged configurations. In the USA, NASA redefines the lifting body concept HL-20 [11] as well as a new project, SHARP [12]. Also in Germany DLR is being put since the beginning of the nineteen, considerable effort to understand the hypersonic physics of sharp edge configurations (**Fig. 3**). Examples of that are the German national program SÄNGER and in particularly the DLR-F8 waverider concept [13] and the DLR-ONERA project JAPHAR [14]. Earlier in the nineteen it has been envisaged at DLR the potential of ceramic materials to build up sharp configurations thanks the property of thermal conduction along the fibers. At that time first computed results by means of 2-D fluid-thermal-structure coupling pointed out that one reason for the high efficiency of the waverider forebody is associated with its small leading edge bluntness [13]. Preliminary studies based on the Fay-Riddell equation showed that also relatively blunt leading edges with nose radius of 10mm will not allow the use of metallic materials. In this situation, where a hot structure and the use of ceramic materials is required anyway, also much smaller bluntness could be considered in order to increase the aerodynamic performance. The interdisciplinary analysis of fluid-structure interactions was done by coupling a finite-element program with a Navier-Stokes code allowing 2-D simulations. The coupling of the nonlinear thermal and mechanical fluid-structure interactions has been treated time-accurately with view to the structure and as steady state problem for the fluid. This interdisciplinary approach contains the complete interactions between structure and fluid as shown **Fig. 4**, while the results exceed the accuracy of classical flow analysis with prescribed adiabatic or isothermal wall condition and using the assumption of Stanton number independent of the temperature. The figure shows that the temperature rise at sharp edges would not be as large as resulted from a pure radiation adiabatic computation, a finding also reported by NAL [15]. When the material conductivity decreases or the radius R decreases the maximum temperature increases. In case of large radius the influence of the material conductivity is reduced because the temperature gradients in the edge area are small and the distance to surface parts with low aerodynamic heating are long. On the other side for radius smaller than 8mm as the material conductivity increases the maximum temperature remains almost independent of the radius. Indeed, when the radius is very small large heat fluxes go into the structure in the vicinity of the stagnation point being then emitted from the surface to the flow downstream the stagnation point. Further for very high conductivity materials and small radius, a decrease of the maximum temperature becomes nonlinear and hence, a decrease of the maximum temperature can be observed if the radius is reduced.

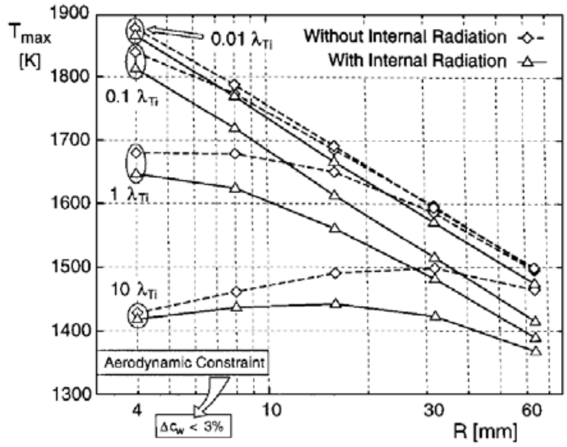

Fig. 4: Maximum temperatures as function of the nose radius and materials of the DLR F8 configuration.

Within the design of the JAPHAR vehicle, the fluid-structure coupling procedure has been extended to allow 3D analysis. A direct comparison of 2D and 3D results indicated that in the stagnation point region the three-dimensional effects are small but the temperature level decreases drastically with increasing leading edge sweep. Further, in comparison to results with radiation adiabatic wall condition it is demonstrated that a temperature reduction of almost 180K may be obtained if fluid-structure interactions are taken into account, being the effect of the fluid-structure-coupling of the same size for perfect gas as for equilibrium air. **Figure 5** shows that the coupling effect is limited to the region of the leading edge radius. Here the temperature level is noticeably reduced and the heat transfer rate increases according to the temperature change. The heat is conducted away through the structure which results in a slightly increased surface temperature level in streamwise direction.

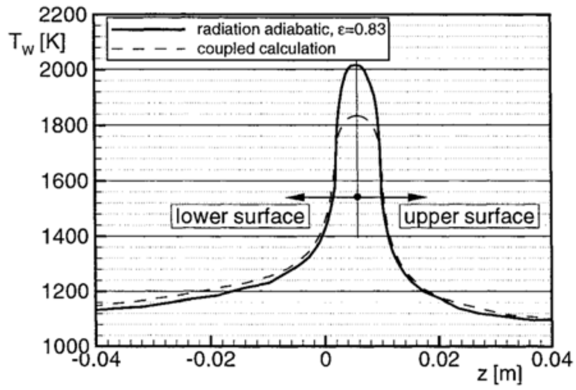

Fig. 5: Influence of the fluid-structure-coupling at the leading edge of the JAPHAR configuration.

