

THE CHALLENGE OF MODELING HIGH SPEED FLOWS

J.M.A. Longo¹, K. Hannemann², V. Hannemann²

German Aerospace Center (DLR)

¹D-38108 Braunschweig, Lilienthalplatz 7, Germany

²D-37073 Göttingen, Bunsenstraße 10, Germany

Jose.Longo@dlr.de (José Longo)

Abstract

Current and expected developments in space transportation have led to growing interest in new space vehicles. These new vehicles require essential improvements over current vehicles in order to ensure economic viability and to fulfill mission and safety constraints. The size and complexity of this problem has led to growing importance of numerical methods for design and optimization involving all disciplines as well as the optimal use of all technical potentialities is necessary. The material presented here reviews the growth and advances achieved at DLR in the last years. The status of the physical modeling, code development issues such as algorithms, grid generation and validation strategy is provided. Viscous, high speed unsteady flows are still restricted to simple problems due to a strong demand in computer resources. The short coming to obtain adequate data for code validation in high speed flow problems is discussed. It turns out that little progress has been realized in numerical modeling since past major difficulties to obtain reliable experimental data in the high enthalpy flow environments still remain. On the other side, CFD based multidisciplinary analysis is emerging as a key discipline in aerospace design. Supported by a continuous, almost linear, grows in computer capacity and performance, this new procedure to conduct configuration analysis is paying off its way. A number of selected applications of multidisciplinary problems with complex physics are presented.

Keywords: Hypersonics, Multidisciplinary Analysis, Chemical Models, Compressible Flows, Turbulence Models.

Presenting Author's biography

José M.A. Longo received his degree in aeronautical engineering from the Catholic University Córdoba of Argentina and his Doctor degree from the Technical University Braunschweig of Germany. Dr. Longo's main scientific interest concern modeling, simulation and design in aerodynamics, gas dynamics and aerothermodynamics. He became head of the Spacecraft Branch of the Institute of Aerodynamics and Flow Technology of DLR at Braunschweig in the year 2000. He has managed several space programs at national level as well as for the European Space Agency; serving also on several scientific advisory boards.



1 Introduction

Computational Fluid Dynamics (CFD) has now matured to the point that it is widely accepted as a key tool for aerospace design. Algorithms have been the subject of intensive development for the past three decades. The principles underlying the design and implementation of robust schemes which can accurately resolve shock waves and contact discontinuities in compressible flows are now well established. However, the effective use of CFD to more complex applications, require more sophisticated algorithms, particularly for high speed flows, since one of the key problems here is the treatment of multiple space and time scales. These arise not only in turbulent flows, but also in many other situations such as chemically reacting flows, combustion, flame fronts and plasma dynamics. Also the strong need to validate the computer codes by comparison with experiments is for high speed flows not an easy task due to the wide spectrum of physical conditions to reproduce in the experiments.

In spite of that, computational simulation is becoming the principal tool of the aerospace design process 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 multi-disciplinary analysis and multi-disciplinary 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 like 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 in which CFD are coupled with the computational analysis of other disciplines of the design is playing an increasingly important role.

2 Scope of the High Speed Flows

Within recent years the development of high speed flight vehicles has brought a number of new design problems into prominence. Most of these problems arise because of extremely high flight velocities, and are characteristically different in some way from the problems which arise for passenger transport in aeronautics. The term “hypersonic” is used to distinguish flow field phenomena and problems appearing at flight speeds far greater than the speed of sound from those phenomena appearing at flight speed which are almost subsonic or moderately supersonic. These new characteristic hypersonic features may be roughly divided into those which arise because the flight Mach number Ma (a non dimensional number $Ma = \text{flow velocity} / \text{speed of sound}$) is large and those which arise because the energy of the flow is large. If the gas involved is rarefied, so that the mean

free path is not negligibly small compared to an appropriate characteristic macroscopic scale of the flow field, the continuum mechanical approach needs to be replaced by the kinetic theory. Rarefied gas flows are encountered in flight at very high altitudes. The changes in the flow physics for a flight vehicle as a function of flight altitude and speed are summarized in Fig. 1 [1].

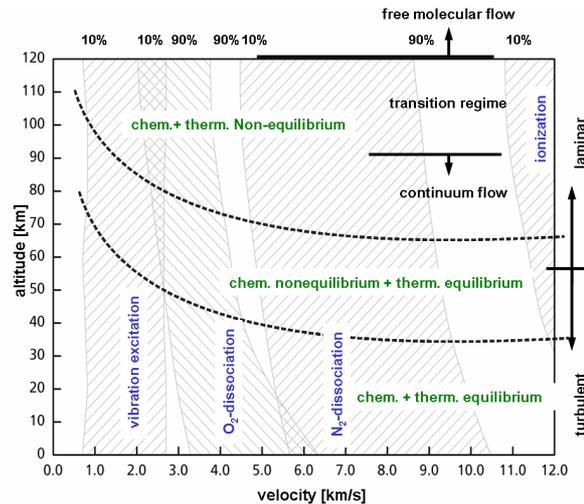


Fig. 1: High speed flow physical / chemical features

Shock waves are the dominant feature of the high speed flows and thus, the techniques of linearization of the flow equations and the use of mean-surface approximations for boundary conditions have a vanishing range of applicability. The entropy gradients produced by curved shock waves make the classical isentropic irrotational approach inapplicable. Boundary layer interaction phenomena become important in hypersonic flow. The new features of a physical or chemical nature appearing here are mostly connected with the high temperatures generally associated with the extremely strong shock waves present in such flows as schematically presented in Fig. 2.

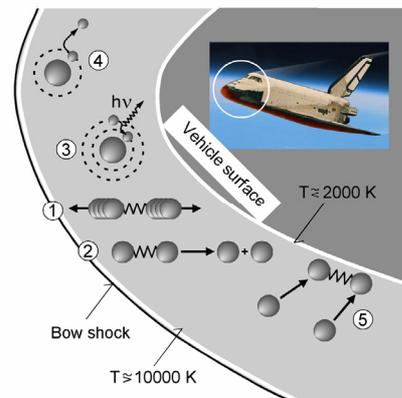


Fig. 2: Schematic description of the high temperature effects behind a compression shock of a vehicle performing an atmospheric entry

At high temperatures in air or in other gases of interest vibrational degrees of freedom in the gas molecules may become excited (1), the molecules may dissociate into atoms (2), the molecules or free atoms may ionize (3), and molecular or ionized species unimportant at lower temperatures may be formed (4). Any of these processes require characteristic times and relaxation phenomena appear. At sufficiently high temperatures the gas may radiate, providing a method for the transfer of energy which is negligible at lower temperatures. With the presence of different molecular or ionized species in large gradients of concentration, temperature and pressure, the processes of diffusion become important. Finally, there are phenomena connected with the interaction of gas particles (or dust particles) with solid surfaces which require for example the consideration of the accommodation coefficients of rarefied gas theory, catalytic recombination of dissociated atoms on the surface, and ionization of the surface material (5).

While the knowledge of the global pressure field is necessary for any estimation of local physical effects, there is a strong interaction between both. Indeed, local physical phenomena may not only strongly influence local details of hypersonic flow fields, but in extreme cases might control the nature of the entire flow, like in the case of an interplanetary re-entry as is schematically indicated in **Fig. 3**. There the flow field of the space vehicle behind the bow shock is characterized by several local phenomena which dominate over the global features. Examples of these local phenomena are viscous interaction; laminar-turbulent transition; radiation and ablation; leeside and base flows; real gas effects and low density flows among others [2].

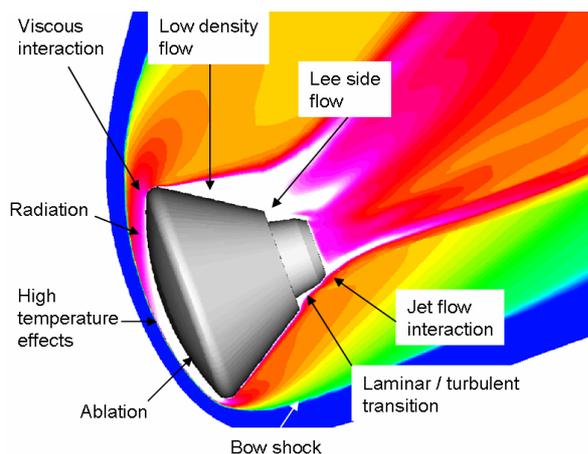


Fig. 3: Physical phenomena around and on the surface of a vehicle performing an atmospheric entry

In the following sections key issues on mathematical and physical modeling, numerical modeling and validation data for hypersonic flows are given. Future areas of research to enhance accuracy, reliability, efficiency, and robustness are discussed based on a number of applications examples for hypersonic vehicles.

3 Mathematical Modeling

A gas flow may be modeled at either the macroscopic or the microscopic level. The macroscopic model regards the gas as a continuous medium and the description is in terms of the spatial and temporal variations of the familiar flow properties such as the velocity, density, pressure, and temperature. The macroscopic properties are the dependant variables in these equations, while the independent variables are the spatial coordinates and time. Those macroscopic properties may be identified with average values of the appropriate molecular quantities at any location in a flow. They may therefore be defined as long as there are sufficient numbers of molecules within the smallest significant volume of a flow. If these conditions are satisfied, the results from the molecular model can be expressed in terms of the familiar continuum properties. The Navier-Stokes equations provide the conventional mathematical model of a gas as continuum fluid. These equations express the conservation of mass, momentum, and energy in a flow. Further, as soon as Mach numbers above one are reached a shock can be part of the flow field solution, which is essentially a discontinuity in all flow properties. Contact discontinuities are exact solutions of the Euler equations, i.e. the Navier-Stokes equations without viscosity.

With increasing Mach number the compressibility of the fluid has to be taken into account, so that conservation equations for mass, momentum and energy have to be solved simultaneously. Special attention should be put on the conservation of mass for high temperature flow problems since it means the conservation of the resulting species once the gas dissociate. Further, are required to close the equations the fluid properties expressed in an equation of state and expressed by the transport properties like viscosity and thermal conductivity. However it must be point out that the conservation equations do not form a determinate set unless the transport properties, shear stresses and heat flux can be expressed in terms of the lower-order macroscopic quantities. It is the failure to meet this condition, rather than the breakdown of the continuum description, which imposes a limit on the range of validity of the continuum equations. Indeed, the transport terms in the Navier-Stokes equations of continuum gas dynamics fail when gradients of the macroscopic variables become so steep that their scale length is of the same order as the average distance traveled by the molecules between collisions, the so called mean free path. The nondimensionalised parameter used to characterize the continuum and rarefied flow regime is the Knudsen number Kn ($Kn = \text{mean free path of particles } \lambda / \text{characteristic length scale } L$). A wrong practice is to try to define a single overall Knudsen number for the complete flow field. The correct approach is to specify a local Knudsen number using as length scale of interest a macroscopic local

gradient, i.e. $L = \rho / (d\rho / dx)$ and the local mean free path. The error in the Navier-Stokes results is significant in flow regions with local Knudsen number exceeding the value 0.1, while for values ~ 0.2 the continuum model must be replaced by a molecular one.

Assuming a perfect gas, a stationary flow field can be characterized by a set of 3 non dimensional numbers, namely the Mach number Ma , the Reynolds number Re ($Re = \text{density} * \text{velocity} * \text{length scale} / \text{dynamic viscosity}$) and the Prandtl number Pr ($Pr = \text{dynamic viscosity} * \text{specific heat capacity at constant pressure} / \text{thermal conductivity}$). While the Prandtl number is usually kept constant, the Mach and Reynolds numbers describe the global motion of the fluid. Focusing at first on the flow velocity, high speed flow is linked to high Mach and high Reynolds numbers without to forget the unsteady character that the problem could show. A couple of examples of these flows are presented in Refs. [3-5]

3.1 Turbulence Modeling

Turbulent flow is one of the oldest unresolved fluid dynamics problem and a major source of uncertainty in the prediction of heating in hypersonic flows. Its importance for vehicle design lies in the fact that turbulent thermal loads, as displayed in Fig. 4, may be 3 times larger than laminar ones.

