MODELLING OF HYPERSONIC FLOW PHENOMENA

J.M.A. Longo
German Aerospace Center (DLR)
Lilienthalplatz 7, D-38108 Braunschweig, Germany
Tel.: +49 531 295 2643 Fax: +49 531 295 2320 Email: Jose.Longo@dlr.de

Forewords

Computational Fluid Dynamic is becoming a mature discipline in aerospace sciences. It is no longer only an analysis tool but ready to be used in all stages of design as well. The materials here presented review the growth and advances done in the last 10 years for hypersonic flow phenomena in continuum regime. The status of the physical modelling, code development issues such as algorithms, surface and field grid generation and validation data is provided. A number of applications examples for ascent vehicles and atmospheric re-entry vehicles are presented based on lessons learned at the German Aerospace Center, DLR, as well as experiences collected from past European technology programs. The materials here presented highlight future areas of research to enhance accuracy, reliability, efficiency, and robustness on modelling hypersonic flow phenomena.

1. Introduction

Current and expected developments in space transportation have led to growing interest in new space vehicles. Several expendable and partially or fully reusable concepts are discussed or already planned. These new vehicles require essential improvements over current vehicles in order to ensure economic viability and to fulfil mission and safety constraints. Therefore, a close interaction of all involved disciplines as well as the optimal use of all technical potentialities is necessary. Simultaneously design cycle times have to be reduced. The size and complexity of this problem has led to growing importance of numerical methods for design and optimization. Computational Fluid Dynamic (CFD) is the strategic tool which one day will enable to reduce dramatically the design and development-time required for new vehicles. A number of developments have contributed to this situation: increased robustness of CFD codes, lower computational costs, improvements in hardware as well as grid generation and more user-friendly post-processing tools.

High fidelity CFD results are today based on the solution of the Navier-Stokes equations, including appropriate physical models to account for viscous-turbulent and high-temperature effects. Although CFD can provide accurate prediction of vehicle surface pressure and temperature, it faces some challenges unique to the high-temperature, hypersonic environment. Equations for chemical and thermal non-equilibrium must be included. In the

Paper presented at the RTO AVT Lecture Series on "Critical Technologies for Hypersonic Vehicle Development", held at the von Kármán Institute, Rhode-St-Genèse, Belgium, 10-14 May, 2004, and published in RTO-EN-AVT-116.

boundary layer a combination of the viscous, thermal and diffusive transport phenomena that are functions of the chemical state of the gas must be modelled to simulate accurately the convective heat transfer. However, the potential of CFD is not restricted to model properly the vehicle's environment; 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. Surface and volume grids are often difficult and/or time consuming to generate. Perturbations to the configuration are not easily accommodated in the grid generation process, remaining one of the most time consuming aspects of numerical simulations.

Validation and error estimation are critical challenges within CFD because uncertainties in predicting vehicle performance increase design margins (and therefore add weight lowering vehicle performance). CFD error estimation is predominantly based on code validation experience. In any given design application, the code is validated against the experimental data that most closely match configuration and flight parameters. Under the ideal circumstances, 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. Furthermore, for geometrically complex configurations, obtaining a grid converged solution is either not possible or is precluded by higher demand in computer resources.

In the following sections key issues on physical modelling, numerical modelling and validation data for hypersonic flows are given. Future areas of research to enhance accuracy, reliability, efficiency, and robustness on modelling hypersonic flow phenomena are discussed based on a number of applications examples for ascent and atmospheric re-entry vehicles.

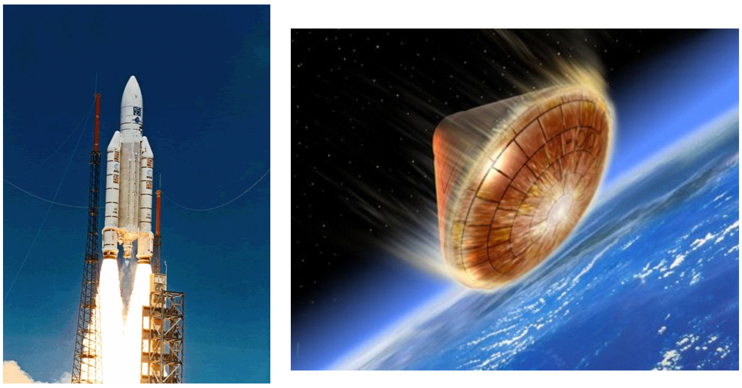

Figure 1.1 Start of the European launcher ARIANE 5 and atmospheric re-entry of the European ARD capsule (ESA courtesy).

2. Physical Modelling

In the last ten years, there has been a considerable progress in the modelling of hypersonic flow phenomena but still there is a need of improvement on several key issues [13, 19]. While the aim is to resolve at once the flow field surrounding a vehicle moving at hypersonic speed, for the purpose of the present discussion the difficulties to be confronted are classified into eight types. They are real gas effects; wall catalysis and ablation; radiation cooling effects; shock-wave boundary-layer interaction and transition; sneak flows; nozzle flows; turbulence and base flows; and propulsion. In the following, it is given only a short description of these issues without to establish any kind of ranking between them and without to forget that they are strongly coupled. For a deeper discussion, the interested reader is referred to the here cited literature.

2.1. Real gas effects (thermo-chemically reacting gas effects)

A real gas implies the existence of a gas in equilibrium, non-equilibrium, frozen or all of those states. This includes the possibility that a real gas can look identical to a perfect gas or a chemically-frozen gas. Real gas effects are important in hypersonic flows in terms of both their influence on aerodynamic performance and their effect on thermal loads. The use of a perfect gas assumption for re-entry hypervelocity problems may lead to a wrong estimation of the aerodynamic performance and the longitudinal trim, like in the case of the first flight of the US-Orbiter. Furthermore, real gas thermo-chemical non-equilibrium processes are also important for the determination of aerodynamic convective and radiative heating. They also affect the leeward side and base flow topology of the vehicle. The magnitude of real gas effects or the degree of gas dissociation affecting a vehicle depends on the vehicle shape and the flight path, i.e. altitude and velocity. Recently post-flight analyses using wind tunnels and CFD for vehicles like the winged US-Orbiter and the ARD (Europe) and OREX (Japan) capsules have identified real gas effects as the cause for the observed differences in aerodynamic performance [07, 55] or thermal environment [45] or both [51,67]. However, CFD and wind tunnels predicted only qualitatively those changes. For chemically frozen as well as chemical-equilibrium flows, there is only need for ground-based experiments for turbulent flows since no closure model is used in the Navier-Stokes equations in case of laminar flow. Between those limits, in the region of reacting-gas flows, CFD results can largely vary upon the thermo-chemical and turbulence models used [16, 11]. Consolidated data for CFD validation, obtained in high enthalpy facilities are today emerging [25 (**Fig. 2.1.1**)]. However, while high enthalpy wind tunnels like the HEG (DLR) or F4 (ONERA), among others, at the present offer the only means of producing both the total enthalpy and pressure levels representative for flight conditions beyond Mach 10, they do not precisely simulate flight environments. Indeed, they have a strongly limited operational range and therefore, their free stream conditions must be previously characterized.

