Some Applications of Computational Aerodynamics to Support Guided Weapon Design and Development

Scott SHAW, Bernard DENCHFIELD, Jon ARMSTRONG, Ben HINCHLIFFE, Jed HOLLOM
MBDA UK Limited
UNITED KINGDOM

scott.shaw@mbda-systems.com

ABSTRACT

Understanding the external aerodynamics of a missile airframe is important to establishing overall system performance. The aerodynamics of the configuration influences kinematic performance, stability, control and manoeuvrability of the weapon and its safety during carriage and release. Characterizing the aerodynamic loads generated by a missile airframe is therefore a fundamental aspect in the development of the missile system.

Within the MBDA-UK aerodynamics department, the use of computational fluid dynamics to characterize the aerodynamics of the airframe is ubiquitous. The use of computational fluid dynamics through the missile development life cycle is illustrated by three use-cases related to the carriage and launch of an internally carried air-to-air missile launched from a platform bay. The three use cases (i) Free air aerodynamics (ii) characterization of the aero-acoustics of payload bays and (iii) interaction between the platform and missile during release illustrate different aspects of the engineering use of CFD analyses at MBDA and reveal the inherent tensions between the development of analysis tools to support engineering practice and the development of the science of engineering simulation.

1.0 INTRODUCTION

Understanding the external aerodynamics of a missile airframe is important for establishing overall system performance. The aerodynamic characteristics of the configuration influence kinematic performance, stability, control and manoeuvrability of the weapon and its safety during carriage and release. Characterizing the aerodynamic loads generated by a missile airframe is therefore a fundamental aspect in the development of the missile system.

1.1 Historical Perspective

In the late 1980s, it was imagined that future computing performance increases would not only allow us to run many more existing Computational Fluid Dynamics (CFD) simulations much more quickly, but also increase the fidelity of those calculations. The vision was for a "virtual wind-tunnel" enabling rapid numerical analysis and assessment of imaginative novel airframe configurations. It was even widely expected that such a numerical simulation capability would eventually remove the need for wind tunnels entirely.

The example multi-block structured mesh shown in Figure 1-1 illustrates a typical industrial application of (Euler) CFD for missiles dating from around 1988. For the (limited) flight conditions of interest, results obtained at that time were considered acceptable for engineering purposes; proving to be better than semi-empirical methods, whilst also quicker and cheaper than wind-tunnel tests.

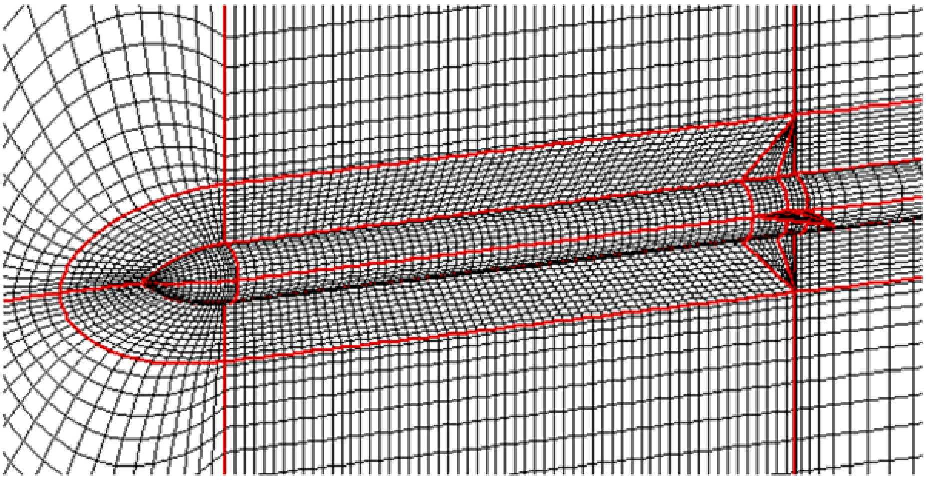


**Figure 1-1: Weapons Multi-Block Suite structured mesh
(38,146 cells, Euler solution required around 1 CPU hour on Cray-1S, c.1988)**

As available computing power increased, however, so too did industrial ambitions for exploring ever more complex airframe geometries (increasingly beyond the scope of semi-empirical methods) at more challenging flight conditions. This expansion in usage exposed the fundamental limitations of all CFD methods arising from the inherent numerical and physical modelling approximations needed to represent a continuous world using discrete points. So began the seemingly endless quest for mesh resolution and flow solution fidelity, with each new generation of computational capacity encouraging ever-finer meshes and increasing flow solver complexity. The original vision of the "virtual wind-tunnel" remains tantalisingly just beyond reach.

The realisation that simply increasing computing power will not overcome fundamental limitations of current industrially usable CFD methods is now driving renewed research and development efforts in several competing directions. Ironically, the continuing need for real wind tunnels is also becoming increasingly apparent. The relatively low entry bar for producing unsubstantiated CFD predictions has increased the perceived value and credibility of physical test results. Far from replacing the wind tunnel, it is clear that successful CFD method development remains critically dependent upon high quality experimental data, and close collaboration between theoreticians and experimentalists.

1.2 Use of Computational Fluid Dynamics in Design

Historically the use and importance of CFD within aerodynamic design can be organised into three main eras, illustrated in Figure 1-2.

1.2.1 Analysis of Single Configurations, Out-of-iteration

In its early use, the time required to prepare and perform CFD analyses was too long for the results to be used within a single design iteration. Results generated by analysis were useful in helping to providing a detailed understanding of observed performance but did not contribute significantly to design decisions.

