AIAA SciTech Forum
11–15 & 19–21 January 2021, VIRTUAL EVENT
AIAA Scitech 2021 Forum

10.2514/6.2021-0955

A Grand Challenge for the Advancement of Numerical Prediction of High Lift Aerodynamics

Jeffrey P. Slotnick*
The Boeing Company, Seattle, Washington, 98124

Dimitri J. Mavriplis†
Department of Mechanical Engineering, University of Wyoming, Laramie, WY 82071

This paper presents an ambitious Computational Fluid Dynamics (CFD) grand challenge problem, including a proposed set of increasingly complex and connected challenge problems, to advance the technological state-of-the-art in the numerical prediction of commercial airplane low-speed, high-lift aerodynamic characteristics and performance. It describes the critical need for a vastly improved computational capability for high-lift airplane design, system development, and product certification, highlights current technology gaps and shortcomings, and details key research and development focus areas where significant progress is required. A key goal of this effort is to energize the aerospace CFD/Aerodynamics communities by coordinating and collaborating across multiple levels of government, industry, academia, and other technology providers to accelerate the use of efficient and robust computational tools to ultimately create products with increased aerodynamic performance that are environmentally cleaner, more fuel efficient, and ensure safe flight while reducing non-recurring product development cost and risk.

I. Introduction

Physics-based numerical simulation continues to expand into all phases of the aerospace product lifecycle, including conceptual design and trade studies, detailed configuration development, product certification, and in-service support. Model-based engineering, through the increasing use of numerical simulation within the "digital thread", promises to reduce non-recurring product development costs and risk, create products that are environmentally cleaner, more fuel efficient, safer, etc., and enable designs closer to the optimum.

However, numerical simulation strategies in general, and computational fluid dynamics (CFD) in particular, currently are not sufficiently accurate and/or reliable to be used with confidence throughout the flight envelope, severely limiting the potential impacts these technologies could have on both the product development and certification phases. This shortcoming is affirmed in the CFD Vision Report [1], published in 2014 which states that, "…in spite of considerable successes, reliable use of CFD has remained confined to a small but important region of the operating design space due to the inability of current methods to reliably predict turbulent-separated flows." While this quote references the relative success that CFD has enjoyed in predicting aircraft cruise configurations with minimal flow separation, the report discusses advances that will be required in various technological areas in order to extend the applicability of CFD to the full flight envelope with confidence and fully realize the potential of numerical simulation technologies on the aircraft product life cycle.

The regions at the edges of the operational flight envelope are well known to present particular challenges for numerical simulation methods [2-4]. The takeoff and landing phases of flight are characterized by increased wing camber through modified aerodynamic surfaces, and often exhibit small amounts of flow separation in nominal operation. However, much larger amounts of flow separation

---

[*] Boeing Technical Fellow, AIAA Associate Fellow; Co-chair CFD2030 Integration Committee; email: jeffrey.p.slotnick@boeing.com

[†] Professor, AIAA Associate Fellow; Chair CFD2030 Integration Committee; email: mavriplis@uwyo.edu

characterize airplane performance at or beyond maximum lift, at angles of attack encountered during stall maneuvers. Accurate numerical simulation of aircraft configurations that exhibit large regions of turbulent separated flow, particularly in landing and/or takeoff ("high-lift") mode operating at low-speed, off-design flight conditions, is not readily available. Thus, numerical prediction of low-speed aerodynamics of transport aircraft in landing and/or take-off configuration remains a key pacing item in the ability of aircraft manufacturers to rapidly develop products that operate efficiently near the edges of the flight envelope where important aerodynamic and structural constraints, such as buffet and flutter margins, must be properly considered earlier in the design process. This is becoming increasingly important particularly for novel configurations that depart significantly from standard "tube with wings" designs.

Furthermore, approximately two-thirds of all certification flight testing to address requirements in Subpart B (Flight) of 14 CFR Part 25 of the Federal Aviation Regulations [FARs] occurs at low-speed, off-design conditions [5]. The use of physics-based numerical simulation to support aircraft certification is part of a focus area known as Certification by Analysis (CbA), which is currently a domain of growing interest [6]. Although CbA is being used as a key driver in the development and validation of numerical simulation tools to predict real airplane performance characteristics, a critical application of the computational toolset is to reduce the amount of developmental pre-certification flight testing required to refine airplane configurations to meet certification requirements, allowing accelerated product time-to-market. Also, addressing shortcomings in numerical prediction methods for airplane certification, particularly with CFD, will allow the use of these methods to optimize the aerodynamic configuration much earlier in the design phase, where the bulk of the total product development cost is often committed. Reliance on the analysis for making critical configuration design decisions places added importance on the overall confidence of the simulation results, which translates into a greater emphasis on the development and incorporation of rigorous uncertainty quantification techniques into the simulation process.

II. Motivation

The development and demonstration of accurate and reliable numerical simulation techniques for high-lift aircraft configurations for both product development and certification offer the potential to radically change costs, risk and outcomes throughout the entire product lifecycle. However, significant investments in CFD technology development, coupled with the deployment of advanced emerging High Performance Computing (HPC) resources, will be required to meet these goals. In order to express the value that revolutionary simulation capabilities may enable, one must adopt a forward looking vision which is not constrained by the limitations of current tools and practices. In this paper, the formulation of a Grand Challenge (GC) is used for these purposes.

Historically, GC problems have often been used to illustrate the benefits that potential advances in computational science and engineering can bring to their respective application discipline. The CFD Vision 2030 report [1] included four GCs designed to cover the range of NASA aerospace applications and to demonstrate the potential new frontiers that improved simulation capabilities could enable in aerospace engineering. In general, the GCs must be ambitious and even may not be attainable in the stated time frame. However, a well formulated grand challenge problem can serve to identify the added value enabled by more capable simulation tools, which in turn can be used to justify the investments required to develop these new capabilities. At the same time, the GC can be used to identify the barriers to progress in the current state-of-the-practice, and to categorize these into technical barriers and logistical barriers.

Technical barriers are considered key gaps and shortcomings in CFD technology that remain unsolved, and that do not have readily available solutions. For these barriers to be overcome, focused and sustained effort in the science and development of CFD methods must occur. Examples of technical barriers include efficient numerical algorithms (e.g. higher-order schemes, etc.) and accurate modeling of flow physics (e.g. turbulence modeling, etc.), among others.

Logistical barriers refer to gaps or shortcomings in the development and/or application of CFD technology where solution(s) are readily available, or could be methodically generated, given the proper amount of funding and technical expertise applied to the particular problem. For instance, improved software engineering approaches for better design, use, and reuse of CFD code is generally known, but only focused effort to restructure solvers using the latest techniques, employing key experts, is required. Similarly, porting current CFD solvers to emerging heterogeneous HPC hardware such as GPUs is relatively well understood, but requires a large investment in software development that must be prioritized.

Finally, GC problems can serve to focus a diverse set of stakeholders on a common objective. For CFD technology to advance sufficiently to successfully address the grand challenge problem, key resources, such as technical experts and funding, must be identified and focused over likely extended periods of time. As mentioned in the CFD Vision 2030 report, efforts must be undertaken to "…develop, foster, and leverage improved collaborations with key research partners and industrial stakeholders across disciplines within the broader scientific and engineering communities." Clear definition of the barriers that prevent the use of CFD technology to address the grand challenge problem will help concentrate technical and scientific organizations to collaborate and coordinate efforts to advance progress towards a shared goal. For the solution of the high-lift grand challenge problem, examples of areas that already maintain various levels of focus include CFD solver technology, meshing technology, disciplinary coupling, uncertainty quantification, flow visualization, and data/knowledge extraction, often through community discussion groups, conference special sessions, and CFD prediction workshops. However, providing additional focus on these, and in other related areas, involving key stakeholders within the relevant science and engineering communities, will be required to advance CFD capabilities for high-lift aerodynamics prediction.

III. Grand Challenge

To ultimately enable the use of high-fidelity CFD methods and tools specifically for future commercial airplane product development, particularly at off-design conditions, the proposed Grand Challenge (GC) is a critical low-speed (high-lift) maneuver typically performed for airplane certification. Several increasingly complex technical sub-challenges (SCs) are also defined, which provide meaningful and realistic targets to demonstrate tangible progress towards the GC. The GC, and spectrum of proposed SCs leading up to it, are illustrated in Figure 1 and are described in greater detail in the sections below.

For the GC, the airplane maneuver under consideration is a variant of a wind-up turn (WUT) typically performed to satisfy regulatory requirements[1] [7]. For this GC, we focus on the "excess-thrust" windup turn where, in a coordinated turn, both the aircraft angle of attack and normal load factor are increased while maintaining a constant altitude and airspeed [8]. Longitudinal stick is used to control the airplane angle of attack or normal acceleration by changing the elevator deflection, lateral stick is used to control roll rate by changing the wing aileron and/or spoiler deflection, and the throttle is used to control airspeed by changing engine thrust. Rudder pedal deflection is not used, although the rudder may deflect from flight augmentation systems, such as yaw dampers. In this maneuver, the airplane is in a flap-extended landing (or take-off) configuration with landing gear retracted, and the banked turn is initiated at a moderate altitude (generally 15,000 – 20,000 feet AGL) and subsonic Mach number (M~0.35-0.40). As the turn progresses, the stick is continually pulled aft, increasing angle-of-attack and normal load factor, and effectively tightening the turning radius. Heavily dependent on pilot technique and atmospheric conditions, target parameters for a WUT include holding airspeed constant (to within +/- 5 knots), and holding altitude constant (to within +/- 500 feet). A key success metric is to demonstrate a smooth increase in stick force with increasing load factor, measured as stick force per 'g'. Maximum load factor typically does not exceed 2g.

