Separated Flow: Some Challenges and Research Priorities for Missile Aerodynamics

N. Taylor
MBDA UK
GBR
nigel.j.taylor@mbda-systems.com

ABSTRACT

This paper provides insights into the challenges associated with using CFD to predict separated flows of relevance to missile aerodynamics. Both the physical phenomena being modelled and the practices associated with CFD use are addressed. The scope of separated flow environments routinely encountered in industrial missile aerodynamics contexts is outlined and examples are provided to illustrate some of the attendant challenges. The topic of solution verification is then considered, citing recent experience gained in the AVT-316 Missile Facet as a backdrop, before outlining the challenges of bridging the gap between verification and prediction. Although grounded in the context of missile aerodynamics, many of the topics raised are potentially of much broader relevance. In view of this, several examples are provided to illustrate the types of mutual benefit that are most readily gained by working with stakeholders from the wider aerospace community.

1.0 INTRODUCTION

When a missile – or for that matter any flight vehicle – flies through air, it generates a region (or regions) of separated flow. The way in which flow separation occurs plays a crucial role in determining the vehicle's aerodynamic performance. The importance of this is impressed upon aerodynamicists at a very early stage in their undergraduate careers – usually by explaining the role of the Kutta condition (as applied to two-dimensional aerofoils) and the treatment of trailing vorticity (in theories associated with the generation of lift and drag for three-dimensional wings). The adaptation and extension of such fundamental principles to broader contexts – including those appropriate to missile aerodynamics – is an endeavour that can last for the duration of their professional lives.

Since they have no requirement to care for a human payload, the operating envelope of a missile can be appreciably larger than that of other flight vehicles. The variety of missiles currently in service or in development is also uncommonly broad – covering operation at subsonic through to hypersonic Mach numbers, over short or long durations, at altitudes extending from below the Earth's surface to beyond the stratosphere and attitudes well beyond those experienced by manned aircraft. Moreover, since maximising aerodynamic efficiency is not always critical to their operational effectiveness, the extent and diversity of the separated flows missiles can generate is also very large indeed. Consequently, when the organising committee of this symposium invited me to prepare a presentation on the subject of "Missile Aerodynamics: challenges and needs for improved separated flow prediction and understanding", I immediately sought a basis for de-scoping my remit.

I decided to retain the focus on prediction and, by implication, the use of Computational Fluid Dynamics (CFD) in circumstances where no physical data is available. This has meant relegating a host of other important topics – including those associated with the need to be able generate the separated flows of interest in controlled environments, to acquire suitably detailed and time-accurate physical measurements and handle the resulting (large quantities of) data – to brief passing comments. In many cases, they are not addressed at all. It should not be inferred that the combined requirements for improved ground test facilities, testing techniques, instrumentation, flow visualisation and data analysis procedures are less important than those associated with CFD, rather that I have found insufficient time and space to do them justice herein.

CFD prediction, too, is a broad and complex subject in itself. For reasons that will be outlined below, I have decided to focus, initially, on one particular aspect of it: verification. Moreover, I have approached this from the perspective of a practising industrial missile aerodynamicist, rather than someone engaged primarily in research or code development. Clearly, there is considerable overlap in the demands of these roles: my intent is, amongst other things, to provide additional insight as to how open communication and close working can be extremely beneficial to all stakeholders. Indeed, I believe it to be a pre-requisite for success in all of the stages required to develop a predictive CFD capability.

Section 2 sets the scene by providing an impression of the scope of separated flow environments that are routinely encountered in industrial missile aerodynamics contexts, together with an outline of the attendant requirements that may be placed on their prediction. In Section 3, various aspects of CFD verification are addressed, with particular reference to the ongoing activities of the NATO STO AVT-316 Missile Facet – the subject of the following paper in this symposium.[1] The gap between verification and prediction is then bridged in Section 4, before providing some closing remarks in Section 5.

2.0 THE SCOPE OF THE CHALLENGE

Flow separation occurs when a boundary layer separates from the physical surface to which it was previously attached. In itself, flow separation is thus a rather localised phenomenon. However, its effects, together with their aerodynamic significance, need not be – even in circumstances where the separated boundary layer reattaches a short distance downstream (as may occur in the vicinity of shock wave interactions, for instance). Once separated, the flow may behave in a variety of ways, acting as a shear (or mixing) layer between different regions of the flowfield or rolling-up to form large vortical structures, for instance. In most cases, the separated flow is turbulent and contains dynamic, coherent features of varying scales. There are many possibilities, augmented further by the potential for various forms of interaction – between multiple flow features and/or airframe components, even multiple airframes (an example of which will be described in Section 2.3). Indeed, because of the close physical proximity of the different components forming the outer mould lines of a missile airframe, separated flow features rarely act in isolation (for long).

