A Schlieren photograph of the Ariane-5 launcher in the wind tunnel at FOI in Sweden (speed Mach 0.85)

Modelling Launcher Aerothermodynamics

– A Vital Capability for Space Transportation

Richard Schwane, Johan Steelant, Wilhelm Kordulla
ESA Directorate of Technical and Quality Management, ESTEC, Noordwijk,
The Netherlands

David Perigo, Oliver Gloth, Henry Wong, Yang Xia, Roland Schmehl
Advanced Operations Engineering Services BV, Leiden, The Netherlands

Marc Toussaint
Directorate of Launchers, ESA, Paris

Daniele Barbagallo
ESA Directorate of Launchers, ESRIN, Frascati, Italy

Europe's access to space today relies primarily on the Ariane family of heavy-lift launchers and the future Vega launcher for smaller payloads. The successful development of such launchers critically depends on finding reliable solutions in the challenging areas of propulsion and aerothermodynamics, which are the key elements of any launch vehicle. The processes involved are highly complex because these disciplines strongly interact with the other build elements such as structures, thermal protection, acoustics, and guidance, navigation and control. To illustrate the aerothermodynamic challenges, we need only look at the various phases in the flight of a generic vehicle containing technology elements from both Ariane-5 and Vega.

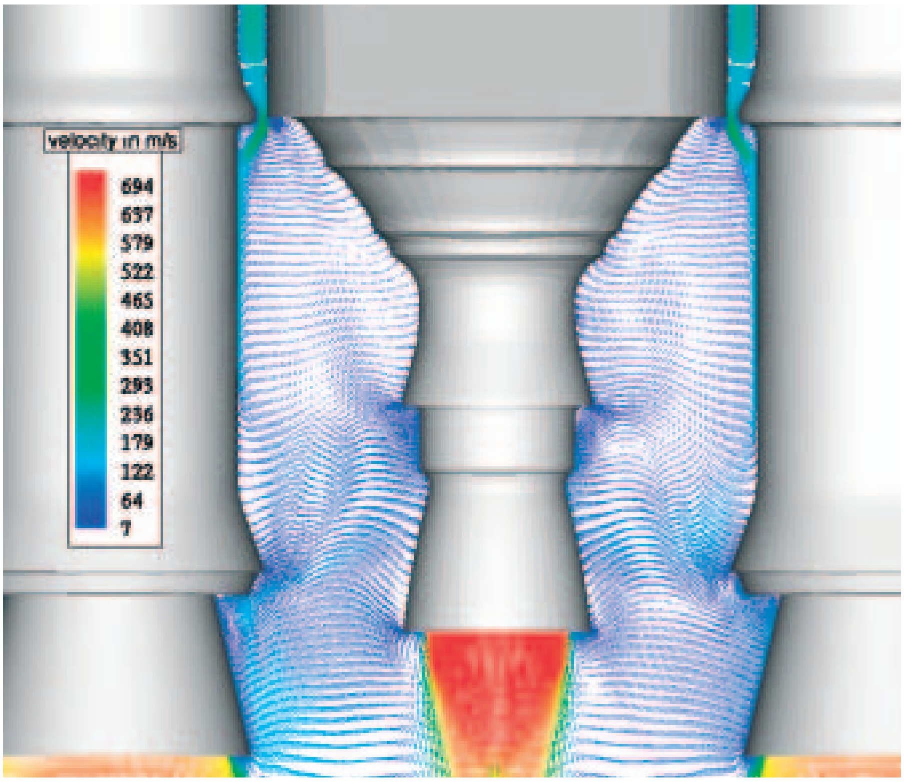
Introduction

In the past, the aerodynamic and aerothermodynamic development work on vehicles such as the Space Shuttle or Ariane-4 has been based almost exclusively on engineering and empirical methods and experiments. In the last two decades, however, Computational Fluid Dynamics (CFD) has matured to a point where it can make meaningful contributions when fast analytical modelling requires that the physical problem be strongly simplified, and testing is either extremely expensive or cannot recreate the real problem. Increasingly, CFD-based methods offer the only feasible solution for accelerating and refining the space-transportation design process whilst still maintaining sufficient confidence in the results for safe design decisions to be made.