1.2.2 Analysis of Single Configurations, Within-iteration

In the mid 1990s dramatic reductions in the cost of computing, primarily due to the development of high-performance computing infrastructure based upon commodity hardware and message passing communication paradigms, significantly increased the utility of CFD computations. Results could now be obtained at acceptable cost within a timeframe that allowed insights from the computations to be used within the design iteration. Design decisions were now informed by analysis.

1.2.3 Analysis of Multiple Configurations, Within-iteration

In this final era, which began towards the middle of the first decade of this century, further reductions in the time and cost of performing CFD simulations allowed multiple configurations to be assessed within each design iteration. The use of CFD simulation in aerodynamic design was transformed from a supporting activity to the main means of driving design.

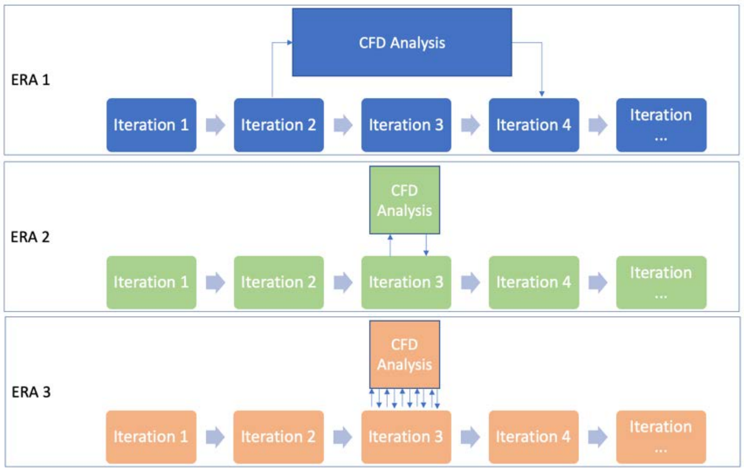


Figure 1-2: Historical Usage of CFD

In recent times, the situation has become less clear. Reynolds-Averaged Navier-Stokes (RANS) based simulations continue to be widely used to characterize multiple configurations within single design iterations. Indeed, further reductions in the time and cost of RANS simulations is allowing them to be used earlier and earlier in the design activity. At the same time, the advent of scale-resolving methods such as large and detached eddy simulations has provided increased physical fidelity but at much greater cost and results obtained in this way are used either out of iteration or sparingly within iteration.

In order to illustrate some important aspects of how MBDA-UK uses computational aerodynamics, and where we believe developmental effort can have greatest impact, we confine our discussion to three key phases of the launch of an internally carried air-to-air missile. The three phases are shown in Figure 1-3. The remainder of the paper deals with these three phases in turn. In the next section, we discuss the computation of the so-called free-air aerodynamics, the aerodynamic characterization of the weapon in free flight far from the platform. We then consider the case of the weapon in its carriage position before finally considering the case where the displacement effect of the aircraft on the local airflow around the weapon is significant, weapon-platform interaction. The paper then concludes with our recommendations on how developers can best use their efforts to support our industrial processes.

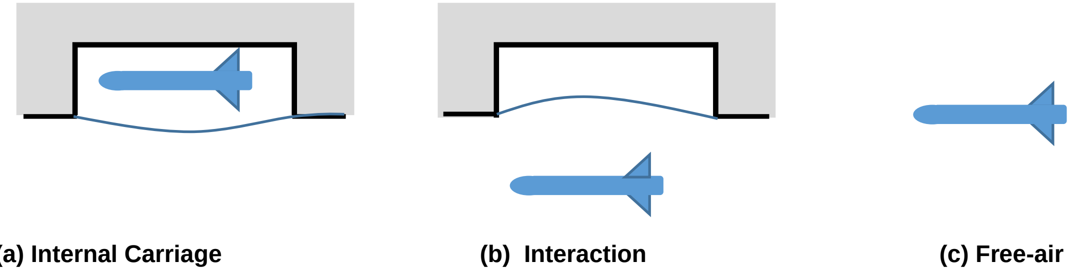


Figure 1-3: Key phases during the launch of an internally carried weapon.