One reason for the potential enormity of this scope is the diversity of the mission requirements a missile may be designed to address. The ways in which requirements for range (or endurance) and manoeuvrability are combined with the constraints associated with carriage and launch can have profound effects on the choice of airframe layout. For instance, requirements for elevated manoeuvrability increase the likelihood of separation occurring forward of the base of a missile and the trailing-edges of its aerodynamic control surfaces. Indeed, the attendant non-linear augmentation of the aerodynamic loads can be extremely valuable in this regard. The existence of various forms of excrescence – hangers, cable ducts, umbilical connectors and telemetry aerials, for instance – provide additional potential sources of separated flow. If an air-breathing propulsion sub-system is incorporated, additional potential sources of flow separation are introduced – both for the external and internal air paths (think of buzz in supersonic air intakes, for instance). The use of lateral jets or other forms of thrust vectoring system add to the potential complexities that may need to be addressed. As alluded to above, the physical arrangement of the various airframe components is also important since strong – and performance defining – separated flow interactions can occur as the air makes its passage downstream: the effects of vortices shed from the forebody on the control fins, or of dynamic distortion in the ducting ahead of turbomachinery components, for instance. Further complications may arise during carriage and/or launch, since the local environment may itself contain – or induce – further regions of separated flow.

Recognising that missiles may be required to operate over a wide range of Mach numbers and altitudes (including conditions where, even in the freestream, the air may no longer be considered to act as a continuum), the range of potential separated flow scenarios to be predicted is immense. Suffice it to say that the net effect is usually that flows of interest are both varied and extremely complex, containing several regions of separated flow, some of which may be subject to strong mutual interactions.

Although the above overview is far from comprehensive, it should be clear that the challenges facing the application of CFD to the prediction of separated flows on missiles are substantial, both in their scope and complexity. By means of illustration, we could select any of the above topics to expand upon. However, resisting the temptation to explore the more obviously complex scenarios, three simpler examples are outlined, briefly, below. As will be seen, despite their relative geometric simplicity, they each still pose some very substantial challenges.

2.1 A Simple Axi-symmetric Forebody

We start by considering the separated flow generated by an axi-symmetric forebody at incidence. The situation has been familiar to researchers for many years: in the late 1970’s and early 1980’s, extensive experimental campaigns were conducted over a wide range of Mach and Reynolds numbers and for various nose profiles, in order to gain better insights into the underlying flow physics.[2] Figure 23-1 presents the results of a subsequent Detached Eddy Simulation (DES) computation, reported in 2006. [3] This gives an impression of the complexity of the flow structures computed on the leeward side of the forebody and shows that the vortices shed from both sides of the forebody adopt an asymmetric pattern that alternates from one side to the other as the distance from the nose increases.

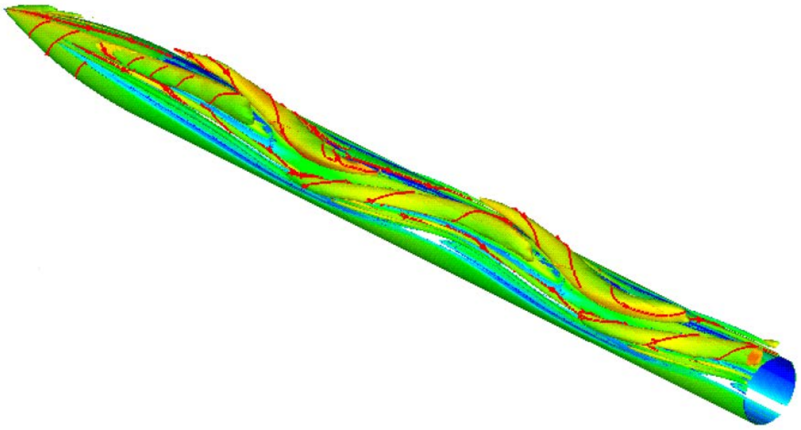

Figure 23-1: Time-averaged vortex sheets (Q-criterion, via DES) on an axi-symmetric forebody Mach 0.2; total incidence, σ =45° (Adapted from [3])

While the computed pattern of asymmetry was reported to follow that observed in experiments reasonably well over the forward part of the forebody, the quality of agreement deteriorated at stations further downstream. This is illustrated in Figure 23-2, which shows how the local side force coefficient (CY) varies with the non-dimensional distance from the nose, (X/D). (It also shows that solutions of the Reynolds-Averaged Navier-Stokes (RANS) equations, obtained using Spalart-Allmaras one-equation turbulence models – with and without rotation corrections – failed to capture any asymmetry in the shed vorticity.)