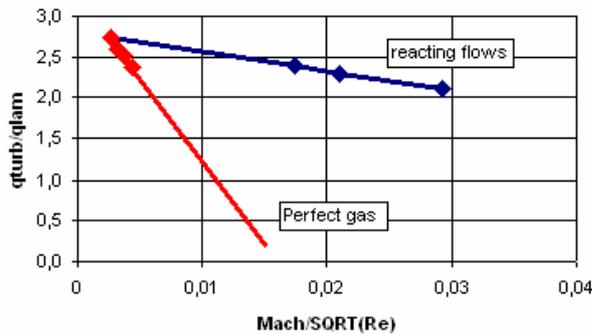


Fig. 4: Turbulent/laminar heat flux ratio on vehicle surface as function of flight Mach and Reynolds numbers

Although the Navier-Stokes equations are valid to describe turbulent and transitional flows without any additional turbulence model, the so-called direct numerical simulations (DNS) for complex 3D configurations at high Reynolds numbers is not feasible even with the most powerful computers of today. Therefore, transition and turbulence models have to be incorporated into the simulation process. Apart for algebraic turbulence models, the result of this is that the number of equations to be solved is increased depending on the chosen model. The number of additional equations ranges from one using one-equation turbulence models up to 7 for Reynolds stress models. The basis for turbulent flow computations are the so-called Reynolds Averaged Navier-Stokes equations (RANS). In general RANS

methods attempts to solve the time-averaged flow, i.e. all scales of turbulence must be modeled. In general, the development of the turbulence models is first of all related to incompressible flows (the majority of technical flows) neglecting the energy equation and is subsequently applied and enhanced for compressible flows over adiabatic walls. However, there exists no specific turbulence model for hypersonic flows. The adjustment of the available turbulence models for hypersonic flows is an ongoing work. At DLR several one and two equations turbulence models are currently applied in an ongoing benchmark study for selected supersonic and hypersonic flows past relatively complex 3D configurations.

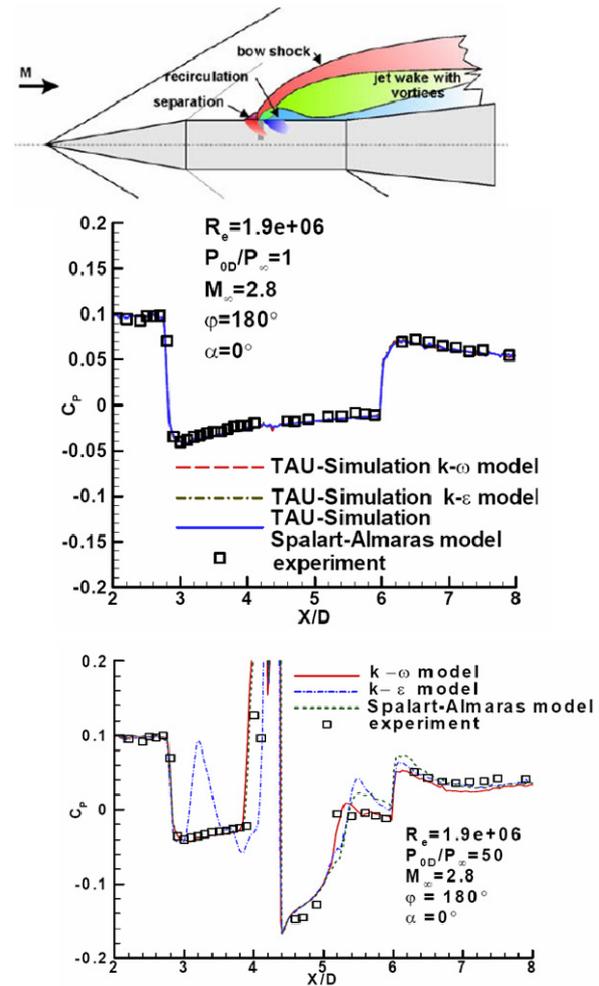


Fig. 5: Computed surface pressure distribution for a vehicle without and with active reaction control (RCS) using different turbulence models. Top: scope of the simulation. Middle: RCS off. Bottom: RCS on.

For external flows, the study shows that the simulations obtained with the 2 equation $\kappa-\omega$ turbulence model compare better with the experimental data than other models. High angle of attack, $AoA=24deg$, turbulent supersonic flows with $Ma=1.75$ and $Re=6.10^6$, around a twice cruciform configuration missile are presented in Ref. [6]. In spite of the fact that none of the applied models accurately

reproduces the experimental forces, the results with the Wilcox $\kappa-\omega$ model compare better with the experiments. Jet flow interaction with a supersonic external free stream flow for a missile at $Ma=2.8$, $Re=2.10^6$ is reported in Ref. [7]. As Fig. 5 shows, the study puts in evidence the superiority of $\kappa-\omega$ to predict pressure distributions for separated flows, while for attached flows almost all turbulence models provide the same answer. Furthermore while it is still a research topic how to apply wall functions to temperature boundary layers in compressible flow, the investigation discussed in Ref. [8] shows that for hypersonic low altitude flows at $Ma=6$, the experimental distribution of heat fluxes is better captured by using a $\kappa-\omega$ model. This conclusion remains valid also for a hypersonic high enthalpy flow at $Ma=8$ and high angle of attacks $AoA=45deg$, around a re-entry glider presented in Ref. [9]. Here also the heat flux distribution obtained with a $\kappa-\omega$ turbulence model is in better agreement with the experimental values than all other models.

Internal flows seems to be less sensitive than external flows to turbulence modeling. In such cases one equation turbulence models seem to perform equally well than two equations models as is shown in Fig. 6. The computed pressure distribution along the bottom wall of a supersonic combustion chamber using one and two equation turbulence models are rather similar.

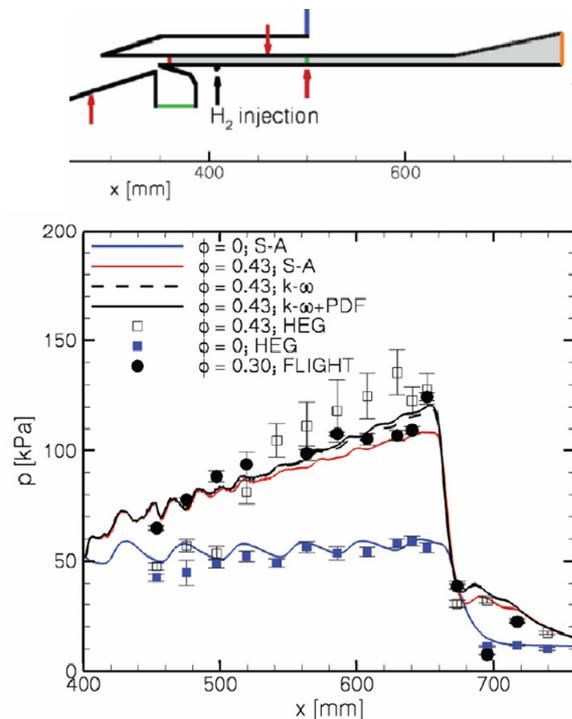


Fig. 6: Pressure distribution along the bottom wall of a supersonic combustion chamber. Top: Sketch of the simulation. Bottom: Numerical and experimental values (HEG: ground facility; S-A: Spalart-Allmaras turbulence model; PDF:Probability-Density-Function)

In some cases, low sensitivity is found with respect to the type of turbulence model used. The numerical investigations for a generic compression intake reported in Ref. [10] are performed at flow conditions typical for a scramjet combustion chamber with a static temperature of $T=1300K$, 1 bar static pressure and a Mach number of $Ma=3$. It is demonstrated, that both mixing and combustion efficiency are significantly increased due to the turbulence of the incoming boundary layer but the impact of the type of model used to mimic turbulence is very low.

In general RANS models lead to good prediction of attached flows but they often fail to provide accurate results for separated flows in particular when the flow is unsteady since the large scales for those flows are very dependent on the geometry. Better results could be obtained with Large Eddy Simulation (LES), since such models attempt to model the smaller, more homogeneous scales, while resolving the larger, energy containing scales. However, the resolution needed for CFD investigations using Large Eddy Simulation models is also far too large in order to be used for practical problems. Only solutions for generic flat plate or ramp flows are available today. Present efforts are oriented to hybrid Large Eddy turbulence models, so-called Detached Eddy Simulation (DES) models, which combine the advantage of RANS models close to the wall and LES models in the core flow. In Fig. 7 numerical solutions of DES for a turbulent supersonic base flow at $Ma=2.46$ are presented. Base flows, a complex type of flow which usually occurs in the wake of missiles, rockets or vehicles at high angle of attack are very important since an accurate estimation and understanding of the effects in this region enables the possibilities to improve vehicles aerodynamic stability [11].

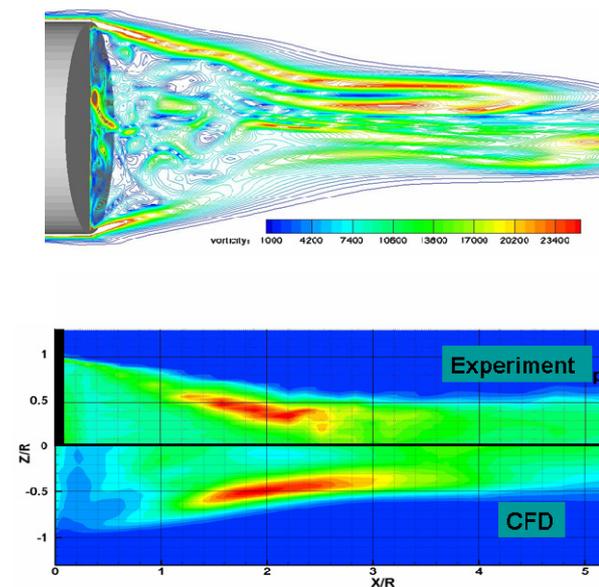


Fig. 7: Computed turbulent vorticity behind a cylinder in a $Ma=2.46$ flow as DES solutions (top). Bottom: Contours of turbulent kinetic energy, experimental (above) and numerical (below)

Another flow problem of major interest in high speed vehicle design is the so-called shock-wave boundary-layer interaction. The shock waves originated by the vehicle flying at hypersonic speed are the origin of interference phenomena resulting, first from the intersection of shocks, and second from their interaction with the boundary layer developing on the vehicle surface. The later is very often observed on the control surfaces of the vehicles, like aileron and flaps. Such interactions may induce separation of the boundary layer. In high-enthalpy hypersonic flows, the subsequent reattachment of the separated shear layer gives raise the heat transfer which can be far in excess of those of an attached boundary layer. The transition modeling process remains rather difficult since transition can be triggered by sources of different nature, involving in all cases complex mechanisms. On deflected control surfaces semi-empirical criteria are used to predict the onset of transition. Numerical results for a hypersonic re-entry glider at $Ma=6$, $AoA=40deg$ have been obtained in Ref. [12] by coupling the Navier-Stokes equations with an empirical transition correlation model. The laminar part of the flow is there analyzed for characteristic boundary layer parameters such as the momentum thickness based Reynolds number. The turbulence model used in this work is the Spalart-Allmaras model extended for predictions of heat transfer in compressible flows. The transition criterion is based on the momentum thickness based Reynolds number divided by the Mach number at the edge of the boundary layer. The results compare rather well with available experimental data.