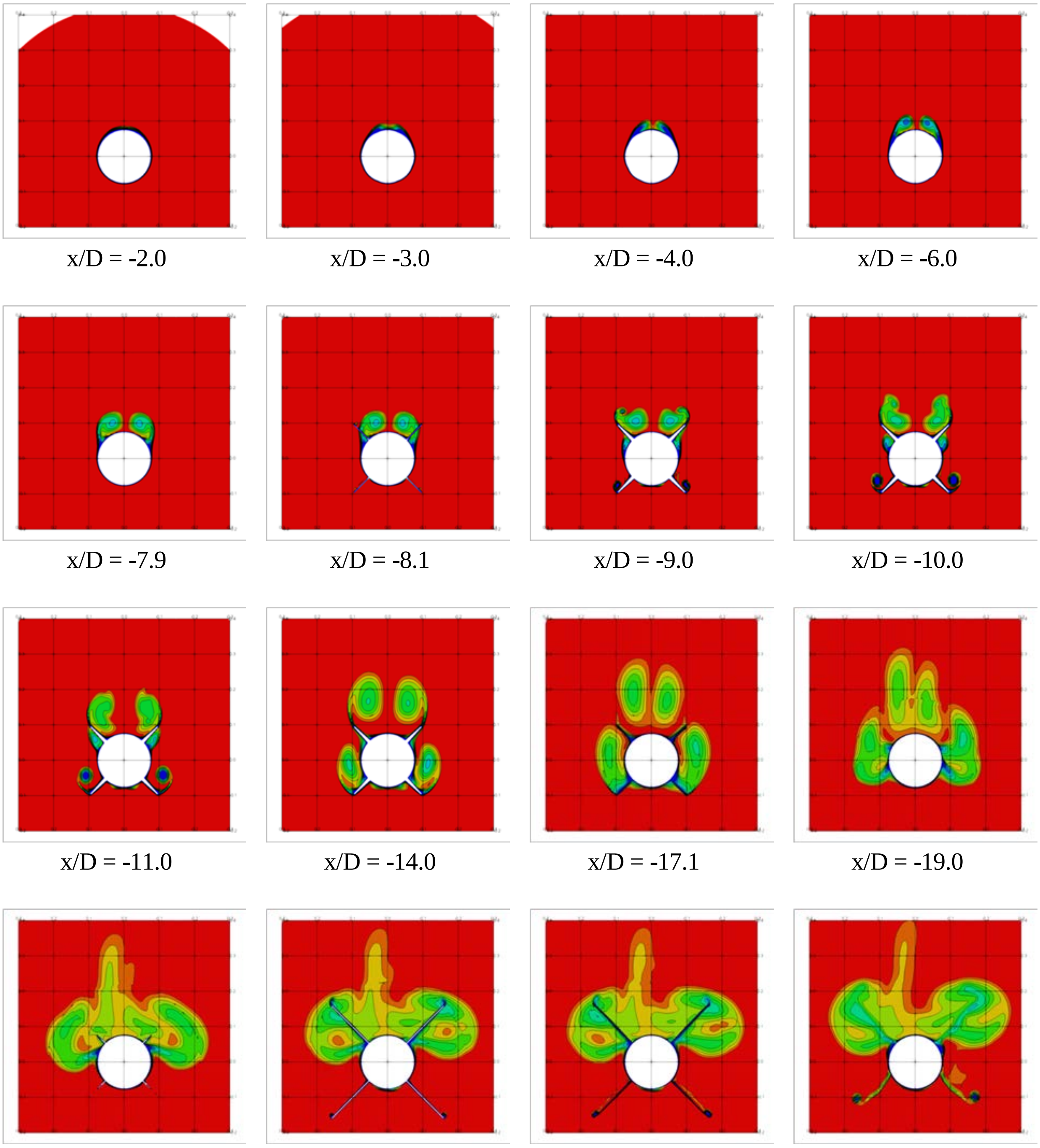
2.0 FREE-AIR AERODYNAMICS

The use of CFD to compute the free-air aerodynamics of missile configurations occurs throughout the development life cycle. In early stage pre-conceptual design, the need for detailed aerodynamic models is less important and typically, only a small number of cases need to be computed. However, the number of configurations that need to be considered in order to arrive at a balanced design is large. As the development process continues, the number of configurations that need to be considered diminishes rapidly, but the number of cases that must be evaluated for each configuration increases significantly. Finally, in detailed design and during integration with a platform only a very small number of configurations need to be considered, but many hundreds of evaluations may be required to populate the aerodynamic model. Large numbers of CFD evaluations are therefore required at all stages of the development process and consequently the cost of a single calculation can be a significant driver for time and cost.

Consequently, MBDA-UK typically employs simulations based upon the solution of the steady Reynolds Averaged Navier-Stokes equations together with a turbulence closure. In common with much of the external aerodynamics community, the Spalart-Allmaras [1] and Menter SST [2] turbulence models and their developments are used.

Shaw et al [3] present a detailed assessment of the application of RANS turbulence models for the MBDA open test case OTC1 [4] that illustrates the challenges faced computing the supersonic flow around missile configurations. Although the OTC1 test case is geometrically simple, the resulting flow field is rich involving multiple shockwaves, boundary layers, wakes, vortices and their interactions. Figure 2-1 shows contours of total pressure ratio obtained at various locations along the axis of the OTC1 missile viewed from front to aft at a nominal flow condition of Mach number $M = 1.4$, total incidence $\sigma = 15^o$ at sea-level conditions. The results were obtained by solving the RANS equations with a turbulence closure based on the SA-neg turbulence model.

The flow development over the forebody closely matches that expected for slender bodies. Despite a significant thickening of the leeside boundary layer, there is little evidence of crossflow separation over the nose ($x/D > -2.0$) and it is not until $x/D = -3.0$, a calibre aft of the nose body junction that crossflow separation first becomes evident. The separated shear layers roll up to form a pair of coherent counter rotating vortices in the leeside wake of the body. The vortex cores grow in size and the mutual interaction between the vortices and body causes them to be pushed away from the feeding sheet. The resultant stretching produces a non-circular core. The effects of the slight flow asymmetry produced by the roll angle are clearly visible with the starboard side separation occurring slightly later than on the port side. This difference in separation location is evident in the relative position of the vortex cores. There is some evidence of secondary separation below the primary vortices.



x/D = -2.0 | x/D = -3.0 | x/D = -4.0 | x/D = -6.0
x/D = -7.9 | x/D = -8.1 | x/D = -9.0 | x/D = -10.0
x/D = -11.0 | x/D = -14.0 | x/D = -17.1 | x/D = -19.0
x/D = -21.1 | x/D = -21.9 | x/D = -22.1 | x/D = -23.0

Figure 2-1: Evolution of the flow around the OTC1 missile at the nominal flow condition, contours of normalized total pressure (SA-neg model, structured mesh).