Various reasons were cited as potentially contributing to these disparities in the net side force (specifically wall interference effects in the wind tunnel – which were not accounted for in the computations – and that the boundary conditions applied in the CFD were for a forebody of infinite length). However, unfortunately, no evidence of solution verification is provided in the paper, so it is not possible to ascertain the extent to which the published results are spatially (or temporary) converged. This leaves a number of important questions unresolved. For instance, is the agreement between the polarities of the measured and computed values of CY over the forward portion of the nose fortuitous? Would the CFD computations be able to reproduce (or provide further insight into the source of) the switching in polarity of the total side force coefficients that has been observed in other wind tunnel test data?[4] How would the agreement between computation and measurement be affected by the introduction of control fins towards the aft of the body? Such questions have profound implications for the ability to characterise the aerodynamic stability of the airframe.

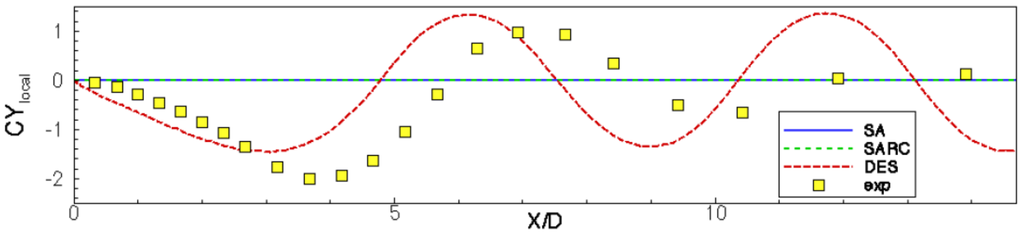

Figure 23-2: A comparison of measured and computed longitudinal distributions of local side force coefficient on an axi-symmetric forebody body Mach 0.2, total incidence, σ =45° (Adapted from [3])

At such low dynamic pressure (the data reported in [3] is for Mach 0.2), the magnitude of the aerodynamic moments generated by the airframe will be of limited practical significance to most autopilots: some form of thrust vectoring is often employed to effect control in these circumstances (adding further complexity to any CFD tasking). Consequently, the accurate characterisation of vortex asymmetry is far more important at higher Mach numbers. This remains a significant challenge for users of CFD and ground test facilities alike.

2.2 The CFD_OTC1 Test Case

We now consider another simplified airframe, this time of a generic missile possessing cruciform wings and fins. Prediction of its aerodynamic characteristics at Mach 1.4, a total incidence (σ) of 15.0° and an aerodynamic roll angle (λ) of 2.5° is the current focus of activity for the AVT-316 Missile Facet.[1] This flight condition was selected on the basis of past experience of applying CFD to similar airframe configurations where, when seeking solutions RANS equations, difficulties demonstrating predictable dependencies on mesh resolution had been encountered. The computed results could also be sensitive to the chosen modelling approach, for example the choice of turbulence model or whether the flow was modelled as being steady or unsteady. An example of the results from a recent DES computation are presented as Figure 23-3.

When compared with Figure 23-1, the additional complexity of the computed flow topology on the leeward side of the airframe is immediately apparent. While this disparity in complexity may be explained, in part, by the fact that Figure 23-1 presents time-averaged data, whereas Figure 23-3 shows an instantaneous snapshot of the computed events, the CFD_OTC1 airframe clearly generates multiple vortices, shed in different ways from the forebody, wings and fins. The airframe also generates a number of shock waves (not evident in the figure). As [1] illustrates, the computed interactions of the vortices – both mutual and with various airframe components – play a crucial role in determining the airframe’s overall aerodynamic characteristics, in particular its local static roll stability.

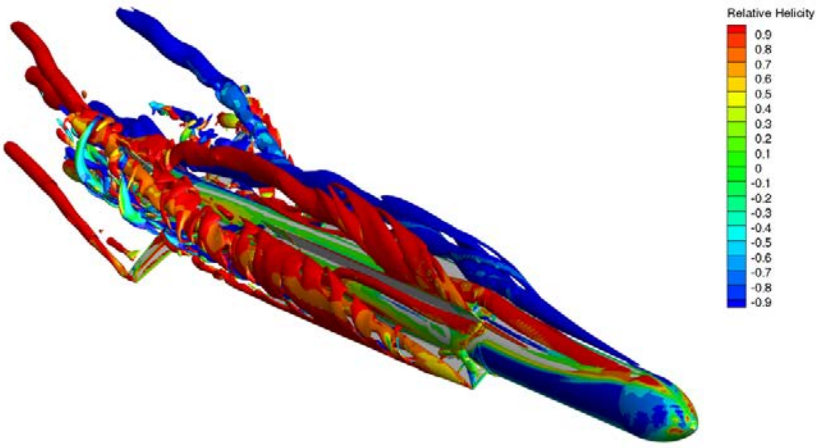

Figure 23-3: An example of the instantaneous Q-criteria iso-surfaces, coloured with relative helicity computed for the CFD_OTC1 test case at the mandatory flight condition ([1])