The above described numerical observations have been experimental corroborated in the frame of the German national program on re-usable technologies TETRA, with results obtained in arc-jet facilities [17] and finally re-confirmed using numerical results obtained with the DLR most developed coupling tool. While the above discussed coupling problems have been more or less manual driven, in the frame of the IMENS project (a project embedded in the German national program ASTRA) a full integrated CFD tool has been developed and extensive validated using arc-jet facilities results for hypersonic fluid-/thermal-structure problems [18]. The new coupling procedure is a loose coupled approach, in which the solutions are performed using different schemes, as shown in **Fig. 6**. In particular the CFD solver TAU [19] of DLR calculates the surface heat flux, and then its solution is interpolated using MpCCI and set as boundary condition of the structural commercial solver ANSYS. The structural solver gives the temperature associated with the applied heat flux, which is then interpolated and set as boundary condition to the flow solver. TAU computes the associated heat flux. This is called Dirichlet-Neumann iteration and is applied with a relaxation of the temperature for the steady state case, and looped over time for the unsteady simulation over the trajectory. This procedure is repeated iteratively until a convergence of the algorithm has been reached. Typically, an accurate solution is achieved within 3 to 5 iterations for steady and in 2 to 3 for the unsteady simulations depending on the time step and the variation of the temperature.

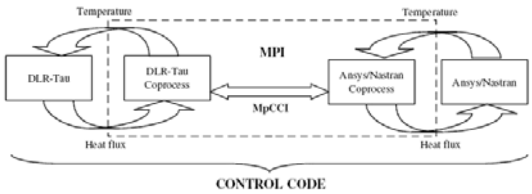

Fig. 6: DLR fluid-/thermal-structure coupling procedure IMENS.

However at that time no experimental evidences in flight were available to compare and the extreme heat impacting the sharp edges could jeopardize the material integrity in case of a space transportation vehicle. Furthermore, at the interfaces of the panels, chamfers may be created which interact with the surrounding flow. All these effects have to be investigated in detail with help of flight experiments before to assess the overall aerodynamic performance of the sharp concept approach. Hence, the objective of the SHEFEX-I experiment has been the assessment of the potential of sharp edged configurations applying the three point strategy: numerical analysis - ground based facilities - flight experiment [20]. The motivation has been neither to perform a re-entry experiment nor to fly at the thermal boundary of modern high temperature materials but to prove in flight that the temperature peaks at the edges of the ceramic-composite panels are lower than those predicted based on a radiation equilibrium hypothesis.

Driven by the facetted concept, two main criteria have been used to define the aerodynamic shape of the SHEFEX forebody. To have as many as possible facetted panels and to represent as many as possible configuration details of space vehicles, like concave and convex chamfers, wedged compression corner to mimic control surfaces and sharp unswept leading edges. It turns out the configuration of SHEFEX-I is a non-symmetric one as is shown in **Fig. 7**.

Fig. 7: SHEFEX-I forebody

From an aerothermodynamics point of view, a secondary goal of the experiment has been the acquisition of flight-data for shock-wave boundary-layer interaction particularly for turbulent boundary-layer flows embedded in a cold hypersonic free stream. Accurate aerodynamic prediction is critical for the design and optimization of hypersonic vehicles. While laminar hypersonic flows are today well mastered with CFD, turbulence modeling remains a major source of uncertainty in the computational prediction of hypersonic aerodynamic heating [21]. While comprehensive reviews of hypersonic shock/turbulent boundary-layer interaction experiments have been published in the past years [22-25], none of then address the impact of the facility turbulent nozzle-flow on the measured results and therefore it is difficult to assess the final accuracy of the existing turbulence models. While in absence of significant flow separation most of the turbulence models predict adequately surface pressure [26-28], heating may be largely over predicted, in some cases by more than one order of magnitude. This situation is even worse for massive shock-wave boundary-layer separation flow problems. In such cases flow features as separation and reattachment location may differ strongly from one model to the other [29-31]. Shock-wave boundary-layer interaction is also a problem where DLR is also here being put considerable effort since the times of the HERMES development [26-31]. In the frame of SHEFEX-I experiment a wedged compression corner has been defined in the lower surface of the configuration to promote hypersonic shock-wave boundary-layer interaction but no attempt to determine accurately transition was planed. Further, no wind tunnel test campaign to assess the aerodynamic shape was planed before the flight. The aerodynamic layout of the launcher as well as the aerothermodynamic definition of the experiment has been performed based only on CFD Euler- and Navier-Stokes calculation, applying the DLR TAU code [32] on unstructured grids. Driving design factors for both, the re-entry experiment vehicle and the launch vehicle, have been the experiment's requirements. These included a controlled and stable descent flight without any spin-up and at best, zero angle of attack (AoA), especially in the axis containing the experiment asymmetry. To meet these requirements, considerable modifications to the original concept for the complete vehicle have been essential as displayed in **Fig. 8**. Since small changes from a flight proven configuration significantly affect the ascent behavior, particularly the vehicle's stability and performance, the investigations have considered the complete vehicle system from launch to impact and work out a compromise between experiment goals and technical feasibility.