This maneuver is particularly challenging for current generation computational tools commonly used within industry because it requires seamless integration of efficient, accurate, and robust flow modeling methods with key multi-disciplinary simulation capabilities. In particular, technical requirements include the ability to model:

- Unsteady, time-accurate flow physics at off-design flight conditions, which are dominated by various types of turbulent flow separation (smooth-body, shock/boundary layer, juncture, wake,

---

[1] 14 CFR 25.143(g). When maneuvering at a constant airspeed or Mach number (up to V_{FC}/M_{FC}), the stick forces and the gradient of the stick force versus maneuvering load factor must lie within satisfactory limits. The stick forces must not be so great as to make excessive demands on the pilot's strength when maneuvering the airplane, and must not be so low that the airplane can easily be overstressed inadvertently. Changes of gradient that occur with changes of load factor must not cause undue difficulty in maintaining control of the airplane, and local gradients must not be so low as to result in a danger of over-controlling.

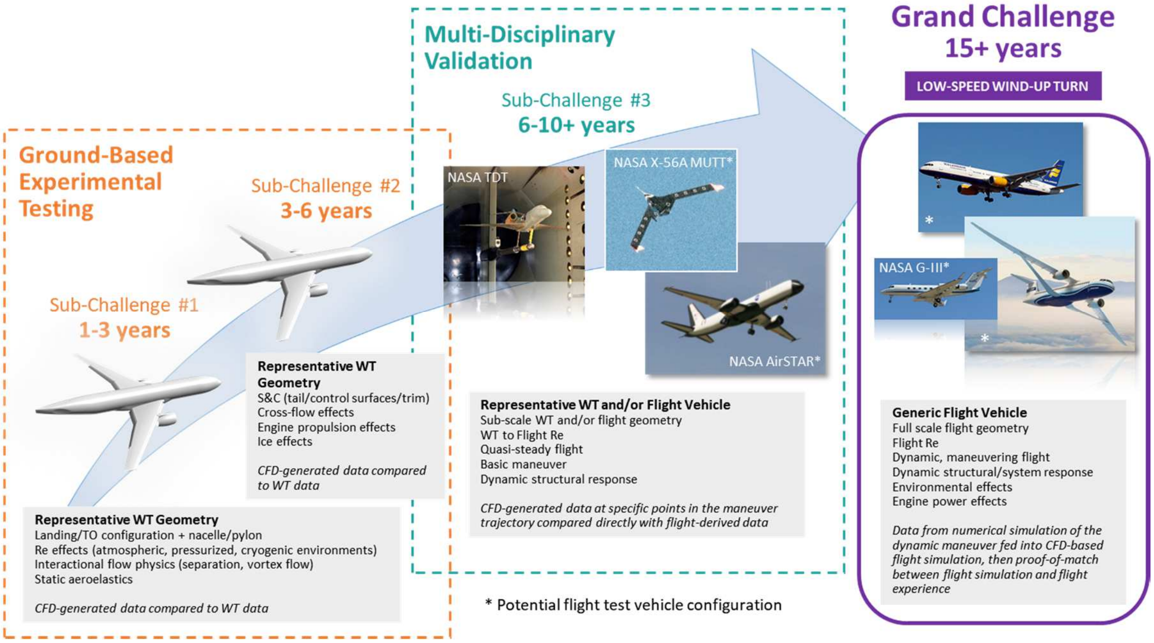

Figure 1. Proposed Grand Challenge and associated Sub-Challenge problems

etc.), laminar-to-turbulent transition occurring over a range of locally-varying Mach numbers (from subsonic to transonic) at high (flight) Reynolds numbers, and multiple, interacting vortices from various aircraft components.

- Dynamic, aero-elastically-deforming geometry that is linked to the airplane structural response
- Coordinated turns utilizing control surface deflections (e.g. elevator/aileron) through a control feedback loop
- Proper system integration to capture stick force and pilot response (hinge moments, flight computer, stick force feedback, etc.)
- Powered engine effects that adequately capture relevant flow-field/engine interactions (e.g. cross-flow effects) and provide realistic thrust at all phases of the airplane maneuver
- Atmospheric/environmental effects (gusts, etc.)
- Quantification of model uncertainties

It is envisioned that aerodynamic data for this GC will be obtained from flight testing of a full-scale vehicle associated with the WUT maneuver, as discussed above. Independent of which suitable airplane configuration is chosen, a detailed CAD definition of the configuration geometry is required, along with key aerodynamic and structural response data, to support physics-based numerical modeling. The ultimate measure of success in performing the WUT maneuver is pilot assessment of the smooth increase in stick force with increasing load factor (or stick force per “g’). The WUT maneuver is usually performed in flight test, but can also be performed in a piloted flight simulator. Since the successful completion of the WUT is heavily influenced by pilot evaluation and the variability of the airframe response to environmental disturbances, among others, a comparison between the aerodynamic parameters (e.g. forces/moments, etc.) extracted from the computational simulation for the time-dependent, dynamic WUT maneuver directly with flight-derived data is not sufficient to establish CFD as a suitable replacement for flight testing. Rather, a proof-of-match between the *flight simulator* results utilizing the aerodynamic parameters derived from the computational analysis and those utilizing actual flight test data is the most effective approach to build confidence and show credibility with regulatory agencies. In practice, the piloted flight simulator has specific hardware to control hydraulics and electrical devices to mimic the appropriate airplane control device forces (e.g. stick and pedal), which are tuned to match the relevant flight test data. It is the combination of controller hardware in the flight simulator, coupled with the aerodynamic response characteristics, that is key to properly representing the integrated effect of the external aerodynamics, dynamic structural response, and environmental effects on final pilot assessment.

To supply the required aerodynamic parameters for the flight simulator assessment, accurate and efficient predictive computational technologies, developed and matured over time and properly integrated into a robust multi-disciplinary analysis capability, are required to successfully address the grand challenge. To this end, a series of sub-challenges (SCs), also depicted in Fig.1, are proposed to provide an effective pathway to demonstrate incremental technical progress towards the successful simulation of the GC. The SCs, as detailed below, target specific elements of the required numerical predictive capabilities, and rely on focused ground-based and flight-test campaigns to demonstrate steady progress in analysis capability towards the overall final GC.

SC-1 – High Lift Flow Physics.

The first sub-challenge problem is focused on assessing and improving the ability of CFD methods and tools to accurately and efficiently predict the effects of Reynolds number and the integrated effects of various types of flow separation (e.g. smooth-body, juncture, etc.) and vortex flows (e.g. from chine, slat/flap edges, etc.) on external aerodynamic performance. Results from CFD simulations would be compared to experimental data collected from testing of a nominal landing and/or takeoff configuration in multiple wind tunnel test facilities using both semi- and full-span wind-tunnel models. Wind-tunnel testing at atmospheric, pressurized, and cryogenic facilities should be sought to obtain a wealth of data over a large range of Reynolds number. Experimental data for assessing model aero-elastic deformation and the effects of laminar-to-turbulent flow transition will also form an important part of the validation data set. Innovative off-body measurement techniques, such as Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV), may also be utilized to characterize off-body flow phenomena to complement standard force/moment and surface pressure data.

SC 2 – Airplane Discipline Assessments.

Building on SC-1, SC-2 serves to focus attention specifically on CFD prediction in the following areas: engine propulsion effects, stability and control characteristics, icing, and the effect of cross-flow on aerodynamic performance. Accurate predictive capability in all of these areas will be required to prepare for the GC simulation. For engine propulsion effects, it is envisioned that a propulsion simulator (e.g., Turbine Engine Simulator or possibly Electric Motor Powered Aero-Engine Simulator (EMPAS) [9, 10]) will be installed on a wind tunnel configuration and tested in a suitable wind tunnel facility. Data from that configuration may be used to assess propulsion-airframe interactions, asymmetric thrust effects, and perhaps nacelle inlet distortion, provided the appropriate test data is collected. Stability and control characteristics studied would include tail trim and the effects of control surface effectiveness, among others. The effects of leading-edge slat ice shapes on aerodynamic performance should also be assessed, since characterization of icing effects is becoming a critical aspect of airplane high-lift system design and certification, particularly at aerodynamic stall.

SC 3 – Disciplinary Coupling.

SC-3 is specifically envisioned to assess the integrated improvements in the CFD predictive capabilities considered in SC-1 and SC-2 within a multi-disciplinary environment, either through ground-based testing, or using a representative and suitable sub-scale flight-test configuration, or both. Ground-based testing and sub-scale flight testing offer complementary advantages, with the former enabling tight control on test conditions and environmental factors, while the latter allows for full coupling of most or all of the required disciplines. In both cases, key requirements include the public availability of an accurate CAD geometry definition and a suitable structural Finite Element Model (FEM). For ground-based testing, aeroelastic analysis validation using dynamically scaled models in specialized facilities such as the NASA Transonic Dynamics Tunnel (TDT) may be targeted. For sub-scale flight testing, the NASA experimental Airborne

Subscale Transport Aircraft Research (AirSTAR) test-bed could be considered for a sub-challenge problem. AirSTAR is a remotely-piloted 5.5% dynamically scaled, generic transport aircraft configuration which was designed to study flight control law development pertaining to dynamics modeling and control beyond the normal flight envelope [11]. Another possibility would involve the NASA X-56A Multi-Utility Technology Testbed (MUTT), which could be utilized to specifically study the coupling of external aerodynamics with large structural deformations [12]. For this sub-challenge, numerical simulations may be performed in a steady-state, quasi-steady, or fully time-dependent manner, depending on the nature of the experimental data that is made available for validation and the maturity of the multidisciplinary simulation tools.

IV. Research and Development Focus Areas

The accurate and credible numerical simulation of a high-lift wind-up turn at flight Reynolds number represents a daunting task, given the state of current-day numerical simulation technologies. One of the defining characteristics of the GC WUT and associated SC problems is that they occur at the edges of the flight envelope, where current CFD technology is particularly lacking. Additionally, the SC and GC problems are highly multidisciplinary, requiring a level of multidisciplinary analysis that is well beyond the capabilities of current methodologies. At the same time, in order to provide a credible alternative to flight testing, highly detailed geometric and structural models will be necessary, coupled with robust error estimation and uncertainty quantification strategies, which will require significant advances in the current state-of-the-art.