The immediate effect of the wings is to detach the forebody vortices from their feeding sheets (see for example x/D = -8.1) resulting in an outwards movement of the vortex core. The forebody vortices continue to grow becoming more circular in shape. At the same time, additional vortices begin to form along the tips of the wings. The development of the windward and leeward tip vortices is very different. On the leeward side, the behaviour of the tip vortex is dominated by its interaction with the forebody vortex. The vortices are co-rotating and initially well separated, the individual cores remain coherent and rotate about one another. As the vortices age further, their cores spread by viscous diffusion, the ratio of core diameter to separation distance increases and the vortices begin to merge. This process is relatively rapid and results in the forebody and tip vortex merging into a single vortex, see for example x/D = -11.0. Once the merging of the forebody and wing tip vortices on the leeward-side has completed the vortex grows rapidly and develops an elliptical shape. On the windward side the vortex development is similar to that observed for low aspect ratio wings, and once it has formed the vortex grows rapidly and develops an elliptical shape.

Downstream of the wings there are two features of note. Firstly, there is evidence of a strong interaction between the leeward and windward vortices. It appears that the vortices rotate around one another, although the structures are indistinct. In addition, the wake structure that develops behind the wings (x/D = -17.1) interacts with the free vortices and is entrained into the vortical flow (x/D = -19.0). By the time the flow reaches the missile fins, the wing wake structure is no longer distinct. The interaction of the remnants of the forebody vortex with fin 4 appears strong with the fin tip appearing to pass through the core (x/D = -21.9, x/D = -22.1).

The flowfield associated with the OTC1 test case is typical of that for many missile configurations. Indeed vortices and their interactions are fundamental to the aerodynamic characterisation of nearly all missile airframes. Unfortunately, the prediction of vortex-dominated flows is challenging for conventional turbulence model based approaches due to the sensitivity of the flow to the rate of decay of the vortex. The rate at which the vortex decays is fundamental to the characterisation of its behaviour downstream, in particular its interaction with other features, such as solid surfaces and vortices. In numerical simulations based upon the RANS equations, the decay of the vortex occurs due to the action of:

(i) Diffusion related to the effective viscosity of the fluid.

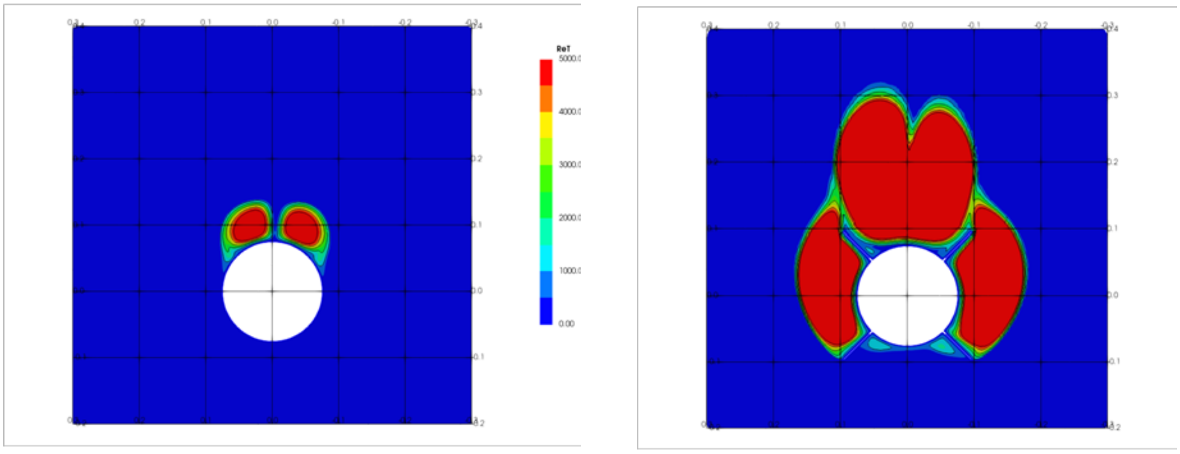
(ii) Dissipation due to truncation error.

(iii) Dissipation due to other features of the numerical method

(iv) Diffusion due to the over prediction of turbulent eddy viscosity by the turbulence model

The action of (ii), (iii) and particularly (iv) can overwhelm the action of the physical viscosity resulting in vortices that decay and grow more rapidly than is observed in physical experiments, for further discussion see Bradshaw [5].

Dissipation due to truncation error and other features of the numerical model can often be addressed by careful construction and local refinement of the computational mesh. However, dealing with over prediction of viscosity by the turbulence model is more problematic. The fundamental problem is clearly illustrated in Figure 2-2 which shows computed contours of turbulent Reynolds number, the ratio of the turbulent eddy viscosity to the molecular viscosity, for the OTC1 test case at two streamwise stations. We observe that in the region of the vortices, the turbulent eddy viscosity is 3-4 orders of magnitude greater than that of the molecular viscosity. This seems unreasonable on physical grounds and results in computed vortices that rapidly dissipate as they travel downstream. In the case of the OTC 1 test case, this affects downstream vortex-vortex interactions that changes the strength and geometry of the vortex field close to the fins.



(a) x/D = -7.9 **(b) x/D = -16.9**

Figure 2-2: Evolution of the flow around the OTC1 missile at the nominal flow condition, contours of normalized total pressure (SA-neg model, structured mesh).