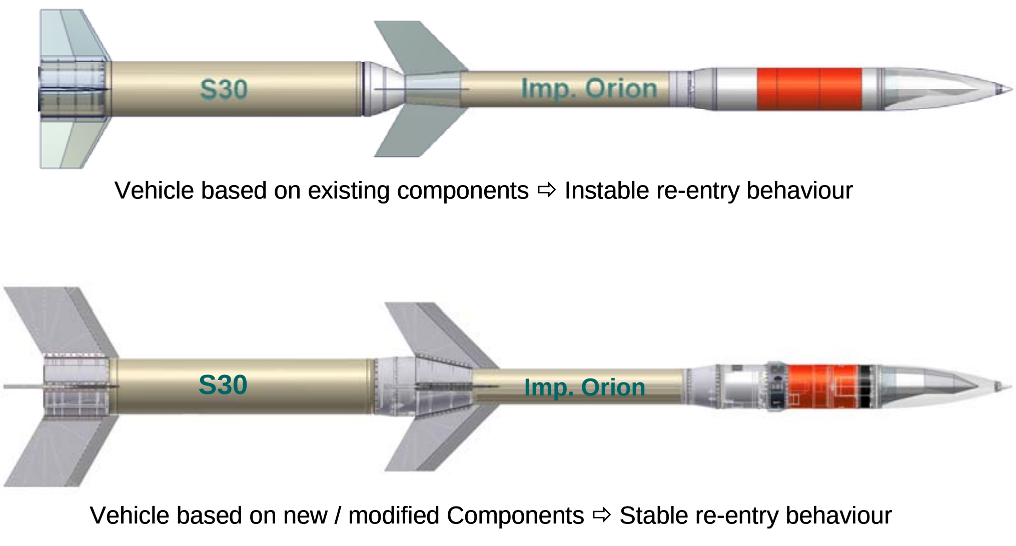

Fig. 8: Aerodynamic changes of the launcher layout.

Further, the obtained surface values of pressure, temperature and heat flux have been basis for the choice of the flight instrumentation, the positioning of the sensors and for the layout of structure and TPS. In particular, pressure ports and heat flux sensors have been placed almost along the configuration X-Z plane, allowing capturing the main flow features on wedged compression corner. Pre-flight data for a flight Mach number of M=7.5 anticipates the changes in flow topology upon the shock-wave boundary-layer interaction correspond to laminar or turbulent boundary-layer flows as is shown in **Fig. 9**. The calculations cover the complete Reynolds number range of 70.000 < Re < 3.500.000, while depending on the altitude the boundary layer has been assumed to be fully laminar or fully turbulent.

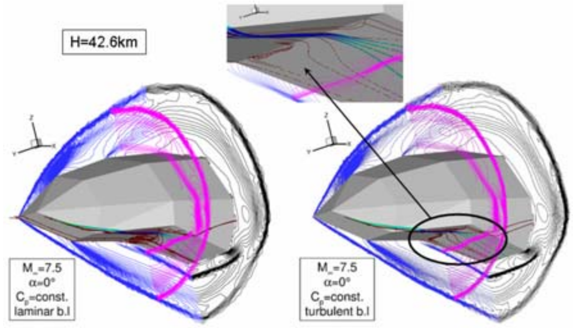

Fig. 9: Pre-flight computations on the impact of the boundary-layer state on near surface flow field.

Summarizing, SHEFEX-I is not a new reentry vehicle but a hypersonic experiment on a new concept for future aerospace vehicles which should enhance vehicle performance due to the hypersonic exploitation of the aerodynamic sharp configuration concept. The main purpose of the flight experiment has been the validation of analytical predictions, allowing the cross checking with ground testing data and acquisition of flight testing on some structural design features like sharp leading edges and seal system for the development of future space transportation systems. Designed following an unconventional way, is SHEFEX-I one of the first concepts resulting from a multidisciplinary approach. Finally, SHEFEX-I is a pathfinder project of an emerging strategy on a kind of sky based facility, as economical way to acquire knowledge in the physics of hypersonic fly.

III. Vehicle Architecture and Mission

The SHEFEX launcher was a two-stage solid propellant sounding rocket system conceived primarily for ballistic microgravity experiments. The launch vehicle consisted of a Brazilian S30 motor as first stage and an Improved Orion motor as second stage. During the ascent the facetted forebody was protected by an ogive nose cone. Between the facetted SHEFEX experiment and the second stage were two cylindrical modules which housed the recovery system, the main electronics, the data acquisition devices, the power supply and the cold gas ACS (**Fig. 10**). Since the facetted body has no control devices the second stage remained attached to the forebody until the end of the experiment to provide flight stability through its fins.