Following the approach taken in the CFD2030 Vision report, these critical technological gaps, that are exposed through the formulation of the GC problem, can be categorized into five fundamental technology areas, namely, Physical Modeling, Geometry/Grid, Algorithms, Multidisciplinary Coupling and Uncertainty Quantification. A sixth category on HPC is also included to emphasize the important role that next generation computing will play in enabling advanced CFD capabilities required to address the GC. This categorization is used as a foundation to construct a multi-tier roadmap driven by the technical and logistical barriers identified through the consideration of the high-lift windup turn Grand Challenge problem (see Figure 2). This roadmap is not as comprehensive as that developed in the CFD2030 report, but is more targeted for the specific GC problem considered herein. This also results in additional emphasis on certain aspects that were not as prominent in the CFD2030 roadmap such as uncertainty quantification.

The roadmap is organized into two distinct sections. First, from top to bottom, we identify the key sub-challenge configurations that will be utilized as either part of ground-based experimental testing or with sub-scale flight testing leading to the simulation of the GC and beyond, as depicted in Fig. 1. Next, we identify a set of technology demonstrations and milestones which serve as way-points and opportunities to assess CFD development research and technology integration efforts utilizing these challenge configurations. For example, we anticipate that comprehensive wind tunnel testing of a representative high-lift configuration over the next several years will provide extensive data to assess simulation technologies (e.g. turbulence modeling, Adaptive Mesh Refinement (AFR), integrated aero-servo-elastic response, etc.) to enable simulation capabilities (e.g. $C_{L,\ max}$, stall speed determination, dynamic maneuvers, etc.) in an efficient way (e.g. overnight turnaround for production CFD tool, real-time multi-fidelity database, etc.).

Second, along the bottom half of the roadmap, the six critical technology categories are detailed. Within each category (with the exception of HPC), a series of swim-lanes are defined to highlight important computational technologies expected to play a critical role in the numerical simulation for the GC. The swim-lanes are color-coded by Technology Readiness Level (TRL). Green indicates a technology at a high TRL level, which is ready for production use. Yellow indicates a technology at a medium TRL level, which has been demonstrated for a representative application. Red indicates a technology at a low TRL level, which is at a basic research or early validation stage. The end of each swim-lane is shown with a green arrow, denoting that the specific technology item is at a sufficiently mature level to be readily used at the indicated period along the time-line. Many swim-lanes show arrows that extend beyond 2040, which signifies that elements of the specified technology will likely require a significant amount of development

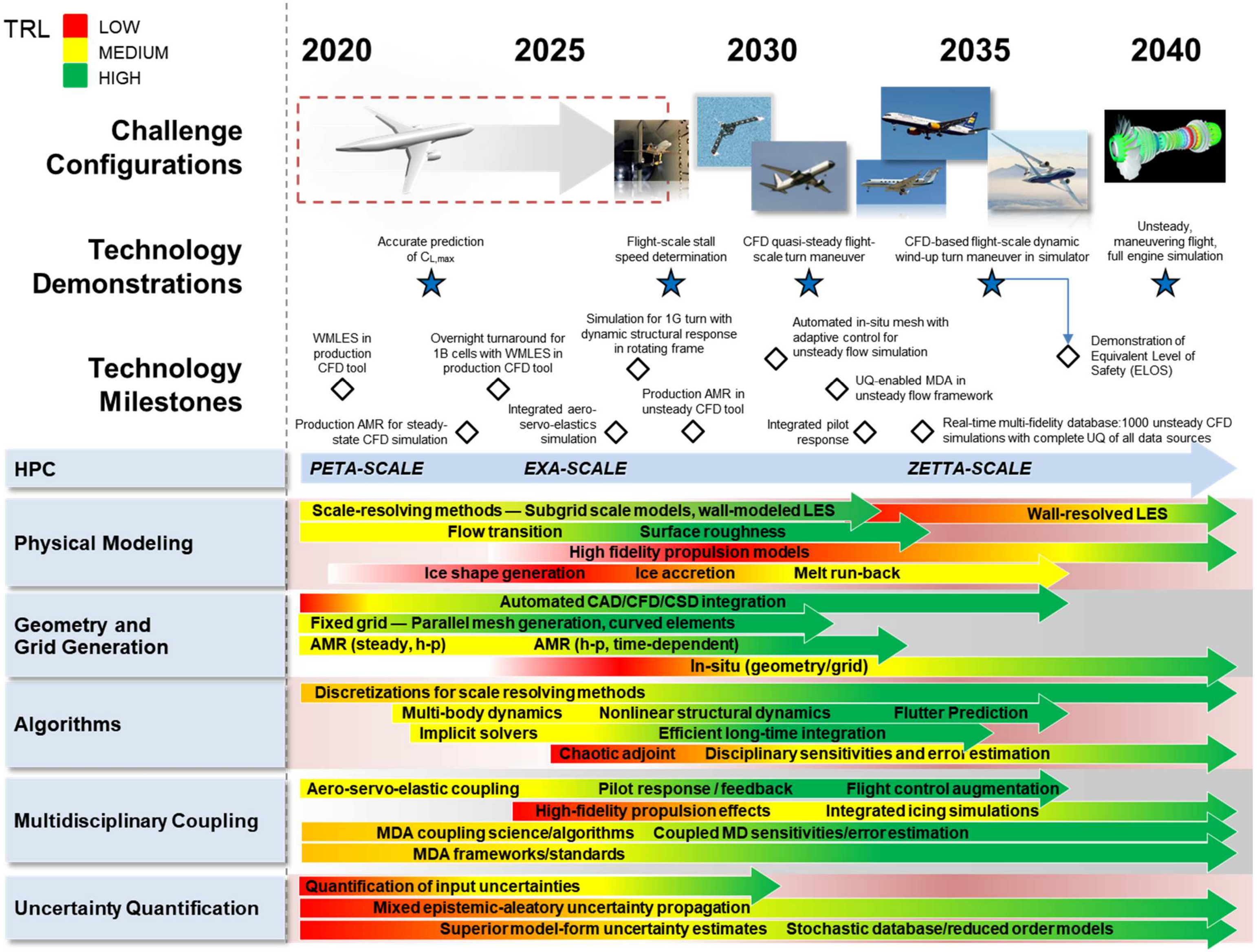

Figure 2. High Lift Grand Challenge Roadmap

focus and attention over many years to properly mature for application to the GC. In many cases, technologies will mature to a high TRL level, but would be expected to be supplanted by more advanced technology over time. For instance, fixed-grid mesh generation, already at a fairly high maturity level, will likely give way to AMR technology, both for steady and time-dependent analyses, over the next decade. Then, as AMR matures, this capability is expected to be integrated into the numerical simulation itself, where mesh generation is not a separate step, but is part of the solution methodology. In the following subsections, we discuss the required advances and associated investments in the technology development areas in more detail.

A. *Physical Modeling*

Clearly, a CFD capability which can successfully capture all of the relevant flow physics at the edges of the flight envelope must be developed to be able to tackle the WUT GC. The ability to accurately compute flows with smooth body separation as well as separated flows with sharp corners is critical for predicting control surface effectiveness, maximum lift ($C_{L,\max}$), and stall characteristics. Furthermore, the prediction of accurate moments and detailed loading will be required throughout the flight envelope including post-stall conditions with regions of massive flow separation. Although current industrial CFD simulations rely predominantly on steady-state Reynolds-Averaged Navier-Stokes (RANS) or unsteady RANS (URANS) methods, there is growing evidence that turbulence scale-resolving methods will be required to enable the accurate simulation of flows with large regions of separation [13]. This puts an emphasis on the development of improved sub-grid scale (SGS) models, including dynamic SGS turbulence models for Large Eddy Simulations [14]. At the same time, the high computational cost of fully (wall) resolved LES methods for flight Reynolds numbers favors the adoption of wall-modeled LES (WMLES) approaches, which in turn emphasizes the need for investments in improved wall models along with their validation. We note that scale-resolving methods encompass a wide variety of discretization approaches, including finite-volume [15], high-order finite-element [16] and even Lattice Boltzmann (LBM) methods [17]. In each case, these models must be developed for their specific accompanying discretizations and validated accordingly.

The ability to predict transition at flight Reynolds numbers takes on additional importance for certification type maneuvers such as the WUT. Here, the effect of surface roughness on transition due to surface contamination, ice crystal formation and ice accretion must also be incorporated. Clearly, fully resolving the laminar and transitional regions within an LES simulation will remain impractical for the foreseeable future and some level of modeling will be required. Although there has been growing interest in transition models for industrial calculations [18, 19], these models are primarily intended for use with RANS simulations and emphasis must be focused on the development of transition models for scale-resolving methods in the presence of wall modeling.

For the GC and associated SC problems discussed in this paper, flow physics are not the only areas in need of physical modeling. For example, a range of progressively low-to-high fidelity propulsion models can be envisioned for the WUT GC problem. At the low end, current engine deck models can be used to provide suitable inlet/outlet flow boundary conditions for powered flight simulations. However, for more complex cases such as high cross-flow or post-stall conditions, accurate engine response to dynamic inlet distortion is required. At the highest fidelity level, one could envision a complete engine simulation [20] coupled to the external CFD simulation, although the incorporation of detailed combustion effects would likely not be warranted. However, there are various scenarios including engine-out conditions with windmilling, where the most effective approach may be to simulate the fan and model other engine components such as the core. The formulation of a useful hierarchy of propulsion models and the development of interfaces and requirements for such models are areas that could be led effectively through government and industry collaboration.