Several efforts have been made to remedy this behaviour including attempts to sensitize the model equations to curvature and rotation. The use of such palliatives results in significant changes to the structure of the vortices and their downstream interactions that in turn result in significant increases in rolling moment. This is illustrated in Figure 2-3 which shows comparisons of rolling moment coefficient computed using several models and several codes.

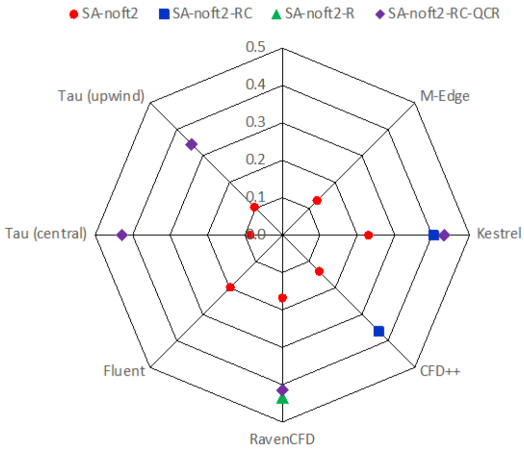


Figure 2-3 Influence of rotation correction on computed rolling moment coefficient (unstructured mesh).

Scale resolving methods, see for example Tormalm [6] and Barakos [7], appear to offer a resolution to this issue. Figure 2-4 provides a comparison of the convergence histories of the peak turbulent Reynolds number for calculations employing the Spalart-Allmaras model and two scale-resolving IDDES methods. The IDDES methods appear to behave in a much more physically meaningful way, resulting in much stronger vortices and a significant (more than double) rolling moment coefficient.

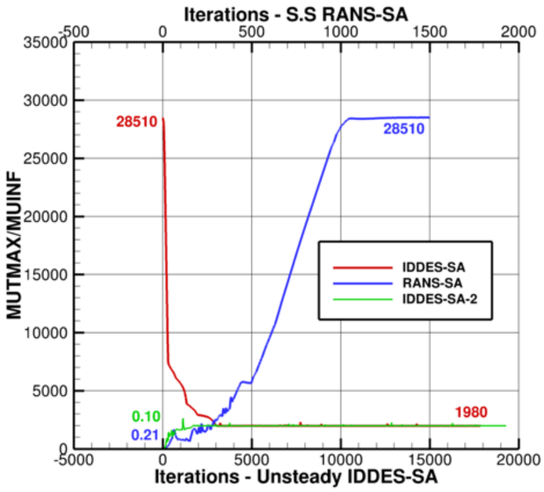


Fig. 2-4 Iterative convergence of maximum turbulent Reynolds number (ARL, unstructured mesh).

The apparent improvements offered by scale resolving simulations is promising and has motivated significant recent efforts amongst the CFD research community at the expense of more traditional turbulence modelling approaches. However, it must be recognised that with even relatively poorly resolved detached eddy simulations costing $O(1000)$ more than related RANS calculations and requiring days of wall-clock time rather than hours, the use of scale resolving simulations in aerodynamics would turn the clock back to the first or second eras discussed in the introduction. Such approaches are therefore unlikely to displace RANS as the main means of producing aerodynamic data in the short to medium term.

Instead, we argue that there is an industrial need for development efforts to be focussed on understanding and extending the utility of RANS based models for vortical flows. Adopting some of the ideas expounded by Spalart [8], we believe that this effort should be both systematic and more openly empirical.

Although progress has been made in sensitizing standard turbulence models to the effects of streamline curvature and system rotation, the models still perform inadequately for vortex-dominated flows. A more systematic approach based upon the solution of the Reynolds stress transport equations seems to offer the most promising avenue for definitive improvement.

We should also realise that the search for a single, all-purpose turbulence model is likely to end in failure and disappointment. Being more systematic in our modelling is therefore not sufficient in itself. Instead, industry must actively embrace a more openly empirical approach in which we work together with research and development partners to calibrate models for our specific needs. This approach is not new; industry has long used semi-empirical aerodynamic predictions built upon simple theoretical considerations. The problem has been that as the models have become more complicated the knowledge and understanding required to calibrate them has typically exceeded that available within an individual company. Moreover, commercial imperatives have prevented effective transfer of experimental data from industry to the research and development community. If we are to overcome current limitations, a new collaboration is required. To facilitate this collaboration on calibration there is a need for high-quality validation *and* calibration experiments to be performed.

3.0 INTERNAL CARRIAGE

When dealing with the weapon in its carriage position we are particularly concerned with characterizing the aero-acoustic environment within the weapon bay. Although the dynamic pressure within the bay is low when the weapon bay doors are open, the aerodynamic environment is harsh due to the unsteady flow associated with the resulting cavity flow. This has implications for the platform, the weapon and any adjacent stores within the payload bay. Tonal behaviour at lower frequencies can be important for the structural integrity of the missile airframe while higher frequency broadband noise can limit the useful service life of internal sub-systems particularly energetic materials (for example the warhead and rocket motor) and electronics

The mechanism governing the aero-acoustic environment within a cavity is shown schematically in Figure 3-1. As the flow passes over the leading edge, it separates, creating a shear layer between the relatively stagnant air contained within the cavity, and the fast flowing air outside it. A feedback mechanism is created with the rear wall as pressure perturbations are communicated back upstream, eddies form in the shear layer near the leading edge, growing in size as they travel downstream. This leads to a highly unsteady flow regime.