The Launch Sequence

T_0 – 12 hours: Launcher Roll-out

The launcher is transported from the controlled environment of the integration hall to the launch pad, and into what may possibly be a 'hostile' environment due to rain or gusty winds. Aerodynamically, this may already be a problem because if a cylindrically shaped object, such as a tall building or launch vehicle, is exposed to sufficiently strong cross-winds, vortices formed in the wake of the structure cause pressure fluctuations, resulting in a fluctuating force on the object.



Snapshot of an unsteady flow field at the Ariane-5 launcher's base

CFD methods and simplifying analytical modeling are therefore used to determine critical wind velocities for the launcher structure on the launch pad, through the simulation of vortex shedding effects coupled with structural dynamics. This is a substantial help in determining critical roll-out and launcher installation conditions.

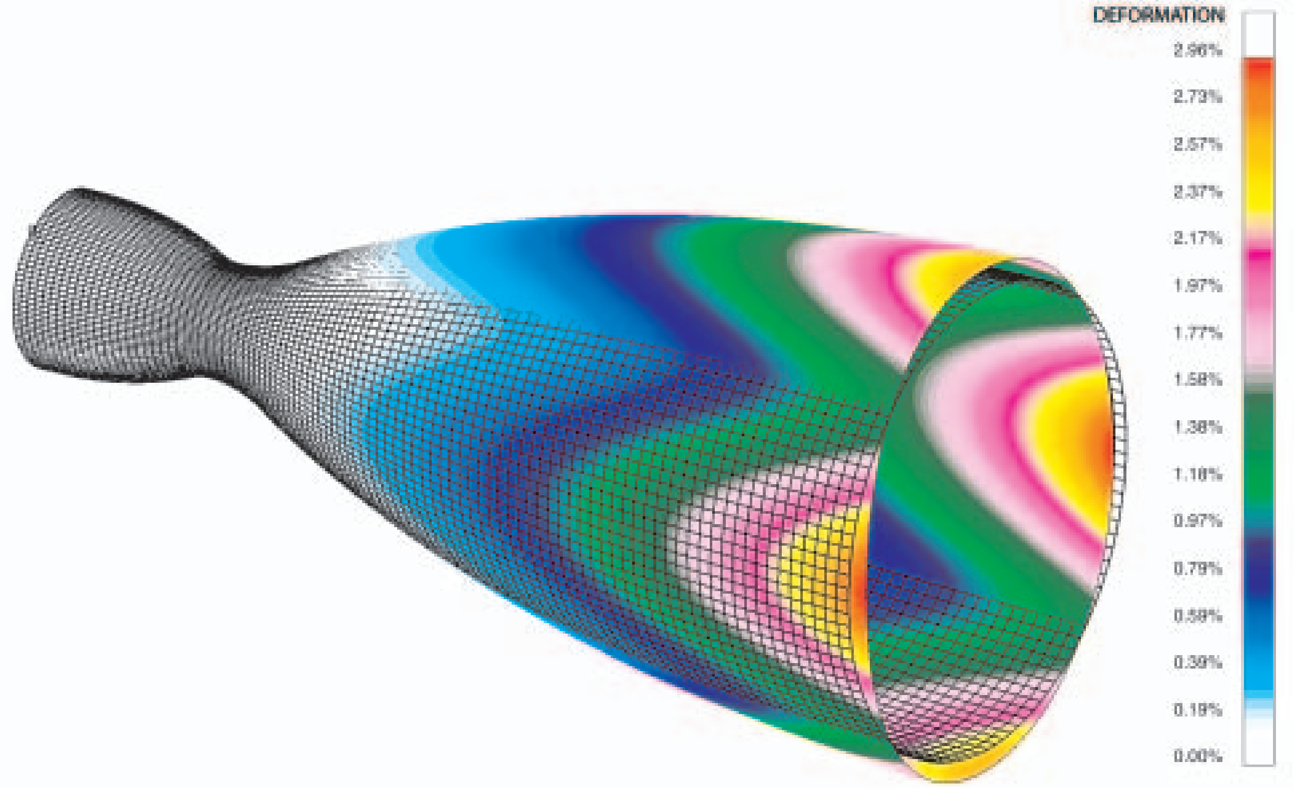
T_0 – 7 seconds: Main-Engine Ignition

After main-engine ignition, the development of the engines' exhaust plumes and the blast wave characteristics determine the acoustic environment before lift-off, which has a direct impact on the acoustic vibration levels acting on the payload. As more and more stringent acoustic requirements are imposed on the launchers, aerodynamic optimisation of the launch-pad area by diverting the high noise level of the engine plume away from the launch vehicle becomes more important. CFD methods are currently used, together with subscale testing and empirical modelling, to predict the effect of the complex flow in the ducts that divert the exhaust plumes on the launch pad.