Aircraft icing plays an important role in the design and certification of high-lift systems, and will need to be considered increasingly as an integral component in the simulation process [21]. The determination, inclusion and gridding of static ice shapes involve various physical modeling assumptions and constitute the entry-level approach for the consideration of icing effects on performance [22]. At the higher-fidelity end, icing simulations entail a variety of physical models including ice crystal formation, droplet impingement, ice accretion and melt run-back [23]. While current modeling approaches exist for all of these icing aspects, continual improvement in the fidelity of these models along with suitable validation must be undertaken in order for icing simulations to be incorporated with scale-resolving simulations and to keep pace with advances in other simulation technologies.

B. Geometry/Grid

The ultimate goal of Geometry/Grid technology development is to enable the generation of accurate discretizations for computational fluid dynamics (CFD grids) or computational structural dynamics (CSD models) which conform to the underlying geometry in a fully automated manner. Automation takes on additional importance in the context of certification by analysis, where repeatability is essential and where the dependence of the quality of the simulation outcome on user input must be minimized. The automation process naturally begins with the link between the CAD defined geometry and the numerical discretizations, i.e. the CFD grid and the FEM structural model. This coupling must be tight and fully automated, including the ability to add geometric complexity or to de-feature the geometry in a traceable manner. This association is important because it represents the essential link between the numerical simulation capability and the digital thread of the product development process. Although commercial software vendors are making significant progress towards linking numerical simulation tools with digital product life-cycle management, the ability to incorporate cutting-edge or specialized simulation tools from outside vendors or from government and/or in-house code development efforts remains a bottleneck that will need to be addressed.

The geometric complexity and the required accuracy of the simulations for our GC problem will result in the requirement of generating highly resolved and detailed CFD meshes and CSD models. The routine generation of computational meshes with hundreds of billions of cells will be required in the near future for these types of problems. Alternatively, high-order discretizations may be used with fewer mesh elements, although these discretizations require curved mesh elements in the vicinity of the underlying geometry in order to fully realize their accuracy potential. In CFD as in CSD, both approaches will likely remain competitive for specific applications. Nevertheless, investments in large scale parallel mesh generation with distributed access to CAD will be required going forward.

Ultimately, adaptive mesh refinement (AMR) can relieve the burden of generating highly resolved initial meshes while providing more accurate, consistent and repeatable solutions. Although impressive advances in AMR have been demonstrated for industrial RANS problems over the past 5 to 10 years [24, 25], significant issues remain in terms of optimal error estimation for refinement criteria and application to time-dependent problems. Indeed, dynamic AMR will be critical for capturing flow features such as wakes or vortical flows, which may impinge on downstream components, in a truly predictive manner. Although there have been some successes in dynamic AMR CFD, particularly for rotorcraft simulations [26], these techniques must become more widely adopted and tailored for the problem at hand, which will likely include scale-resolving methods and highly complex moving geometries. For higher-order methods, dynamic AMR must be combined with dynamic solution order refinement to enable h-p refinement [27]. Investments in areas such as dynamic refinement criteria and distributed CAD access for surface point placement must also be addressed [28].

As mentioned at the outset, the ultimate goal is the development of a fully automated process where the CFD mesh and CSD model are generated directly from the CAD geometry resident in the product development digital thread, and adapted throughout the dynamic simulation process with no user intervention. This “in-situ” mesh generation can be expected to become reality through the logical progression from parallel mesh generation to static AMR and finally dynamic AMR.

C. Algorithms

Algorithmic advances will be crucial for achieving the GC and associated SC problems, as more complex and highly resolved simulations increasingly expose the deficiencies in current-day algorithms. Within the realm of CFD, the development of suitable discretizations for scale-resolving methods should be emphasized. Currently both second-order accurate finite-volume methods [15] as well as higher-order accurate finite-element methods [16] are considered viable approaches for industrial LES, and this trend can be expected to continue. However, in both cases the development of low dissipation and non-linearly stable discretizations is important for optimizing the accuracy, efficiency and robustness of LES simulations [29, 30]. For structural dynamics, highly resolved geometrically nonlinear models will be required. Additionally, these must include the ability to simulate multibody dynamics necessary for accurately modeling control surface motion, actuation and linkage to flight-control systems, as is often utilized for rotorcraft problems [31]. Furthermore, the increasing size and detail of the required structural models means that more emphasis towards efficient parallel assembly and execution of these models will be required.

One of the defining characteristics of the GC/SC problems is that they entail dynamic simulations over relatively long time periods. Therefore, algorithmic advances for time-integration schemes must also be investigated. This should include both frequency-domain and time-domain approaches. Advances in the efficiency and accuracy of these methods, as well as control of the accumulation of error over long time periods should be sought. Although many LES approaches currently rely on explicit time-stepping schemes, due to the wide range of resolution required for detailed models, a strong case can be made for the use of implicit time-stepping schemes for scale-resolving methods, particularly in the presence of high-order temporal discretizations [32]. Thus, advances in efficient parallel implicit solvers (linear and nonlinear) will remain a strong driver for both CFD and CSD disciplines.

Advanced techniques for sensitivity analysis and error estimation at the disciplinary level will need to be developed in order for these quantities to feed into a general UQ framework for complex simulations. Disciplinary linear or local sensitivity analysis is commonplace today for RANS simulations using discrete adjoint and tangent methods [33]. However, for scale-resolving methods, which result in chaotic solution behavior, linear sensitivity approaches diverge and obtaining meaningful sensitivities at acceptable computational cost for chaotic problems remains an open problem today [34]. Similarly, current techniques for estimating discretization error must be expanded or replaced with methods that can estimate discretization error for time-dependent problems. Even at the disciplinary level, multiple sources of error must be considered, starting with time-varying spatial discretization error, possibly in the presence of AMR/h-p refinement, coupled with temporal discretization error and algebraic error resulting from inexact solution of the implicit systems at each time step [35]. More focused error estimates for engineering quantities-of-interest (QoI) will be essential for typical metrics of success which arise in the GC problem and certification by analysis. Although it is well known today how these can be formulated through the weighting of general error estimates with adjoint derived field quantities [36, 37], the inability to efficiently compute adjoint sensitivities for chaotic problems will severely limit our ability to compute QoI-based error estimates with scale-resolving methods. Therefore, sensitivity analysis and error estimation at the disciplinary level will remain an important focus area in the near future.

D. Multidisciplinary Coupling

One of the defining characteristics of the GC and associated SC problems is that they are all highly interdisciplinary. Thus a new emphasis must be devoted to the realization of efficient and robust multidisciplinary simulations with quantifiable error estimates. Clearly, most simulations at flight Reynolds numbers must include, at a minimum, aerodynamic and flexible structural effects for realistic prediction abilities. For the simpler SC problems, this may manifest initially as static aero-structural simulations, for example as in steady heading trim problems, although extension to dynamic aero-elastic simulations will rapidly follow as focus turns increasingly to time-dependent problems. Furthermore, the objective of simulating dynamic flight maneuvers will require the development of coupled aero-servo-elastic simulation capabilities with the inclusion of movable control surfaces. Finally, the integration of aero-servo-elastic simulations with a flight-control-system model will enable the calculation of pilot feedback and response, and ultimately the simulation of flight-control augmentation effects, all of which represent critical metrics in the certification process.

At the same time, other disciplines will come into play. For example, the simulation of propulsion effects at off-design conditions will require the incorporation of high-fidelity propulsion models into the multidisciplinary framework. These may be as simple as prescribed inlet/outlet conditions as determined from engine deck models, or more complex models which involve the simulation of specific engine components (i.e. rotating fan for engine-out windmilling). This represents a relatively new area that will require focused investment to develop and validate. Additionally, the fact that aircraft icing constitutes an integral part of high-lift system development and certification means that the various icing physical models discussed previously will need to be coupled into the broader multi-disciplinary analysis (MDA) facility.

Therefore, a flexible and standardized MDA framework must be developed to enable the incorporation of a growing set of widely disparate disciplines. To enable tight coupling of diverse disciplines and codes, data standards need to extend to include memory resident information and coding structures, and must be designed to support the computation of coupled sensitivities and error estimates based on the corresponding disciplinary computed values. To some degree, the development of such as framework is already underway, partly in response to recommendations put forth in the CFD2030 report [38]. However, this effort must be continued and broadened to include the full range of different disciplines envisioned herein. Such efforts are best led by government (e.g. NASA in this case) in collaboration with university researchers and industrial concerns.

Special attention must also be given to the science of multidisciplinary coupling at high fidelity. Currently, there is a dearth of formal methodologies to guarantee the stability and accuracy of coupled high-fidelity simulations. The development of libraries and procedures that enable accurate and stable couplings, capable of handling disparate disciplinary fidelities, must be pursued. Such properties often require the satisfaction of conservation principles to which close attention must be paid. Ultimately, solvers using discretizations of a given accuracy (in space and in time), when coupled to other solvers, must ensure that the accuracy of the component solvers is preserved and that the coupling procedure does not give rise to numerical errors that may manifest themselves through solution instabilities [39]. Progress in tightly coupled nonlinear solution techniques, which can be fully supported through the MDA framework must also be achieved, for example using disciplinary preconditioned global or monolithic Newton-Krylov methods [40, 41].

At the same time, the MDA framework must support the calculation of coupled multidisciplinary sensitivities and error estimates, based on disciplinary supplied values. Here many of the coupling techniques and solution strategies devised for multidisciplinary analysis can be leveraged for the coupling of sensitivities and error estimates, although additional considerations for issues such as the estimation of coupling errors must be taken into account. The end-result will consist of high-fidelity multidisciplinary sensitivities and error estimates, which can be used as input for global uncertainty quantification strategies.