The work of the AVT-316 Missile Facet has provided several insights into the scope and complexity of the challenges that can be faced by users of contemporary CFD codes in securing verified solutions. These will form a convenient backdrop for the material presented in Section 3, below. However, before moving on, two further observations are noteworthy:

(i) The difference in computational power brought to bear in generating the results presented in Figures 23-1 and 23-3, is substantial. While direct comparisons of the resources – or algorithms – used are not available in the literature, a simple perspective may be gained by comparing the size of the computational meshes that were used: the data presented in Figure 23-1 was computed using a structured mesh containing 2 million cells; the structured overset mesh of Figure 23-3 contained 140 million. Rather than trying to quantify the improvements in a more systematic manner, or make projections about future timelines, the simple point here is that the supporting technologies – both hardware- and software-related – are developing at a rapid pace. While there is debate as to the precise route that related technological developments will take, there is no doubt that the levels of computational power available to aerodynamicists will continue to increase for the foreseeable future. Although this constitutes huge opportunity, it also poses a number of short-term challenges, including those associated with prioritising research and development plans (as expressed in [5], for instance.)[*]

(ii) Unlike the situation described in [3], where the CFD computations are compared with wind tunnel data, the CFD_OTC1 Test Case was, from the outset, developed to start its life “blind”, i.e. without physical test data. This decision was made intentionally, partly to reflect the situations encountered during the early stages of the product development cycle (i.e. prior to wind tunnel testing having taken place). Additional objectives were to deliberately expose some of the challenges faced in arriving at suitably verified CFD solutions and to help guide decisions to be made about what physical data would be required to support subsequent validation activities.

2.3 Weapon Bay Carriage and Release

Moving away from vortex dominated flows, we now consider a very different type of separated flow, namely the aero-acoustic environments encountered in an open weapons bay. Again, these have been the

---

[*] In this context, it is interesting to note that, by virtue of the relative scale (and lower Reynolds number) of the airframes of interest, the application of DNS to missile aerodynamics is, potentially, a more immediate prospect than it is for manned combat or commercial transport aircraft.

subject of extensive research and a growing bank of information is available in the open literature.[6] Rather than attempting to provide a comprehensive overview of the residual challenges for CFD prediction posed by these types of flow, a perspective informed by recent local experience is provided below.

In seeking to obtain physical measurements to support the development of a validated prediction process, a number of limitations of the established techniques for generating such data – i.e. wind tunnel testing – became increasingly apparent. In no particular order, the areas of potential concern included: the need to characterise unsteady effects on the overall forces and moments (only steady-state data can be measured with confidence via conventional strain gauge balances); challenges associated with dynamic scaling and, in conjunction with flow unsteadiness and aero-elastic effects, the attendant potential for augmented variability in release trajectories; the influence of any intrusions into the weapon bay flow associated with missile model support structures.

Thus, we (MBDA and a team of researchers at the University of Glasgow) found ourselves in a position where, having pioneered the use of Scale Adaptive Simulations (SAS) as a more computationally efficient alternative to DES for these computations[*] and coupling the CFD code to structural and release trajectory prediction models,[7] we were struggling to find an affordable way of obtaining suitable physical measurements to help validate the results we could now compute.

Figure 23-4: (Wind-off) Photographs of the weapon bay rig installation at the High Speed Blower Facility at BAE SYSTEMS, Warton (from [8]).
Left: side-view, from port. Air flow would be from left-to-right, exhausted from a nozzle (light blue) under the weapon bay (green, shown here with doors closed) the support structure for which is painted grey.
Right: view from below the weapon bay, looking upstream. Visible are the rectangular nozzle (blue), weapon bay (open, but with doors removed; green) and the missile (off-white). The technician and traffic cone provide an impression of scale.

---

[*] While the attendant savings are still of value in focussed investigations involving only a limited number of computations – especially those seeking to produce data for low frequencies – the real significance of these savings should be viewed in the context of the more extensive computational campaigns required to support the characterisation of aero-structural behaviour over the entirety of the intended operational envelope.

In the event, we were able to secure valuable physical measurements via a collaborative activity which saw MBDA work with BAE SYSTEMS and Harris Release Systems to demonstrate the carriage and release of a full-scale, structurally representative missile model from a bespoke weapon bay rig, incorporating a live ejector unit.[8] Even though the rig was designed to be used with an existing ground test facility, the photographs in Figure 23-4 show that this was no small endeavour.