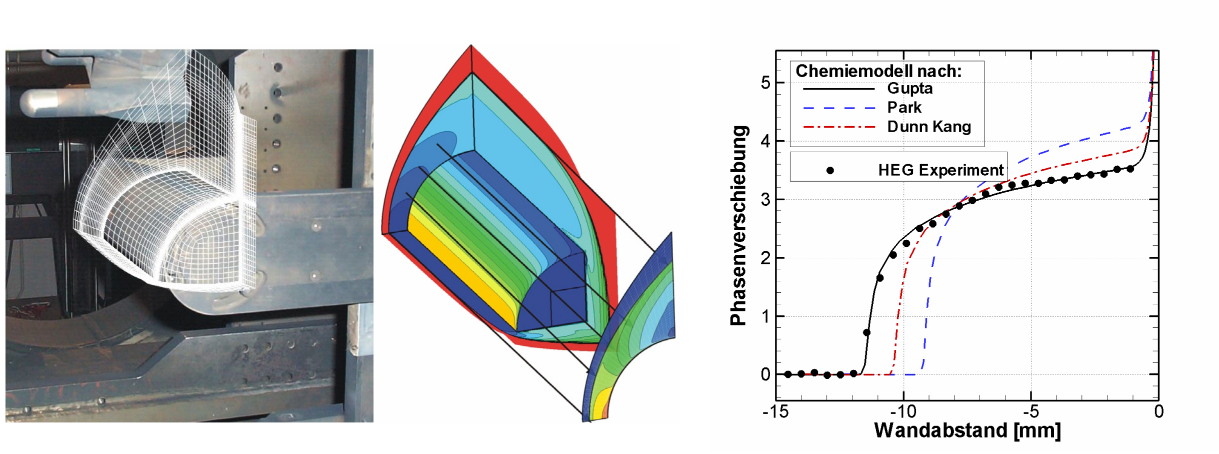

Figure 2.1.1 Shock standoff of a cylinder-model measured in the HEG high enthalpy facility compared against CFD solutions obtained with different transport coefficient models. Left: wind tunnel model with numerical grid superimposed; middle: CFD pressure field and surface solution; right: measurements (symbols) and computed values(dotted and continuous lines).

2.2. Wall catalysis and ablation

The study of the catalytic behaviour of aerospace materials is not a matter for fluid-dynamics specialists but its knowledge and consideration for the design of thermal protection systems (TPS) may severely affect the vehicle performances. Indeed, until now the design of TPS is made under the hypothesis of full catalytic wall conditions. It means that the flow is considered to be in an equilibrium state on the surface and that the catalytic properties of the material are always overestimated. This first approximation can be acceptable for vehicles that allow large margins in thermal loads, in practice non-reusable ones. However, having in mind that reduced wall catalysis can decrease the heat transfer by a factor of 2 or 3, the above assumption results in strong limitations for reusable space transportation systems and in particular for vehicles with high wing load where the additional TPS-weight increases unnecessarily the wing load factor. For reusable vehicles, the minimum requirement for re-usability is that the operational envelope of the material reliably exceeds the conditions it is exposed to during re-entry. Navier-Stokes results using global catalytic models for configurations with TPS based not only on SiO_2 but also SiC are today available [08, 33]. While ceramic materials proved so far to have excellent oxidation characteristics for typical re-entry profiles, insufficient knowledge of the true local parameters may lead to “active” oxidation, i.e. rapid surface degradation. The most important parameters defining the oxidation mode are the local temperature, pressure, gas composition and flow conditions. Furthermore, the plasma-surface interaction, and particularly the specific surface recombination reaction of atomic oxygen, depends strongly on the chemical state of the surface material. In fact, the reactive flow interaction should be different if the material is already oxidized or not [26, 35]. New detailed catalytic models that account for active/passive oxidation effects are now emerging [24 (**Fig. 2.2.1**)].

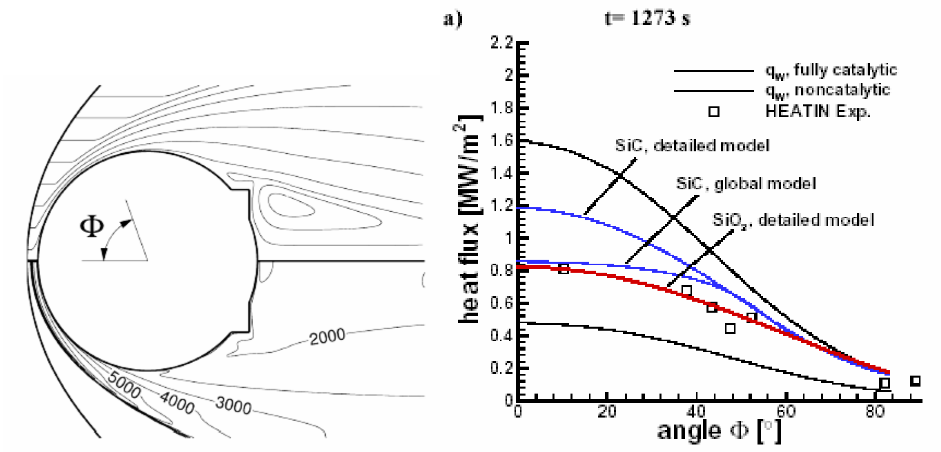

Figure 2.2.1 Computed heat flux densities and SiC-recombination coefficients at the surface of the MIRKA capsule compared with heat flux densities values measured during re-entry flight. Left: capsule geometry and reference coordinates; right: measurements (symbols) and computed values (dotted and continuous lines). (Courtesy IRS).

Ablation of aerospace materials is also not a matter for fluid-dynamics specialists but until today it is a low cost TPS solution and therefore used in almost all capsule-shape vehicles. Ablation during re-entry is dictated mostly by radiative heating. Convective heat-transfer rates are nearly zero because the outward flow of ablation products prevents the conductive heat flux, due to the hot regions, to reach the wall. The product gas of ablation forms a layer, called ablation-product layer, which prevents that the hot shock layer gas reaches the wall. The ablation-product layer absorbs a portion of the radiative flux directed toward the wall. In order to accurately predict the extend of ablation of the heatshield, one must accurately predict the thickness of the ablation-product layer and the thermochemical state therein. For cone-shaped nose tips, one-dimensional analysis provides rather accurately results. However, for capsule shapes besides problem like chemical and turbulence models, bursting process is the most critical one. Indeed, the burst particles may penetrate deeply into the inviscid region of the shock layer, vaporize, and absorb or emit radiation therein. Models of such environment are not yet available [05, 34]

2.3. Radiation-cooling effects and thermal-fluid coupled analysis

Thermal surface effects and their implications on hypersonic vehicle design are being a matter of more attention by the scientific community. An overview of the overall problem is found in [27]. Surface radiation cooling can be considered as the basic cooling mode of high-speed vehicles operating in the earth atmosphere at speeds below approximately 8 km/s. Under this condition, typical for the re-entry flight from a low earth orbit, emission and absorption processes in the air stream can be neglected. Radiation cooling involves very interesting fluid mechanical problems with strong implications for the design of hypersonic vehicles. One should take into account the important coupling between the radiation-cooled surface, the flow field and the heat loads. Indeed, the thermal state of the vehicle surface affects or interrelates with the boundary layer, the shock interaction phenomena and if thermal and/or chemical non-equilibrium is present, also with the catalytic surface recombination. Today is possible the simulation of surface radiation cooling effects in laminar flows by means of CFD. For turbulent flows, heat-transfer as well as mass-transfer is mainly regarded by choosing a constant turbulent Prandtl number. On the experimental side, arc-jet facilities operate at very low Reynolds numbers and experimental validation of that phenomenon in high-enthalpy wind tunnels is not possible as long as the surface of the model is not previously heated. However, basic experimental works on the influence of the surface temperature and the surface-near temperature gradient on the properties of turbulent flows are now emerging [47].

Besides radiation cooling, thermal-fluid coupled analyses are also important. Particularly for geometries with small radius, like leading edges, gaps or slots, coupled thermal-fluid solutions exhibit notoriously reduced levels of heating. Indeed, the radiation adiabatic wall assumption is a good hypothesis everywhere except at the edges of gaps where the differences on temperature between coupled and uncoupled solutions may amount almost a factor 2. While first CFD results of coupled thermal-fluid analysis have been reported in the past [18], today considerable progress are being made in the development of the coupled computational tool [38, 42 (**Fig. 2.3.1)]**.