E. Uncertainty Quantification

Uncertainty Quantification (UQ) is the process of characterizing all major sources of uncertainty in a model or experiment, and quantifying their effect on the analysis outcomes [42]. For complex multidisciplinary simulations, where sources of uncertainty are manifold, robust UQ is essential for building credibility and confidence particularly when certification issues are at stake. Obviously, sensitivity analysis and error estimation, as discussed in the previous sections, are central to any UQ effort. However, in this section we focus on the development of non-intrusive UQ strategies which leverage available single or multi-disciplinary sensitivities and error estimates when available. In general, UQ involves four major components: the identification of sources of uncertainty, the characterization of their statistical form, the propagation and aggregation of uncertainty through models, and finally the analysis of uncertain results. For the GC problem, uncertainty sources include input parameters such as freestream flow conditions including atmospheric turbulence effects, geometric variations, structural specifications etc., as well as numerical sources of uncertainty. Numerical sources include time-dependent discretization errors both in space and time, algebraic errors and multidisciplinary coupling errors, all of which have been mentioned previously. Additionally, the effect of model-form errors resulting from approximate physical models (i.e. such as a turbulence model) plays an important role. The characterization of these error sources and classification into aleatory or epistemic types is an important step that helps determine suitable strategies for propagating and aggregating these error sources for obtaining final simulation output uncertainties. Much of the required future investment in UQ will be focused on the development of suitable propagation and aggregation methods for uncertainties in large complex multidisciplinary simulations. This includes building UQ frameworks for driving ensemble runs of numerical simulations using optimal sampling strategies in parameter space, leveraging available single or multi-disciplinary sensitivities and error estimates, and building reduced-order models (e.g. polynomial chaos, Kriging models, etc.) with built-in uncertainties of simulation quantity of interest (QoI)s or certification metrics. For the WUT GC problem, the ultimate goal is the development of a stochastic flight database, which can be used as input to a pilot-operated flight simulator, with credible uncertainty estimates at any location in the database. The first step towards a more robust UQ capability consists of applying existing well-known UQ strategies to the single and multi-disciplinary simulations as they are developed for the SC and GC problems. However, new fundamental approaches will also be required to address some of the significant issues that must be overcome in order to provide realistic uncertainties for the GC problem. These include the ability to deal with large parameter spaces, techniques for suitable interpolation and extrapolation from validation data, realistic assessment of the effect of model-form uncertainties within large complex simulations, and rare statistical even predictions (i.e. UQ tails) which are paramount for safety considerations. Finally, with wind tunnel and flight-testing still expected to play a key role in the design and certification process, the development of methods to merge and assimilate CFD and multidisciplinary simulation data with other multi-fidelity experimental/computational data sources to create an integrated database, including some measure of confidence level and/or uncertainty of all (or individual) portions of the database, will be required.

V. Community Coordination and Collaboration Opportunities

Success in addressing the GC problem will require coordinated activities within the relevant scientific and engineering communities that align with advancing technologies in the focus areas described above. Ongoing efforts in CFD validation for high-lift aerodynamic prediction, for instance, could be more directed towards addressing specific shortcomings in important capabilities required to solve the GC problem. Likewise, refocusing workshop and other community collaboration activities to tackle specific aspects related to the GC problem would also be particularly effective. Several current and potentially new areas for coordination and collaboration to accelerate progress towards the GC problem are described below.

A. High Lift Common Research Model Ecosystem

The High Lift Common Research Model (CRM-HL) is an industry-representative high-lift transport geometry envisioned to provide a pre-competitive open platform in which to develop and mature technologies in several areas [43]. A community-sourced set of resources available from within a broad collaborative environment or "ecosystem" specifically involving the CRM-HL geometry has been established. The primary focus of the CRM-HL ecosystem is to advance CFD methods and tools through careful validation with comprehensive experimental test data. Over the next several years, a series of experimental test campaigns are planned using geometrically-consistent reference models [44] to be fabricated by a number of international organizations. Specific technical requirements [45] will drive CFD validation efforts. Experimental data will be collected in a variety of wind tunnel facilities capable of testing at atmospheric to cryogenic conditions. Additionally, significant efforts in exploring and developing new flow measurement techniques are expected to provide a wealth of high-quality on- and off-body flow data not generally available during standard wind tunnel testing of industrial high-lift configurations. Experimental data from these tests are envisioned to be used to validate critical CFD tools and technologies required to successfully address SC-1 and SC-2.

B. Prediction Workshops

Since the early 2000's, CFD prediction workshops have been instrumental in focusing community energy to address the hard, and previously intractable, problems in CFD prediction of aerodynamic performance of industrial aerospace configurations. Starting with the Drag Prediction Workshop (DPW) series in 2001 [46], a number of workshops based on the DPW model have been organized and executed over the last several decades. It is expected that CFD prediction workshops will be the primary mechanism by which the aerospace CFD community interacts with industrial aerodynamic design and product development experts to advance and mature key computational capabilities, particularly to tackle SC-1 and SC-2. Addressing and overcoming the technical shortcomings that currently exist in CFD methods and tools are ultimately required for demonstrating the high-lift GC. Several of the established workshop activities are described below, emphasizing the connection between these specialized efforts with the predictive capabilities required to successfully address the GC problem.

The High Lift Prediction Workshop (HLPW) series [47] focuses on the numerical prediction of aerodynamic performance for swept, medium-to-high-aspect ratio wings in airplane landing and take-off configuration. There have been three workshops to date. HLPW-1 was held in Chicago, IL in 2010 and utilized the NASA Trapezoidal Wing ("Trap Wing") configuration [48]. HLPW-2 was held in San Diego, CA in 2013 and featured the DLR-F11 model configuration [49]. HLPW-3 was held in Denver, CO in 2017 and used the JAXA Standard Model (JSM) configuration, as well as an early version of the CRM-HL [50]. Efforts within HLPWs have studied the predictive capability of CFD methods on a range of geometrically complex high-lift configurations from a simplified multi-element wing (Trap Wing) to more complete configurations that have modern wing architectures with industrially-representative engine installations (F11, JSM, CRM-HL). CFD results have been collected from participants utilizing various flow solver methodologies (e.g., Reynolds-Averaged Navier Stokes (RANS), Lattice Boltzmann Method (LBM), etc.) using various mesh discretization strategies (e.g., fixed grid, Adaptive Mesh Refinement (AMR), etc.) at realistic flow conditions, and compared with wind tunnel test data at several Reynolds numbers. Comparisons between experimental and computational results, obtained with and without wind tunnel walls modeled, have shed light on the ability of current CFD technology to predict aerodynamic forces and moments along the entire configuration lift curve, particularly with focus on maximum lift ($C_{L,\ max}$). The upcoming fourth HLPW will benchmark community predictive capabilities on the CRM-HL landing configuration as tested in the QinetiQ 5-metre facility in 2019 [51]. Flow solutions using emerging Large Eddy Simulation (LES)-based turbulence treatment methods (e.g., hybrid RANS/LES, wall-modeled LES) and maturing AMR and higher-order discretization methods will help to assess current capabilities. Future workshops will exploit greatly expanded collection of experimental data at high Reynolds numbers (e.g. cryogenic facilities) focusing on aeroelastic deformation, more realistic engine effects, and the inclusion of representative ice shapes. The focus on high-lift flow physics and multi-discipline coupling will directly support efforts described for SC-1 and SC-2, respectively.

The Geometry and Mesh Generation Workshop (GMGW) series [52] focuses attention on the advancement of geometry processing and computational mesh generation tools and techniques to specifically address shortcomings outlined in the CFD Vision 2030 report [53]. GMGW-1 was held in collaboration with HLPW-3 in 2017, and focused effort on geometry and mesh requirements for a simplified version of the CRM-HL configuration [54]. GMGW-2 was held in San Diego, CA in 2019 and addressed exascale meshing and re-meshing of the simplified CRM-HL configuration, as well as re-meshing of a notional aircraft parametric model [55]. In 2020, an AIAA Special Session was held to study how changes in the mesh affect solution accuracy and convergence for a CFD flow solver on a two-dimensional slice of the CRM-HL geometry [56]. In the future, GMGW-3 will be co-located with HLPW-4 in the summer of 2021 and will further focus attention on assessing meshing issues related to complex geometry, particularly for the complete CRM-HL landing configuration. For both GMGW-3 and HLPW-4, a prominent area of emphasis will be exploring Adaptive Mesh Refinement (AMR) strategies for the CRM-HL landing configuration, and the effect of mesh refinement on solution accuracy. Effective mesh adaptation strategies, particularly for unsteady, dynamic flow simulations, are expected to be critical elements in successfully demonstrating the GC problem.

The Propulsion Aerodynamics Workshop (PAW) series [57] focuses community attention on CFD prediction of engine inlet and nozzle flows. PAW-1 was held in Atlanta, GA in 2012 and focused on a serpentine duct (S-duct) geometry from ONERA [58]. PAW-2 was held in Cleveland, OH in 2014 and focused on flow control for the S-duct geometry [59]. PAW-3 was held in Salt Lake City, UT in 2016 and PAW-4 was held in Cincinnati, OH in 2018. Both workshops utilized the Internal Flow Control Prediction Technology (IFCPT) S-duct geometry [60, 61]. Scheduled for January 2021, PAW-5 will consider an inlet close to a ground plane and a NASA convergent nozzle configuration. These workshops featured increasing use of unsteady RANS CFD methods in validating against an extensive set of high-quality experimental data. The fundamental CFD validation work performed by members of the propulsion community through these workshops leads directly to the improved computational methods that will be required to accurately model flight propulsion flow-fields and assess their impact on external aerodynamics within a dynamic maneuver.

The Aeroelastic Prediction Workshop (AePW) series [62] focuses attention on the assessment of computational aeroelasticity methods and tools to predict static and dynamic aeroelastic phenomena and the responses of these phenomena on relevant geometries. AePW-1 was held in Honolulu, HI in 2012 and compared CFD results to experimental data for two supercritical wing geometries, as well as to data collected using the High Reynolds Number Aerostructural Dynamics (HIRENASD) model tested in the European Transonic Wind Tunnel (ETW) [63]. AePW-2 was held in San Diego, CA in 2016 and focused attention on aerodynamic flutter using the Benchmark Supercritical Wing (BSCW) configuration [64]. Current efforts are focused in several areas: simulations on configurations with large deflections, simulations of the X-56 MUTT flight test vehicle, simulations at high angle-of-attack using the BSCW configuration, and simulations at hypersonic conditions. Future emphasis on the aeroelastic deformation and structural response for highly-loaded dynamically-scaled airplane configurations at subsonic conditions could be addressed in this community. This type of effort, and efforts like it, would help enable the integrated fluid-structure interaction (FSI) predictive capabilities required to support successful completion of the high-lift GC.