Ignition of the main engine also involves high structural loads on the expansion nozzle due to a transient asymmetric flow field within the nozzle. Detailed knowledge of the physics of such unsteady flows allows one to arrive at a nozzle design that combines high efficiency with low operational risk. Due to the expense of conducting full-scale tests, such understanding is usually derived from a combination of subscale testing, empirical methods and sophisticated CFD methods.

T_0 + 0-10 seconds: Solid Booster Ignition and Lift-off

Lift-off takes place immediately after the ignition of the solid-rocket boosters. During the very first flight phase, especially if an early pitch-over manoeuvre is necessary to reach a planned trajectory, interactions between the hot main-engine and booster plumes and the ground facilities, e.g. launch tower and mobile gantry, have to be taken into account. A combination of empirical, analytical and numerical methods is used to assess critical parameters in this area.



CFD modelling of the deformation, or so-called 'ovalization' of a Vulcain-2 engine nozzle due to asymmetric internal pressure loads

Flow/structure coupling makes an important contribution to the structural noise level that is transmitted to the payload. Computational tools are used to describe the interaction of launcher oscillations with the flow for the most important structural modes.

During the first flight phase, high heat loads are observed in the nozzle near the exit due to incipient separation of the flow in performance-optimised nozzles. Control of such heat loads again requires detailed knowledge of the nozzle flow, since a large separation could cause post combustion and overheating close to the nozzle's exit. Experimental and numerical methods are used to determine the extent and severity of flow separation for this first highly over-expanded flow phase.

T_0 + 30-50 seconds: Launcher in the Transonic Flight Regime

The point where the launcher goes supersonic represents another critical phase. The local flow-field around the launcher becomes highly complex, including localised regions of supersonic flow terminated by strong shock waves with associated high pressure variations (see Schlieren photograph of Ariane-5 on title page).

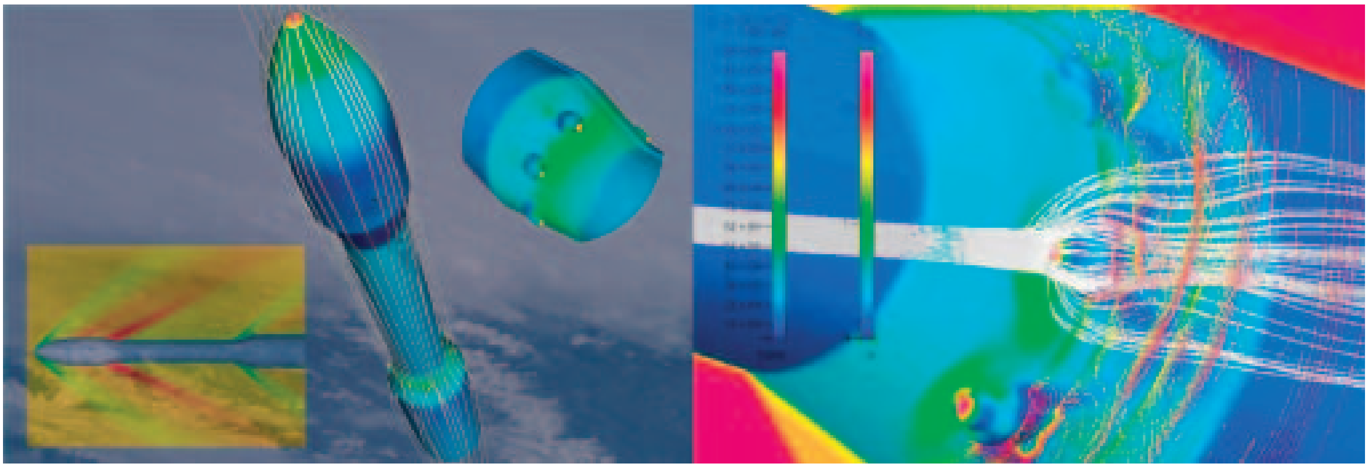
The flow-field is then highly sensitive to small changes in flight direction (pitch and yaw) and significant asymmetric loads may result from such motions. Moreover, the shock waves around the vehicle exhibit oscillations, and the resulting typical high frequency shock motion is another source of severe acoustic noise, against which measures must be taken, either by changing the shape or by protecting instruments or mechanisms located near the disturbances.