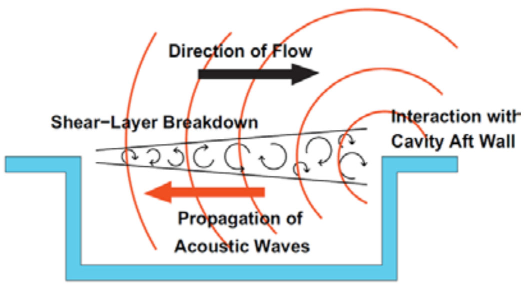


Fig. 3-1 Schematic of the flow over an empty cavity.

The inherently unsteady nature of the problem and the importance of length and time scales similar to those of the largest turbulent eddies requires the adoption of a scale-resolving strategy to the modelling of turbulence. In the simulations discussed in this paper, a direct eddy simulation model has been employed.

Some example results for the empty cavity are shown in Figure 3-2 and Figure 3-3. Figure 3-2 shows instantaneous contours of density gradient. The complexity of the flow and its inherent unsteadiness are clearly visible. The schematic shown in Figure 3-3 shows the potential insights that can be obtained from such visualisations.

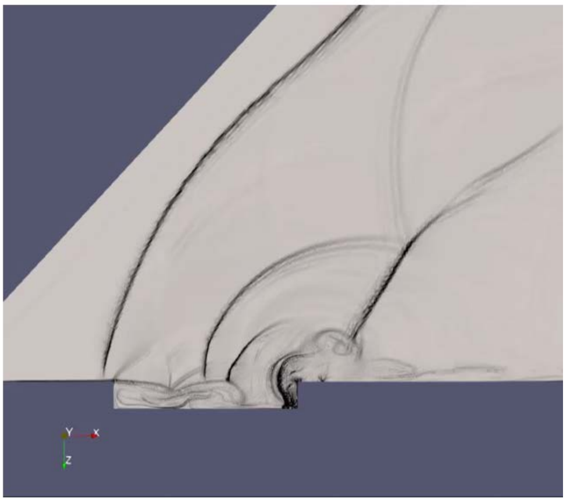


Fig. 3-2 Instantaneous density gradient contours (empty cavity).

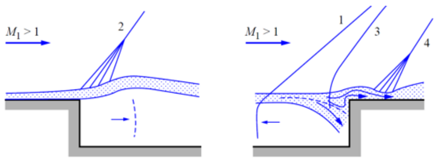


Fig. 3-3 Sketch of inferred flow behaviour (empty cavity).

In order to characterise the aero-acoustic environment within the bay and around the missile, time averaged contour plots of overall sound pressure were created by extracting pressure slices at each time-step and calculating space discretised values with Equation 1.

$$OASPL = 10\log_{10}\left(\frac{P_{rms}^{2}}{P_{ref}^{2}}\right) \quad (1)$$

An example result is shown in Figure 3-4 for the empty payload bay at a Mach number M = 0.85.

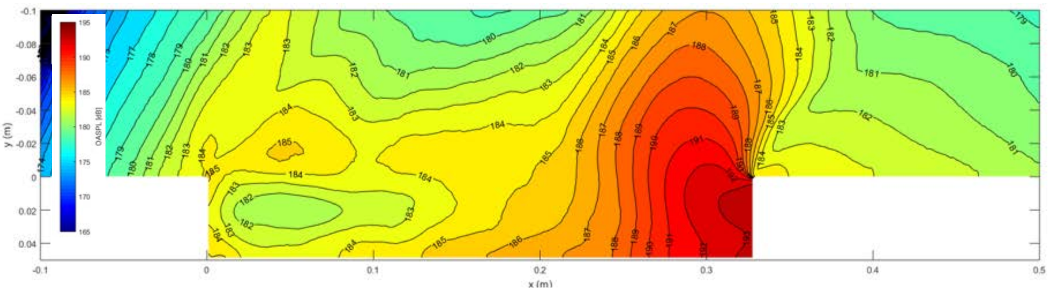
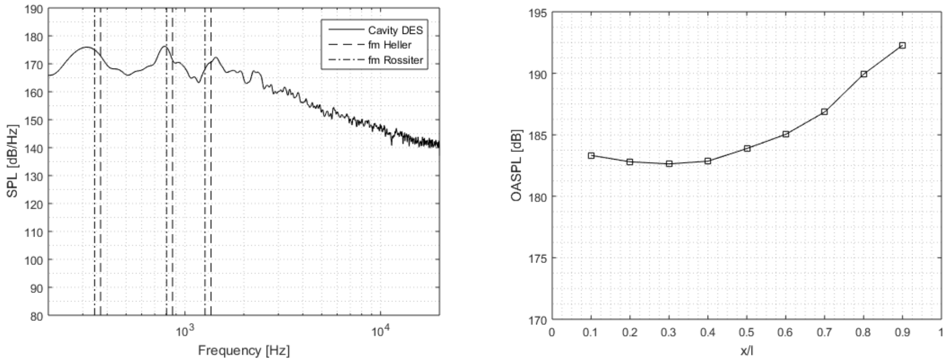


Figure 3-4 Computed OASPL in an empty payload bay.



(a) Computed PSD

(b) Computed OASPL (cavity ceiling)

Figure 3-5 Computed Power Spectral Density (PSD) and Overall Sound Pressure Level (OASPL) for the empty payload bay.

Figure 3-5 shows the computed power spectral density and the overall sound pressure level on the cavity ceiling.