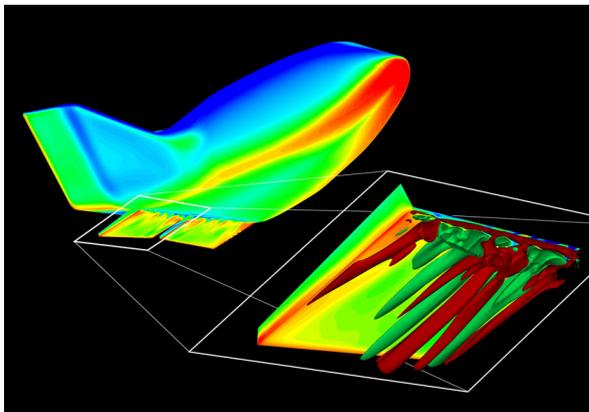


Fig. 8: Numerical simulation of Görtler vortices on a flap of a hypersonic vehicle at $Ma = 6$, $AoA = 40deg$

Further, as is demonstrated in **Fig. 8**, the resulting concave curvature in stream wise direction of a deflected control surface induces a centrifugal imbalance within the viscous layer resulting in stream wise vortices also called Görtler vortices. Under hypersonic conditions Görtler vortices, which can be induced with a very low amount of stream wise curvature, are responsible for a premature transition and 25% more heating on the controls surfaces than in a case of transitional flow without stream wise

vortices. Stream wise vortices on a compression ramp at hypersonic conditions, respectively supersonic boundary layers, have been numerical studied and compared with wind tunnel data in Ref. [13]. The results allow insight into the perturbed flow field, the associated flow topology and the influence of different flow parameters that affect the stream wise vortices. For laminar cases and cases with fixed transition vortex effects on the heat transfer are observed with and without artificial vortex excitation.

High enthalpy, low Reynolds number, transitional flow is a recently identified problem that requires more research effort to be understood. Reference [14] presents surface pressure and heat flux measurements on a re-entry hypersonic glider performed in a shock tunnel facility at two operating conditions at total specific enthalpies of $H=12$ and 22 MJ/kg and unit free stream Reynolds numbers varying from $Re=2$ to 4.10^5 . The comparison of the measured normalized surface pressures with the values predicted by CFD shown differences of 10% or less. The most significant difference of the computed and measured heat flux was obtained for a 30deg deflected body flap. For this configuration the measured heat flux on the flap was approximately twice as high as the computed values assuming fully laminar flow. Apart from an overshoot observed in the reattachment region, such a large discrepancy was not observed for the flap deflection angle of 20deg. The high heat transfer level on the 30deg body flap could be reproduced by performing a computation with laminar fore body flow and fixed transition in front of the hinge line. One reason for the high heat flux on the 30deg flap might be that compared to the 20deg flap the larger extent of the separated flow is responsible for the amplification of perturbations present in the flow. Very little knowledge is currently available in the literature about separated hypersonic and high enthalpy flows and more detailed investigations are necessary to clarify this point.

3.2 High Temperature Effects Modeling

High velocities of a fluid relative to a body surfaces bordering the flow field are equivalent to high kinetic energy which will be transferred into high temperatures if the flow is decelerated behind shocks or in boundary layers. For example, immediately behind the bow shock an atmospheric entry or re-entry vehicle, extreme non-equilibrium conditions are reached and translational temperatures may achieve values as high as 10^4 K. With increasing temperature, the temperature dependency of the heat capacities, viscosity and thermal conductivity of the gas mixture has to be modeled. Even the composition of the gas mixture becomes temperature (and pressure / density) depended, if the energy is high enough to enable chemical reactions. In case of air, dissociation of the molecules into atoms happens as well as formation of nitric oxide. At even higher temperatures the particles are ionized and the electrical conductivity of such

thermal plasmas makes the flow field sensitive for magnetic and electric fields. The calculation of the mixture properties can be derived from kinetic theory combined with quantum mechanical models for the particles, but this introduces many parameters (collision integrals and spectroscopic data) which have to be measured and approximated based on experiments. For example, a numerical strategy for the assessment of radiative energy transport in an Argon plasma flow is presented in [15]. The major drawback in such cases is the massive CPU resource needed. Also, both equilibrium and non-equilibrium of thermo-chemical processes cause subtle changes in the flow behavior and sonic line position, and can even lead to vehicle static instability. Indeed, for the Mars atmosphere, thermal excitation of vibrational modes in CO₂ could cause a rapid change of the effective ratio of specific heats [16], leading to displacements in sonic line locations.

The high temperature (thermal and chemical) effects do not follow a change of the temperature instantaneously but relax to an equilibrium state on their own time scales. Due to the high flow velocities the parts of the flow field, in which the gas mixture is in a non equilibrium state, can be significant even in stationary cases. The local state of the gas mixture depends on its history or way through the flow field. Therefore, additional equations for each important species, chemical and thermal kinetics and diffusion terms have to be implemented to account for the non equilibrium effects [17], where the number of additional equations to be solved can easily exceed the number of basic equations for laminar (5) or turbulent (6-12) flow. On a body boundary additional assumptions are made to close the equations. The simplest are called ‘fully catalytic’ (assuming equilibrium of the local gas mixture at the local temperature and pressure) and ‘non catalytic’ (vanishing gradients of the partial densities at the boundary). A more accurate and expensive modeling of the catalytic behavior via finite rate chemistry at the surface is possible [18], but most predictions are made with the two extreme cases to highlight the bandwidth of the catalytic influence. The exact catalytic properties of a technical, hot surface are difficult to determine and can vary during experiments due to chemical reactions between gas and solid phase. Even more complicated becomes the situation at a surface, if ablation has to be modeled with additional species leaving the surface into the fluid and the diminishing of the body material changes the geometry of the configuration.

The importance of a species is related to its role inside the chemical reaction mechanism and inside the mass and energy balance equations as well as depended on special technical interest like tracer species or environmental impact. Unfortunately the required species data is difficult, if not impossible to be measured directly, especially at higher

temperatures of several thousand Kelvin. Therefore, despite the deduction from first principles a validation of these models is still an ongoing work to be done in close cooperation with detailed experiments. Indeed, the numerical analysis of chemically reacting hypersonic flows is still subject to large uncertainties caused by the different models for relaxation processes. Reference [19] presents measurements using a cylinder as configuration, carried out at Mach number $Ma = 8$ and total specific enthalpies of $H=22.4$ MJ/kg and 13.5 MJ/kg using air as a test gas. As **Fig. 9** indicates, variations of up to 20% in the prediction of the static pressure in the free stream and the shock stand-off distance are observed. While for this particular case, a modified rate model by Gupta, compared to the other chemistry models used for the study like Park and Dunn & Kang, led to a better agreement of numerical and experimental results, the uncertainties encountered in the numerical simulations underline the necessity of precise validation experiments on various geometries and flow conditions.

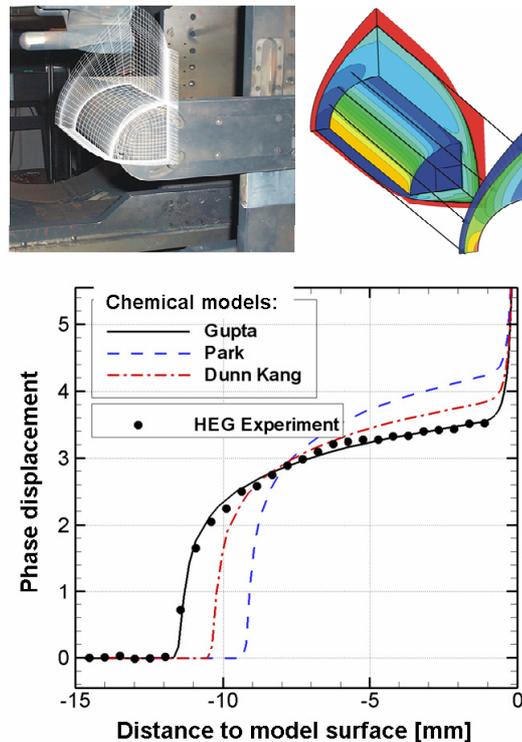


Fig. 9: Validation of chemical models for high enthalpy flows. Top: wind tunnel model (left) and CFD simulation (right). Bottom: location of the bow shock as function of the different chemical models.

3.3 Combustion Modeling

Also of interest is the simulation of combustion, most obvious in the case of supersonic combustion but as well in the case of subsonic combustion in rocket propulsion systems. The handling of mixtures (fuel and oxidizer), chemical reaction mechanisms and the compressibility influence due to high temperature gradients are similar. Additional models for multiphase flows (liquid components are injected into

the combustion chamber) spray generation, evaporation and condensation have to be integrated for combustion chamber simulations. Physical models for propulsion applications include turbulence, chemistry, and boundary-layer transition. Among these, turbulence is the critical item and drives the fidelity of the calculations. Current turbulence models used in propulsion are of the one- and two equation types.

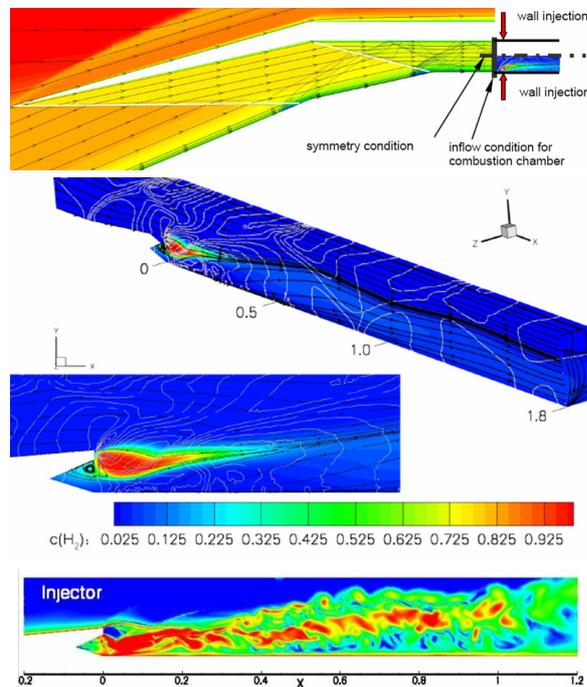


Fig. 10: Computation of supersonic combustion. Top: Intake and combustion chamber. Middle: hydrogen mass fraction contours and streamlines in a perspective plane and the plane of the injector. Bottom: DES vorticity contours of hydrogen mass fraction

The primary requirement is the prediction of injector performance, combustor wall temperatures and heat loads, overall combustion chamber performance and description of the complex, multiphase environment of the combustion chamber. The complexity of the combustion process occurring in a fluid medium stems from the many physical processes of different types, different natures, different temporal and spatial scales, and different degrees of being describable by deterministic models. Second, all of these processes usually are strongly coupled, making it difficult to simplify the problem. In addition, multiphase reacting flow models are affected by the turbulence and chemistry model used; turbulence and combustion interaction; two-phase flow coupling; the spray combustion model; the vaporization rate; atomization; particle size. Indeed, multiphase combustion modeling is still largely an art that relies heavily upon empirical correlations. Although there are codes available that contain an impressive array of combustion models, they are prohibitively expensive to use for any realistic three-dimensional geometry and flow

conditions. It is envisioned that as the next generation solvers become available and a factor of 10 increase in computational efficiency is realized, multiphase combustion calculations will become more frequent. As in other areas, lack of data for validation, coupled with the high cost of obtaining these measurements, is the major roadblock in determining the effectiveness of the models. Indeed, the prediction of injector/combustor flows requires a robust and highly efficient numerical platform that can be used for both steady state and transient calculations. In addition, a comprehensive set of physical models and sub-models needs to be incorporated into the codes. These include volume-of-fluid methodology for tracking immiscible gas and liquid phases, Monte Carlo techniques for tracking of finite-size droplets or particles, equilibrium and finite-rate chemistry models, liquid atomization models, droplet collision and break-up models, sub- and super-critical droplet vaporization models, turbulent dispersion models, and turbulent chemistry interaction models.