Experience has shown that in this flight phase, the simulation and control of the interaction of the main engine plume and the ambient flow field requires the greatest attention because of the generation of side loads and large local heat loads. Uncontrolled side loads can also lead to malfunctioning of the gimbal system. However, since the complex aero-thermodynamics of separating flow in expansion nozzles and the coupling to the ambient flow field are not yet entirely understood, nozzles have to be designed with large margins and therefore cannot be fully optimised for both sea-level and rarefied atmospheric conditions.

More sophisticated three-dimensional computational results, e.g. wall stream-lines and velocity vector plots near the nozzle exit (see accompanying figure), as well as experimental data obtained both on the ground and in flight, are essential for an in-depth understanding of the flow physics and ultimately for shape improvements that reduce the loads to an acceptable level. Such results will be employed in future work on buffeting reduction through launcher shape optimisation.

T_0 + 30 - 120 seconds: Ovalization of Launcher Nozzles

Currently, analytical models are generally used in nozzle design to, for example, evaluate structural fatigue and nozzle



Pressure distribution and Mach-number contours in plane of symmetry and streamlines past retro-covers for Vega

deformation known as 'ovalization'. However, due to the necessary simplifications for the complex coupled flow field, such models require the application of large safety factors, leading to non-optimal performance. Benefiting from today's greatly improved computer performances with cost-efficient high-performance parallel-processing systems, the enhanced validation levels of CFD methods are becoming more and more important. For engine components, such as nozzles, recent coupling of flow with heat transfer as well as structural dynamics is leading to better understanding of the limiting structural loads, and hence to reduced design margins.

Analytical models can only predict nozzle stability limits for separated flows. Very recent computational analysis has provided the first evidence that fully attached flow can also excite nozzle ovalization. The accompanying illustration shows the CFD-predicted structural deformation response of a thin-walled nozzle under an internal pressure load.

An exact quantitative analysis is currently being pursued, but already at this stage the conclusion that safety factors cannot replace a thorough understanding of the aerothermodynamic flow properties seems justified.

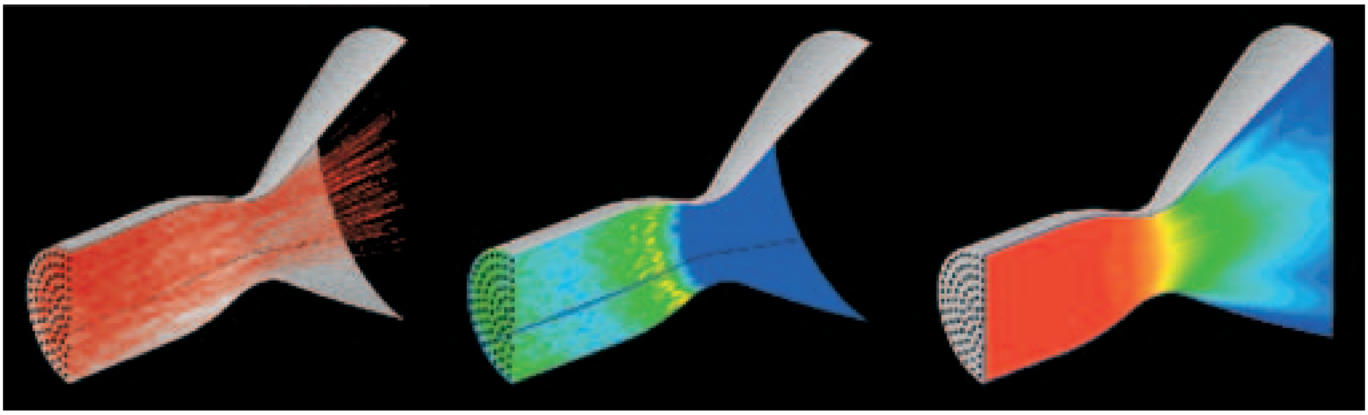
T_0 +120 seconds: Flight at Maximum Dynamic Pressure

Due to its increasing velocity and the decreasing ambient pressure, the launcher reaches a point on its trajectory where the ambient flow results in the highest dynamic loads on the launcher's structure. It typically occurs at roughly twice the speed of sound, and not only the main structure of the launcher, but also protuberances such as wiring tunnels, fuel pipes and retro-rocket covers have to withstand such loads. Such local flow effects are strongly viscous-dominated and cannot be accurately modelled in sub-scale wind-tunnel experiments. CFD modelling of the flow has become an indispensable tool for the extrapolation of wind-tunnel results to actual flight conditions.