The results of such simulations, together with related computations of the payload bay with the missile in its carriage position provide help to characterise not only the aerodynamics of the problem but also the response of the missile to the unsteady loads. Working with other engineering disciplines within the business, we are able to understand how the weapon bay environment affects the structural integrity and life of the missile. Moreover, the insights into the hydrodynamic and acoustic phenomena offered by the simulations provide important information that can help us to devise effective flow management strategies to mitigate or eliminate the consequences of the noisy environment in which the missile must operate.

4.0 PLATFORM-MISSILE INTERACTION

Our final example is associated with the launch of a missile from the platform payload bay. The objective of this activity is to establish the conditions over which a weapon can be safely and successfully launched or jettisoned from the platform. The process is built on a 6 degrees of freedom rigid body representation of the missile and an aerodynamic model that employs the free-air aerodynamics of the platform and missile and their mutual interactions. The aerodynamic model can be generated using wind tunnel measurements, results from CFD simulations or more usually using a combination of data from both sources. Aerodynamic interference between the platform and missile are produced using a grid survey in which data are obtained for a combination of Mach numbers, missile attitudes (total incidence and roll), control deflections and positions relative to the aircraft.

In determining the aerodynamics using computational fluid dynamics, a quasi-unsteady approach is adopted. Steady computations based upon solution of the Reynolds-averaged Navier-Stokes equations are performed for each location relative to the aircraft platform. The need to consider the relative position of the missile to the platform requires that the mesh contain both bodies and developing an appropriate meshing workflow is key to managing the time and cost required to establish the aerodynamic database. Formerly a remeshing approach was used, in which a single mesh was generated for each combination of missile attitude and position relative to the platform aircraft. More recently, the overset meshing capability available within the flow solver that we use has been exploited to simplify the meshing problem. In this approach, a mesh containing the platform aircraft is generated independently of the mesh around the missile. The two meshes are then overlaid and an appropriate interpolation region established in the overlap between the background (platform) and foreground (missile) meshes. This process is shown schematically in Figure 4-1. Calculations are then performed on the two meshes and information is exchanged in the overlap region. This process has proven to be both simple to manage and robust.

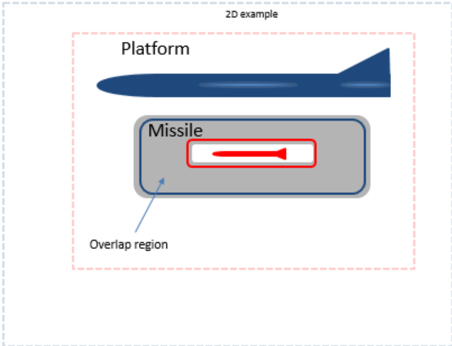


Fig. 4-1 Schematic of the Overset Meshing Strategy.

The resulting computed data form the basis of the aerodynamic model used by the six-degrees of freedom trajectory model. An example trajectory envelope is shown in Figure 4-2 for a generic missile with geometric and mass properties representative of those of a current generation air-to-air missile jettisoned from the open-access fifth-generation fighter aircraft described in [9]. The nominal trajectory (red line) is indicated, together with a large number of related trajectories that show the sensitivity of the launch to input data (for example variability in the mass properties of the missile or the initial accelerations imparted by the ejector unit) and model form uncertainties.

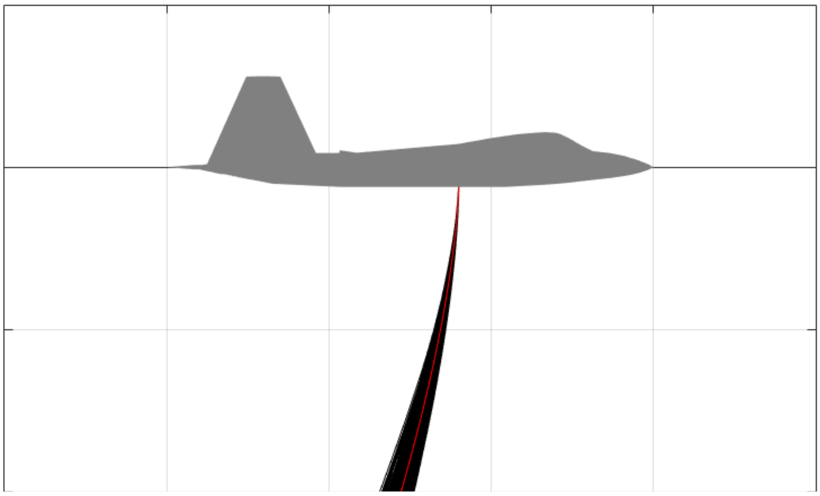


Fig. 4-2 Jettison of a generic missile from a representative platform.

More recently, there has been interest amongst the research and development community in performing computations that couple the unsteady aerodynamics of the problem with a rigid body trajectory simulation. Loupy and Barakos [10] used the Glasgow University solver by HMB3 and a Scale-Adaptive Simulation to simulate a representative missile configuration being launched from a shallow rectangular cavity at a Mach number of 0.85 and the Reynolds number based on the cavity length of 6.5 million. Loupy and Barakos' work shows that in principal such simulations are within the realm of possible, indeed in later work Loupy has performed simulations which include the effect of aeroelasticity [11]. The results also illustrate the potential costs of employing such simulations in the assessment of safe release. Even in the case where the inertia of the missile dominates the dynamics one must still perform 10s of simulations to achieve statistically converged trajectories.