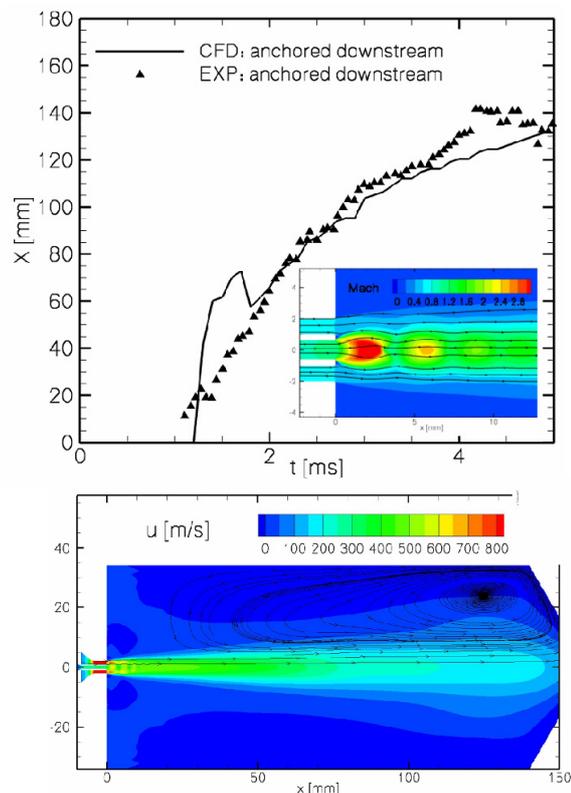


Fig. 11: Numerical simulation of a micro-combustor ignition. Top: experimental and numerical temporal evolution of the flame. Bottom: steady state prior ignition (velocity & Mach)

For hydrogen / air combustion, different detailed finite rate chemistry models are presented in Ref. [20], where the flow is considered to be a reacting mixture of thermally perfect gases and a transport equation is solved for each individual species. The advantage of the approach presented is its high flexibility. Extensions such as multi temperature models to handle thermal non-equilibrium effects are easily

possible. Knowing the mixture composition and the thermodynamic state of the individual species the properties of the reacting gas mixture are computed using suitable mixture rules. For turbulent flows the viscosity is then derived from the applied turbulence model. Thermal non-equilibrium flows are computed by solving an additional transport equation for the vibrational energy of each molecule in non-equilibrium. An assumed Probability-Density-Function (PDF) model is used to model the influence of turbulent fluctuations on the species source terms from detailed chemistry schemes. Two additional transport equations for the variance of temperature and the sum of the variances of species concentrations are solved to completely describe the PDF at each point of the flow field. Representative computed results for a 3-dimensional air-breathing combustor flow field using a modified 9 species and 17 steps Jachimowski reaction rate set are given in **Fig. 10** [20]. Further, the prediction of combustion characteristics and efficiency as well as the thermal and mechanical loads on a rocket combustor structure requires the development and validation of dedicated models. **Figure 11** present numerical results simulating an ignition-process of an H_2/O_2 mixture, in a micro-combustor [21, 22]. Transient flow phenomena such as ignition or combustion stability problems involve a large range of different time scales and require robust time accurate modeling capabilities.

4 Numerical Modeling

The above described models have their impact on the accuracy of flow simulations but require a large amount of resources (memory and cpu time). The necessity to find efficient algorithms and implementations for high end computers becomes crucial and a challenge of its own, if the technical applications require parameter studies of unsteady, three dimensional flow fields. While the parameter variation and the spatial resolution can benefit from parallel computing, the causal time dependence of unsteady phenomena prohibits a parallelization of this part of the problem. Although the validity range of the Navier-Stokes equations has physical limitations for very small scales and low densities, the numerical solution should provide on larger scales (behind a shock) and in regions of higher densities the correct answers. Therefore, the numerical method needs a well defined handling of shocks and low density (near vacuum) states. Further, high speed flows at high Reynolds numbers indicate very thin boundary layers on body surfaces. The only efficient method so far to resolve the high gradients of these boundary layers is to discretise with a different spacing normal and tangential to the wall. The necessary aspect ratio of the grid cells can be of the order of 1 to 10000. Such a direction splitting has to be accounted for in the numerical scheme. In addition, local preconditioning is another algorithmic enhancement that is currently under development to address problems associated

with convergence and truncation errors in very low velocity flows. The currently investigated preconditioning methods still have draw backs for supersonic flow fields with locally embedded subsonic regions. Also, the equations that represent chemical and thermal non-equilibrium flows contain source terms that may add stiffness to the numerical scheme used for the solution of the governing equations. This is particularly true when the chemical time scale is relatively smaller than the fluid dynamic time scale, and is typically solved implicitly with either explicit or implicit time-stepping methods. Although compared with implicit schemes, explicit schemes are extremely expensive for the highly stretched grids associated with high Reynolds number viscous simulations (the explicit time step scales as the square of the mesh size for pure diffusion model problems), they are widely used due to their relatively easy coding. The most extensively used explicit method is the Runge-Kutta time stepping together with residual smoothing to extend its stability limit. The solution of the 3D Navier-Stokes equations has become more acceptable by the aerospace industry with the advancements in multigrid algorithms that have significantly accelerated the convergence to steady state over a single-grid algorithm. This technology is still not fully developed for the treatment of chemical source terms but the prospects exist for considerable further enhancements to the convergence rate.

Indeed, the effective use of CFD for viscous hypersonic reacting flows is a challenge. Key requirements for efficient solution algorithm includes sharp capturing of strong shocks, robustness in regions of strong flow expansion, high resolution of viscous regions, efficient treatment of adverse grid and flow situations in the case of complex 3D geometries, and effective integration of stiff equations introduced by the large chemical source terms. Some of the most important algorithmic advances for the computation of hypersonic flows have been in the development of upwind and non-oscillatory schemes for improved shock capturing. While central difference schemes with upwind-biased or non-oscillatory dissipation operators are included in this class of algorithms upwind schemes, flux-difference-splitting or flux-vector-splitting, undoubtedly have become the main spatial discretization techniques adopted into nearly all major research and commercial codes. Further, new high-order, conservative and efficient methods for conservation laws are emerging. For structured grids, high order compact-difference discretization schemes require more operations per grid point but they offer higher accuracy. On unstructured grids, the computational effort required for techniques like Essentially Non Oscillatory and Weighted Essentially Non Oscillatory schemes is still larger than the benefit in accuracy. Discontinuous Galerkin or Multi-Dimensional Spectral differences methods for high Reynolds number problems are today in infancy.

In combination with all these models, having a special relation to high speed flows, other models to address more general problems are needed as well. For example, the time accurate simulation of a stage separation needs a chimera technique with automatic hole cutting to model the viscous flow fields around each obstacle and their interaction. A simulation of vortices generated from one vehicle and transported over a long distance to another vehicle or the analysis of sound wave generation in the near field of a body followed by the sound propagation to the far field are examples, which will show the benefit of higher than second order methods. Furthermore, all general purpose codes of today need a fine spatial resolution to locate and evolve discontinuities like shocks and contact surfaces correctly. When such features are traveling through the flow field of interest, a time accurate simulation benefits from the possibility of an automated local refinement and de-refinement strategy of the code as is shown in Fig. 12 from Ref. [23].

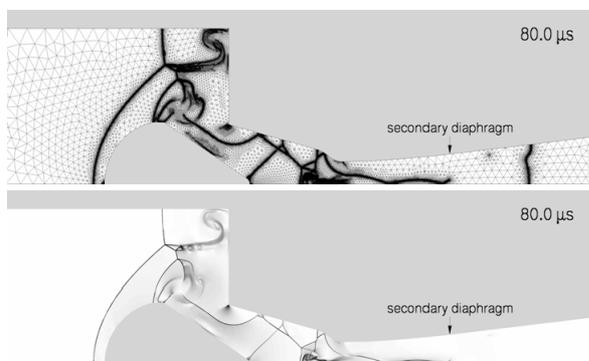


Fig. 12: unsteady numerical simulation of the reservoir region of a shock tunnel. Top: local grid adaptation. Bottom: numerical reconstruction of a Schlieren picture

In contrast to steady state simulations the region, where a fine spacing is needed, is no longer only depended on the local flow field, but also due to the propagation of unsteady features on neighboring regions within reach during the next physical time step. The prediction of the feature movements resulting from the adaptation indicator becomes more important the quicker (in terms of solver iterations) a feature is leaving the refined region. This has to be accounted for in case of high speed flows and especially when using implicit methods, where the CFL condition no longer bounds the wave propagation to the local cell size.

4.1 Numerical Scheme

For more than a decade ago, DLR initiated its CFD policy “one code for all type of flows”, beginning in the nineties of the last century with the CEVCATS code [24] and its extensions, and later with the FLOWer code [25]. Today DLR’s work-horse code for CFD in hypersonic is TAU [26], an unstructured method based on a dual mesh approach, which is well suited for hybrid grids thus allowing the use of mixed-

element meshes composed of tetrahedron, prisms, hexahedra, and/or pyramids. Building the regular grid parts with hexahedra and/or prisms allows for high aspect ratio cells with their edges aligned to the wall-normal and the wall tangential directions. This near-wall grid topology is known from the established structured methods to be well suited for accurate and efficient resolution of boundary layers. With a given surface discretization composed of triangles and/or quadrilaterals the generation of structured-type sub-grids over viscous aerodynamic surfaces can be done automatically by using a front method and is not too complex, because the extend of the sub-grids can be restricted to thin overall heights estimated from a maximum boundary-layer thickness. In order to employ a multigrid technique the agglomeration approach is used to obtain coarse grids by fusing together the fine grid control volumes, which are again described by the same metrics. Therefore, the coarse grid solution can be computed with the same approach as on the finest grid. By considering whirl fluxes in flux balance the code is capable to account for arbitrary rigid body motion of the grids. Additional terms accounting for the geometric conservation allow also for free stream consistency when mesh deformation is applied. As the Chimera technique has been recognized as an important feature to efficiently simulate maneuvering bodies, it has been also integrated into the TAU-Code. In the context of hybrid meshes the overlapping grid technique allows an efficient handling of complex configurations with movable control surfaces or other bodies in relative motion. For the data exchange in grid overlap regions linear interpolation based on a finite element approach is used. The search algorithm for donor cells is based on the alternating digital tree data structure.

For the calculation of low-speed flows, preconditioning of the compressible flow equations is implemented. The inviscid fluxes are calculated either by a Roe- or AUSM-type 2nd-order upwind scheme, or by employing a central method with scalar dissipation. Different upwind solver such as Van Leer, AUSM/Van Leer and AUSMDV are implemented. This last algorithm is an evolution of the AUSM scheme and is in fact a combination of a flux-difference-splitting scheme (AUSMD) and a flux-Vector-splitting scheme (AUSMV). As the AUSMV has a higher shock capturing capability, but presents some oscillating problems when confronted to a small velocity perturbation contrary to the AUSMD, a mixed momentum flux is preferred with a switching function depending on the pressure gradient. The resulting scheme, AUSMDV, is accurate, has a robust shock-capturing capacity and less problems in boundary layer (pressure oscillation near walls, usually due to the limiter) than the original AUSM scheme. With the explicit time stepping scheme, each domain can be treated as a complete grid when employing the domain decomposition method for parallel computing. During the flux integration data

have to be exchanged between the different domains several times.

In order to enable the correct communication, there is one layer of ghost nodes located at each interface between two neighboring domains. Edges of the dual grid connect the ghost nodes with the regular ones. These edges are those, which have been cut by the grid partitioning algorithm. The cut edges are part of both domains. Since a ghost node of one domain is a regular node of the corresponding neighbor domain the ghost-node values are to be updated by the regular node values before an operation on cut edges is performed. In order to allow for the use of distributed memory computers MPI is employed for exchanging the data for the update of node values. The number of ghost nodes compared to that of the regular nodes depends on the relation between the number of domains and the size of the global (non-decomposed) grid. Since (massive) parallel computing is needed for reduced turnaround it is as more important as larger the size of the computational global grid is. For large (grid-) scale problems a reasonable number of points and edges remain in each grid partition and as a result the relation to the number of cut edges and ghost nodes remain small, such that the overhead introduced by parallelization does not dominate the performance. In any case, for moderate parallel computations (e.g. 8 domains or less) the overhead remains small or negligible. In order to efficiently resolve detailed flow features, a grid adaptation algorithm for hybrid meshes based on local grid refinement and wall-normal mesh movement in semi-structured near-wall layers is applied. This algorithm also allow for de-refinement of earlier refined elements thus enabling the code to be used for unsteady time-accurate adaptation in unsteady flows. To extend the range of applicability of TAU to hypersonics or high enthalpy flows additional modifications and extensions have been introduced into the code, enabling e.g. simulations of re-entry vehicles including chemical reactions of air as a five component gas [27]. These modifications range from stabilization of the solver for high Mach numbers over additions for thermo-chemical equilibrium flows to the consideration of non-equilibrium gases. Due to the latter additional conservation equations for the partial densities and the vibrational energies of the species are introduced in the code and to close the system, models for the state of species as well as fits for their viscosity and the resulting heat conductivity are taken into account. Furthermore mixture rules, diffusion and detailed chemistry based on the Arrhenius-Ansatz with thermal coupling after Park are implemented together with thermal relaxation after Landau-Teller to allow for full thermo-chemical non-equilibrium simulations. With respect to boundary conditions walls with full, finite or non catalytic surfaces can be considered as well as radiation-adiabatic walls or porous walls.