The above figure shows CFD modelling results for a Vega-shaped vehicle travelling at Mach 2 at an altitude of approximately 14 000 metres (angle of attack 3 deg and roll angle 45 deg).

T_0 + 240-244 seconds: Ignition of Upper Stage Engine

Typically 4 minutes into the flight, at hypersonic speeds and in rarefied atmospheric conditions, the first stage



Low-pressure oxidiser pre-flow in the combustion chamber and nozzle of the Aestus engine reaches the end of its lifetime and is pyrotechnically separated from the upper-stage. Following a possible ballistic flight phase, the upper-stage engine is ignited in vacuum, involving the injection of pressurised liquid propellants into the combustion chamber, contact ignition, and an increase of pressure resulting in steady reacting flow.

Upper-stage engines as Aestus have been designed by means of experiments including the simulation of ambient vacuum conditions for the cold pre-flow and extensive use of system-level simulation tools. To ensure a stable transition to nominal engine operation for a wide variety of initial conditions, customised 3D-CFD methods are used to analyse the dynamics of the initial low-pressure flow processes in the engine's feed system, combustion chamber and nozzle. Experience has shown that the transient priming of the fuel dome and the oxidiser pre-flow in the combustion chamber of the engine require special consideration to guarantee smooth and reliable operation.

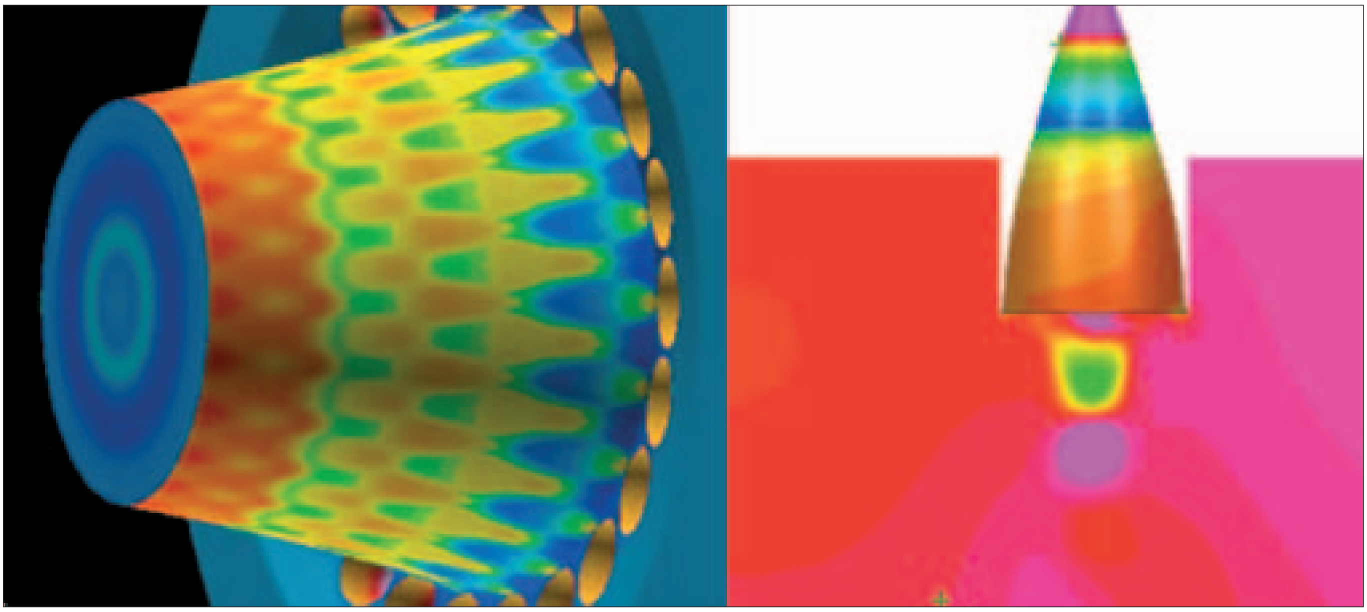
The start-up sequence also involves an oxidiser pre-flow phase to ensure well-defined flow conditions in the combustion chamber prior to fuel injection and contact ignition. This non-reacting low-pressure flow expands into vacuum and is characterised by a severe drop of oxidiser temperature and complex two-phase flow phenomena. Propellant and engine temperatures have a key influence on the pre-flow; however, these parameters can vary significantly, depending on the mission profile. Due to the complexity of experimental testing at vacuum conditions, 3D-CFD analysis has recently been employed to assess this parametric influence in a cost- and time-efficient way and identify critical limits, such as icing of oxidiser, local accumulation on chamber walls, or backflow into the fuel dome.

