1) HIGH FIDELITY MULTI-DISCIPLINARY SIMULATION TOOLS FOR ENABLING DIGITAL ENGINEERING OF MILITARY AIRCRAFT

Digital Engineering is a concept recently gaining ground in government and industry acquisition circles. The definition of Digital Engineering is not settled yet but may include some or all of the concepts Digital Thread, Digital Twin, Model Based Systems Engineering, Model Based Source Selection, as well as others. All of these concepts are intended to address the issues of rising costs and unreasonably long development cycles for next generation air vehicle acquisition. They will require system level multi-disciplinary simulation capability and the ability to populate aerodynamic, structural loads, and control effectiveness databases for the full envelope of the vehicle in a timely manner.

This presentation describes system level analysis capabilities and reduced-order model building from high-resolution simulations that take several days on 10^4 to 10^5 cores of a supercomputer to fulfill the needed advancements in air vehicle acquisition.

2.1) EVALUATION OF A MODULAR HYBRID AIRSHIP DESIGN USING CFD TECHNIQUES

New interest in airship technology has evolved in the last years particularly in design which combines buoyant lift with aerodynamic lift. Also, Computational Fluid Dynamics (CFD) has been increasingly used to evaluate new aircraft designs. Advancements in computational power allow aerodynamic data to be obtained at a lower cost than when using wind-tunnel or flight tests. Thenceforth, CFD emerges as the obvious option to validate and optimise a new modular hybrid airship design which prototype is under development. The estimated lift drag and moment coefficients of this prototype are the objectives of the study. The methodology, the mesh generation necessary, the CFD simulations of different configurations of the airship, including no aft wings and aft wings at various incidence settings at different angles of attack of the vehicle are presented in this work.

2.2) CFD VALIDATION - WHAT IS IT AND HOW DOES IT AFFECT US?

The term “validation” can have a variety or meanings, depending on the context in which it is being used. When asked for a definition of CFD Validation, the first point of reference for the experienced CFD practitioner is often AIAA-G-077-1998, viz: “The process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.” Many still regard this statement as definitive. Some are unaware of the challenges that have been made to its interpretation or of the recent developments that have been made in order to address the attendant concerns.

This paper will provide a brief overview of the principal developments and explain how a wider appreciation of the issues raised - and a wider contribution to the ongoing debate – may play important roles in stimulating both fundamental CFD development and the ways in which it is exploited by its various “end-users”.

3.1) ZEPHYR AERODYNAMIC CHALLENGES

Zephyr is a High-Altitude Pseudo Satellite, able to fly in the stratosphere for months running exclusively on solar power. Aerodynamic efficiency is critical to Zephyr’s mission as the power storage available is constrained by battery technologies and low aerodynamic drag is crucial to reduce battery mass and increase payload mass. Furthermore, this must be achieved with a very light airframe. After an introduction to Zephyr; the presentation will cover the main challenges encountered evaluating numerical approaches to correctly predict transition mechanisms at low Reynolds numbers and turbulence intensity, the experimental efforts undertaken to design more efficient propeller turbulators for this regime as well as the ongoing collaboration with an academic partner to develop and assess a framework for nonlinear aeroelastic analysis of very flexible aircraft.

3.2) RECENT DEVELOPMENTS TO THE EST PROCESS: SYNCHRONISING OML PARAMETERISATION WITH SIMULATION INTENT

This talk accompanies a paper that will cover the key developments that have been made to EnGAM, a missile concept design environment used within MBDA UK, since last published update (2015). Particular attention will be paid to: the improvements made to geometric parameterisation options to permit increased Outer Mould Line (OML) fidelity; and the measures to ensure those inputs required for CFD simulation are kept in synchronisation during systematic investigations of trades in or optimisation of aerodynamic performance.
4.1) THE DESIGN AND TESTING OF A LARGE VARIABLE-BUOYANCY PROPELLED UAV

‘Phoenix’ is an ultra-long endurance UAV which exploits a variable-buoyancy propulsion system never before used for the propulsion of a large-scale aircraft. It spends half its time as a heavier-than-air aeroplane, and the other half as a lighter-than-air balloon, the repeated transition between which provides forward motion. The helium-filled fuselage provides buoyancy sufficient to make the vehicle lighter than air and ascend like a balloon. Within the fuselage is a separate bag into which air drawn from outside is compressed and thus adds weight sufficient to overcome the buoyancy and descend like a glider. The release of the compressed air returns it to a lighter-than-air configuration and the process is repeated. The power is provided by a rechargeable battery which, in turn, is supplied by an array of lightweight, flexible solar cells distributed on the upper surfaces of the wings and horizontal tail.

4.2) MEASUREMENT OF SHAPE AND PRESSURE OF AN AERODYNAMIC MODEL

This presentation details measurement methods and results combining optical metrology for shape determination and surface pressure measurements utilising pressure-sensitive paint. Future extensions of this technique include measurement of surface pressure and shape of moving structures for fluid structure interaction.