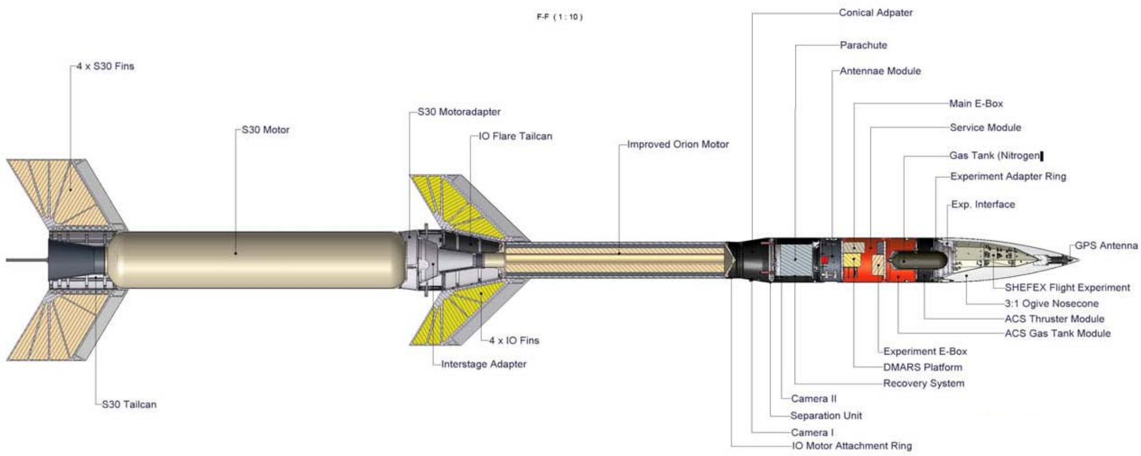

Fig. 10: SHEFEX-I Launcher configuration

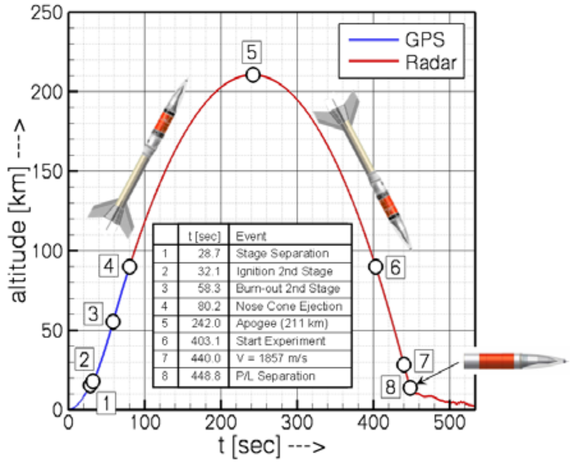

Fig. 11: SHEFEX-I radar and GPS trajectory and time events.

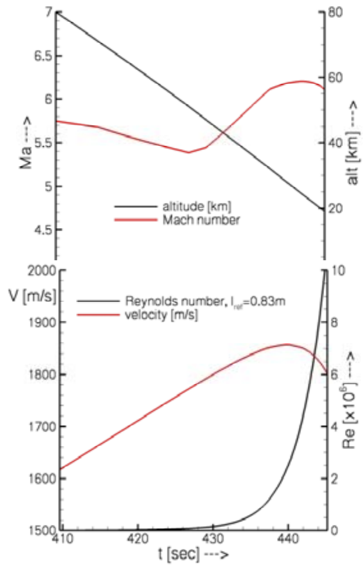

Fig. 12: SHEFEX-I flow conditions during the atmospheric re-entry.

The flown mission is schematically sketched in **Fig. 11**. During flight the vehicle reached an apogee of 211 km which is 35 km lower than the expected altitude. The total flight time was 550 seconds, comprising almost 40 seconds of experimental time for the atmospheric re-entry between 90 km to 20 km. The originally planned altitude of approx. 246 km could not be reached because the mass of the motors was 25 kg higher than expected [33]. The beginning of the SHEFEX experiment was defined to be at an altitude of 90 km. The first atmospheric effects on the acceleration sensors could be observed at 80 km. Here the pitch and yaw angles started to oscillate and unfortunately, contrary to the originally planned flight without spin (no yo-yo de-spinning maneuver was planned since the ACS should eliminates any remaining spin), the roll rate started to increase as the atmospheric influence starting to build up. The vehicle finally achieved a stable flight attitude with a decreasing ellipse-shaped precession around the flight vector, corresponding to maximum values of yaw and pitch rate well below 0.1Hz and maxima in roll rate of almost 1.5Hz short before the end of the experiment. The flight velocities during the atmospheric decent were around 1700 m/s. The maximum velocity of 1857 m/s during re-entry was measured at an altitude of 28.3 km. Originally projected was a velocity in the order of 2000 m/s. The Mach number is relatively constant at a value of approx. 5.6 from 100 km down to 50 km. Then the Mach number increases up to its maximum value 6.2 at 26 km while the Reynolds number increases up to $10x10^6$ at stage separation (**Fig. 12**).

As described more depth in [34] the main sensor for the attitude control has been a stabilized inertial platform. It provides accurate attitude and orientation information. The platform also provides the attitude control signal for the automatic cold gas ACS system. A main microcontroller performs the data acquisition from the platform, the experiment electronics, GPS receiver, ignition and recovery system and the general housekeeping data. The controller also assembles the various data packets in a PCM frame for telemetry and receives the telecommands via dual redundant receivers for correction of maneuvers or selective enabling of lateral and roll axes control. The baseline control strategy is a predefined maneuver according to the predicted nominal trajectory. In the event of a trajectory significantly differing from the nominal, the calculation of the flight vector is done from real-time radar data and the transmission of the maneuver correction is then uploaded via telecommand. The data from DMARS, Radar, GPS, the two onboard cameras and the housekeeping sensors enable a detailed flight mechanic description of the complete flight and together with the instrumentation of the facetted payload a direct comparison in a numerical post-flight analysis is possible. Two ground station, one at the launch place, Andoya, and one as back-up in Alomar, have been used to down load telemetry (GPS and ACS) and video-stream data. Data obtained from 59 sensors, pressure transducers, thermocouples, and heat flux sensors (**Fig. 13**) distributed on the surface of the forebody, constitutes the core of the row-data available for aero- and aerothermodynamic analysis [35]. Unfortunately, as a consequence of the differences in performances between the planned and the real flight-path, the recorded data for the upper part of the trajectory, 90km <H< 60km, is invalid since the magnitude of the measured values there have been lower than the minimum sensitivity of the instrumentation.