The figure on the previous page shows the flash-evaporating oxidiser spray (left), with the vapour flow field ranging from low subsonic conditions in the combustion chamber to supersonic conditions at the nozzle exit (middle), and the deposition of liquid oxidiser on the chamber walls (right). The second phase of this analysis addressed the dynamic evolution of the oxidiser spray during pre-flow, providing essential

The Sequence of Aerodynamic Events during a launch

Time	LV Condition	Flow Problem	Tool*	Expected or Achieved Benefit
T_0-12 hours	LV rolled out	LV exposed to environment	CFD, AM	Launch cost reduction, risk mitigation
T_0-7 sec	EPC ignition	Blast wave EPC start up loads	CFD, EXP	Improved payload environment, risk mitigation
T_0	Lift-off	Plume / ground interaction V2 with incipient separation	CFD, EXP AM	Cost reduction for ground facilities, risk mitigation
T_0 + 30-120 sec	Transonic flow regime Base buffeting impact on EPC	Side loads on EPC nozzle	EXP, CFD, AM	Permit LV performance optimisation
T_0 + 120 sec	Max. dynamic pressure	Loads on structure and protuberances	CFD, EXP	Structural design optimisation
T_0 + 240-244 sec	Stage separation and ignition of upper stage	Ignition of upper stage engine	CFD, EXP	Risk mitigation
T_0 + 244-251 sec	Transients in HM7B start up	Impact of HM7B plume on EPC	CFD	Risk mitigation

* EXP: Experimental Methods CFD: Computational Fluid Dynamics AM: Analytical Modelling



CFD predictions for advanced nozzle concepts: left, pressure contours for a clustered aero-spike nozzle; right, wall-pressure contours for a dual-bell nozzle

information for accurate system-level simulation of the engine start-up phase.

T_0 + 244-250 seconds: Upper Stage Engine Plume Impact on the Main Stage

The interaction of the plume from the starting of the upper-stage engine (e.g. HM7B) with the separated first stage (EPC) needs to be thoroughly investigated to avoid damage by high heat fluxes that could cause residual fuel in the tanks of the first stage to explode and thereby endanger the upper stage. Forces and moments on the first stage also have to be controlled to avoid stage collision due, for example, to interaction with the plume of a gimballed upper-stage engine nozzle.

A computational study was performed to arrive at an improved estimate for the heat fluxes on the EPC dome for steady nozzle operation, assuming a constant position of the two stages relative to each other for the start-up process. The main conclusion of that investigation is that the heat-flux on the first stage fluctuates strongly, and the peaks by far exceed the time-averaged values. Hence, the usually applied steady-state flow simulations substantially under-predict the transient loads encountered. This could have been expected for the time-dependent engine start-up, but is in fact also true for steady HM7B operation. These results highlight the need for careful resolution of time-dependent flow phenomena for design purposes also.

Advanced Nozzle Design

The research activities mentioned so far in the context of the various stages in a launch scenario are mainly concerned with clarifying the aero/aerothermodynamic challenges faced with today's launchers on the ground and in flight. Another essential task is to support the development of new launcher technologies aimed at overcoming existing limitations, for example in propulsion.

Conventional nozzles for main-stage application typically expand the combustion products to pressure levels well below sea-level conditions in trying to optimise the overall nozzle performance for the entire flight envelope. Unfortunately, the higher ambient pressure at sea level results in a recompression of the plume and in potentially unstable flow separation within the nozzle. A direct consequence of such behaviour is the increased and unacceptable level of side-loads exerted on the nozzle structure. This limits the performance that may be derived from conventional nozzle designs by limiting the maximum area ratio that can be safely employed.