While it was not possible to use the latest optical surface or off-surface flowfield measurement techniques with this test arrangement, it was possible to furnish the test equipment with considerably more instrumentation (primarily dynamic pressure transducers, strain gauges and accelerometers) than would have been possible in a smaller wind tunnel model. Since the basic proportions of the weapon bay were scaled from those of the M219 wind tunnel model,[9] it was also possible to obtain read-across from existing wind tunnel test data. Thus, while the physical measurements do not provide data of comparable spatial detail to that afforded by CFD computations, they constitute valuable extensions to the data currently available for assessing the suitability of the modelling assumptions embedded in numerical simulations. Since the outcomes of initial comparisons between the measured and computed data associated with these tests have not yet been reported in the open literature, further comment on them is beyond the scope of the current paper. The purpose for including this material herein is merely to illustrate (i) that despite the improvements that continue to be made in measurement techniques and technologies, it is not always possible to secure physical measurements of all quantities of interest in numerical simulations; and (ii) the lengths to which it can be necessary to go in order to secure the physical measurements required to support the validation of CFD computations.

3.0 MATTERS ARISING FOR CFD VERIFICATION

Having provided some perspectives on the scope of the challenges facing the missile aerodynamicist seeking to use CFD to predict separated flows, we now move focus to the topic of CFD verification. We will do so under the following headings: (i) Solution Verification and (ii) Code Verification. Many of the challenges are not unique to missile aerodynamics applications. Again, the material presented is not intended to provide a comprehensive account. Instead, for convenience, emphasis is placed on identifying matters that have been echoed by recent AVT-316 Missile Facet experience with the CFD_OTC1 Test Case, as reported in [1]. Finally, recalling that the remit of this paper is to identify challenges and research priorities, the material presented focusses on open questions and issues, rather than on providing a systematic justification of their place or importance in the wider scheme of things.

3.1 Solution Verification

Solution verification is usually considered to pertain to the quantitative estimation of the numerical accuracy of solutions to the governing equations produced by a CFD code. A key aspect of this is the reduction of spatial (and, where appropriate, temporal) discretisation error to the point at which these errors may be estimated with adequate confidence. In a nutshell, this is the goal of mesh convergence (etc) studies; it is also fundamentally linked to the process of building a computational mesh in the first place. The responsibility for undertaking these tasks (usually) lies firmly with the CFD user; both can be extremely taxing and are recognised as being so.

The experience of the Missile Facet to date in this regard is illuminating in at least two respects. Firstly, at the time of writing, it has not yet been demonstrated how the various contributions to discretisation error may be reduced to the point at which the effect of other modelling assumptions – the choice of turbulence model, for instance – on the computed total rolling moment coefficient, C_l, can be studied reliably at the mandatory flight condition. Secondly, it would appear that the acute sensitivity of the computed value of C_l to the computational mesh is confined to a reasonably small region of the flight envelope.

That the requirements for solution verification are dependent on the nature of the flow being modelled is not surprising. Nor, given the complexity of the flow topology and its apparent sensitivity to the flight conditions, is the observation that current *a priori* meshing guidelines do not appear to be particularly reliable – after all, this is one of the attractions of adaptive mesh refinement. Assuming discretisation error may simply be reduced by increasing mesh resolution and/or increasing the order of the numerical schemes, these problems may be expected to reduce with the passage of time (due to the availability of increased computing power and capacity; improved algorithms, etc.). It is possible to anticipate a future in which the additional elapsed time and cost associated with these larger computations would be sufficiently small as to render fretting further about them redundant. However, in the meantime, the effort involved in reducing discretisation error can be considerable (especially given the diversity of airframes and flow regimes of potential interest) and the risks of it either going undetected or being misinterpreted are not insignificant.

3.2 Code Verification

Code verification is usually considered to pertain to the assessment of: (a) the adequacy of the numerical algorithms to provide accurate numerical solutions to the governing equations used in the mathematical model; and (b) the fidelity of the computer programming to implement the numerical algorithms used to solve the discrete equations. Responsibility for the latter (i.e. (b)) lies squarely with the CFD code developers. While the same also applies to (a), the situation is a little more nuanced.

The AVT-316 Missile Facet found that limiters – mathematical devices introduced to improve the stability and robustness of the numerical solution process – were affecting the intensities of and trajectories being followed by some of the vortices in the flow. To those who were aware (their effect on vortices is rarely reported in the fixed wing literature – [10] being a notable exception), this was not particularly surprising. However, investigation of the associated effects was only possible because one facet member had direct access to code developers who, in turn, had access to implementations of various limiters and were able to allow their influence on the computed results to be visualised. They were also able to re-visit the types of systematic study undertaken during code development (namely comparing model outputs with exact solutions) to illustrate the attendant sensitivities in a context more closely related to the CFD-OTC1 Test Case.

There tends to be better awareness in the user community of the related numerical problems that can occur in the vicinity of shock waves. These become more aggressive at higher Mach numbers – [11,12] provide perspectives on the lengths to which it can be necessary to go in order to prevent visible corruption of the resulting flow solutions. However, despite their potential importance, there is currently generally little provision in the way of features to draw users' attention to the influence that limiters (or other such numerical algorithms) may be having on their flow solutions. Like other sources of discretisation error, they may be expected to become less of a problem in the future. However, in the meantime, while making suitable additional information available for the user to monitor might appear to be contrary to the aims of process simplification and automation,[*] it is important. Without it, there is a risk that the effects associated with limiters (&c) may be confused with other less immediately tangible aspects of the solution – those associated with turbulence modelling, for instance.