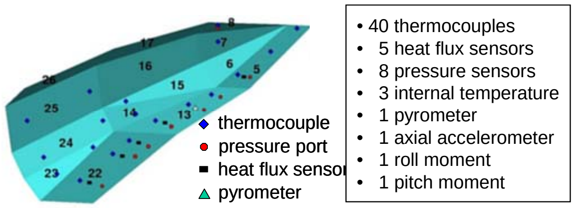

Fig. 13: SHEFEX-I payload instrumentation.

IV. Post-flight Analysis

Pre-flight aerodynamic investigations including computational fluid dynamics (CFD) analysis of the re-entry stability, indicated a major problem in the pitch axis which is the axis containing the experiment asymmetry. During descent into denser layers of the atmosphere, the experiment body induces lifting forces which considerably exceed the original control capability of the attitude control system and leads to an uncontrollable increase in AoA. An extensive aerodynamic study was hence performed to find an optimum position for both, the centre of pressure (CoP) and the centre of gravity (CoG). **Figure 14** points out that the flight value of the angle of attack of α=0.75° is very close to the numerical result of α=0.88° obtained when re-computing the aerodynamic behavior for the resulted flight Mach number of M=6.16. Such result confirmed that today, for cold hypersonic flow problems, one can aerodynamically design a flight vehicle in shorter time and requiring less budget, almost based entirely on CFD since the capabilities offered by the numerical tools clearly surpass those provided by wind tunnels.

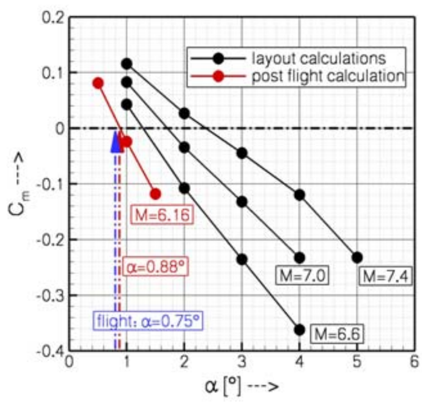

Fig. 14: Comparison of the trimmed angles of attack, M=6.16, alt=20km, flight CoG.

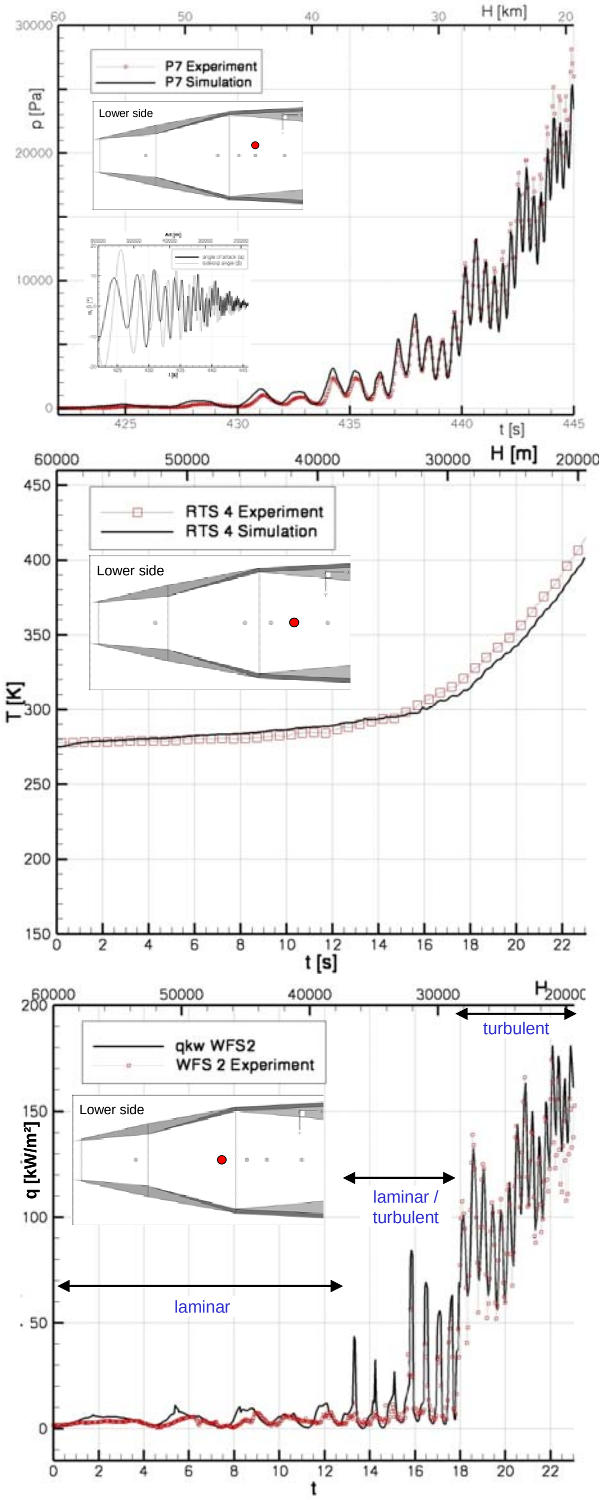

Fig. 15: Example of CFD post-flight reconstruction of pressure (top), temperature (mid) and heat-flux (bottom).