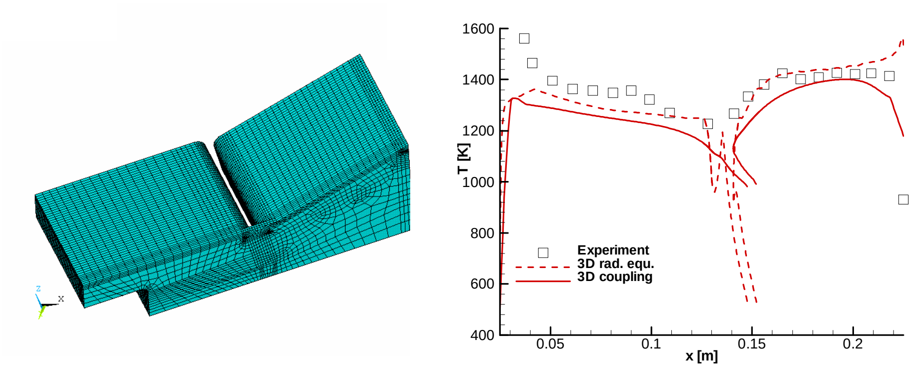

Figure 2.3.1 Surface temperature distribution measured in the arc jet facility L3K for a wedge compression corner model with closed gap compared with computed values obtained without and with fluid-thermal coupled analysis. Left: model geometry; right: measurements (symbols) and computed values (dotted and continuous lines).

2.4. Shock-wave boundary-layer interaction and Transition

The shock waves originated by the vehicle flying at hypersonic speed are the origin of interference phenomena resulting, first from the intersection between shocks, and second from their interaction with the boundary layer developing on the vehicle surface. Such interactions may induce separation of the boundary layer. In high-enthalpy hypersonic flows, the subsequent reattachment of the separated shear layer gives rise to the heat transfer which can be far in excess of those of an attached boundary layer. Laminar/turbulent-transition prediction in hypersonic flows remains one of the most important unresolved fluid dynamic problems [54, 61]. Because of their great practical importance, these phenomena have been extensively studied during the past 50 years and are still the subject of investigations due to their extreme complexity. Indeed, its importance for vehicle-design lies in the fact that, transitional thermal loads are about 3 times larger than laminar ones, exceeding also the fully turbulent thermal loads. While the transition phenomenon can be triggered by sources of different nature like flow instability, stream wise vortices, surface roughness or TPS-misalignments, it involves, in all cases, complex mechanisms affected by, among others, Mach and Reynolds number, real gas and wall catalysis effects. On deflected control surfaces, the resulting concave curvature in stream wise direction induces a centrifugal imbalance within the viscous layer resulting in stream wise vortices also called Görtler vortices [69]. 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.

Since the transition modelling process remain rather complex, semi-empirical criteria have been used to predict the onset of transition. Most of these criteria were obtained and calibrated in cold gas facilities, using wind tunnel models with distributed surface roughness [04, 09]. For cold hypersonic flow-problems, transition-onset prediction using CFD coupled with semi-empirical criteria based on boundary-layer stability has been recently reported [57, 43]. Furthermore, CFD Navier-Stokes codes have been successfully applied to predict Görtler vortices under laminar and turbulent conditions [40]. However, the extension of those research efforts to cover the hot hypersonic flow regime is not easy without flight measurements [03]. Indeed, there is today a relative large amount of experimental data available from several European and US facilities to assess the efficiency and heating of control surfaces for Mach numbers up to 10. The situation in the high enthalpy flow regime is not the same, there the amount of available experimental information is scarce and the comparison between CFD and WT results is not completely free of questions [53, 15]. Here also the unsteady character of the interacting process requires more attention since it seems to be a key factor. High enthalpy, low Reynolds number, transitional flow is a recent identified problem that requires more research effort to be understood [25 (**Fig. 2.4.1**)].

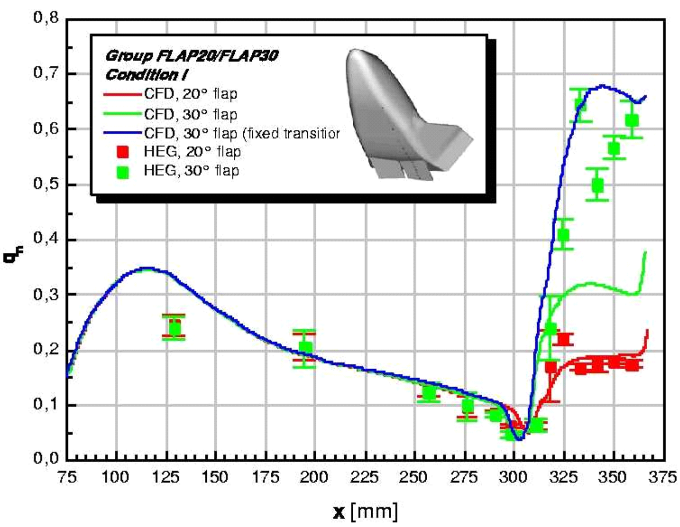

Figure 2.4.1 Heat-flux distribution measured for an X-38 model in the high enthalpy tunnel HEG (symbols) for two different flap deflections compared with CFD predictions done assuming, for the 20^o flap deflection, laminar flow, and for the 30^o case laminar flow and transition fixed at the flap hinge line (i.e. assuming turbulent flow on the flap).

2.5. Sneak flows

Many of the aerodynamic heating problems encountered by practical re-entry vehicles are not represented by small-scale wind tunnel models employing idealized and smooth loft line surface contours and are extremely complex to be resolved by CFD. They are the motivation of what today is known as local aerothermodynamics, and as examples one can mention leakage effects, bleed flow around control surfaces and surface distortion as [60 (**Fig. 2.5.1**)].

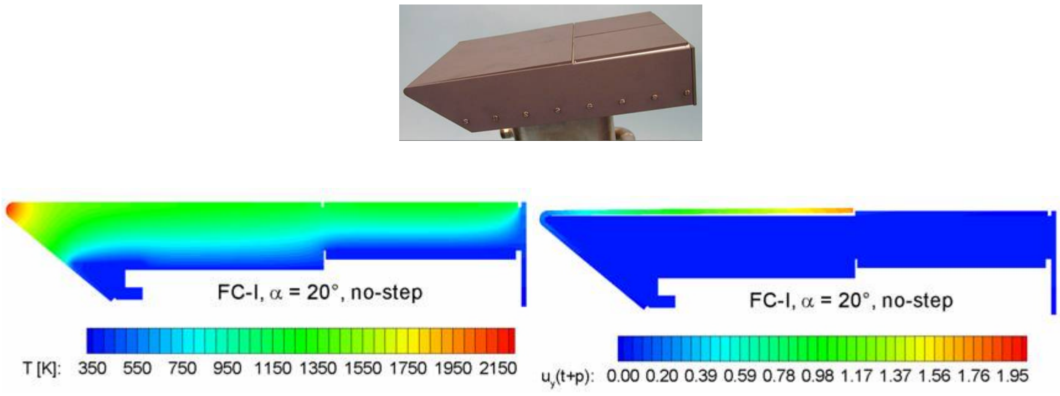

Figure 2.5.1 Temperature distribution measured in the arc jet facility L3K for a generic nosecap model and computed simulations without and with leakage effects. Top: wind tunnel model with first row of TPS behind the generic nosecap; bottom left: CFD solution without leakage between nosecap and TPS; right: CFD solution allowing leakage underneath the nosecap.

Due to imperfections of the outer surface of flight vehicles, resulting from manufacturing accuracy and/or mechanical and thermal loads during flight, leakage flows are present often underneath the TPS tiles or into cavities such as reacting controls systems (RCS) or leading edge panels. In the case of aerodynamic control surfaces, it is impossible to seal completely the gap between the control surface and the configuration. The resultant bleed of high-energy airflow between the adjacent control surfaces may cause critical heatshield design problems not entirely because of the high heating rates but also because of the lack of radiation relief of the surfaces caused by very low radiation view factors between adjacent surfaces [70]. The solution of those types of problems, which can jeopardize the vehicle integrity, requires, even today, the use of specific transposition to flight modelling techniques with large security margins. On the experimental side, the arc-jet facilities allow testing only partial configurations and it is not possible to match simultaneously all flight parameters. CFD solutions, on the other side, suffer not only the lack of flexibility of the grid generation technique to handle properly the geometrical configuration details but also at the solver level, new developments are required to handle micro-scale phenomena [06].