The first Icing Prediction Workshop (IPW) [65] is schedule for June 2021 in Washington, D.C., and will establish ice shape comparisons between CFD results from both 2D and 3D simulation codes with test data for cases where experimental ice shapes are available. The goal is to establish benchmark datasets to assess icing simulation tools and practices, and develop best practices. Over time, the workshop organizers seek to better understand the interaction of the flow solver with the physics of ice generation and accretion, particularly for complex industrial applications.

Although the NASA sponsored Computational Methods for Stability and Control (COMSAC) symposium was held back in 2003 to assess the use of CFD to predict stability and control derivatives [66], there has been little activity in this area until recently, with the announcement of the first Stability and Control (S&C) Prediction Workshop to be held in January 2021. The goal of this workshop is to develop best practices for the prediction of S&C derivatives using both RANS and hybrid RANS/LES methods (e.g. DES), and to assess the limitations of these CFD methods when best practices are applied. Data collected from a recent test of the High-Speed CRM (CRM-HS) in the ONERA-S2MA facility [67] will be used to assess the ability of CFD methods to predict side force, yawing moment, and rolling moment at small angles of sideslip. This initial assessment of CFD predictive capabilities in sideslip will be an important step in better understanding the aerodynamic effects associated with large body-driven vortex formation, and its impact on asymmetric wing loading. Accurately predicting these effects will be crucial in addressing similar effects in a dynamic, roll-induced WUT maneuver.

C. Potential Future Activities

In addition to community-driven workshop events that advance the state-of-the-art in CFD predictive capabilities over a fairly wide range of individual discipline areas germane to simulating the GC problem, other areas in the CFD development and validation pipeline require more focused attention to prepare for GC application. Specifically, efforts to integrate and validate multi-disciplinary analysis capabilities are critically important if the GC problem is to be solved in the proposed timeframe. For this purpose, we propose several new focused activities below.

i. "Digital Flight" Workshop

To accelerate the development of multi-disciplinary analysis capabilities, we envision a new community focused on "Digital Flight" type workshops, which could be used to benchmark and assess current capabilities, perhaps using a building-block approach, to identify technical gaps and difficulties in discipline coupling areas. For instance, results of simulations for a representative time-dependent structurally rigid aerodynamic maneuver with only fixed control surfaces could be compared to appropriate test force/moment/rate data. Then the same maneuver could be repeated but with the added complexity of accurately simulating structural response. Finally, a time-dependent flight controller model/capability could be added to simulate flight-control augmentation effects as influenced by aerodynamic forces and structural deformations. Initially, these activities could simply focus on comparison between computational technologies to not only evaluate and prove out disciplinary integration solutions, but to demonstrate and mature the underlying frameworks developed and required for effective industrial use. Later, validation of these integrated solutions to assess accuracy, robustness and establish best practices, particularly in a time-dependent sense, would be performed.

ii. Validation of Multi-Disciplinary Analysis Capabilities

Currently, a significant gap exists between ground-based wind tunnel testing performed to validate single-disciplinary (e.g. external aerodynamics) capabilities on static, representative airplane configurations at moderate Reynolds numbers (like the CRM-HL), and flight testing performed to collect data in a dynamic flight environment on complete, aeroelastically-deforming airplane configurations at flight scale. As a result, systematic validation of coupled, multi-disciplinary analysis capabilities must be performed to develop and mature CFD technologies able to accurately address the GC WUT maneuver. Ground-based testing in specialized facilities, which have capabilities to obtain validation test data for coupled disciplines, ideally in a dynamic flow environment, offers the possibility of obtaining lower cost and higher quality data for validation of multidisciplinary simulations. Similarly, sub-scale model flight testing also offers reduced cost and potentially more elaborate validation data compared to full-scale flight testing. These are described in more detail below.

a. Ground-based testing

Ground-based testing of geometrically-representative configurations in specialized facilities to generate experimental data in dynamic "maneuver-like" flow scenarios offers the possibility of generating high quality validation data in a controlled environment. One of the key advantages is that the overall flow environment, and the geometric configuration, along with any potential structural deformations, are much more easily quantified. As such, design of suitable testing campaigns to study specific discipline coupling (e.g. aero-structural) are much more feasible when certain aerodynamic effects, such as onset flow or engine thrust, can be tightly controlled. However, in general, current facilities can simulate two out of three required disciplines, i.e. aero-elastic problems or rigid controlled maneuvers, but not all three (aero-servo-elastic) disciplines together. The specialized facilities that do exist, for example the Transonic Dynamics Tunnel (TDT), 12-Foot Low-Speed Tunnel (LST), and 20-Foot Vertical Spin Tunnel (VST) at NASA Langley Research Center, can be used to simulate specific flight events, but do not currently have the capability to perform testing of flight maneuvers like the WUT for aeroelastically-scaled high-lift airplane configurations at flight scale Reynolds numbers. However, some recent thinking along these lines indicates that it may be possible to utilize these facilities by modeling the WUT as a steady increase in configuration pitch (which equates to a steady increase in wing loading) using an available forced-oscillation rig. Details on model size, Reynolds number, and other critical parameters would need to be further explored. However, making bespoke modifications to existing facility infrastructure to enable testing for aero-servo-elastic maneuvers may not prove to be cost effective. Therefore, the construction of new facilities specifically designed to perform systematic and integrated multi-discipline testing, based on requirements from this GC problem and from other critical applications, may ultimately be required to bridge the validation gap discussed herein.

b. Subscale Flight Testing

Sub-scale testing of a flight vehicle perhaps offers the most direct route to obtaining sufficient data suitable for multi-disciplinary simulation validation. Flight testing of a sub-scale vehicle, if instrumented appropriately, would provide direct aerodynamic performance characteristics (force, moments, derivatives) from a dynamic maneuver, which could be chosen to mimic the WUT. For analysis validation purposes, having both a high-fidelity CAD geometric definition of the vehicle Outer Mold Line (OML) and a complete structural model of the flight vehicle available would be required to model the dynamic aeroelastic deformation that the configuration would experience in a representative maneuver. Significant disadvantages exist, however, in utilizing flight-based data for validation purposes. First, available flight demonstrator/research vehicles, like the X-56A MUTT, are often not geometrically representative of existing commercial airplane configurations, and the lower, sub-scale Reynolds numbers reflected in their flight test data may present significantly different flow characteristics compared with full-scale data from the flight of a production airplane, particularly for the high-lift system. Also, the time-dependent engine flow conditions experienced in flight may not be easily known or determined, adding to the uncertainties for accurate modeling of critical propulsion effects in a dynamic maneuver. Finally, atmospheric effects such as wind gusts may also be difficult to quantify, further adding to the overall uncertainties associated with any flight data utilized for CFD validation purposes. Nevertheless, sub-scale flight testing presents complementary advantages compared to ground-based testing, and the lower cost of sub-scale flight testing, combined with the possibilities of more elaborate instrumentation, make them an attractive alternative to full scale flight testing

c. Full Scale Flight Testing

Ideally, a full-scale flight test campaign using a representative transport aircraft configuration would be required to provide final validation data to characterize real airplane aerodynamic performance. A full-scale flight test represents the ultimate reality or truth value of what the virtual WUT GC numerical methods attempt to simulate. The drawbacks of full-scale flight testing are well known. In addition to the drawbacks mentioned for sub-scale testing, these include first and foremost the associated cost, but also the use of a representative configuration CAD geometry, including structural details, which can be publicly released without impinging on proprietary issues. Another challenge with full-scale flight testing is the potential difficulty in obtaining detailed measurements required for validation of numerical simulations, which would be over and above data that is typically obtained for certification testing. These include, at a minimum, substantial unsteady surface pressure measurements, enhanced flow visualization (e.g. flow cones), detailed structural deformations, particularly to quantity high-lift element positioning (e.g. slat, flap gap and overlap) relative to the main wing, control surface hinge moments, data to characterize time-dependent engine propulsion conditions, and a detailed assessment of wind and atmospheric effects, among others. Nevertheless, there have been various NASA supported full-scale flight tests at least partly designed for CFD validation over the years, including the B-737 high-lift experiments in the early 1990's [68], and the F16XL [69], to name a few. More recently, the VicToria project at the German Aerospace Center (DLR) has undertaken an elaborate flight test campaign using a highly instrumented A320 aircraft specifically for validation of multidisciplinary numerical simulation capabilities [70]. A validation flight-test campaign could make use of current NASA Armstrong research flight vehicles, perhaps as part of existing or planned tests, or could seek out a specific new platform for the WUT GC, perhaps with a larger purpose in support of certification by analysis. Clearly, designing and deploying a targeted flight-test campaign to support efforts to address the WUT GC problem is a long-term endeavor that must be planned years in advance and which will require substantial advocacy to become reality.

iii. Access to HPC Resources

Initial solution approaches to the GC problem may take on different forms, from a single dynamic maneuver simulation to the training of accurate reduced-order models using multiple high-fidelity solutions. In all cases, the successful simulation of the GC problem will require leading-edge high-performance computational (HPC) resources. Perhaps more importantly, early and sustained access to leading-edge HPC resources will be required by researchers, developers and stakeholders in order to develop and validate the technology necessary for the GC problem. Access to HPC is critical in the software development stage, where new algorithms must be tested at scale before significant development resources are committed, and where porting and testing of software to emerging HPC architectures must be done on a continual basis. Furthermore, validation activities for these complex multidisciplinary simulations will require significantly more capable HPC resources than what is currently available to most community groups. Although various government agencies maintain leading-edge computational facilities, many of these resources are devoted to capacity computing, i.e. processing large numbers of smaller runs, rather than capability computing, where a small number of jobs that make use of a significant portion of the entire machine are prioritized. In order to accelerate development towards the GC problem, a mechanism for granting community access to large-scale computing resources must be developed. For example, within NASA, this could be achieved by augmenting and reserving a portion of the NAS Pleaides system for large-scale runs devoted to the various GC and SC problems under consideration. Other mechanisms such as the INCITE program within the DoE have been successful at granting early access to leading-edge HPC for specific problems of importance [71]. Generally, access can be granted through a competitive proposal process, where proposals are evaluated based on their technical suitability, their relation to one of the GCs and their computational readiness to run at large scale on the available hardware. Facilitating HPC access may involve innovative approaches, such as the pooling of resources between different government agencies, including NASA, DoD and DoE within the US, and possibly with other international agencies.