4.2 Geometrical Representation

A CFD analysis is particularly helpful when predicting surface properties in localized surface areas of topological and or geometrical complexity. In those cases, CFD faces several challenges. Today an extensive array of block-structured methods, unstructured grid methods, and hybrid schemes are available, although no single method has emerged as the preferred approach. Numerical schemes for block structured methods, including patched and overset grids, are the most efficient methods for high Reynolds number simulations, but are the most labor intensive in terms of grid generation. On the other side, hybrid grid methods have emerged as the methods of choice for high Reynolds number viscous flow on complex 3D configurations because of their speed and low need of user interaction in the grid generation process.

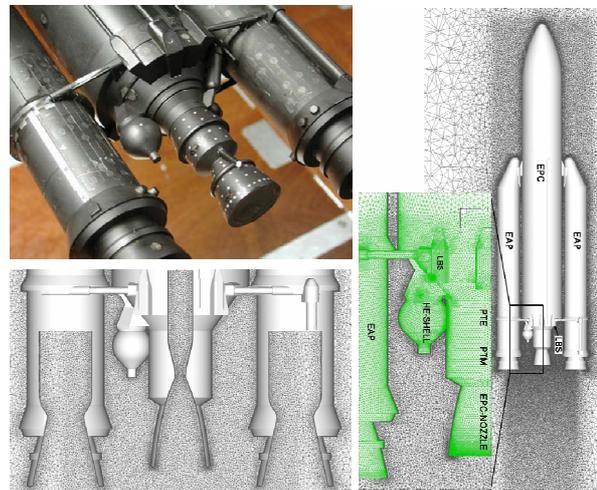


Fig. 13: unstructured grid around a launcher configuration. Right: grid around the complete launcher with zoom in nozzle area. Left: geometry to be modeled (top); grid detail at the nozzle (bottom).

The discretization of a complete launcher vehicle resembling the European Ariane-5 with 2 solid boosters aside, including many small details and connection fittings as shown in Fig. 13 [28] demanded almost 10 working days. In this approach, a structured layer of points is developed near the surface and connected to the outer field through an isotropic distribution of unstructured grid elements. In comparison with structured grids, unstructured grid generation requires approximately twelve times less effort but the performance of numerical algorithms for unstructured grids are about ten times slower than those for structured ones. For structured as well as unstructured grid generators, configuration details like antennas, inspection doors, gap and slots are not easy to accommodate unless special considerations and foresights are applied in the initial grid generation process. In many cases, geometrical simplifications in the configurations introduce errors in the predicted aerodynamic and aerothermodynamic quantities

comparable to those arising from numerical accuracy or lack of physical modeling [29]. Surface representations based on Computed Aided Design systems, even though these tools have gained great sophistication, remain cumbersome and restrictive and require skills that are not generally available in researchers and designers. As alternative, today most professional computer graphics applications available for desktop use offer methods based on Non-Uniform Rational Basis-Splines (NURBS).

4.3 Validation Data

Validation and error estimation are critical challenges for CFD because uncertainties in predicting vehicle performance increase the design margins (and therefore add 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, code 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. Indeed, for geometrically complex configurations, obtaining a grid converged solution is either not possible or is precluded by higher 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 because the physical models are function of flow parameters and their gradients, which are function of 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 [30, 31].

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 conditions such as the precise definition of the test set-up 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 conditions in a wind tunnel is not possible. As is shown in Fig. 14, flight Reynolds numbers and flight Mach numbers associated with high enthalpy flows are critical to simulate; 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 [32].

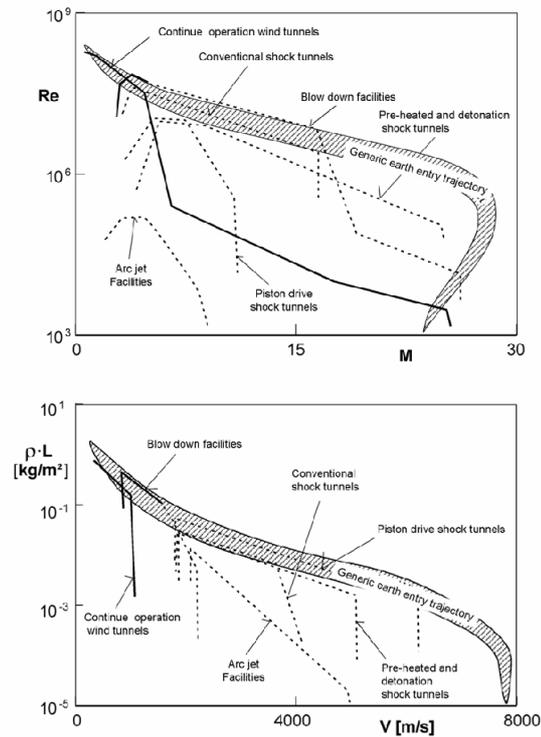


Fig. 14: Wind tunnel operating ranges. Top: Diagram based on Reynolds and Mach numbers. Bottom: Diagram based on density and velocity.

Also, a new trend is emerging for low cost physical-modeling and technology validation in flight based on the use of sounding rockets. Examples are the HyShot [33] and SHEFEX [34] flight experiments. In flight-measurement constitute the only way to obtain data for prediction tools validation and calibration under real conditions and therefore, they are irreplaceable for CFD validation. However, flight measurements are expensive, 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 phenomena that cannot be directly measured may contain important uncertainties [35].

5 Multidisciplinary Problems

Numerical optimization has several advantages over the traditional system design approach. It is potentially faster; more likely to achieve a truly optimal design; force the design-team to specify the design problem carefully and completely; provide insight into the nature of the design space and operating points. Virtually all system components must perform efficiently over a range of operating conditions. While several algorithm have been developed that can efficiently perform aerodynamic shape optimization

[36], optimization at a single operating point invariably leads to poor off-design performance. Therefore, the optimization problem must be posed such that a range of operating conditions and off-design performance requirements are included in either the objective function or in the constraints [37]. It is clear that numerical optimization will in general not proceed directly from problem specification to the optimal design. Rather, the problem specification will evolve iteratively based on feedback provided by the optimization results. In particular, the development of high speed flying machine needs powerful tools for the design of an aerodynamic shape with a guarantee for reliable flight controllability along the complete flight envelope [38]. Thus, the optimization problem becomes a multipoint-multiple-discipline problem and hence the numerical tool shall allow multiple discipline analysis. In the following sections, some examples are given.

5.1 Fluid-Thermal/Structure Coupling

Radiation adiabatic wall condition is an engineer assumption which provides rather good results in aeronautic design. Although by this means the amount of heat accumulated in the structure is not computed, emission of radiation from hot surfaces and absorption on cooler surfaces has to be accounted for. Depending on the fluid composition and optical thickness the transport of energy within the flow field via radiation can become significant at higher temperatures. The nature of radiation energy transport (time scales, global influence) is so different to the fluid mechanical transport that a separate simulation tool is necessary and has to be coupled with the CFD. Furthermore, the heat transfer within a body becomes important, if the structural connection between hot and cold parts of the surface leads to changed distributions of the surface temperature (e.g. local peaks are smoothed). This kind of thermal coupling between CFD and computational structure mechanics (CSM) becomes important in high temperature flows in addition to the typical force/deformation coupling of aero-elastic problems.

A numerical investigation of a generic control surface for a space vehicle is reported in Ref. [39]. The numerical solutions are obtained with the TAU code coupled via a surface interpolation routine to a structural solver. The investigation confirmed that due to strong coupling effects between fluid and structure, the temperature peaks predicted by stand-alone computational fluid dynamics solutions are not observed either in the coupled solution or in the experimental data as is shown in Fig. 15. Further, characterization of the thermo-chemically frozen flow around a generic nose model of a hypersonic vehicle, followed by thermally and thermo-mechanically coupled analysis to investigate the physical effects of heat conduction inside the structure has been recently published in Ref. [40]. The investigation could reproduce the experimental data, showing good

agreement with the temporal temperature behavior during the experimental tests as is shown in Fig. 16.

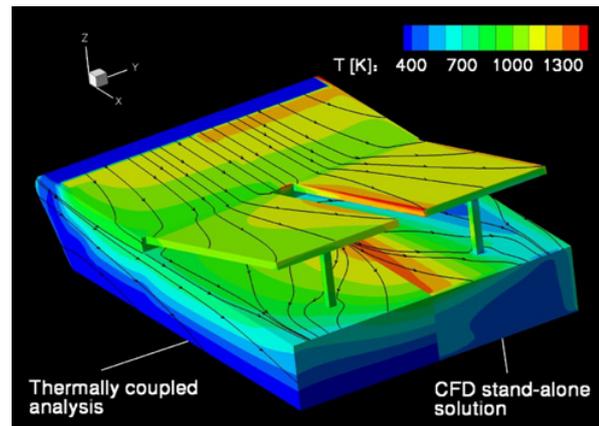


Fig. 15: Numerical simulation of fluid-/thermal-structure coupling effects on a generic flap at $H=13\text{Mj/kg}$, $Ma=7$, $AoA=15\text{deg}$ and 20deg flap deflection.

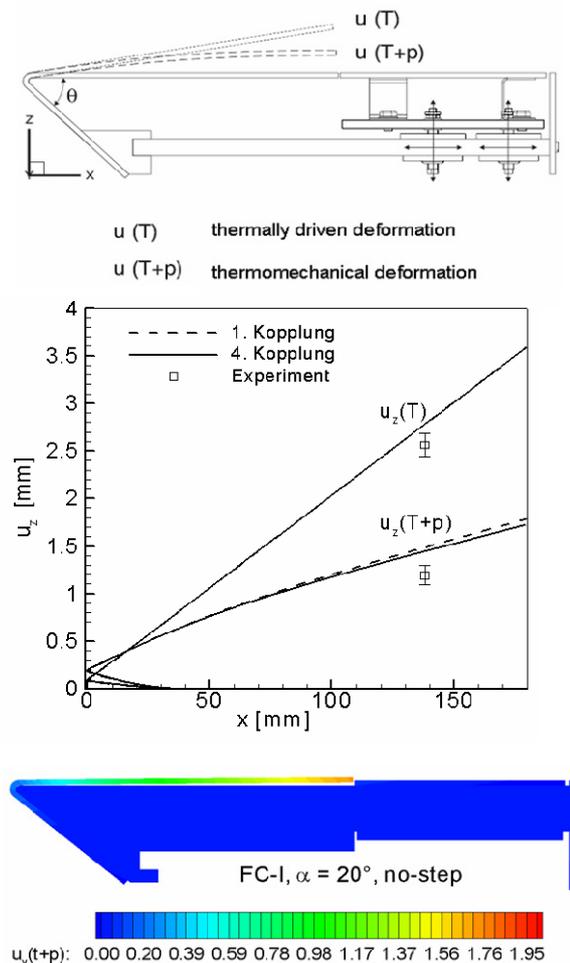


Fig. 16: Computed fluid induced thermo-mechanical deformations by $Ma=7$, $H=13\text{Mj/kg}$ and $AoA=20\text{deg}$. Top: Sketch of the experiment. Middle: Numerical and experimental thermal and thermo-mechanical deformations. Bottom: Thermo-mechanical deformation distribution along the plate.

While the above mentioned examples were obtained for generic models, coupled fluid-thermal simulations along the flight path of hypersonic vehicles become necessary to reliably select the material and design the thickness of the thermal protection system in earlier stage of a the vehicle design [41]. In such cases, to save computer resources the time changes of the flow pattern may be handled as steady but the time change of the temperature profile inside the vehicle should be determined by unsteady simulations. As shown in **Fig. 17** there are important differences in maximal temperature when considering the time evolution of the temperature inside the vehicle, as well as when considering different type of insulations.