2.6. Nozzle flows

One of the major concerns of the aerospace industry is the overexpansion regime of future high area-ratio nozzle, in particularly flow separation and consequences on side-loads and thermal loads. The most significant parameter for aerodynamic performance of propulsion engines is the area ratio of expansion of the nozzle. Presently the area ratio is limited by the need for attached flow conditions for all ambient pressures that occur during ascent of a vehicle. Further increase of the area ratio would result in flow separation for the highest ambient pressures, which, based on present structural properties of nozzles, may lead to mechanical failures due to large side loads generated by asymmetric flow separation in the nozzle [62, 59]. 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 the pressure distribution on the vehicle surfaces surrounding the exhaust plume [28, 48]. The supersonic jet emanating from the nozzles interacts with the external main flow around the body. In the interaction zone a turbulent mixing layer, a re-circulation region and a shock system, plume shock/barrel shock, and reattachment regions with considerable heat loading may be formed. These perturbations, mostly of unsteady character, lead to interaction forces that must be accurately predicted in order to obtain the desired vehicle performance. Simulations of such phenomena in wind tunnel are not simple. It is difficult, when not impossible, to match all variables at the same time, i.e. flight-path parameters and jet-flow parameters. Here CFD can provide an important contribution in the elaboration of extrapolation models for flight conditions. However, it requires the use of unsteady formulations for both turbulence and chemistry models [10, 20 (**Fig. 2.6.1**)].

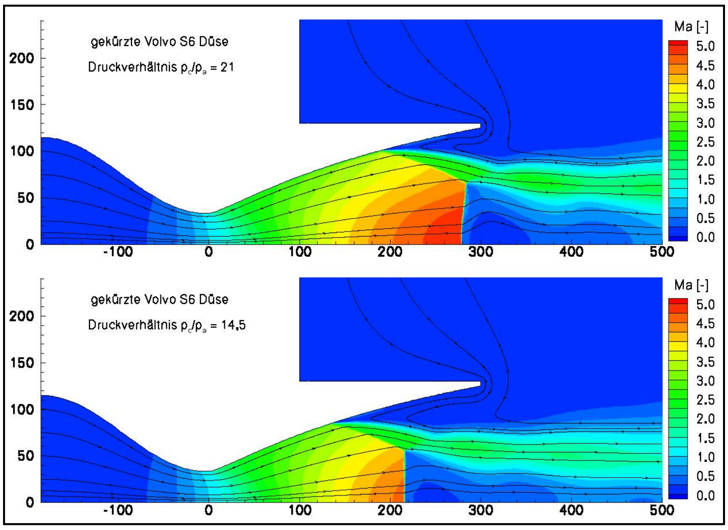

Figure 2.6.1 Time accurate numerical simulation of nozzle flow. Pressure field, Mach disk and stream lines visualization for two different time steps.

2.7. Turbulence and base flows

Turbulent flow is one of the oldest unresolved fluid dynamics problems and hence, there is no need to address its importance for a vehicle design here. In addition, turbulent flows are present in almost all of the previously described unresolved issues and therefore, any improvement in the knowledge of this phenomenon will positively influence the understanding of the other problems. Improvements are achieved via direct numerical simulation and wind tunnel experiments equipped with non-intrusive measurement devices. Based on those results new turbulence models are under development. However, one can say that except in transonic and, moderately, supersonic flows, experimental results for hypersonic turbulent flows are scarce. In addition, attention should be given to unsteady flow phenomena [Sinha]. Today there is a lack of physical understanding on turbulent reacting supersonic as well as hypersonic flows particularly if such flows have unsteady character. Most of the experimental facilities require artificial devices to trigger turbulent flow (turbulence generators) and is hard to guarantee that the flow becomes turbulent everywhere. Re-laminarization of the boundary layer due to flow expansion may happen on configurations exhibiting local flow separations. On the CFD side, the turbulence models used are, very often, straightforward extensions of models constructed for incompressible flow. Indeed, the Baldwin-Lomax model [02], one of the most popular turbulence models in the aerospace community due to its simplicity, was developed for 2D transonic flow with small separations but it is used to describe 3D hypersonic flows with massive flow separations! [12].

The flowfield and heat transfer rate distribution in the base region of space vehicle is difficult to predict because crossflow, vortex flows, turbulence, inviscid shear flows, entropy layers and temporal instabilities affect the behaviour of such flows. In high-enthalpy flows, these effects are combined with chemical reactions. There are various possibilities to model such flows. These range from Reynolds Average Navier-Stokes to Large Eddy Simulation, to Direct Numerical Simulation. The Reynolds Averaged Navier-Stokes approach (RANS) attempts to solve for the time-averaged flow. This means that all scales of turbulence must be modelled. Since the large scales for separated flows are very dependent on the geometry, RANS models often fail to provide accurate results for these flows in particular when the flow is unsteady. However, they lead to good prediction of attached flows. On the other side, Direct Numerical Simulation attempts to resolve all scales of turbulence, i.e. from the largest to the smallest, the grid resolution requirements are very high, and scales rapidly with Reynolds number. Large Eddy Simulation attempts 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 far too large in order to be used for practical problems [65, 71]. Recently hybrid turbulence models, so-called Detached

Eddy Simulation (DES) models are emerging combining the advantage of RANS models close to the wall and LES models in the core flow [41 (**Fig. 2.7.1**)].

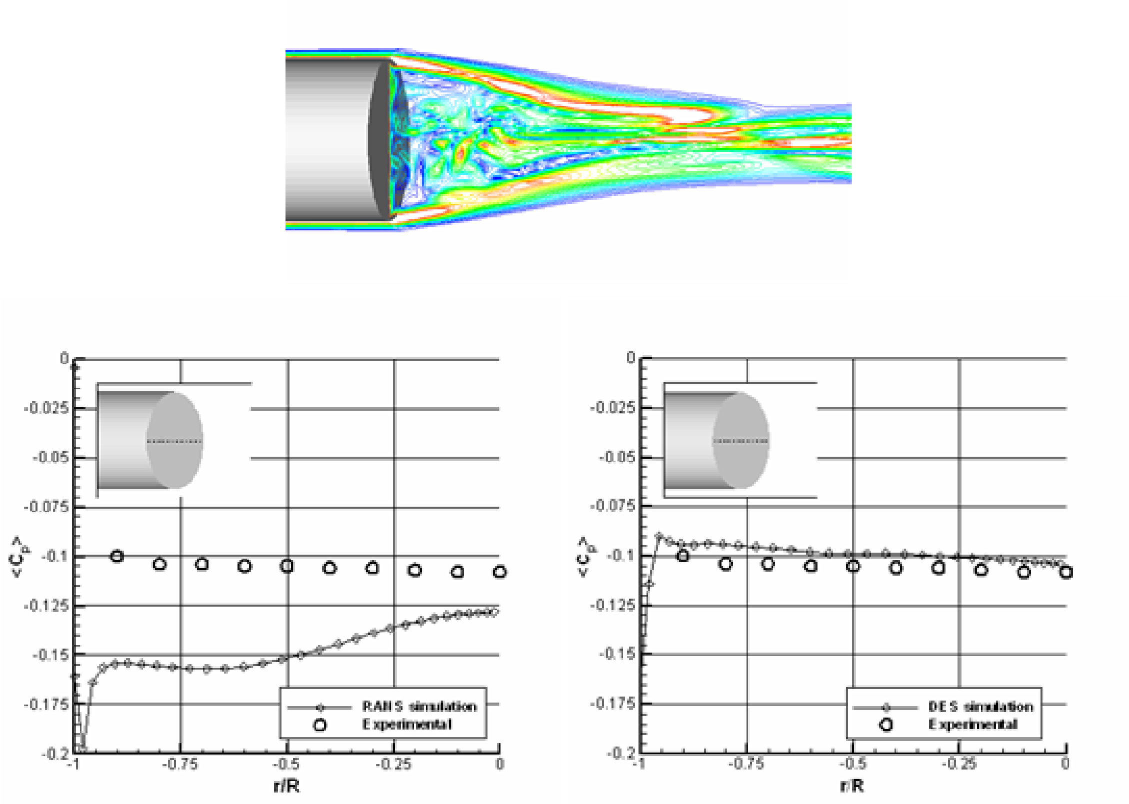

Figure 2.8.1 Numerical simulation for a supersonic turbulent wake of cylinder. Top: local distribution of the turbulent kinetic energy. Bottom: base pressure measured (symbols) and computed (continuous line) using different turbulence models. Left: Reynolds average Navier-Stokes. Right: Detached Eddy Simulation.