For the assessment of the uncertainties, numerical rebuilding in former projects pointed out that the applied standard atmospheres may have a significant influence on the aerodynamic behavior on the aerodynamic coefficients. The effects on pressure coming from changes in the standard atmosphere may be larger than those provided by turbulence models, the chosen boundary layer state and / or the chemistry effects. For the SHEFEX-I experiment, a maximum uncertainty of 10% in pressure resulted which is relatively large if the difference between laminar and turbulent pressure distribution is considered [36]. Further, a couple of careful investigations of the accuracy of the data recording and of the potential sources of errors for the numerical simulations like influence of sideslip, differences between 2D and 3D simulations, impact of ablation, influence of atmosphere model, and so on have been carried out [37]. It can be summarized that the payload data and the flight data recording during the experiment deliver synchronous values which are very important for the post-flight analysis using numerical simulations. The sensitivity study shows that there are a lot of parameters and modeling conditions which have an impact on the numerical analysis of SHEFEX-I. Advanced aerothermodynamic analyses have been also performed to determine the influence of sensor response time, sensor position, and sensor projection [38]. The response time of the mantle thermocouple plays a very important rule for the post-flight analysis and has to be measured in future experiments. Besides the analysis shows that sensor position within the TPS affects enormously the temperature measurement. A small change of the position causes a distinct temperature variation. The heat flux distribution is analyzed on the basis of two different cases with respect to the mounting condition of the senor (with and without projection). The results pointed out that the projection length of the heat flux sensor is an important parameter which has to be taken into account during the analysis. Finally, numerical simulations using a time accurately coupled fluid-/thermal-structure interaction calculation have been done for the re-entry range from 60km to 20km, representing totally 23 flight seconds. Due to restricted computational resources, the nonlinear thermal and mechanical fluid-structure interactions has been treated, as like in the past, time-accurately with view to the structure and as steady state problem for the fluid. Using a moderate coarse grid for the payload of 3.5×10^6 points, 1.5×10^6 iterations have been necessary to complete the numerical simulation. Each 0.05 flight second required 460 computational time steps for the flow solution while convergence of the solution has been achieved after 1000 fluid-structure coupled iterations. The numerical data generated amount almost 4TBytes. **Figure 15** shows exemplarily that considering all investigated sources of errors the numerical results for pressure, temperature and heat flux distributions of the analyzed re-entry range offer a very good agreement with the experimental flight data. Indeed, such results are a clear evidence of the potential of multidisciplinary design and analysis CFD tools for the conception and design of future hypersonic flight vehicles.

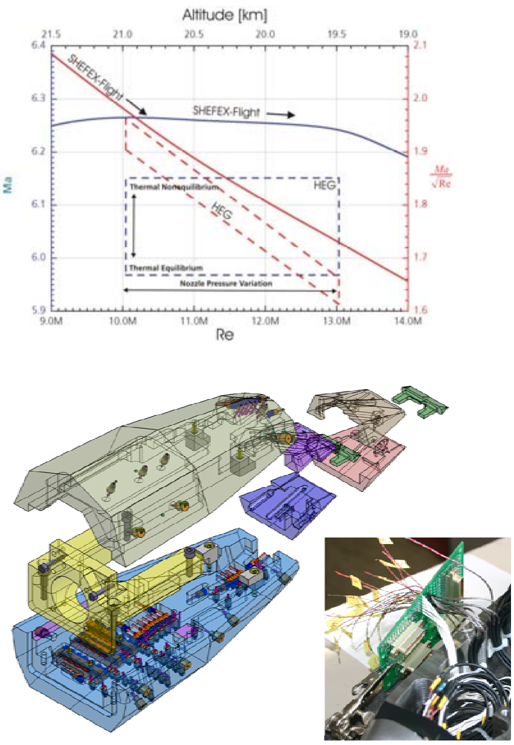

Fig. 16: Comparison between SHEFEX flight-path and HEG operational range (top). Model instrumentation (bottom).

Current activities are related to the post-flight rebuilding by means experimental data in the high enthalpy facility HEG of DLR. The flight path of flown by SHEFEX-I fits almost rather well inside the operational range of the facility, allowing numerical and ground based data correlations with flight measurements (**Fig. 16**). A hardly instrumented wind tunnel model, scale 1:2, has been manufactured and will go for tests at the time of the present paper. Instrumented with 164 gages (pressure ports, thermocouples and heat flux sensors), the SHEFEX-I wind tunnel model mimics al the sensors using during flight plus several more allowing a detail analysis of the flow.