5.1) A RAPID 3D AERODYNAMIC PREDICTION METHOD FOR BLENDED-WING-BODY CONCEPTS

The focus of this paper is the demonstration of a rapid, lower fidelity method for the conceptual design and analysis of unconventional transonic cruise transport aircraft. Since the application of interest is the design analysis of transonic cruise wing flows, the Viscous Full Potential method, which couples the solution of the Full Potential equations for compressible rotational inviscid flow with the integral boundary layer equations, is wholly adequate for a design method. The method has been applied to the prediction of some challenging transonic test cases. The superior aerodynamics of the Blended-Wing Body (BWB) has the potential to reduce the fuel consumption also by ingesting the boundary layer (BLI) and reduce noise since the exhaust noise is not reflected by the wing. Some preliminary optimisation studies of the BWB at transonic regimes using rapid aero method are presented.

5.2) EXTERNAL AIRCRAFT LOAD VARIABILITY DUE TO CORRELATED STRUCTURAL DESIGN UNCERTAINTIES

Aeroelastic calculations responsible for computing structural loading during design optimisation are functions of both the loading conditions and the aircraft structure itself; the latter being responsible for determining the deformation of the aerodynamic surfaces under load. This presents a dilemma for designers, as to perform such calculations, the structure requires definition. This results in structural design uncertainty as inability to designate the aircraft structure precisely, prior to performing aero-elastic analyses, implies the resulting forces on the structure are uncertain due to imprecision in the structural definition. In this work we propose a generic method for generating samples of design uncertainties exhibiting variable coupling and use this to show that different correlations between uncertain design variables can have important consequences on the resulting uncertain loads and downstream analyses, such as optimisation.

6) AERODYNAMIC TOOLS FOR RAPID PREDICTION AND DESIGN OPTIMISATION

Some years ago, methods such as full-potential codes, coupled with integral boundary layer methods, were extensively used for transonic wing design. Since then improvements in computer speed and memory capacity have led to a shift away from these low-order methods to high-order methods such as RANS, DES etc, in which more complex geometries and separated flow can be represented. Recently there has been considerable interest in reducing design-cycle time. Viscous-coupled full-potential codes are quick to set up and are fast enough to allow complete drag polars to be rapidly calculated with very modest computational resources. They are, therefore, ideal for use in assessing the relative merits of competing designs and for initial optimisation of wing-sections and planform. The design will then be more mature at the stage where the use of costlier high-order methods is required. This paper shows that low order does not imply low accuracy for calculation within the separation boundary and that these methods can also be used to predict the separation boundary. Their use with unconventional wing geometries and in establishing manufacturing tolerances is also discussed.

7) AERODYNAMIC DESIGN OPTIMIZATION METHODS, TOOLS, AND APPLICATIONS

CFD-based aerodynamic design optimization has matured significantly in the last few years, thanks to the refinement of CFD solvers, mesh deformation, sensitivity computation, and optimization tools. In this talk we will review the state-of-the-art methods for each of these components, and give an overview of the open-source tools recently made available for aerodynamic shape optimization. A wide variety of applications will be presented, including the optimization of a supercritical airfoil starting from a circle, a web application that optimizes airfoils within a few seconds, full configuration aerodynamic and aerostructural optimization, and aeropropulsive optimization.

8.1) THE HIGH-LIFT COMMON RESEARCH MODEL - A UK PERSPECTIVE

The Common Research Model programme, which began in the early 2000s, is a global initiative to improve the understanding and aid development of aerodynamic prediction tools. A common geometry of an aircraft in the cruise configuration was provided as a means for comparison of both theoretical and experimental data, generated via various computational methods and a number of wind tunnels worldwide. The success of this programme has led Boeing and NASA to initiate a similar exercise for an aircraft in its high-lift configuration, known as the High-Lift
Common Research Model or CRM-HL. With the aid of a UK government grant, QinetiQ and Boeing will be manufacturing two UK-based CRM-HL models that will be available in 2021 for use by UK industry and academia.

This presentation will describe the current status of the CRM-HL initiative from the UK perspective and the aims for the UK-based models.

8.2) GEOMETRY MODELLING & DIGITAL THREADS

Increased exploitation of advances in computational techniques and technologies pervades every aspect of contemporary engineering practice - as evidenced by emergence of the terms Digital Thread, Digital Twin and even the Digital Economy. The desire to improve upon current geometry modelling practices is no exception. This paper will outline the various roles played by geometry modelling throughout industrial product development lifecycles. In so doing, it will identify the opportunities to be afforded by a more extensive exploitation of a suite of emerging “disruptive” geometry modelling techniques.

9.1) ELECTRIC MOTOR POWERED AERO-ENGINE SIMULATOR EMPAS

Whilst testing commercial jet engine aircraft in wind tunnels, simulation of the effects of the power plants is either not achieved, or simulated using fans which are driven by air or hydraulically powered turbines. With the advent of high power density motors, this project created an electric motor powered fan, and its use is described in this presentation.

9.2) HIGH ORDER METHODS FOR HIGH FIDELITY INDUSTRIAL CFD

Not available at time of print.

10.1) CIVIL AIRCRAFT WIND TUNNEL FEATURE RICH TESTING AT THE EDGE OF THE ENVELOPE

Not available at time of print.