2.8. Propulsion

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. The primary requirement from CFD 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 at hot-fire conditions. 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 [17, Kassal]. Indeed, multiphase combustion modelling 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, if a factor of 10 increases 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 these models [56]. 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, droplet collision and break-up models, sub- and super-critical droplet vaporization models, turbulent dispersion models, and turbulent chemistry interaction models.

3. Numerical Modelling

CFD has the greatest prospects of the aerospace community by means of it, it might be possible to optimize one day the wind tunnel and the costly flight experiments. However, there is still a long way to go, to achieve this goal. In the last years, only modest progress has been made on the algorithms used in hypersonic CFD codes. However, significant progress has been achieved on surface and flow field discretization using structured and unstructured grid methods, and significant gains in efficiency are being obtained through the increasing use of parallel computers. Computers are the tools of CFD and they still drive its progress. At present the major contributions of CFD in a design process are to provide at early stages of the project with preliminary data; to contribute to the analysis of flow anomalies detected in ground based facilities and / or during flight experiments and to provide data for such flow regimes where no ground-based facility is available. Also foreseeing in a coming future is the application of CFD in multidisciplinary design optimization tasks. The following sections shortly discuss issues on grid generation, computational algorithms, multi-design optimization and hardware requirements. For a deeper discussion, the interested reader is referred to the here cited literature.

3.1. Grid Generation

Surface modelling and grid generation appear to be the most significant contributors to the total time required to generate a CFD solution. 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. Even though tools have gained great sophistication, they remain cumbersome and restrictive, and require skills that are not generally available in researchers and designers. The efficiency and accuracy of the interfaces between different Computed Aid Design systems used for surface modelling must be improved as well as the treatment of often-imperfect data. Today an extensive array of block-structured methods, unstructured grid methods, and hybrid schemes are now 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 labour intensive in terms of grid generation. Indeed, structured grid generation for relatively complex configurations requires a minimum of six to twelve weeks of effort, even if CFD and grid generation specialists are working together [50]. 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 relative speed and low need of user interaction in the grid generation process. 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 (**Fig. 3.1.1**). In comparison with structured grids, unstructured grid generation requires for the same configuration approximately twelve times less time but at the present the performance of the numerical algorithms for unstructured and/or hybrid grids are about ten times slower than those for structured ones [23].

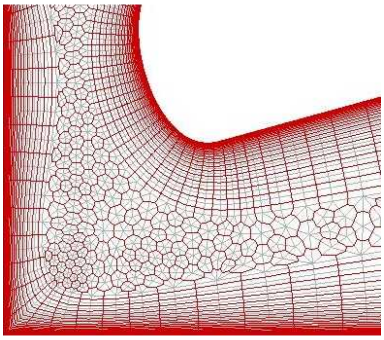

Figure 3.1.1 Hybrid grid for the analysis of internal flows. The number of different boundary parts for the created mesh is fixed by the geometry-data file. Initial point distributions on surfaces and far field boundaries can be controlled by input parameters. Growing rates, shapes of boundary layers etc, are also user-controlled input parameters.

3.2. Computational Algorithms

From a numerical point of view, there are unique challenges to the hypersonic flow environments. A simple test case includes flow domains with strong contrasts [39 (**Fig. 3.2**)]. 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-differences or flux-vector-splitting, undoubtedly have become the main spatial discretization techniques adopted into nearly all major research and commercial codes. As CFD is being used more routinely, the need for maximizing accuracy, efficiency and robustness for a wide variety of problems remains the foremost concerns. In spite of the progress achieved, deficiencies like post-shock overshoots and pressure oscillations along the transverse direction in the boundary layer continue being a problem [37]. In addition, numerical schemes of order of accuracy greater than two are necessary to handle properly, wave propagation problems. In general, the flow equations are solved to steady state by properly posed, time dependent problem and marching the solution to large time with steady state boundary conditions. The algorithms that are in use for time-stepping can be classified as either explicit or implicit schemes. 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 they relatively easy coding. The most extensively used explicit method is the Runge-Kutta time stepping one, originally introduced to the solution of the Euler equations together with residual smoothing to extend its stability limit [29]. 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 [44]. 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 [30]. 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, such as the stagnation region of a blunt body. Finally, 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 [32].

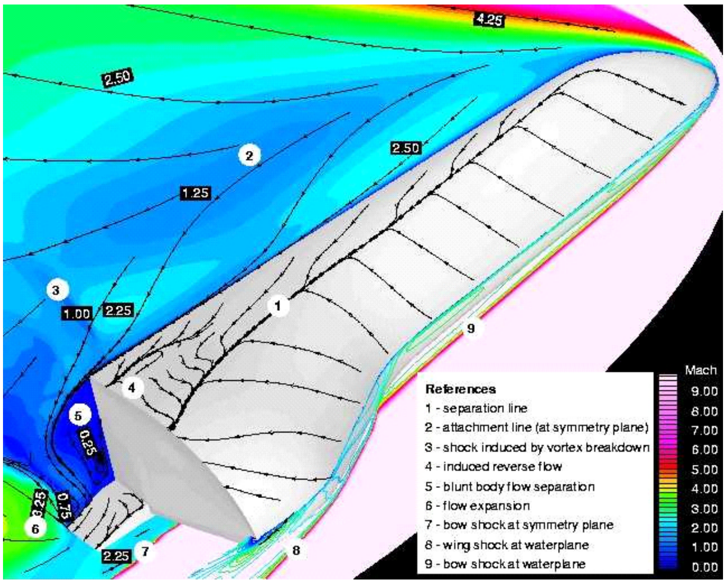

Figure 3.2.1 Numerical simulation for the flow field around a vehicle moving at hypersonic speed, Mach = 10, 40^o angle of attack. Flow domains that vary from subsonic stagnation regions, with high-pressure and temperature, behind a strong bow-shock wave to high-speed low-density, high Knudsen-number, regions at the lee side of the configuration are observed. Embedded shocks and vortices and a large separated base flow complete the picture.

3.3. Multi-design optimization and Hardware

Faster numerical schemes, programmed to take advantage of today's highly clustering computer systems (above 800 nodes) in combination with faster and highly user friendly grid generation systems are required. The great challenge of CFD is to provide solutions in the order of minutes so that CFD may be an intimate and integral part of the design process. In general multi-design applied methods can be divided based on their orientation into multidisciplinary methods and disciplinary methods. Multidisciplinary methods are used mainly in the pre-design stage for a rough assessment of the vehicle characteristics and a preliminary interaction of the involved disciplines. Their computational effort is low because of the relatively simple models used for aerodynamic, structure, etc [64, 22]. However, most methods for hypersonic flows are a kind of disciplinary methods. Disciplinary methods are used mainly in the design stage for an exact calculation and improvement of the vehicle characteristics. Their computational effort is significant because of the sophisticated models used. In addition, classical optimization algorithms used can be divided based on their underlying strategy into algorithms with heuristic strategy and algorithms with deterministic strategy. Algorithms with heuristic strategy work with help of the coincidence. The parameters are changed unsystematic within the constraints and only the parameters, which fulfil the demands formulated through the objective function best, are stored. Algorithms with deterministic strategy work with information about the gradients of the objective function. The gradients and with this the direction, where an improvement of the objective function can be expected, are calculated with finite differences or adjoint equations. The parameters, which fulfil the demands best, are located by parabolic extrapolation or Newton method. This continues until gradients cannot be found anymore. Algorithms with heuristic strategy guarantee a certain determination of global extreme but require a significant effort. Algorithms with deterministic strategy cannot guarantee a certain determination of global extreme in case of complex parameter spaces but require less effort [66 (**Fig. 3.3.1**)]. The adaptation of proven computational capabilities combined with new approaches to produce the next generation of computational tools for design and analysis is being considered today. A limitation in this approach is the lack of a standardized, robust, efficient data format to be used in the exchange of information between the various stages of a numerical simulation, i.e. grid generation, solver and post-processing, using either in-house or commercial simulation tools. To overcome that problem efforts like the CFD General Notation System, to provide a standard for recording and recovering computer data associated with the numerical solution of the equations of fluid dynamic are emerging [31, 36, 58].