V. Conclusions

The given paper summarised the SHEFEX-I hypersonic flight experiment of the German Aerospace Centre, DLR, and the status of the post-flight analysis. It gave a detailed overview about the overall philosophy and the background of the experiment. For the cold hypersonic regime, the study demonstrates the capabilities and accuracy that offer today CFD tools for the design and analysis of flight machines. The results envisage the future of the multidisciplinary numerical simulation and its predominantly increasing role in aerospace for virtual design and qualification of hypersonic vehicles as the computational resources continuing growing, at least, at the same rate as today and / or in the past. Additionally, the paper highlighted the main outcomes and drawbacks of the flight. The up today evaluation of the flight data emphasize that relatively straightforward experiments like SHEFEX-I with their enormous amount of scientific data allows to reduce the risk on developing future demonstrator vehicles. These types of flight experiments are an essential source of knowledge at a budget level well below the required in most of cases for experimental campaigns in so-called "industrial facilities". Indeed, SHEFEX-I has been a pathfinder project of an emerging strategy on a kind of sky based facility, as economical way to acquire knowledge in the physics of hypersonic flow and re-entry technology. Today SHEFEX has evolved in a long term hypersonic-flight program of DLR where the currently prepared SHEFEX-II mission is not a re-flight of the first experiment but a vehicle re-designed and extended by an active control system, which will allow active aerodynamic control during the re-entry phase.

VI. References

[1] J.M.A. Longo, K. Hannemann, V. Hannemann. The Challenge of Modeling High Speed Flows. (Invited Lecture) Proceedings of the Federation of European Simulation Societies, EUROSIM 2007 Symposium, Lujbiana, Slovenien, September 2007.

[2] S. Labbe, L. Perez, S. Fitzgerald, J.M.A. Longo and M. Rapuc. X-38 NASA/DLR/ESA-Dassault Aviation Integrated Aerodynamic and Aerothermodynamic Activities. Aerospace Science and Technology 3 (1999) 485-493.

[3] I.A. Johnston, M. Weiland, J. Martinez-Schramm, K. Hannemann, J.M.A. Longo. Aerothermodynamics of The ARD: Post-flight Numerics and Shock-Tunnel Experiments. AIAA Paper 2002-0407, January 2002.

[4] K. Hannemann, J.M.A. Longo. Aerothermodynamik und Wiedereintritt. Handbuch der Raumfahrttechnik, Edt. Profs. W. Ley u. K. Wittmann, Carl Hanser Verlag München Wien, 2007, pp. 114-128.

[5] J.M.A. Longo. Aerothermodynamics – A critical review at DLR. Aerospace Science and Technology 7 (2003) 429-438.

[6] "Waverider Results", Aviation Week & Space Technology, November 10, 2003, pp. 17.

[7] J. Steelant, A. Mack, K. Hannemann and A.D. Gardner. Comparison of Supersonic Combustion Tests with Shock Tunnels, Flight and CFD. Proceedings of the 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Sacramento, July 2006.

[8] R. Kimmel. Aerothermodynamic Issues for HIFiRE. AIAA Paper AIAA-2008-4034, June 2008.

[9] H. Weihs, J.M.A. Longo, J. Turner. The Sharp Edge Flight Experiment SHEFEX II, a Mission Overview and Status. AIAA 15th Space Planes and Hypersonic Systems and Technologies Conference, April 2008.

[10] H.J. Allen, A.J. Eggers. A Study of the Motion and Aerodynamic Heating of Missiles Entering the Earth's Atmosphere at High Supersonic Speeds", NACA RM-A53D28, Aug. 1953.

[11] J. Reuther et al. A Reusable Space Vehicle Design Study Exploring Sharp Leading Edges. AIAA paper 2001-2884, June 2001.

[12] J. Arnold, S. Johnson, P. Wercinski. SHARP: NASA's Research and Development Activities in Ultra-High Temperature Ceramic Nose Caps and Leading Edges for Future Space Transportation Vehicles. Proceedings of the 52nd International Astronautical Congress, IAF paper 01-V.5.02, France, October 2001.

[13] D. Strohmeyer, Th. Eggers, M. Haupt. Waverider Aerodynamics and Preliminary Design for Two-Stage-to-Orbit Missions, Part 1. Journal of Spacecraft and Rockets, Vol. 35, No. 4, July-August 1998, pp. 450-458.

[14] Th. Eggers, Ph. Novelli. Design Studies for the Development of a Dual Mode Ramjet Flight Test Vehicle. Deutscher Luft- und Raumfahrtkongress, DGLR Jahrestagung, Berlin, Germany, Sept. 2001.

[15] K. Hozumi, S., Nomura. Experimental Investigation of Aerothermal Problems Associated with Hypersonic Flight of HST. ICAS paper 92-4.9.3, Sept. 1992.

[16] Th. Eggers, Ph. Novelli, M. Haupt. Design Studies of the JAPHAR Experimental Vehicle for Dual Mode Ramjet Demonstration. AIAA paper 2001-1921, April 2001.