10.2) DEVELOPMENTS TOWARDS FLUID-STRUCTURE INTERACTION SIMULATIONS IN BAE SYSTEMS CORPORATE CFD SUITE

This presentation outlines the new capabilities for fluid structure interaction and aeroelasticity that have been developed in BAE Systems. A brief summary of the on-going developments will be made followed by a demonstration of the capabilities on validation test cases.

11) AERODYNAMICS AND AEROELASTICITY METHODOLOGIES FOR FUTURE CONCEPTS IN VERTICAL LIFT

Vertical lift concepts for future applications across the military and civilian sectors transcend traditional helicopter designs. Military specifications call for new configurations with increased forward flight speeds, increased manoeuvrability, and significant reductions in drag, as well as operation in challenging environments. Civilian designs call for many of the same improvements, with concepts ranging from small delivery drones to urban air mobility. To meet the plethora of new design goals, engineers must be able to apply high-fidelity methodologies earlier in the design cycle.

This presentation will discuss aerodynamic methodologies to meet these advanced design goals, including adjoint optimizations, dual-solver hybridization, and first-principles-based reduced-order modelling. Development of aerodynamic methodologies to more accurately predict these integrated technologies are needed for design, but also to understand the underlying physics. The ability of these aerodynamic tools to improve interdisciplinary predictions that are inherent to vertical lift, such as aeroelasticity and aeroacoustics will be discussed.

12) RECENT PROGRESS IN ADVANCED WIND TUNNEL BOUNDARY SIMULATION

Recently, a technical workshop was sponsored by the Applied Vehicle Technology (AVT) panel within the Science and Technology Office (STO) of NATO: Advanced Wind Tunnel Boundary Simulation. The purpose of the workshop was to identify state-of-the-art integration of CFD with ground testing for wind tunnel wall and support interference corrections and validation campaigns. The use of CFD for wind tunnel wall interference in wind tunnel testing is critical with continuing interest in increased model size, higher subsonic and near-sonic Mach numbers, and near-full-scale component testing. Support interference and test section aerodynamic characterization are also significant features of wind tunnel tests requiring characterization for proper interpretation of wind tunnel results. Although modeling fidelity may differ, validation campaigns also require resolution of wind tunnel boundaries to appropriately represent measured flow physics.

This presentation summarizes workshop results and plans for a second AVT workshop scheduled for May 2021.

13.1) THE NATIONAL WIND TUNNEL FACILITY

The purpose of NWTF is explained, and some examples of successful joint projects arising from it are presented. The future role and need for NWTF is discussed.

13.2) BOUNDARY LAYER INGESTING ELECTRIC DUCTED FAN INSTALLATION AERODYNAMICS AND OPTIMIZATION

Fully integrated electric ducted fan (EDF) propulsion aerodynamics for novel air-vehicle design concepts are being investigated in this research. Embedded propulsion systems offer overall system benefits as propulsion mounting components can be omitted, reducing both wetted area and system weight while allowing for novel and more compact airframe designs. The overall aim of this research is to quantify the interdependence of body and propulsion system for the definition of design guidelines for future applications of air-vehicles with closely integrated propulsion systems employing boundary layer ingestion (BLI). Detailed characterization of the boundary layer and flow field is therefore undertaken through combination of experimental and computational techniques. Experimentally validated...
computational fluid dynamics (CFD) simulations will allow for derivation of suitable performance metrics used in aerodynamic design optimization. Experimental studies of different inflow designs and corresponding CFD simulations will be presented alongside alternative performance metrics for use in design optimization.

14.1) APPLICATION OF TURBULENT ADJOINT TO AERODYNAMIC DESIGN PROBLEMS

Ability to accurately compute the gradients of design objectives with respect to design parameters is a key requirement for gradient-based optimization algorithms. Adjoint-based methodology is often employed for this purpose because it allows all the gradients (for a single objective function) to be obtained at a cost comparable to that of a CFD analysis. This paper will present the results of a recent work on developing a fully turbulent adjoint capability in the Simcenter STAR-CCM+ CFD solver. The goal is to allow for accurate computation of sensitivities in industrial-scale flow problems, where turbulence plays a major role. The current implementation is based on the Spalart-Allmaras turbulence model, but the developed framework allows for an easy extension to more complex models. The validation study will demonstrate the impact of the turbulent adjoint on the accuracy of the computed gradients for the ONERA M6 wing, both in subsonic and transonic conditions. The impact of the addition of the turbulence adjoint on the accuracy and the computational cost of the aerodynamic shape optimization of the entire wing will be also evaluated against a frozen turbulence approach.

14.2) TBC

15) CFD PREDICTION FOR HIGH LIFT AERODYNAMICS: RECENT PROGRESS AND EMERGING OPPORTUNITIES

Numerical prediction of commercial airplane low-speed characteristics at take-off and landing is a difficult problem and is pacing the development of advanced CFD methods and tools. This talk will highlight the challenges involved in high-lift prediction, review recent progress in CFD applied to High Lift, and discuss emerging opportunities, particularly in new international CFD validation activities.