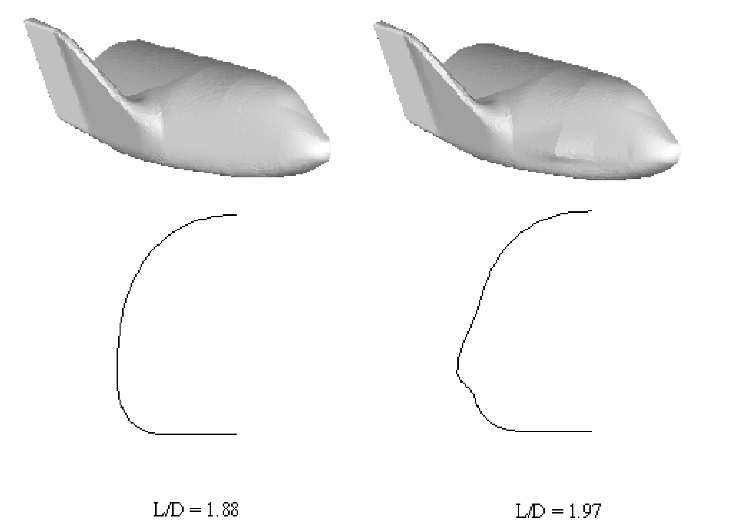

Figure 3.3.1 Optimization of the subsonic L/D value with restricted geometry changes for a generic re-entry vehicle. Left: original configuration; right: optimized configuration. The resulting configuration shows an added geometrical bump that improves the L/D in 5%.

From the hardware point of view, the requirements are processing speed, storage capacity, pre-processing capability, graphic post-interchange, and communication bandwidth for the exchange of data and for the use of remote facilities. Computer speed has grown in the last decade by an order of magnitude every five years, a trend that will continue. Central memory has historically growth with computer speed as about one byte per flops. Mass storage and the associated search and data base software however, have grown slower. Indeed, semi-archival storage should be at least 1000 times larger as today.

4. Validation Data

Although CFD simulations are widely conducted in industry, government and academia, there is less agreement on procedures for assessing their credibility. There is no fixed level of credibility or accuracy that is applicable to all CFD simulations since the accuracy level required of simulations depends on the purpose for which the simulations are to be used. The two main principles that are necessary for establishing credibility are code verification and validation. Validation is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model [01]. To accomplish its mission, CFD strongly requires good wind tunnel and flight data to be used for code validation and when necessary code calibration too. Only validated CFD codes should be used to predict flow phenomena. According, there are two main sources of CFD errors: the code numerical accuracy and the accuracy of the physical models used in the code. The numerical accuracy of a computer code hardly depends on the correct numerical solution of the necessary set of equations required by the problem. This process of determining that a model implementation accurately represents the developer’s conceptual description of the model and the solution to the model is called code verification. Solutions are regarded as grid independent when the observed changes of the results between two consecutive grid densities are less than the required accuracy. Grid convergence studies, i.e. start computing on a large -fine- grid and then reduce the size of the grid by neglecting every second point in each co-ordinate direction while monitoring the solution, are a necessary but insufficient metric for the estimation of numerical accuracy errors. Comparisons of numerical solutions with experimental data are also necessary but not enough to determine physical modelling errors. Indeed, numerical accuracy and physical modelling errors are coupled because the physical models are function of flow parameters and their gradients, which are function of grid resolution. In addition, for hypersonic problems is very difficult to de-couple effects. For example, surface radiation cooling influences directly the vehicle heat loads but also the state of the boundary layer, i.e. laminar or turbulent, which in turn will have again an effect on the heat loads. Therefore, confidence in CFD predictions depends ultimately on comprehensive comparisons with experimental data with well-defined error bounds (**Fig. 4.1**).

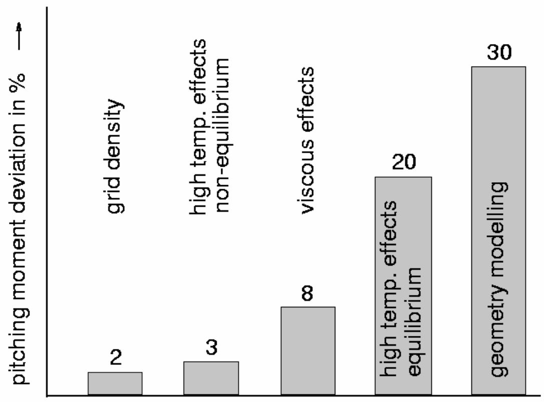

Figure 4.1 Summarizes the impact several source of errors on pitching moment prediction. Viscous effects are addressed by comparing Euler with Navier-Stokes solutions. High temperature effects are obtained comparing solutions for equilibrium flows with perfect gas flows. The change in pitching moment due to non-equilibrium effects exhibits the difference with respect to equilibrium flows. The figure shows that the high-temperature equilibrium-flow effects are extremely important. However, the largest contribution to the error band results from neither the physical modelling nor the discretization error but from the geometrical representation of the respective configuration.

Associated with the verification and validation procedures, one should define code numerical uncertainties and the uncertainty of the physical models respectively. Methods for prediction the magnitude of uncertainty or for bounding the magnitude of the error are required for any verification and validation procedure. The concept of an error band assumes that, within this band, the value of the true numerical solution error will occur with 95% confidence or 95 times out of 100. Therefore, it is important to quantify the error bands and understand the sources and mechanisms of the errors. The impact of these uncertainties is a separate consideration and more the concern of the customer of the CFD predictions but certainly, with such process the current practice of adjusting computations to match experimental findings could be greatly reduced. Best practices for reduction of uncertainties in CFD results are emerging [46, 14].

There are two main sources for CFD validation data: wind tunnel tests and flight experiments. Wind tunnels are the major source of flow data. They are very important because they allow „controlled" simulations and therefore a better understanding of the flow physics. Although wind tunnels are a useful tool for CFD validation and physical understanding of flow phenomena, today it is recognized that the simulation of all flight conditions in a wind tunnel is not possible. Particularly, flight Reynolds numbers and flight Mach numbers associated with high enthalpy flows are critical to simulate. Wind tunnels have also limitations inherent to the type of facility, operation form (open or close, continuous or blow down, etc) and instrumentation used. Furthermore, in the future to enhance data credibility several factors must be quantified as for example: radial and axial flow uniformity; partial flow blockage phenomena; sting interference effects; flow particle levels; vibration excitation and possible departure from equilibrium; flow liquefaction; free-stream disturbance level due to acoustic disturbances radiated from the nozzle; etc. [52, 68, 21].

In flight-measurement constitutes the only way to obtain data for prediction tools validation and calibration under real conditions and therefore, they are irreplaceable for CFD validation [49]. Requests for flight data range from classical like surface pressure, temperature and heat flux, to more specific ones like control surface efficiency, catalytic behaviour of the TPS, knowledge of the real state of the gas surrounding the vehicle, active/passive oxidation or boundary layer flow status (i.e. laminar, turbulent or transitional). However, flight measurements are expensive, they require considerable time for preparation and their complete repeatability is not always possible. Indeed to repeat a same fly 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. Indeed, e.g. measurements to detect boundary layer transition strongly depend on boundary layer contamination, a phenomenon not always easy to quantify. Furthermore, while flight experiments are generally done with sufficient instrumentation to measure local flow phenomena, they are not sufficient to accurately predict global phenomena such as laminar-turbulent boundary layer transition or surface catalysis effects.

5. 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 academy. However, the tremendous loose of momentum in space programs, worldwide in the last decade, has severely affected new developments on modelling of hypersonic flow phenomena. Moderately progress has been only achieved in most of the research areas. While high fidelity

CFD results are today based almost on the solution of the Navier-Stokes equations, they still require validated models to account for viscous-turbulent and high temperature effects. In the past, validation data have been achieved as secondary product of expensive space-transportation programs. Since in the last ten years there has been almost no successful program due to lack of investment, no new experimental data are available. According, a new trend is emerging for low cost technology validation in flight based on the use of old military rockets.