The above discussion provides several further examples of the benefits that can be gained by facilitating close working between different stakeholders: the Missile Facet is demonstrating the value of (i) interactive community benchmarking amongst users (bringing different tools, practices and experience to the table) and (ii) users having interactive dialog with CFD code developers. Recognising that the numerical sources of discretisation error discussed above have potential implications for both the geometrical properties of the stencils being used by the flow solver and their alignment with certain features in the flowfield being

---

[*] This need not be the case: adaptive mesh refinement schemes addressing the progressive reduction of the influence these numerical algorithms during a RANS computation need not increase user workload. However, even here, the benefits of providing facilities for what might loosely be termed "solution integrity checking" are not diminished.

simulated, it is reasonable to expect that additional benefits may accrue from similar interactions with the developers of mesh generation software.

In spite of the potential benefits, establishing opportunities for such diverse communities to work together to address topics pertaining to CFD verification is, in itself, subject to its own challenges. Amongst these is an interest in focussing on what happens next, i.e. what one does when in possession of a suitably verified CFD solution.

4.0 BRIDGING THE GAP BETWEEN VERIFICATION AND PREDICTION

This broad topic will be addressed under the following headings: (i) Model Inputs (and Assumptions) and (ii) Model Outputs. As with the previous Section, it is recognised that many of the challenges are not unique to missile aerodynamics applications. Again, the material presented focusses on some of the open questions and issues, rather than on providing a systematic justification of their place or importance in the wider scheme of things.

4.1 Model Inputs (and Assumptions)

The practising industrial missile aerodynamicist is posed with a variety of challenges and dilemmas in ensuring that CFD solutions are fit for their intended purpose. Many of those not already mentioned arise from a keen awareness of the potential limitations of RANS (currently – primarily for reasons of time and cost – the first port of call in most circumstances) for simulating separated (and therefore unsteady) flow. These are addressed, briefly, under two headings (i) Understanding the Options (ii) Justifying the Approach, below.

4.1.1 Understanding The Options

Of the choices to be made when preparing to use CFD, some of the hardest to make derive from the knowledge that it is not (currently) possible to set out to solve the full Navier-Stokes equations for situations of immediate practical interest. Anticipating the potential implications of using the additional modelling assumptions this necessitates is far from straightforward. For instance, while in the past, most models of vortices were, of necessity, inviscid and conceptually possible to visualise, with the reliance on turbulence models to provide mathematical closure, this is no longer the case with RANS. Great difficulty can be encountered in constructing conceptual frameworks that may be critically assessed in the context of the separated flows being studied. Moreover, according to one of the most respected contemporary developers of turbulence models, “even deep inside [a mature] vortex, the theory of turbulence is incomplete and controversial”.[13]

Similar difficulties persist for variants of the governing equations that resolve – rather than model – components of the turbulence in the flow being simulated. Here the situation can be compounded by the increased complexity of the resulting flows and the need to address the time-dependent aspects of the ways in which small disturbances are propagated: many of the larger-scale phenomena observed in separated flows depend on the formation of coherent structures from localised instabilities in the flow. (Recall, also, that separation is, itself, a highly localised event.)

When viewed from this perspective, establishing the extent to which the models employed (together with the manner of their implementation) influences these processes is a particular concern. To illustrate this point, in Large Eddy Simulations (LES), where the effect of small scales on the resolved-scales is modelled and is referred to as sub-grid-scale modelling, conventional Favre filter-based LES formulations and their sub-grid-scale models do not represent variable-density physics at sub-grid-scales. Studies using a recent re-formulation,[14] which permits direct representation of variable-density sub-grid-scale dynamics, has indicated that the sub-grid-scale baroclinic torque (which represents the differential acceleration of a variable-density fluid element) plays an important role in the formation of various instabilities encountered in compressible reacting flows, including those crucial to the functioning of high speed combustors. There is clearly much still to be learned here – even in scenarios like those discussed in Section 2.1 and 2.2, from which, for various reasons, it is customary not to include the effects of heat addition or reacting flow.

As the computational power available to researchers continues to increase, ambitions to attempt the simulation of increasingly complex flow phenomena will be realised. It can be difficult to keep pace with the latest developments, let alone undertake critical assessments of their utility. Appeals for improved user awareness and education, for all aspects of CFD application, are becoming more prominent in the literature – see e.g. [15]. Interaction between stakeholders at all technology readiness levels – from fundamental researchers and code developers, through to industrial users – will play a key role in responding to such requests. Moreover, by raising awareness and improving insight, such interactions may yield better-focussed stimuli for future research and development.