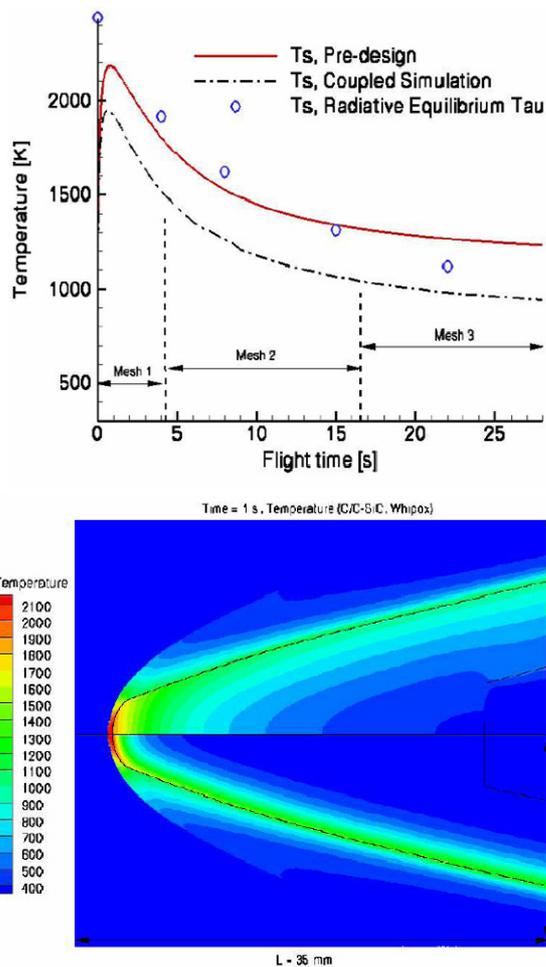


Fig. 17: fluid-/thermal-structure coupling simulation for a generic vehicle along a flight path at Ma=6 and low altitude. Top: Surface temperature evolution as function of time and different hypothesis. Bottom: Instantaneous temperature profile inside the vehicle

Another example of application is, as presented in **Fig. 18**, the assessments of shock-wave boundary layer interaction on control surfaces of hypersonic vehicles and associated surface heating, a complex 3-dimensional problem which requires coupling of flow field solutions along the flight path with the structure

and gas radiation under non-equilibrium conditions [42]. Further, coupled fluid-thermal simulations along flight path can help to design in-flight experimentation since the researcher may assess the impact of the sensor position (with respect the vehicle surface) on the measured quantity in earlier stages of the experiment-design. For a hypersonic vehicle flying at Mach 6, varying the depth location of a temperature sensor inside the isolation by only 1mm causes variation in temperature of 120K(!) as is shown in **Fig. 19** from Ref. [43].

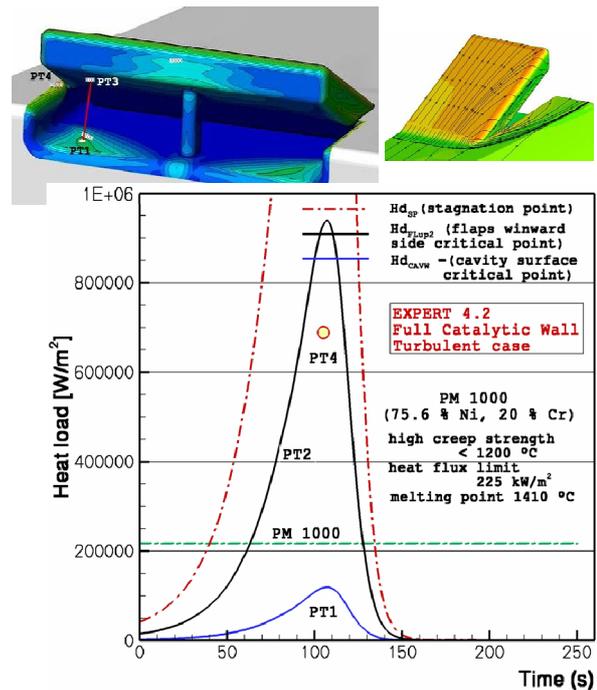


Fig. 18: fluid-/thermal-structure coupling simulation along a flight path for a capsule at Ma=17, 40km altitude. Top: Computed temperature distribution (left) and flow topology (right). Bottom: Heat flux time evolution at different locations of the vehicle

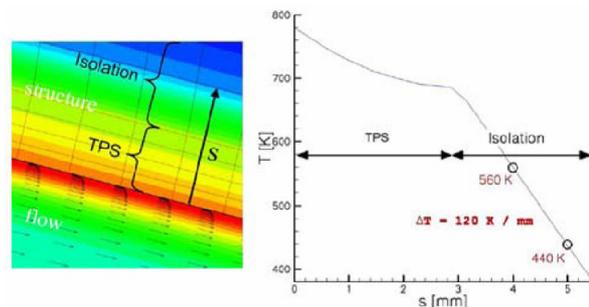


Fig. 19: Computed fluid-/thermal-structure coupling along a flight path for a generic vehicle at Ma=6, 20km altitude. Left: Temperature profile in the flow and inside the vehicle Right: Instantaneous temperature evolution inside the vehicle

Finally, investigation of the fluid-structure interaction for the nozzle section of a launcher vehicle under transonic wind tunnel conditions is reported in Ref. [44]. A fluid-structure coupling procedure between DES flow simulations and FEM data is carried out and ran over seven periods of nozzle oscillations. For the FEM part a structural grid is generated by using the basic natural frequencies of a real nozzle. A rich modal response of the nozzle shape on the unsteady flow field is investigated in detail and a strong influence on the surface pressure distribution in this region is shown in **Fig. 20** where the essential interaction between the turbulent flow field and the nozzle structure is demonstrated.

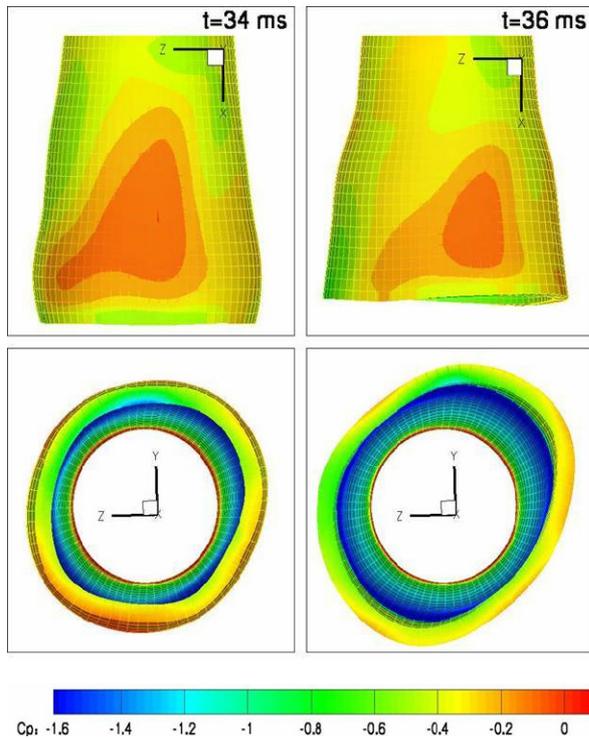


Fig. 20: fluid-/thermal-structure coupling simulation for the nozzle of a launcher at $Ma = 0.8$ during start Pressure distribution and nozzle deformation (expanded view) as a function of time

5.2 Propulsion Coupling

Propulsion integration is a key issue for hypersonic vehicle design. In designing flight vehicles, the engine and its nozzle are usually considered an extra item of the vehicle airframe. However, the larger the flight speed is, the more integrated are the lift system and the propulsion system of the vehicle. Earlier successful pre-design of airbreathing hypersonic vehicles based on CFD solutions of the TAU code have been reported in Ref. [45]. There a blue-print experimental vehicle, **Fig. 21**, considered to fly between $Ma = 4$ and $Ma = 8$ to cover both subsonic and supersonic combustion is presented and discussed.

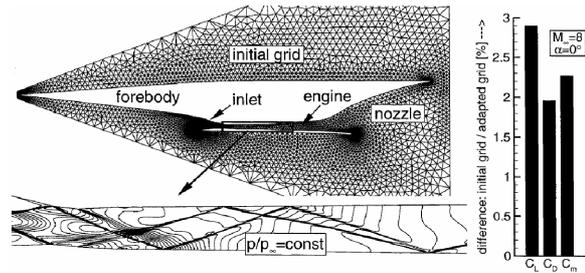


Fig. 21: Computed performance of a supersonic-combustion propelled-vehicle for $Ma=8$, $AoA=0deg$. Left: Computational grid (top) and Isobars inside the engine (bottom). Right: Evolution of the aerodynamic coefficients as function of the grid finest

An important prerequisite for the increase of efficiency of future launchers is the aerodynamic optimization of flows in thrust nozzles. This requires a reliable prediction of stationary and transient phenomena occurring in nozzle flows using CFD tools. The work presented in Ref. [46] concentrates on the numerical investigation of the transition process between separated flow at the wall inflection and fully attached flow for a dual-bell nozzle using the DLR-TAU code. Examples of results are presented in **Fig. 22**. The hysteresis which is important to prevent the nozzle from running into a flip-flop regime was clearly seen in the CFD results. The time of transition from separated to fully attached mode ranges between 4 ms and 14 ms for the considered cases.

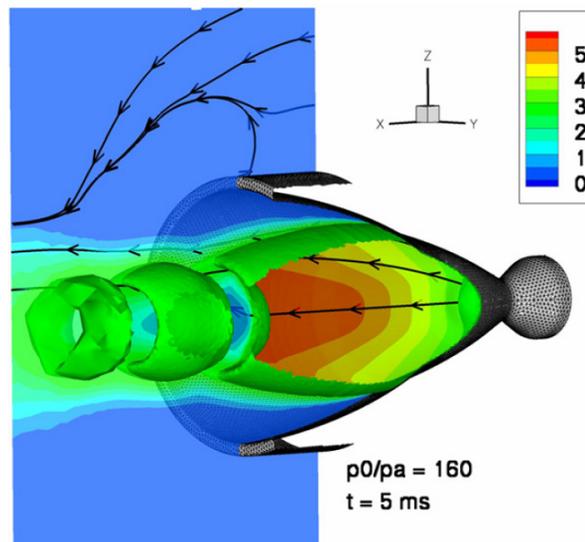


Fig. 22: Snapshot of the time accurate numerical simulation of a flow on rocket nozzle at $Ma=3$, $d(pc/pa) = 20 \text{ ms}^{-1}$ at $t=5\text{ms}$. Mach contours and streamlines on a cut plane

In addition, the exhaust plumes of the vehicle nozzles, main engines or thrusters for control, act as disturbance of the external flow creating an effect that can change favorably or unfavorably the pressure distribution on the vehicle surfaces surrounding the exhaust plume. Examples of such systems are the planar fluidic jet reaction control elements, used in

hypersonic vehicles as reliable and effective means of flight control. Reference [47] presents a design of such system based on CFD. As shown in **Fig. 23**, the outflow channel of a fluidic jet reaction control element and hence the direction of the thrust vector may be controlled by the injection of a secondary jet through one of the slits downstream of the inlet nozzle in the side walls.