In the last ten years, a major problem of concern on the physical modelling side is the availability of experimental data for model validation in the hot hypersonic regime. Probably the most notoriously progresses have been on the side of thermal protection systems. There, new catalytic models accounting for active / passive oxidation and computational tools for fluid-thermal coupled analysis are starting to evolve. Based on the few flight experiments carried out in the past decade, real gas effect could be confirmed. Progresses in nozzle flow, transition and turbulence have been droved by advances in the treatment of unsteady flows. Indeed, unsteady flow phenomena are becoming more attention and since they are present in almost all the hypersonic problems, any improvement in this area positively influence the understanding of the other problems.

On the numerical modelling side, only modest progress has been made on the algorithms used. Up-wind schemes have become the main spatial discretization in hypersonic CFD codes while significant progress has been achieved on surface and flow field discretization. Today an extensive array of structured, unstructured and hybrid grid methods are available. Taking advantage of the tremendously grows on speed and capacity of today's computer systems, multidisciplinary and multi-design optimization methods and 3D simulation of combustion processes are emerging. According, standards for communication between different systems are evolving.

Probably on the side of code verification and validation, one may found the largest progress. Today, there is a common sense that progress in hypersonic modelling would be achieve only by integrating the working teams with specialist of different areas, particularly seating together experimentalist and numerical specialist. In that sense, procedures and recommended practices for assessing the credibility of the CFD simulations are starting to appear.

Acknowledgement

The author is grateful to numerous colleagues, particularly the DLR Spacecraft Branch, who obtained and provided most of the results for presentation in this paper.

References

[01] "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations," AIAA G-077-1988 Guide, January 1988.

[02] Baldwin, B. S., Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA paper 78-257, January 1978.

[03] Bertelrud, A., Budd, J., Sollberger, D., Churchward, R., Smilg,L., Stowers, J., "A Hypersonic Flight Test to Document Laminar Turbulent Transition," AAIAA paper 99-4849, November 1999.

[04] Berry, S.A., Bouslog, S.A., Brauckmann, G., Caram, J.M., "Boundary Layer Transition due to Isolated Roughness: Shuttle Results from the LaRC 20-Inch Mach 6 Tunnel," AIAA paper 97-0273, January 1997.

[05] Bouilly, J.M., His, S., Macret, J.L., "The ARD Thermal Protection System," Proceedings of the AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 1999.

[06] Boyd, I.D., Sun, Q., Martin, M.J., "Simulation of Micro-Scale Aerodynamics," AIAA paper 2003-0440, January 2003.

[07] Brauckmann, G.J., Paulson Jr.J.W., Weilmuenster, K.J., "Experimental and Computational Analysis of Shuttle Orbiter Hypersonic Trim Anomaly," Journal of Spacecraft and Rockets, Vol.32, 1995, No. 5, pp 758-764.

[08] Brück, S., Kordulla, W., Eggers, Th., Orlowski, M., Longo, J.M.A., "The Effect of Catalycity on The Heating of The X-38 Shape, Proceedings of the Eighth Annual Thermal and Fluids Analysis Workshop Spacecraft Analysis and Design," University of Clear Lake, USA, Sept. 1997.

[09] Charbonnier, J.M., Dieudonne, W., Boerrigter, H., "Simulation of Transitional and Turbulent Boundary Layer Flow on Blunted Geometry in Hypersonic Flow, Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 291-297.

[10] Châtelain, A., Hadjadj, A., "Computational Study of a Plume-induced Flow Separation on a Boattailed Afterbody," Proceedings of the 4th European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-487, 2001, pp. 571-578.

[11] Chauveau, S.M., Kelley, D.J., Laux, Ch.O., Kruger, Ch.H., "Vibrationally Specific Modelling of Nonequilibrium Effects in Air Plasmas," AIAA paper 2003-0137, January 2003.

[12] Cousteix, J., Arnal, D., Aupoix, B., Gleyzes, C., "Recent Studies on Transition and Turbulence at ONERA-CERT," AIAA paper 91-0332, January 1991.

[13] Deiwert, G.S., "Classification of Aerodynamic and Aerothermodynamics Issues and Problems," Proceeding of the Hypersonic Experimental and Computational Capability, Improvement and Validation Conference, AGARD AR-319, Vol. I, 1996, pp 1-1 1-6.

[14] Delery, J., "Experimentalist's requirements for a safe methodology in CFD code validation," Proceedings of the Symposium "Integrating CFD and Experiments," Glasgow, UK, September 2003.

[15] Dong, H., Zhong, X., “Numerical Simulation of Transient Growth in a Mach 15 Boundary Layer Over a Blunt Leading Edge,” AIAA paper 2003-1266, January 2003

[16] D’Angola, A., Capitelli, M., Colonna, G., Gorse, C., “Transport Properties of High Temperature Air in Local Thermodynamic Equilibrium,” Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 253-259.

[17] Ebrahimi, H.B., “An Overview of Computational Fluid Dynamics for Application to Advanced Propulsion Systems,” AIAA paper 2002-5130, Sept. 2002.

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

[19] Enzian, A., Devezeaux, D., Mohamed, A., Thivet, F., Tran, Ph., Tribot, J.-P., “Flight Experiments to Address Unsolved Aerothermodynamic Issues for a Future European Reusable Space Launcher,” Proceedings of the AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 1999.

[20] Filimon, A., Lüdeke, H., “Stationäre und zeitgenaue Untersuchungen Turbulenter Düsenströmungen anhand der gekürzten Volvo S6 Düse,” DLR IB 124-2004/08, March 2004.

[21] Fluid Dynamic Panel Recommendations, Proceedings of the Aerospace 2020 Conference, AGARD AR-360, Vol. III, 1997, pp. 15 – 32.

[22] Foster N.F. et al., “Three-Dimensional Aerodynamic Shape Optimization Using Genetic Evolution and Gradient Search Algorithms,” AIAA Paper 96-0555, January 1996.

[23] Frink, N.T., “Tetrahedral Unstructured Navier-Stokes Method for Turbulent Flows,” AIAA Journal, Vol.36, No. 11, 1998, pp. 1975-1982.

[24] Frühauf, H.-H., Infed, F., Fertig, M., Olawsky, F., “Thermal Loads for the Experimental Vehicle X-38,” Proceedings of the 4th European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-487, 2001, pp. 529-535.

[25] Hannemann, “High Enthalpy Flows in the HEG Shock Tunnel: Experiment and Numerical Rebuilding,” AIAA paper 2003-0978, 2003.

[26] Hilfe, G., “Ceramic Thermal Protection Materials – How far can we go? New Aspects on the Oxidation Behaviour During Re-entry Flight,” Proceedings of the AAAF 2nd International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 2001.

[27] Hirschel, E.H., “Thermal Surface Effects in Aerothermodynamics, Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 17-34.

[28] Houtman, E.M., van der Weide, E., Deconinck, H., Bakker, P.G., “Computational Analysis of Base Flow / Jet Plume Interaction,” Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 605-612.

[29] Jameson, A., Schmidt, W., Turkel, E., “Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes,” AIAA paper 81-1259, June 1981.

[30] Kim, S.-S., Kim, Ch., Rho, O.-H., “Multigrid Algorithm for Computing Hypersonic Chemically Reacting Flows,” Journal of Spacecraft and Rockets, Vol.38, No. 6, 2001, pp. 865-874.

[31] Kleb, L.W., Nielsen, E.J., Gnoffo, P.A., Park, M.A., Wood, W.A., “Collaborative Software Development in Support of Fast Adaptive Aerospace Tools (FAAST),” AIAA paper 2003-3978, June 2003.

[32] Kumar, A., Gnoffo, P.A., Moss, J.N., Drummond, J.Ph., “Advances in Computational Capabilities for Hypersonic Flows,” Proceeding of the AGARD Symposium on Future Aerospace Technology in the Service of the Alliance, AGARD CP-600, Vol.3, 1997, pp. 13-1 13-27.