4.1.2 Justifying the Approach

Further to the matters identified above, modelling choices made by industrial aerodynamicists may be influenced by a host of other factors. Usually underpinned by concern for a wider balancing of time, cost and quality, these may evolve during the process of undertaking the computations. For instance, depending on the circumstances in which the computations are being undertaken, decisions made on encountering unexpected problems may be made either tactically (e.g. stop; rely on heuristic judgement; postpone further investigation) or strategically (e.g. continue, possibly with expanded resources, providing regular updates on progress). When embarking on new areas of application, encountering such problems is not uncommon. Indeed, they are not unheard of in more established fields of CFD application, either – the discovery of non-unique solutions, for instance.[16,17]

In the context of this paper, what is important here is that, whatever modelling approach is adopted, estimates of data uncertainty are provided: the supporting justifications, together with their associated traceability, are vital in assessing the level of risk being carried forwards in the development of a missile airframe. It is also fair to say that, if a clear – and affordable – route forward to solving a problem can be identified, there is a much higher chance of a strategic approach being taken towards its resolution. Again, this places various challenges on the protagonists and on their ability to provide evidence to substantiate any claims made.

4.2 Model Outputs

In one sense, any suitably verified CFD flow solution (or successful "modelling outcome") may be regarded as a prediction. Care needs to be taken at this point, however, since there is not unanimity of opinion regarding the associated semantics or taxonomy. The accepted convention is to regard CFD verification as being concerned with the accuracy to which model outputs comply with the governing equations: it says nothing (directly) about how closely these equations (or, by implication, the modelling outputs) correspond to the physical world. The latter is clearly of central concern to the practising aerodynamicist: as explained above, it is vitally important to be able to quantify the confidence held in a prediction and to explain the basis for this. Comparison with physical measurement is clearly of fundamental importance here. However, there is also an important distinction to be made between model outputs that are compared with extant physical measurements and those which are not.[*] With regard to semantics and taxonomy, much hinges on how the term validation is defined and on how the definition is interpreted. These are surprisingly subtle and nuanced subjects [18] and are addressed in more detail in [19].

---

[*] Note that there are similarities here with the difficulties that can be encountered during solution verification, as discussed in Section 3.1, above.

The use of physical measurements to establish the capabilities of mathematical models and to guide the development of new, improved ones is a well-established and fundamental component of scientific method. From the material that has been presented above, it is clear that improvements in the ability to use CFD to predict separated flows will necessitate physical data being measured in greater (spatial and temporal) detail and to greater precision than might otherwise be obtained in wind tunnels or flight test. Over the last two decades or so, a more widespread recognition of similar requirements has led to the development of "validation experiments" – experiments designed with the specific purpose of gathering data to allow the accuracy of CFD computations to be assessed.

Given the scope of the requirement (in terms of the diversity of separated flows of interest) and the constraints imposed by the capabilities of current ground test facilities and measurement techniques, this poses a variety of challenges. Related questions include: How should such experiments be designed? What flow conditions should be studied? What measurements should be taken? Is there a sequence in which they should be conducted? In the author's opinion, [19] such experiments should not be planned without first conducting systematic studies of the flows of interest using CFD. Aside from anything else, recognising that the spatial and temporal resolution of physical measurements is likely to be more constrained than that afforded via CFD, this will help prioritise measurement requirements. In view of the relative difficulty and expense associated with characterising the flow in ground test facilities, such preparatory activities should also include simplified CFD-based sensitivity studies to address the potential impact of any anticipated deficiencies in the planned physical measurements. Adoption of both of these pre-requisites will help establish clear objectives for the experiments, both in terms of their conduct and the subsequent use of the physical data they generate. They might also identify alternative means of making progress towards achieving the strategic goals of assessing and improving the accuracy of CFD computations – possibly in the form of simpler, more readily achievable stepping stones or exploratory activities designed to provide insight rather than detail.

A growing body of experience in undertaking validation experiments is developing in support of various areas of CFD application – see e.g. [20]. Although several examples have relevance to the topic of this paper, to the best of the author's knowledge, none originated specifically to address challenges identified by missile aerodynamicists. More-or-less in parallel, a growing collection of CFD-related workshops have been established, including those convened by the American Institute of Aeronautics and Astronautics. As [21] attests, much has been learned – by participants and observers alike – from these workshops. It is in the context of these types of community-based endeavour that MBDA developed the CFD-OTC1 Test Case. While vortex dominated flows do not necessarily constitute the most complex forms of separated flow physics missile aerodynamicists are required to characterise, the challenges posed by the routine use of CFD to predict them with confidence are considerable. They are also, potentially, shared with stakeholders in other parts of the aerospace industry.