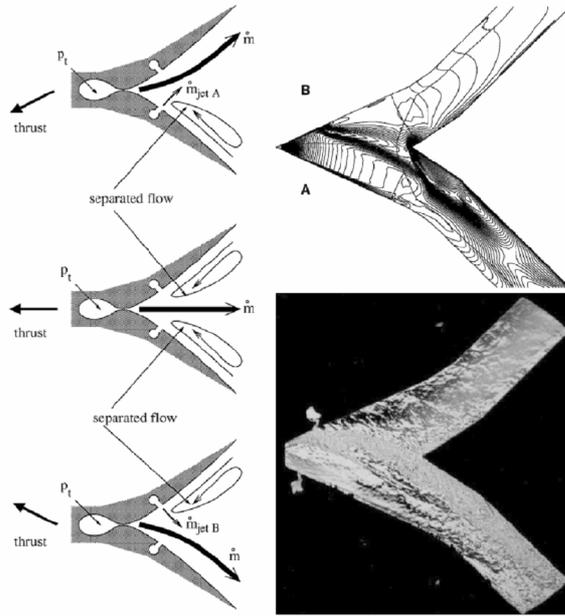


Fig. 23: Computation of a fluid jet reaction control element. Left: principle of work. Right: Density distribution for an active jet, numerical (top) and experimental (bottom)

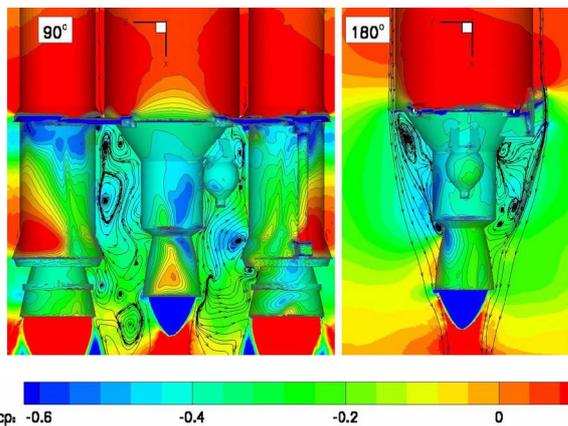


Fig. 24: Snapshot of a time accurate numerical simulation, using DES, for the flow at the basis of a launcher during start, Ma=0.8. Instantaneous pressure contours in 2 planes

One challenge of the investigations of unsteady super- and hypersonic flow fields is the study of turbulent wake flow and the interaction with nozzle sections at modern launcher configurations [48]. Unsteady side-loads, induced by the interaction of flow separation inside of the nozzle and the launcher wake will

strongly influence the design of future main stage propulsion systems. This interaction phenomenon, called buffeting coupling, is one of the main challenges during ascent. **Figure 24** presents preliminary DES results on the investigation of unsteady buffeting coupling phenomena. Transonic and supersonic flows fields for an entire launcher configuration are presented and discussed in Ref. [49]. The investigation showed good agreement with experimental pressure data.

5.3 Fluid – Flight Mechanic Coupling

For steady flows, substantial CFD capability has been achieved over the past decades and Navier-Stokes solvers are intensively used in aerodynamic design. In contrast, although some isolated unsteady flow calculations have been carried out for various classes of problems but almost restricted to sub- or transonic flows, numerical simulations of unsteady flows are certainly not routine, due to the excessive computational effort involved in these calculations, particularly when the flow becomes hypersonic.

A numerical approach for the prediction of the dynamic derivatives is presented in Ref. [50]. The research described there for predicting the damping coefficients utilizes a multi-disciplinary approach, involving flight mechanics and computational fluid dynamics. It resembles existing wind tunnel procedures using forced harmonic motion of the model and transforming the data into the frequency domain via Fourier transformation. The prediction of unsteady viscous flows is done by means numerical solutions of the Reynolds-averaged Navier-Stokes equations in a moving coordinate system. Typical results obtained with this technique for a generic space vehicle are presented in **Fig. 25**.

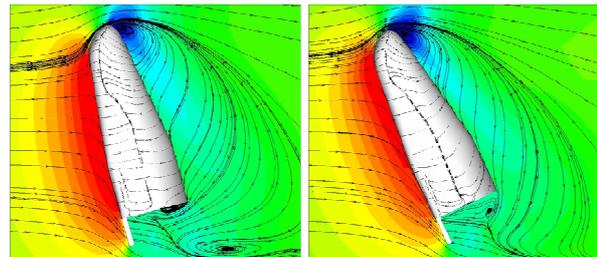


Fig. 25: Instantaneous pressure field computed for a space vehicle approaching to landing, AoA=70 deg., Ma=0.8. Left: Upstroke motion, right: Downstroke

Further, Ref. [51] presents a thermal assessment of a hypersonic payload' carrier designed to deliver a small payload of about 400g at an altitude up to 115km, flying through the dense atmosphere at hypersonic speed. CFD real-time unsteady solutions have been obtained by coupling the DLR-TAU Navier-Stokes solver with the 3DOF trajectory program. To keep the computational effort low only the first 3 seconds of flight, which are the most critical one for the problem considered, are accounted for. No fluid-structure coupling but radiation equilibrium with an emission

coefficient of 0.8 is considered in this case. **Figure 26** shows the changes in heat flux resulting at the surface of the vehicle, as a function of flight time, when the vehicle travels from the earth surface to the free space. The numerical solutions are assessed by comparing results obtained in grids on different densities; applying different wall boundary conditions and using different time steps for the time marching process.

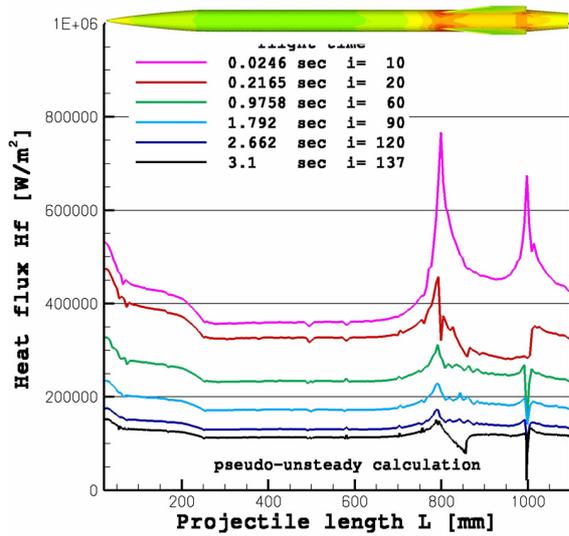


Fig. 26: Time accurate simulation of the resulting surface thermal loads for a generic vehicle during the first 3 sec climbing flight at $Ma=6.3$, from sea level

5.4 Fluid – Electromagnetic Field Coupling

The application of plasma devices for controlling and enhancing aerodynamic phenomena encountered on atmospheric flight vehicles is actively being investigated. These devices operate using the generation of plasma within the flow field, generally through an electrical discharge or injection of an electron beam, and then using either the effects of energy deposition itself or through application of electrostatic or Lorenz forces to modify the aerodynamic phenomena of interest. In such cases, a separate simulation tool is necessary and has to be coupled with the CFD tool if unsteady electromagnetic fields and their interaction with the flow field should be investigated. But to account for the influence of a given magnetic field on a flow of thermal plasma, modifications within the flow solver are sufficient like source term formulation of the magneto gas dynamic equations, as is shown in **Fig. 27** from Ref. [52], where the TAU code is extended by additional source term formulations for the electromagnetic forces.

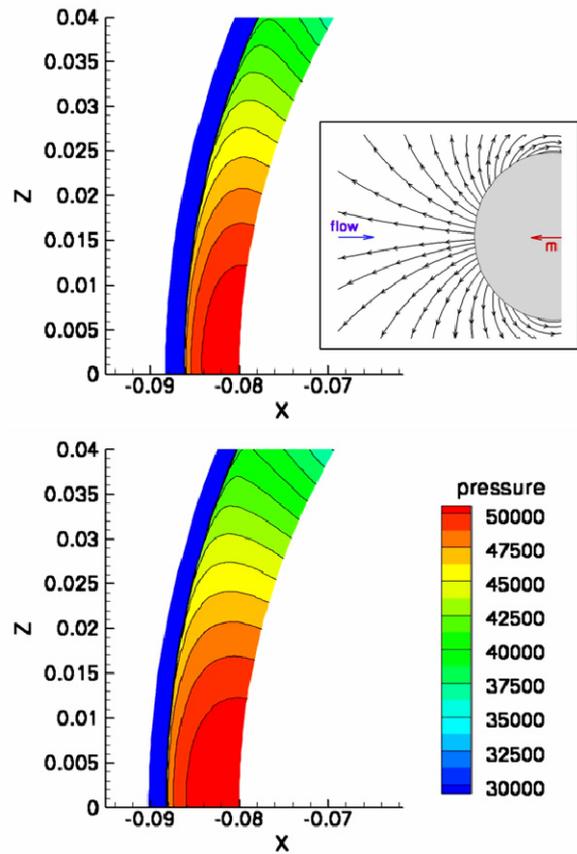


Fig. 27: Computed effects of a dipole magnet field inside a blunt body embedded in a dissociated hypersonic flow. Pressure fields without (top) and with (bottom) activated magnetic field

5.5 Continuum – Rarefied Flow Coupling

Another situation in which the coupling with a complete different solver is crucial for a good simulation of a high speed flow field is at very low densities, e.g. high altitudes during earth reentry; above ~ 100 km. There the mean free mean path of the gas particles is of the same order as or greater than the characteristic length scales of the vehicle ($Kn > 0.1$).

Then the fluid can no longer be described as a continuous medium and the Navier-Stokes equations are not valid for such flow fields. Direct simulation Monte Carlo (DSMC) methods calculate statistics of traced representative particles and can be used to determine flow fields characterized by higher Knudsen numbers. Examples for such flow fields are a nozzle expansion into vacuum with focus on the back flow region as displayed or the wake of a space vehicle at low densities or if the region of interest has a very small scale (e.g. internal structure of a shock or the tip of a flat plate). Today state of the art is the coupling of Navier-Stokes solvers with DSMC codes to resolve flow fields with vary Knudsen number, as is indicated in **Fig. 28**, where the flow expanding from a nozzle at high altitude is computed with the Navier-Stokes equations inside the nozzle (continuum regime) while the plume flow is modeled with DSMC.

Important here is that at the interface no discontinuities in the solutions are observable. New trends are calling now for the extension of the DSMC codes to lower Knudsen numbers, allowing computation of hypersonic flows at equivalent flight altitudes of about 70km. Experiments in ground facilities suggest that when there is significant vibrational non-equilibrium within the shock layer, the vibrational energy does not accommodate to the surface temperature when the molecules impact the surface.

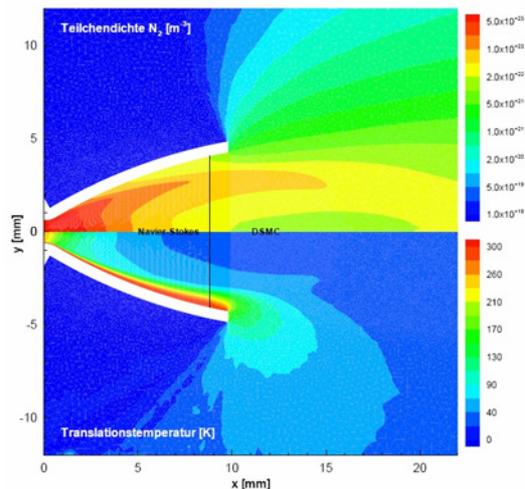


Fig. 28: Numerical simulation of the plume expanding almost in vacuum. Navier-Stokes and DSMC combined computation. Top: Electron density contours. Bottom: Translational temperature contours

6 Summary

The expected developments in space transportation have been historically the motivation for growing interest in Computational Fluid Dynamic for aerospace sciences. Today CFD is becoming a mature discipline, widely used in industry, government and universities. Unsteady flow phenomena are attracting more attention and since they are present in almost all the hypersonic flow problems. A major problem of concern on the physical modeling side is the availability of experimental data for model validation in the hot hypersonic regime. While high fidelity CFD results are today based on the solution of the Navier-Stokes equations, they still require validated models to account for viscous-turbulent, high temperature and combustion effects.

On the numerical modeling side, up-wind schemes have become the main spatial discretization in hypersonic CFD codes while high order compact schemes are emerging. Significant progress has been achieved on surface and flow field discretization using unstructured and hybrid grid methods but the efficiency and accuracy of the interfaces between different Computed Aided Design systems used for

surface modeling must be improved as well as the treatment of often imperfect data.

Significant gains in efficiency are being obtained through the increasing use of parallel computers. Computer speed has grown in the last decade by an order of magnitude every five years, a trend that will continue. Taking advantage of the tremendously growth in speed and capacity of today's computer systems, multidisciplinary and multi-design optimization methods and 3D simulation of combustion processes are today possible. Accordingly, standards for communication between different systems are evolving together with procedures and recommended practices for assessing the credibility of the CFD simulations.