[33] Kurotaki, T., Ito, T., Matsuzaki, T., Ishida, K., Watanabe, Y., “CFD Evaluation of Catalytic Model on SiO2-Based TPS in Arc-heated Wind Tunnel,” AIAA paper 2003-155, January 2003.

[34] Laub, B., “The Apollo Heatshield: Why Performance Exceeded Expectations,” Proceedings of the AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 1999.

[35] Laux, T., Auweter-Kurtz, M., “Untersuchung des Passiv-Aktiv-Übergang in Sauerstoffplasmen,” DGLR-Jarestagung, Leipzig, DGLR-JT2000-207, Sept. 2000.

[36] Legensky, S.M. et al., “CFD General Notation System (CGNS): Status and Future Directions,” AIAA paper 2002-0752, January 2002.

[37] Liou, M.-S., “Ten Years in the Making – AUSM – Family,” AIAA paper 2001-2521, June 2001.

[38] Liu, D.D., Chen, P.C., Tang, L., Chang, K.T., “Hypersonic Aerothermodynamics / Aerothermoelastics Methodology for Reusable Launch Vehicles/TPS Design and Analysis,” AIAA paper 2003-897, January 2003.

[39] Longo, J., “Aerothermodynamics – A critical review at DLR,” Aerospace Science and Technology 7, 2003, pp. 429-438.

[40] Lüdeke, H., “Computation of Görtler Vortices in Separated Hypersonic Flows,” Proceedings of the ICCFD'2000, Kyoto, Japan, July 2000.

[41] Lüdeke, H., “Detached Eddy Simulation axialsymmetrischer Nachlaufströmungen mit den DLR TAU code,” Proceedings of the DGLR Fach Workshop der STAB, Göttingen, Germany, November 2003.

[42] Mack, A., Schäfer, R., Gülhan, A., Esser, B., “IMENS Flowfield Topology Changes due to Fluid-Structure Interaction in Hypersonic Flow using ANSYS and TAU,” Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Eds. Breitsamter, Laschka, Heinemann, Hibig, Vol. 87, 2003, pp. 196-203.

[43] Malechuk, A.M., Edwards, J.R., Hassan, H.A., “Simulation of Transitional Flow over a Elliptic Introduction Cone at Mach 8 using a One-Equation Transition / Turbulence Model,” AIAA paper 2003-1132, January 2003.

[44] Mavriplis, D.J., “Directional Agglomeration Multigrid Techniques for High-Reynolds Number Viscous Flows,” NASA / CR-1998-206911, ICASE Report No. 98-7, January 1998.

[45] Mehta, U. B., “Synthesis of Contributed Simulation for OREX Test Cases,” NASA/TM-1998-112238, July 1998.

[46] Mendenhall, M.R., Childs, R.E., Morrison, J.H., “Best Practices for Reduction of Uncertainty in CFD Results,” AIAA paper 2003-411, January 2003.

[47] Menaa, M., “Heat Transfer in a Compressible Mixing Layer,” Proceedings of the AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 1999.

[48] Nakamura, T., Kanekou, M., Men’shova, I., Nakamuraß, Y., “Numerical Simulation on Aerodynamic Interaction between a Side Jet and Flow around a Blunt Body in Hypersonic Flow,” AIAA paper 2003-1135, January 2003.

[49] Ogasawara, Ko., Fujii, K., Morito, T., “Aerothermodynamic Flight Experience in OREX and HYFLEX,” Proceedings of the AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 1999.

[50] Papadopoulos, P., Venkatapathy, E., Prabhu, D., “Current Grid Generation Strategies and Future Requirements in Hypersonic Vehicle Design, Analysis and Testing,” Proceeding of the 6th International Conference on Numerical Grid Generation in Computational Field Simulation, Greenwich, USA, July 1998.

[51] Paulat, J.C., “Atmospheric Reentry Demonstrator Post Flight Analysis - Aerodynamics,” Proceedings of the AAAF 2nd International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 2001.

[52] Paulson, J.W., Miller, Ch.G., “Aerothermodynamic Testing Requirements for Future Space Transportation Systems,” Proceedings of the Space System Design and Development Testing Conference, AGARD CP-561, 1995, pp. 10-1 10-22.

[53] Perraud, J., Arnal, D., Dussillols, L., Thivet, F., “Studies of Laminar-Turbulent Transition in Hypersonic Boundary Layers at ONERA, Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 309-316

[54] Poll, D.I.A., “Implications of 3-D transition mechanisms on the performances of space vehicles,” Proceedings of the Second European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-367, 1994, pp. 175-182.

[55] Prabhu, D.K., Papadopoulos, P., Davies, C.B., Wright, M.J., McDaniel, R.D., “Shuttle Orbiter Contingency Abort Aerodynamics, II: Real-Gas Effects and High Angles of Attack,” AIAA paper 2003-1248, January 2003.

[56] “Proceedingss of the 2nd International Workshop Rocket Combustion Modelling: Atomization, Combustion and Heat Transfer,” Ed. O.J. Haidn, Lampoldshausen, Germany 2001.

[57] Radespiel, R., Brück, S., Lüdeke, H., “Computation of Transitional Re-entry Flows Over 3D Control Surfaces,” Proceedings of the 2nd AAAF International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 2001.

[58] Rahaim, Ch.P., Oberkampf, W.L., Cosner, R.R., Dominik, D.F., "AIAA Committee on Standards for Computational Fluid Dynamics – Status and Plans," AIAA paper 2003-0844, January 2003.

[59] Reijasse, Ph., James Ph., Vuillermoz, P. "Aerothermodynamics Issues for Future Liquid Rocket-Engine Nozzles," Proceedings of the 4th European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-487, 2001, pp. 555-562.

[60] Schaefer, R., Mack, A., Burkard, E., Guelhan, A., "Fluid-Structure Interaction on a Generic Model of a Re-entry Nosecap," Proceedings of the 5th Congress on Thermal Stress and related Topics, ICTS, Virginia, USA, June 2003.

[61] Schneider, S.P., "Flight Data for Boundary-Layer Transition at Hypersonic and Supersonic Speeds," Journal of Spacecraft and Rockets, Vol. 36, No. 1, 1999, pp. 8-20.

[62] Schwane, R., "Unsteady Numerical Simulations of the Flow in a Truncated Ideal Contour Nozzle under Free Shock Separation Conditions," AIAA paper 2003-3676, June 2003.

[63] Sinha, K., Maheshy, K., Candlerz, G.V., "Modelling shock unsteadiness in shock / turbulence interaction," AIAA paper 2003-1265, January 2003.

[64] Sobieszczanski-Sobieski, J.; Haftka, R.T., "Multidisciplinary Aerospace Design Optimization: Survey of Recent Developments," AIAA Paper 96-0711, January 1996.

[65] Spalart, P., R., "Strategies for Turbulence Modelling and Simulations," 4th International Symposium Engineering Turbulence Modelling and Measurements, Corsica, May 1999.

[66] Tang, W., Orlowski, M., Longo, J.M.A., Giese, P., "Aerodynamic Optimization of Re-entry Capsules," Aerospace Science and Technology 5, 2001, pp. 15-25.

[67] Tran, Ph., Soler, J., "Atmospheric Reentry Demonstrator Post Flight Analysis: Aerothermal Environment," Proceedings of the AAAF 2nd International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 2001.

[68] Vennemann, D., "Hypersonic Aerodynamic / Aerothermal Test Facilities Available in Europe to Support Space Vehicle Design," Proceedings of the Space Systems Design and Development Testing Conference, AGARD CP-561, 1995, pp. 28-1 28-14.

[69] Whang, Ch.W., Zhong, X., "Leading Edge Receptivity of Görtler Vortices in a Mach 15 Flow Over a Blut Wedge," AIAA paper 2003-790, January 2003.

[70] Wong, H., Kremer, F., "Numerical Assessment on the Heating of the Rudder / Fin Gap in X-38 Space Vehicle," Proceedings of the Third European Symposium on Aerothermodynamics for Space Vehicles, ESA SP-426, 1998, pp. 77-86.

[71] Xiong, Z., Lele, S.K., "Simulation and Analysis of Stagnation Point Heat Transfer under Free-Stream Turbulence," AIAA paper 2003-1259, January 2003.