Conclusions

In this paper, we have described a CFD grand challenge problem which consists of the accurate, efficient, and robust numerical prediction of a dynamic WUT maneuver for a commercial airplane in high-lift configuration at a representative low-speed flight condition. Leading up to the GC WUT maneuver, a series of sub-challenge problems have also been proposed to incrementally advance the development and validation of important computational technologies. For these challenges, we have defined a series of technology demonstrations and milestones, along with required advancements in key technology areas, within a comprehensive, integrated roadmap. Insights into potential research areas to address the technology requirements have been discussed, along with current CFD prediction workshops and potential future community collaboration activities, required to accelerate progress towards the GC. A significant gap in available data exists for benchmarking and validating multi-disciplinary capabilities, particularly for the aero-structural-controls coupling vital for successful simulation of the WUT maneuver. To address these gaps, efforts must begin now to collect new ideas to enable innovative ground-based testing for multi-discipline coupling/method validation, as well as to design and plan comprehensive flight-test campaigns where high-quality sub-scale and flight-scale data can be collected and utilized to validate numerical results.

Acknowledgments

The authors would like to thank the following for their technical insights and for fruitful discussions in many of the areas discussed in this paper: Greg Laskowski (Dassault Systemes), M. S. Anand (Rolls-Royce Corporation), Gorazd Medic (Raytheon Technologies), Doug Kothe (Oakridge National Laboratories), Paul Bolds-Moorehead, Doug Lacy, Timothy Mauery, Rob Hoffenberg, Adam Malone, Donald Cook, Kolleen Hood, Philippe Spalart (ret.), Byram Bays-Muchmore and Robb Gregg (The Boeing Company), Walt Silva (NASA Langley Research Center), and Cord Rossow and Stefan Goertz (DLR).

References

1. Slotnick, J. P., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., and Mavriplis, D. J., "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences", NASA/CR–2014-218178, March 2014.
2. Slotnick, J. P., and Heller, G., "Emerging Opportunities for Predictive CFD for Off-Design Commercial Airplane Flight Characteristics," 54th 3AF Conference, Paris, France, 2019.
3. Spalart, P. R., and Venkatakrishnan, V., "On the role and challenges of CFD in the aerospace industry", The Aeronautical Journal, Vol. 120, No. 1223, pp 209-232, January 2016.
4. Abbas-Bayoumi, A., and Becker, K., "An Industrial View on Numerical Simulation for Aircraft Aerodynamic Design", Journal of Mathematics in Industry (2011) 1:10, DOI 10.1186/2190-5983-1-10.
5. Robert D. Gregg III, Chief Aerodynamicist, Boeing Commercial Airplanes, AIAA Forum 360 Panel Discussion on Certification by Analysis, Aviation Conference, June 2018.
6. American Institute of Aeronautics and Astronautics. *When Flight Modelling is used to Reduce Flight Testing Supporting Aircraft Certification.* AIAA R-154-202x (to be published).
7. https://rgl.faa.gov/Regulatory_and_Guidance_Library/rgFAR.nsf/0/701A79191CD71B1286257DC7005718DA?OpenDocument&Highlight=25.143
8. Duke, E. L., Jones, F. P., and Roncoli, R. B., "Development and Flight Test of an Experimental Maneuver Autopilot for a Highly Maneuverable Aircraft", NASA TP-2618 September 1986.

9. Pembery, M., et al., "Electric Motor Powered Aero-engine Simulator – EMPAS", RAeS Aerodynamics Conference, July 2018.
10. Jorgens, C., "Electric Motor Powered Aero-Engine Simulator (EMPAS): Technology Validation Test Summary and Results", to be presented at AIAA Aviation, Washington, DC, June 2021.
11. Jordan, T. L., "Airborne Subscale Transport Aircraft Research Testbed - Aircraft Model Development", AIAA Paper 2005-6432, AIAA Guidance, Navigation, and Control Conference and Exhibit, San Francisco, CA, August 2005.
12. Ryan, J. J., Bosworth, J. T., Burken, J. J., and Suh, P. M., "Current and Future Research in Active Control of Lightweight, Flexible Structures Using the X-56Aircraft", AIAA Paper 2014-0597, 52nd Aerospace Sciences Meeting, January 2014, National Harbor, MD, https://doi.org/10.2514/6.2014-0597.
13. Duraisamy, K., Spalart, P., and Rumsey, C. L., "Directions of Turbulence Modeling Research in Aeronautics", NASA TM 2017–219682, November 2017.
14. Meneveau, C., and Lund, T. S. "The dynamic Smagorinsky Model and Scale-dependent Coefficients in the Viscous Range of Turbulence", Physics of fluids, 9(12), 3932-3934, 1997.
15. Moin, P. and Verzicco, R., "On the Suitability of Second-order Accurate Discretizations for Turbulent Flow Simulations," European Journal of Mechanics-B/Fluids, Vol. 55, 2016, pp. 242-245.
16. Wang, Z. J., Fidkowski, K., Abgrall, R., Bassi, F., Caraeni, D., Cary, A., Deconinck, H., Hartmann, R., Hillewaert, K.,Huynh, H. T., et al., "High-order CFD methods: Current Status and Perspective", International Journal for Numerical Methods in Fluids, Vol. 72, No. 8, 2013, pp. 811-845.
17. Chen, S. and Doolen, G. D., "Lattice Boltzmann Method for Fluid Flows", Annual Review of Fluid Mechanics, Vol. 30, 1998, pp. 329-364.
18. Langtry, R. and Menter, F., "Transition Modeling for General CFD Applications in Aeronautics AIAA Paper 2005-522, presented at the 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno NV, January 2005. https://doi.org/10.2514/6.2005-522.
19. Coder, J., "Further Development of the Amplification Factor Transport Transition Model for Aerodynamic Flows", AIAA Paper 2019-0039, AIAA SciTech 2019 Conference, January 2019.
20. Anand, M. S., Laskowski, G. M., Suder, K. L., Medic, G., and Paliath, U., "Vision 2030 Aircraft Propulsion Grand Challenge Problem: Full-engine CFD Simulations with High Geometric Fidelity and Physics Accuracy", to be published at AIAA SciTech, 2021.
21. Leopold, D., "Flight in Icing Regulatory Evolution and the Influence on Aircraft Design", SAE Technical Paper 2019-01-1958, 2019, https://doi.org/10.4271/2019-01-1958.
22. Lynch, F.T., and Khodadoust, A., "Effects of Ice Accretions on Aircraft Aerodynamics," Progress in Aerospace Sciences, Vol. 37, No. 8, 2001, pp. 669–767.
23. Wright, W. B., "Validation Process for LEWICE by Use of a Navier-Stokes Solver", AIAA-2016-4349, June 2016, https://doi.org/10.2514/6.2016-4349.
24. Alauzet, F., Loseille, A., Marcum, D., and Michal, T. R., "Assessment of Anisotropic Mesh Adaptation for High-Lift Prediction of the HL-CRM configuration", AIAA Paper 2017-3300, AIAA Aviation, June 2017. https://doi.org/10.2514/6.2017-3300.
25. Park, M. A., Barral, N., Ibanez, D., Kamenetskiy, D. S. Krakos, J. A., Michal, T. R, and Loseille, A., "Unstructured Grid Adaptation and Solver Technology for Turbulent Flows", AIAA Paper 2018-1103, AIAA Scitech, January 2018. https://doi.org/10.2514/6.2018-1103.
26. Sankaran, V.,Wissink, A. , Datta, A., Sitaraman, J., Jayaraman, B., Potsdam, M., Katz, A., Kamkar, S., Roget, B., Mavriplis, D., Saberi, H., Chen, W. B., Johnson, W., and Strawn, R., "Overview of the Helios Version 2.0 Computational Platform for Rotorcraft Simulations", AIAA Paper 2011-1105, 49th AIAA Aerospace Sciences Meeting and Exhibit, Orlando FL, January 4-7, 2011.
27. Brazell, M., Kirby, A. and Mavriplis, D., "A High-order Discontinuous-Galerkin Octree-Based AMR Solver for Overset Simulations", AIAA Paper 2017-3944, 23rd AIAA Computational Fluid Dynamics Conference, Denver CO, June 2017. https://doi.org/10.2514/6.2017-3944.
28. Haimes, R. and Dannenhoffer, J., "EGADSlite: A Lightweight Geometry Kernel for HPC", AIAA Paper 2018-1401, AIAA SciTech, January 2018. https://doi.org/10.2514/6.2018-1401.