The results shown in Figure 4-2 illustrate the importance of understanding the sensitivity of the trajectory to uncertainty in the process used to establish a safe release. The nominal trajectory is not enough. This observation reveals a key aspect of the industrial use of simulation in decision-making processes – the need to model and manage risk.

As the use of engineering simulation to make technical and business decisions broadens, it is becoming increasingly important to employ an effective engineering simulation risk model. This model accounts not only for the accuracy of the modelling used, but also for the sensitivity of results to model input and model form uncertainties together with the criticality and risk associated with the decision being made. This is fundamental to using modelling and simulation to support the assessment of safe release of a weapon from a platform where the criticality of a wrong decision may have catastrophic consequences for the platform and its pilot. Individual heroic calculations that couple multiple physical models to perform single highly accurate simulations are unlikely to be helpful in this situation. They are too expensive and too time consuming to be useful for the many hundreds of evaluations that are required to properly characterize and understand the sensitivity of the model outcome to model input and model form uncertainties.

From an industrial end user perspective, significant progress is required to improve the community's understanding of error and uncertainty in CFD simulations. The tools at hand lack the generality and robustness that industry requires to be able to make effective use of simulation results. This leads to the need to take large margins that compromise potential performance and a requirement for extensive physical trials. Heroic simulations that couple multiple physical models, in expensive time accurate calculations will be important, but their impact is likely to be limited to helping to understand the sensitivity of trajectories to individual physical effects (for example flow unsteadiness or structural flexibility) and quantifying associated uncertainties.

5.0 CONCLUSIONS

In this paper, we have presented three examples related to the carriage and release of an air-to-air missile from a weapons bay that illustrate some of the challenges related to the use of computational fluid dynamics in an industrial context. The examples illustrate the inherent tensions between industry's need to make more effective use of existing models, largely based on the solution of the Reynolds-averaged Navier-Stokes equations, and the perceived focus of the research and development community on increasingly accurate (at the expense of time and cost) simulations, whether of the basic fluid mechanics or by coupling multiple physical models.

With respect to the modelling of the fundamental fluid physics we recognise that in some situations, such as the example of unsteady cavity flows described in the paper, scale-resolving simulations employing detached or large eddy simulations are required. Nevertheless, in most of our applications RANS models should suffice. Turbulence modelling improvements are required for the vortical flows that are of direct interest to missile aerodynamicists. We advocate a strategy that combines a more systematic approach to the modelling of turbulence using the Reynolds Stress Transport equations as a basis. However, this should be combined with a more openly empirical approach that recognises the futility of seeking a single model that can be applied in all circumstances. This strategy should be based upon a new collaboration between industry and researchers that would see the community work together to design and conduct appropriate calibration experiments.

More generally, there is a need for the community to bring renewed focus to the development of the understanding and infrastructure to characterise error and uncertainty in simulation results. There is a related need to develop robust, simple processes that allow such data to be used to develop effective engineering risk management models. It is our belief that in most cases having a more robust understanding of the error and uncertainty associated with a large number of less accurate results is likely to be more beneficial than a handful of calculations performed with highly accurate models.

6.0 ACKNOWLEDGEMENTS

The authors would like to thank Michael Anderson, Erdem Dikbaş, Jim DeSpirito, Christian Schnepf, and Magnus Tormalm for use of the data contained in Figures 2-3 and 2-4 which appeared originally in Reference [3].

7.0 REFERENCES

[1] Spalart, P.R., Allmaras, S.R.,. "A One-Equation Turbulence Model For Aerodynamic Flows," AIAA Paper 92-0439, January 1992.

[2] Menter, F.R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal Vol. 32 No. 8, pp. 1598-1605, August 1994.

[3] Shaw S., Anderson, M., Barakos, G., Boychev, K., Dikbaş, E., DeSpirito, J., Loupy, G., Schnepf, C. and Tormalm, M., "The Influence of Modelling in Predictions of Vortex Interactions About a Generic Missile Airframe: RANS", paper to be presented at AIAA SciTech 2022, January 2022.

[4] Taylor, N. et al., "The Prediction of Vortex Interactions on a Generic Missile Configuration Using CFD: Current Status of Activity in NATO AVT-316," NATO STO-MP-AVT-307, Paper 24, October 2019.

[5] Bradshaw, P. "Effects of streamline curvature on turbulent flow," AGARDograph 169, 1973.

[6] Tormalm, M., et al., "The Influence of Scale Resolving Simulations in Predictions of Vortex Interaction about a Generic Missile Airframe", paper to be presented at AIAA SciTech 2022, January 2022.

[7] Boychev, K., Barakos, G. and Steijl, R., "Simulations of flows around complex and simplified supersonic store geometries at high incidence angles using statistical and scale resolving turbulence models", paper to be presented at AIAA SciTech 2022, January 2022.

[8] Spalart, P., "Philosophies and fallacies in turbulence modeling", Progress in Aerospace Sciences , Vol 74, pp. 1-15, 2015.

[9] Bykerk, T., Giannelis, N.F. and Vio, G., "Static Aerodynamic Analysis of a Generic Fifth Generation Fighter Aircraft", AIAA Science and Technology Forum, AIAA Paper 22-1951, 2022.

[10] Loupy, G. and Barakos, G., "Modelling of Transonic Shallow Cavity Flows, and Store Release Simulations from Weapon Bays" 35th AIAA Applied Aerodynamics Forum, AIAA 17-3252, 2017.

[11] Loupy, G. "High fidelity, multi-disciplinary analysisof flow in realistic weapon bays", PhD Thesis, Glasgow University, 2018.