Efforts thus concentrate on new nozzle concepts to meet the ambitious cost-reduction goals to maintain and increase the competitiveness of Europe's launchers. This implies the need to consider also thrust-chamber design options that offer a substantial cost reduction, meeting overall performance requirements possibly at the expense of maximum specific impulse. Losses due to low-cost approaches for engine components and architectures may be compensated for by advanced nozzle designs allowing for altitude adaptation of the nozzle area ratio.

Europe has already been successful in expanding the knowledge base relating to advanced nozzle concepts. The figure above shows two such concepts that have been, or are currently being investigated with ESA's industrial partners. They are primarily designed to overcome the previously described separation problem by adapting the plume boundary pressure to that of the ambient environment.

The application of advanced nozzle technology to upper-stage propulsion is also under active consideration due to the reduced packaging volumes inherent in several of the concepts.

Today, European LO_X/LH_2 engine nozzles rely on welded tubular wall structures for actively cooled configurations, such as the HM7B and Vulcain nozzles. This is a simple but not very cost-effective solution due to the special rectangular tubes needed and the extensive welding procedures employed. Alternative design options offer reductions in recurring costs by using a laser-welded sandwich design for actively cooled parts. Further advantages include: a high stiffness and strength/mass ratio; greater flexibility in cooling-channel design to tailor cooling characteristics; more efficient use of the engine's coolant pressure budget for nozzle cooling; and improved modelling accuracy, both from a mechanical and an aerodynamic point of view.

A number of European research programmes are being initiated to look at the effects of both hot-gas and coolant flows on nozzle-wall heat transfer for such novel concepts. One such programme, under the management of the French Space Agency CNES, is focusing on Flow Separation Control Devices (FSCDs), with working-group members drawn both from industry (SNECMA, Volvo Aero Corp., EADS-ST) and research organisations (DLR, ONERA, LEA Poitiers and ESA/ESTEC). A cornerstone of the group's work is the development of a scaling logic from subscale cold testing, via subscale hot testing, to flight configurations. At the subscale level, cold and hot techniques are both deemed vital and neither are sufficient in isolation. Used in conjunction, however, they become a powerful propulsion-system design tool.

While cold-flow experiments are ideal for several of the tasks in hand, for the design of a new rocket engine's thrust chambers and expansion nozzles, accurate prediction models and tools are needed for studying wall pressure, heat transfer and side-loadings. Such models are incomplete without the inclusion of hot test data and so a 'Calorimeter Nozzle Programme' (CALO) was initiated by EADS in 2001 with support from SNECMA and VAC. Three different thrust-chamber configurations, including actively cooled and film-cooled nozzle extensions, were built and equipped with the latest measurement diagnostics.

The work of the FSCD group showed clearly that high-area-ratio conventional nozzles were an undesirable choice if separation was present in the nozzle extension. Consequently, a detailed study of alternative advanced nozzle concepts was made and, based on various trade-offs, the dual-bell nozzle was finally selected as the most promising candidate for future high-performance engine concepts. Further experimental and numerical testing is being performed with respect to this configuration based on previously applied numerical techniques and the further utilisation of the HYP500 wind-tunnel facility at FOI in Sweden. The potential of extending dual-bell testing to hot configurations, in a programme similar to the CALO campaign of EADS and SNECMA, is also being considered.

Conclusion

This article has given a brief overview of the key role that aerodynamics and aerothermodynamic design factors play in determining the efficiency and reliability of operation of any launch vehicle. In summary:

- Aero/aerothermo-dynamics is a discipline that is strongly coupled to most of the other technologies that are needed for launcher design and optimisation.

- The rapid development of experimental and numerical techniques for enhanced understanding of, in particular, three dimensional, time-dependent flow fields, which were not available until just a few years ago, will support rapid improvements not only in the aerodynamic characteristics of launch vehicles, but also in their performance, reliability and cost-efficiency.

Much work remains to be done to further improve physical modelling, for example for turbulent flows, to arrive at efficient 3D time-accurate computational simulations, particularly for flows with large separations, as well as experimental techniques for complex time-dependent flows. Consequently, the mastery of aerothermodynamics problems is considered one of the fundamental factors governing Europe's competitiveness in terms of launcher-technology development.