In summary, computers are driving today's progress in high speed flow simulations.

Acknowledgement

The authors are grateful to the colleagues of the Institute of Aerodynamics and Flow Technology of DLR, Spacecraft Branch, who obtained and provided all the results contained in this paper.

7 References

- [1] K. Hannemann, J. Longo. *Aerothermodynamik und Wiedereintritt. Handbuch der Raumfahrttechnik*, Ed. W. Ley u. K. Wittmann, Carl Hanser Verlag München Wien, 2007, pp. 94-108.
- [2] J.M.A. Longo. *Modelling of Hypersonic Flow Phenomena. 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.
- [3] J.M.A. Longo, H. Lüdeke, S. Brück, M. Orłowski, M. Rapuc, J.-P. Tribot, M. Stojanowski. *X-38: A Test Bed for the CEVCATS-N Code*. Proceedings of the 1st International Symposium "Atmospheric Re-entry Vehicles and Systems", Arcachon, France, March 1999.
- [4] Th. Eggers, O. Bozic. *Aerodynamic Design and Analysis of an ARIANE 5 Liquid Fly-Back Booster*. Proceedings of the 11th International Space Planes and Hypersonic Systems and Technology Conference, AIAA Paper 2002-5197, Orleans, France, 2002.
- [5] Ph. Reynier, J.M.A. Longo and E. Schülein. *Simulation of Missiles with Grid Fins Using an Actuator Disk*. *Journal of Spacecraft and Rockets*, Vol. 43, No. 1, January-February 2006, pp 84-91.

- [6] M. Khalid, A. Dujardin, P. Henning, 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. *Canadian Aeronautics and Space Journal*, Vol. 51, No. 4, December 2005.
- [7] 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.
- [8] C. Dankert and H. Otto. Experimental Investigation and Numerical Simulation on a Missile Radome at Mach 6. *Proceedings of the 15th DGLR Fach Symposium der STAB*, Darmstadt, Germany, 2006.
- [9] B. Reimann and J. Martinez-Schramm. Numerical and Experimental Investigation of a Hypersonic Glider. *Proceedings of the 37th Fluid Dynamics Conference and Exhibit*, AIAA Paper 2007-3769, Miami, USA, 2007.
- [10] A. Mack, J. Steelant, V. Togiti and J.M.A. Longo. Impact of Intake Boundary Layer Turbulence on the Combustion Behavior in a Scramjet. *Proceedings of the 2nd European Conference for Aerospace Sciences (EUCASS)*, Brussels, Belgium 2007.
- [11] V.K. Togiti, H. Lüdeke. Computation of Supersonic Base Flow Using Detached Eddy Simulation. *Proceedings of the 15 DGLR Fach Symposium der STAB*, Darmstadt, Germany, 2006.
- [12] R. Radespiel, H. Lüdeke, S. Brück. Computation of Transitional Re-entry Flows over 3D Control Surfaces. *Proceedings of the 2nd International Symposium "Atmospheric Re-entry Vehicles and Systems"*, Arcachon, France, March 2001.
- [13] H. Lüdeke, R. Radespiel, E. Schülein. Simulation of streamwise vortices at the flaps of re-entry vehicles. *Aerospace Science and Technology* 8 (2004) 703-714.
- [14] K. Hanneman, J. Martinez-Schramm, S. Brück, J.M.A. Longo. High Enthalpy Testing and CFD Rebuilding of X-38 in HEG. *Proceedings of the 23rd International Symposium on Shock Waves ISSW23*, Arlington, USA, 2001.
- [15] S. Karl, D. Fletcher, G. Dégrez, T. Magin, M. Playez. Assessment of Radiative Transport in an Argon Plasma Flow. *Proceedings of the 4th European Symposium on Aerothermodynamics of Space Vehicles*, Caserta, Italy, 2001. pp. 505-511.
- [16] A. Gülhan, B. Esser, K. Hannemann. Mars Entry Simulation in the Arc Heated Facility L2K. *Proceedings of the 4th European Symposium on Aerothermodynamics of Space Vehicles*, Caserta, Italy, 2001. pp. 665-671.
- [17] S. Brück, R. Radespiel, J.M.A. Longo. Comparison of Nonquilibrium Flows past a Simplified Space-Shuttle Configuration. *Proceedings of the 35th Aerospace Sciences Meeting and Exhibit*, AIAA Paper 97-0275, Reno, USA, January 1997.
- [18] S. Brück, W. Kordulla, Th. Eggers, M. Orłowski, J.M.A. Longo. The Effect of Catalyticity on the Heating of the X-38 Shape. *Proceedings of the 8th Annual Thermal and Fluids Analysis Workshop Spacecraft Analysis and Design*. NASA CP-3359, 1997, pp. 13-1 to 13-13-6.
- [19] S. Karl, J. Martinez-Schramm, K. Hannemann. High Enthalpy Cylinder Flow in HEG: A Basis for CFD Validation. *Proceedings of the 33rd Fluid Dynamics Conference*, AIAA Paper 2003-4252, Orlando, USA, 2003.
- [20] Karl, S, Hannemann, K., Steelant, J., Mack, A., Analysis of the HyShot Supersonic Combustion Flight Experiment Configuration, *Proceedings of the 14th International Space Planes and Hypersonic System and Technology Conference*, AIAA Paper 2006-8041, Canberra, Australia 2006.
- [21] S, Karl, K. Hannemann. Application of the DLR TAU-code to the RCM-4 Test Case: Micro-Combustor Ignition. *Proceeding of the 3rd International Workshop on Rocket Combustion Modelling*, Paris, France, 2006.
- [22] S, Karl, K. Hannemann. Application of the DLR TAU-code to the RCM-1 Test Case: Penn State Preburner Combustor. *Proceeding of the 3rd International Workshop on Rocket Combustion Modelling*, Paris, France, 2006.
- [23] B. Reimann, V. Hannemann, K. Hannemann. Computations of shock wave propagation with local mesh adaptation. *Proceedings of the 26th International Symposium on Shock Waves ISSW26*, Göttingen, Germany, 2007.
- [24] R. Radespiel, J.M.A. Longo, S. Bruck and D. Schwamborn. Efficient Numerical Simulation of Complex 3D Flows with Large Contrast. *AGARD 77th Fluid Dynamics Panel Meeting „Progress and Challenges in CFD Methods and Algorithms“*, Seville, Spain, 1995, pp.32-1 to 32-11.
- [25] N. Kroll. National CFD Project MEGAFLOW – Status Report. *Notes on Numerical Fluid Mechanics*, Vol. 60, Ed. H. Körner and R. Hilbig, Vieweg Verlag, Braunschweig, 1997, pp.15-23.

- [26]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.
- [27]A. Mack, V. Hannemann, Validation of the Unstructured DLR-TAU-code for Hypersonic Flows, Proceedings of the 32nd Fluid Dynamic Conference and Exhibit, AIAA Paper 2002-3111, North Oregon, USA, June 2002.
- [28]O. Bozic and H. Otto. Flow Field Analysis of a Future Launcher Configuration during Start. Proceedings of the 5th European Symposium on Aerothermodynamics for Space Vehicles, Cologne, Germany, November 2004.
- [29]J.M.A. Longo, M. Orlowski, and S. Brück. Considerations on CFD Modelling for the Design of Re-entry Vehicles. Aerospace Science and Technology 4 (2000) 337-345.
- [30]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.
- [31]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. Proceedings of the 40th Aerospace Sciences Meeting and Exhibit, AIAA Paper 2002-0407, Reno, USA, January 2002.
- [32]K. Hannemann. High Enthalpy Flows in the HEG Shock Tunnel, Experiment and Numerical Rebuilding., Proceedings of the 41st Aerospace Sciences Meeting and Exhibit, AIAA Paper 2003-0978, Reno, USA, January 2003.
- [33]J. Steelant, A. Mack, K. Hannemann and A.D. Gardner. Comparison of Supersonic Combustion Tests with Shock Tunnels, Flight and CFD. Proceedings of the 14th International Space Planes and Hypersonic System and Technology Conference, AIAA Paper 2006-4684, Camberra, Australia 2006.
- [34]J.M.A. Longo, Th. Eggers, A. Gülhan, J. Turner, H. Weihs. Designing Flight Experiments for Hypersonic Flow Physics. Proceedings of the RTO/AVT VKI Lecture Series "Flight Experiment Hypersonic Vehicle Development", Brussels, Belgium, October 2005.
- [35]J.M.A. Longo. Aerothermodynamics – A critical review at DLR. Aerospace Science and Technology 7 (2003) 429-438.
- [36]W. Tang, M. Orlowski, J.M.A. Longo, P. Giese. Aerothermodynamic Optimization of Re-entry Capsules. Aerospace Science and Technology 5 (2001) 15-25.
- [37]D. Strohmeier, 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.
- [38]C. Weiland, J. Longo, A. Gülhan, K. Decker. Aerothermodynamics for reusable launch systems. Aerospace Science and Technology 8 (2004) 101-110.
- [39]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.
- [40]A. Mack. Aerothermodynamic behaviour of a generic nose-cap model including thermochemical structural effects. Aerospace Science and Technology 11 (2007) 386-395.
- [41]J. Bartolomé Calvo, A. Mack and O. Bozic. Study of the Heating of a Hypersonic Projectile through a Multidisciplinary Simulation. Proceedings of the European Conference on Computational Fluid Dynamics, ECCOMAS CFD 2006.
- [42]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, Paper IAC-06-D2.5.02, Valencia, Spain 2006.
- [43]T. Barth. Advanced Aerothermodynamic Analysis of SHEFEX I. Proceedings of the 1st European Air and Space Conference CEAS. Berlin, Germany, 2007.
- [44]H. Lüdeke, J. B. Calvo, A. Filimon. Fluid Structure Interaction at the Ariane-5 Nozzle Section by Advanced Turbulence Models. Proceedings of the European Conference on Computational Fluid Dynamics, ECCOMAS CFD 2006.
- [45]Th. Eggers, Ph. Novelli, M. Haupt. Design studies of the JAPHAR experimental vehicle for dual mode ramjet demonstration. Proceeding of the 10th International Space Planes and Hypersonic Systems and Technology Conference, AIAA Paper 2001-1921, Kyoto, Japan 2001.
- [46]S. Karl, K. Hannemann. Numerical Investigation of Transient Flow Phenomena in Dual-Bell Nozzles. Proceedings of the 6th International Symposium on Launcher Technologies, Munich, Germany, 2005.

- [47]U. Reisch, R. Meuer. CFD-Simulation of the flow through a fluid element. *Aerospace Science and Technology* 4 (2000) 111-123.
- [48]M. Freym A. Preuss, G. Hagemann, S. Girard, Th. Alziary de Roquefort, Ph. Reijass, R. Stark, K. Hannemann, R. Schwane, D. Perigo, L. Boccaletto, H. Lambare. Joint European Effort Towards Advanced Rocket Thrust Chamber Technology. Proceedings of the 6th International Symposium on Launcher Technologies, Munich, Germany, 2005.
- [49]H. Lüdeke and A. Filimon. Time Accurate Simulation of Turbulent Nozzle Flow by the DLR TAU-code. *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Vol. 92: New Results in Numerical and Experimental Fluid Mechanics V, Springer 2004, pp. 305-312.
- [50]P. Giese, R. Heinrich, R. Radespiel. Numerische Bestimmung von instationären Derivativa für Lifting Bodies mit Hilfe des Navier-Stokes Solvers FLOWer. *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Vol. 72, Vieweg 1999.
- [51]O. Bozic and P. Giese. Aerothermodynamic Aspects of Railgun-Assisted Launches of Projectiles with Sub- and Low-Earth-Orbit Payloads. *IEEE Transactions on Magnetic*, Vol. 43, No. 1, January 2007.
- [52]C. Böttcher, V. Hannemann, H. Lüdeke. Simulation of Magnetohydrodynamic Effects on an Ionised Hypersonic Flow by using the TAU code. Proceedings of the 15. DGLR Fach Symposium der STAB, Darmstadt, Germany, 2006.