5.0 CLOSING REMARKS

Looking to the future, there can be little doubt that the adoption of systematic approaches to the tasks of establishing and improving the capabilities of physical models embedded in CFD tools will be vitally important in establishing improved confidence of the CFD prediction of separated flows. Such improvements are expected to play an important role in reducing missile system development costs and risks – particularly in circumstances where the alternatives (physical testing) are either extremely expensive or high risk (or both).

In seeking to provide a glimpse into the challenges and research priorities associated with the use of CFD to predict separated flows of interest to missile aerodynamicists, this paper has touched on a number of topics that are potentially of much wider interest. Several examples have been provided to illustrate the types of mutual benefit that are most readily gained by working with other stakeholders. The micro-community of the

AVT-316 Missile Facet is proving highly effective in this regard. As we wait to see what else it will uncover during the remainder of its term, the author would like to thank the NATO STO for the role it played in facilitating its creation.

ACKNOWLEGEMENTS

The author would like to thank the organising committee for their invitation to make this presentation at this symposium. He would also like to thank his fellow AVT 316 Missile Faceteers: as befits a missile-related team, there have been no passengers in the pursuit of our objectives and our shared experience has provided a valuable backdrop for this paper. Finally, he would like to register his thanks to Michael Reeves and Stephen Lawson, colleagues at MBDA who played central roles in developing the CFD_OTC1 Test Case.

REFERENCES

[1] Taylor, N.J., et al, “The prediction of vortex interactions on a generic missile configuration using CFD: current status of activity in NATO AVT-316”, STO-MP-AVT-307-24, 2019.

[2] Ericsson, L.E. and Reding, J.P., “Asymmetric vortex shedding from bodies of revolution”, in Tactical Missile Aerodynamics, Prog. in Astronautics and Aeronautics, vol.104, 1986.

[3] Champigny, P., et al, “Numerical Simulation of Forebody Vortices at High Angle of Attack and their Control Using Innovative Systems”, RTO-MP-AVT-135-24, 2006.

[4] Lamont, P.J., “The effect of Reynolds number on normal and side forces on ogive-cylinders at high incidence”, AIAA 85-1977, 1985.

[5] Bush, R., et al, “Recommendations for future efforts in RANS modelling and simulation”, AIAA 2019-0317, 2019.

[6] Lawson, S.J. and Barakos, G.N., “Review of numerical simulations for high-speed, turbulent cavity flows”, Prog. Aerospace Sciences, vol.47(3), pp186-216, 2011.

[7] Loupy, G.J.M., Barakos, G.N. and Taylor, N.J., “Multi-disciplinary simulations of stores in weapon bays using scale adaptive simulation”, J. Fluids and Structures, vol.81, pp437-465, 2018.

[8] Astley-Jones, P., et al., “Making a lot of noise for future weapons bays”, RAeS Conference: RAF Weapons Systems – Past, Present and Future, May 2018.

[9] Henshaw, M., “M219 Cavity Case”, Case 20 in Verification and validation data for computational unsteady aerodynamics, RTO-TR-26, 2000.

[10] Wooden, P.A., et al, “CFD Predictions of Wing Pressure Distributions on the F-35 at Angles-of-Attack for Transonic Maneuvers”, AIAA 2007-4433, 2007.

[11] Candler, G.V., et al, “Development of the US3D Code for Advanced Compressible and Reacting Flow Simulations”, AIAA 2015-1893, 2015.

[12] Ameur, F.B and Lani, A, “Physics-Based Mesh Fitting Algorithms for Hypersonic Flows”, AIAA 2019-1997, 2019.

[13] Spalart, P.R., and Garbaruk, A.V., “The Prediction of Common Turbulence Models in a Mature

Vortex", Flow Turbulence and Combustion, September 2018.

[14] Sidharth, G.S., et al., "Filtered velocity based LES of mixing in high speed recirculating shear flow", AIAA 2016-3184, 2016.

[15] Spalart, P. R and Venkatakrishnan, V., "On the role and challenges of CFD in the aerospace industry", Aeronautical Journal, pp209-230, January 2016.

[16] Masters, D. A., et al, "Impact of Shape Parameterisation on Aerodynamic Optimisation of Benchmark Problem," AIAA 2016-1544, 2016.

[17] Cary, A., et al, "Current practice unstructured grid CFD results for 3rd AIAA High Lift Prediction Workshop", AIAA 2018-1037, 2018.

[18] Roache, P.J., "Perspective: Validation—What Does It Mean?", J. Fluid Eng., vol.131, 2009.

[19] Taylor, N.J., "CFD Validation: What is it and how does it affect us?", RAeS Aerodynamics Conference, October 2019.

[20] Rumsey, C.L., "The NASA Juncture Flow Test as a Model for Effective CFD/Experimental Collaboration", AIAA 2018-3319, 2018.

[21] Morrison, J.H., Kleb. B. and Vassberg, J.C., "Observations on CFD Verification and Validation from the AIAA Drag Prediction Workshops", 2014-0202, 2014.