[17] A. Gülhan, B., Esser, U., Koch. Experimental Investigation of Reentry Vehicle Problems in Arc-Heated Facilities. Journal of Spacecraft and Rockets, Vol.38, No. 2, March-April 2001, pp. 199-206.
[18] A. Mack and R. Schaefer. Fluid Structure Interaction on a Generic Body-Flap Model in Hypersonic Flow. Journal of Spacecraft and Rockets, Vol. 42, No. 5, September-October 2005, pp. 769-779.
[19] D. Schwamborn, Th. Gerhold, R. Heinrich. The DLR TAU-Code: Recent Applications in Research and Industry. Proceedings of the European Conference on Computational Fluid Dynamics, ECCOMAS CFD 2006.
[20] J.M.A. Longo. SHEFEX-I a Low Cost Highly Efficient Flight Experiment. Proceedings of the 5th European Symposium on Aerothermodynamics for Space Vehicles, Cologne, Germany, November 2004.
[21] J.M.A. Longo. Modeling of Hypersonic Flow Phenomena. (Invited Lecture) RTO-EE-AVT 116 "Critical Technologies for Hypersonic Vehicle Development Technology", AC/323(AVT-116)TP/90, Research and Technology Organization (NATO), ISBN 92-837-1138-6, 2006.
[22] G.S. Settles, L.J. Dodson. Hypersonic shock/boundary-layer interaction database. NASA CR 177577, 1991.
[23] G.S. Settles, L.J. Dodson. Hypersonic turbulent boundary layer and free-shear layer database. NASA CR177610, 1993.
[24] G.S. Settles, L.J. Dodson. Hypersonic shock/boundary-layer interaction database: new and corrected data. NASA CR 177638, 1994.
[25] G.S. Settles, L.J. Dodson. Supersonic and hypersonic shock / boundary-layer interaction data-base. AIAA J 1994,32(7): 1377–83.
[26] J.M.A. Longo, R. Radespiel. Flap Efficiency Versus Flap Heating of a Winged Reentry Vehicle - A Challenge For The CFD -. Journal of Spacecraft and Rockets Vol. 33, No. 2, 1996, pp.178-184.
[27] H. Lüdeke, R. Radespiel, E. Schülein: Simulation of Streamwise Vortices at the Flap of Re-entry Vehicles, Aerosp. Sci. Technol. 8 (2004) 704-714.
[28] M. Khalid, A. Dujardin, P. Hennig, L. Leavitt, F. Leopold, M. Mendenhall, S. Prince. Turbulence Model Studies to Investigate the Aerodynamic Performance of a NASA Dual Control Missile at Supersonic Mach Numbers. CASI, Vol. 51, No. 4, December 2005
[29] R. Adeli, J.M.A. Longo, H. Emunds. Flow Field Study of a Supersonic Jet Exiting into a Supersonic Stream. Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 92: New Results in Numerical and Experimental Fluid Mechanics V, Springer 2006, pp. 160-167.
[30] O. Bozic, J.M.A. Longo, H. Otto. Shock-Wave Boundary-Layer Interaction (SWBLI) around Open Flaps of EXPERT Capsule and their Consideration on Heat and Mechanical Loads under Critical Re-entry Flow Conditions. Proceedings of the 57th International Astronautical Congress, IAC Paper IAC-06-D2.5.02, Valencia, Spain, October 2006.
[31] V. Togiti, V., H. Lüdeke. Computation of Supersonic Base Flow Using Detached Eddy Simulation. STAB 2006, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), Springer-Verlag, 2007.
[32] Th. Eggers. Aerodynamic and Aerothermodynamic Layout of the Hypersonic Flight Experiment SHEFEX-I. Proceedings of the 5th European Symposium on Aerothermodynamics for Space Vehicles, Cologne, Germany, November 2004.
[33] J. Turner, M. Hörschgen, A. Stamminger, P. Turner. SHEFEX Hypersonic Reentry Flight Experiment, Vehicle and Subsystem Design, Flight Performance and Prospects. AIAA Paper AIAA-2006-8115, 2006.
[34] J. Turner, M. Hörschgen, P. Turner, J. Ettl, W. Jung, A. Stamminger. SHEFEX – The Vehicle And Subsystems for a Hypersonic Re-Entry Flight Experiment. 17th ESA Symposium on European Rocket and Balloon Programmes and Related Research, Sandefjord, Norway, June 2005.
[35] A. Gülhan, G. Requardt, F. Siebe, U. Koch, B. Esser. Instrumentation and In-Flight Data of the SHEFEX Flight Experiment, Proceedings of the 1st International ARA Days on Atmospheric Re-Entry Systems, Missions and Vehicles, Arcachon, France, June 2006.
[36] Th. Eggers, T. Barth. Aerodynamic Analysis of the DLR Flight Experiment SHEFEX-I. AIAA paper AIAA-2006-8027, 2006
[37] T. Barth. Aero- and Thermodynamic Analysis of SHEFEX-I. Engineering Applications of Computational Fluid Mechanics Vol. 2, No. 1, 2008.
[38] T. Barth. Advanced Aerothermodynamic Analysis of SHEFEX-I. Proceedings of the 1st CEAS European Air and Space Conference 2007, Berlin, Germany, September 2007.