29. Carpenter, M., Fisher, T., Nielsen, E., Parsani, M., Svard, M., and Yamaleev, N., "Entropy Stable Summation-by-parts Formulations for Compressible Computational Fluid Dynamics", Handbook of Numerical Analysis, Vol. 17, Elsevier, 2016, pp.495-524.
30. Murman, S. M., Diosady, L., Garai, A., and Ceze, M., "A Space-time Discontinuous-Galerkin Approach for Separated Flows", AIAA Paper 2016-1059, AIAA SciTech, January 2016. https://doi.org/10.2514/6.2016-1059.
31. Johnson, W., "A History of Rotorcraft Comprehensive Analysis", NASA TP-2012-216012, April 2012.
32. Diosady, L. T. and Murman, S. M., "Scalable Tensor-Product Preconditioners for High-Order Finite-Element Methods", Journal of Computational Physics, Vol 394, Pages 759-776, October 2019. https://doi.org/10.1016/j.jcp.2019.04.047.
33. Mavriplis, D., "Adjoint Methods for Uncertainty Quantification", 38th Advanced Computational Fluid Dynamics. Adjoint methods and their application in Computational Fluid Dynamics, VKI Lecture Notes, Von Karman Institute, Rhode St Genese, Belgium, October 2015.
34. Wang, Q., "Forward and Adjoint Sensitivity Computation of Chaotic Dynamical Systems", Journal of Computational Physics, Volume 235, pages 1-13, February 2013. https://doi.org/10.1016/j.jcp.2012.09.007.
35. Flynt, B. and Mavriplis, D., "Optimal Error Control using Discrete Adjoint Error Estimates in Unsteady Flow Problems", AIAA Paper 2014-1434, 52nd AIAA Aerospace Sciences Conference, National Harbor, MD, January 2014.
36. Giles, M., & Süli, E., Adjoint methods for PDEs: A Posteriori Error Analysis and Postprocessing by Duality. Acta Numerica, 11, 145-236, 2002. https://doi.org/10.1017/S096249290200003.
37. Venditti, D. and Darmofal, D., "Anisotropic Grid Adaptation for Functional Outputs: Application to Two-dimensional Viscous Flows", Journal of Computational Physics, Volume 176, Issue 1, pages 40-69, February 2002. https://doi.org/10.1006/jcph.2001.6967.
38. Gray, J. S., Hwang, J. T., Martins, J. R. R. A., Moore, K. T., and Naylor, B. A., "OpenMDAO: An Open-source Framework for Multidisciplinary Design, Analysis, and Optimization","Structural and Multidisciplinary Optimization, Vol. 59, No. 4, April 2019, pp. 1075-1104.
39. "Multiphysics Simulations: Challenges and Opportunities", Argonne National Lab Report ANL/MCS-TM-321, Report from Workshop sponsored by the Institute for Computing in Science (ICiS), Park City, Utah, June-August, 2011.
40. Mousseau, V. A., Knoll, D. A., and Rider, W. J., "Physics-based Preconditioning and the Newton-Krylov Method for Non-equilibrium Radiation Diffusion", Journal of Computational Physics, Vol. 160, No. 2, 2000, pp. 743-765.
41. Zhang, Z. J. and Zingg, D. W., "Efficient Monolithic Solution Algorithm for High-Fidelity Aerostructural Analysis and Optimization," AIAA Journal, Vol. 56, No. 3, March 2018, https://doi.org/10.2514/1.J056163.
42. Schaefer, J., Romero, V., Shafer, S., Leyde, B., and Denham, C., "Approaches for Quantifying Uncertainties in Computational Modeling for Aerospace Applications", AIAA Paper 2020-1520, AIAA SciTech, January 2020; https://doi.org/10.2514/6.2020-1520.
43. Lacy, D. S. and Sclafani, A. J., "Development of the High Lift Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports," AIAA-2016-0308, 54th AIAA Aerospace Sciences Meeting, San Diego, California, USA, 4-8 January 2016.
44. Lacy, D. S., and Clark, A. M., "Definition of Initial Landing and Takeoff Reference Configurations for the High Lift Common Research Model (CRM-HL)", AIAA-2020-2771, AIAA Aviation, 2020, https://doi.org/10.2514/6.2020-2771.
45. Clark, A. M., Slotnick, J. P., Taylor, N., and Rumsey, C. L., "Requirements and Challenges for CFD Validation within the High-Lift Common Research Model Ecosystem", AIAA-2020-2772, AIAA Aviation, 2020, https://doi.org/10.2514/6.2020-2772.
46. Levy, D., et al., "Data Summary from the First AIAA Computational Fluid Dynamics Drag Prediction Workshop," Journal of Aircraft, Vol.40, No.5, 2003, pp.875–882. doi: https://doi.org/10.2514/2.6877.

47. https://hiliftpw.larc.nasa.gov
48. Rumsey, C. L., Slotnick, J. P., Long, M., Stuever, R. A., and Wayman, T. R., "Summary of the First AIAA CFD High-Lift Prediction Workshop," Journal of Aircraft, Vol. 48, No. 6, 2011, pp. 2068-2079, doi: https://doi.org/10.2514/1.C031447.
49. Rumsey, C. L. and Slotnick, J. P., "Overview and Summary of the Second AIAA High-Lift Prediction Workshop," Journal of Aircraft, Vol. 52, No. 4, 2015, pp. 1006-1025, doi: https://doi.org/10.2514/1.C032864.
50. Rumsey, C. L., Slotnick, J. P., and Sclafani, A. J., "Overview and Summary of the Third AIAA High-Lift Prediction Workshop," Journal of Aircraft, Vol. 56, No. 2, 2019, pp. 621-644, doi: https://doi.org/10.2514/1.C034940.
51. Evans, A., Lacy, D., Smith, I., and Rivers, M., "Test Summary of the NASA Semi-Span High-Lift Common Research Model at the QinetiQ 5-Metre Low-Speed Wind Tunnel", AIAA-2020-2770, AIAA Aviation Forum, 2020 (virtual event), doi: https://doi.org/10.2514/6.2020-2770.
52. http://www.gmgworkshop.com/index.shtml
53. Chawner, J. R., and Taylor, N. J., "Progress in Geometry Modeling and Mesh Generation Toward the CFD Vision 2030", AIAA-2019-2945, AIAA Aviation Forum, Dallas, TX, June 2019, doi: https://doi.org/10.2514/6.2019-2945.
54. Chawner, J. R., Michal, T., Slotnick, J. P., and Rumsey, C. L., "Summary of the 1st AIAA Geometry and Mesh Generation Workshop (GMGW-1) and Future Plans", AIAA-2018-0128, AIAA Aerospace Sciences Meeting, Kissimmee, FL, 2018, doi: https://doi.org/10.2514/6.2018-0128.
55. Woeber, C. D., Masters, J. S., and McDaniel, D. R., "Summary of Exascale and Remeshing Efforts for the Second Geometry and Mesh Generation Workshop", AIAA-2019-3458, AIAA Aviation Forum, Dallas, TX, 2019, doi: https://doi.org/10.2514/6.2019-3458.
56. http://www.gmgworkshop.com/gmgw25.shtml
57. https://paw.larc.nasa.gov/
58. Delot, A., and Scharnhorst, R., "A Comparison of Several CFD Codes with Experimental Data in a Diffusing S-Duct," AIAA 2013-3796, 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, San Jose, CA, 2013, doi: https://doi.org/10.2514/6.2013-3796.
59. Scharnhorst, R., and Delot, A., "Computational and Experimental Results for Flows in a Diffusing S-Duct without and with Flow Control Devices," AIAA 2015-3964, 51st AIAA/SAE/ASEE Joint Propulsion Conference, Orlando, FL, 2015, doi: https://doi.org/10.2514/6.2015-3964.
60. Winkler, C. and Davis, Z. S., "Summary of the 3rd Propulsion Aerodynamics Workshop: S-duct Results," AIAA 2017-4912, 53rd AIAA/SAE/ASEE Joint Propulsion Conference, Atlanta, GA, 2017, doi: https://doi.org/10.2514/6.2017-4912.
61. Babcock, D. A., Tobaldini, L., and Davis, Z. S., "Summary of the 4th Propulsion Aerodynamics Workshop: S-duct Results", AIAA-2019-3845, AIAA Propulsion and Energy Forum, Indianapolis, IN, 2019, doi: https://doi.org/10.2514/6.2019-3845.
62. https://c3.nasa.gov/dashlink/static/media/other/AEPW.htm
63. Heeg, J. et al., "Overview of the Aeroelastic Prediction Workshop", AIAA-2013-0783, 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine, TX, 2013, doi: https://doi.org/10.2514/6.2013-783.
64. Heeg, J. et al., "Overview and Data Comparisons from the 2nd Aeroelastic Prediction Workshop", AIAA-2016-3121, 34th AIAA Applied Aerodynamics Conference, Washington D.C., 2016, doi: https://doi.org/10.2514/6.2016-3121.
65. http://folk.ntnu.no/richahan/IPW
66. Fremaux, M. and Hall, R. M., "COMSAC: Computational Methods for Stability and Control", NASA CP 2004-213028/PT2, April 2004.
67. Cartieri, A., "Experimental Investigations on the Common Research Model at ONERA-S2MA", AIAA-2020-0779, AIAA SciTech Forum, Orlando, FL, January, 2020, https://doi.org/10.2514/6.2020-0779.

68. Yip. L., Vijgen, P., Hardin J. and Van Dam, C., “Subsonic High-lift Flight Research on the NASA Transport System Research Vehicle (TSRV)”, AIAA Paper 1992-4103, 6th AIAA Biennial Flight Test Conference, June 1992; https://doi.org/10.2514/6.1992-4103.
69. Rizzi, A. and Luckring, J., “What Was Learned in Predicting Slender Airframe Aerodynamics with the F-16XL Aircraft”, Journal of AircraftVolume 54, Issue 2, March 2016, https://doi.org/10.2514/1.C033569.
70. Stefan Goertz, et al., “Overview of Collaborative Multi-Fidelity Multidisciplinary Design Optimization Activities in the DLR Project VicToria”, AIAA Paper 2020-3167, AIAA Aviation 2020, June 2020, https://doi.org/10.2514/6.2020-3167.
71. INCITE: Leading the Way in Scientific Computing; https://www.doeleadershipcomputing.org.