Vision 2030 Aircraft Propulsion Grand Challenge Problem: Full-engine CFD Simulations with High Geometric Fidelity and Physics Accuracy

M. S. Anand[1]
Rolls-Royce, Indianapolis, IN

Gorazd Medic[2]
Raytheon Technologies Research Center, East Hartford, CT

Umesh Paliath[3]
GE Global Research Center, Niskayuna, NY

Kenneth L. Suder[4]
NASA Glenn Research Center, Cleveland, OH

Mujeeb R. Malik[5]
NASA Langley Research Center, Langley, VA

Gregory M. Laskowski[6]
Dassault Systemes, Waltham, MA

In 2014 NASA published the outcome of the 2030 CFD (Computational Fluid Dynamics) Vision study: "CFD Vision 2030: A path to Revolutionary Computational Aerosciences" (Ref. [1]). The study provided a comprehensive review of the state of the art of CFD in 2014 for aerospace applications including, but not limited to, numerical algorithms, physics models, MDAO (Multidisciplinary Design Analysis and Optimization) and HPC (High Performance Computing) hardware. The study also proposed four conceptual ideas of Grand Challenge problems that would build on and benefit from advances outlined in the roadmap. The proposed challenges were meant to foster more detailed descriptions of grand challenge problems for specific disciplines. One of the proposed challenges was in the gas turbine propulsion area, focusing on transient full engine simulations. The current paper addresses detailed technical aspects of that challenge, and proposes a plan to approach it in a gradual manner, which includes high fidelity modeling of components, component coupling, and targeted experimental campaigns relying on common research models.

I. Introduction

In 2014 NASA published the outcome of the 2030 CFD Vision study: "CFD Vision 2030: A path to Revolutionary Computational Aerosciences" [1]. The study provided a comprehensive review of the state of the art of CFD in 2014 for aerospace applications including, but not limited to, numerical algorithms, physics models, MDAO and HPC

[1] Associate Fellow, Sub-Systems and Components Engineering, AIAA Associate Fellow
[2] Associate Director, Aerodynamics, AIAA Member
[3] Technology Manager, Aerodynamics & CFD/Methods, AIAA Member
[4] Senior Technologist, Air Breathing Propulsion Analysis
[5] Senior Aerodynamicist, Computational AeroSciences Branch, AIAA Fellow
[6] Director of Fluid Mechanics, SIMULIA, AIAA Member

hardware. The study also proposed four conceptual ideas of Grand Challenge problems that would benefit from advances outlined in the roadmap including "off-design turbofan engine transient simulation". The proposed challenges served as a starting point for more detailed problem descriptions that would benefit from advances in simulation. The objective of this paper is to build upon the NASA 2030 CFD Vision study and provide a detailed overview of what needs to take place to enable accurate and efficient simulation of flow through an aircraft engine at off-design condition for transient operation. Execution of the proposed roadmap would significantly advance aircraft engine development by reducing program cost, reducing program development timelines and enabling design objectives associated with SFC (Specific Fuel Consumption), emissions, noise, weight and durability.

Modern turbofan engines are the workhorse for both narrow body and wide body commercial airlines due to their efficiency and reliability. Air enters the engine through the fan. A portion of the flow enters the core of the engine with the remaining diverted through the bypass duct to improve propulsive efficiency. The flow that enters the core flows through the multi-stage compressor, which increases the total pressure and total temperature. Heat is then added in the combustor, after which the flow is expanded through the turbine, which drives both the fan and compressor. The remaining enthalpy in the flow is converted to thrust as the flow exits the nozzle. There is considerable interest in, and ongoing research into, innovative and disruptive technologies for aircraft propulsion systems, including hybrid and fully electric propulsion systems [2]. Nonetheless, it is expected that for large civil transport aircraft the gas turbine engine will continue to be the main power source for the propulsion system for the foreseeable future [3]. Design of modern turbofan engines has evolved over several generations with each generation receiving increasing benefit from computer simulation. As designs evolve, increasingly accurate and efficient simulation capabilities are required to keep cost down and unlock performance opportunities for new technologies. The aforementioned modules, namely the fan, compressor, combustor, and turbine each present unique design and simulation challenges. Each module is typically designed separately subject to systems level design requirements, with testing in component rigs used to iterate and optimize the module design. Ultimately, all the components are assembled and an engine test is performed. Accurate model of flow through an entire engine requires accurate simulation capabilities for each module. Pioneering efforts to perform full engine simulations were undertaken in the early 2000s for PW6000 [4] and GE90 [5]. Specifically, the PW6000 efforts under the Stanford ASCI full engine simulation program made strides to address the coupling of component simulations. For example the compressor, combustor, and turbine were coupled by using discrete interfaces between LES codes used for the combustor and RANS (Reynolds-averaged Navier Stokes) solvers used for the compressor and turbine [6]. These simulations passed information between components using newly developed coupling platform [7], but the ability to account for the unsteady interactions between components was somewhat limited. In addition, these full engine simulations were often limited to simplified component configurations and were only conducted at the design point operating condition, where the interactions between the components are most likely to be minimal, since each component is very unlikely to have massive separations and unsteady secondary flows.

For full engine simulations to truly complement engine design and testing at OEMs, higher fidelity unsteady component simulations inclusive of component interactions, as well as higher level of geometrical detail in each component, will be required to capture performance at design and off-design conditions. At off-design operating conditions, the physics of boundary layer transition, separation, and re-attachment becomes especially complex due to the 3D secondary and turbulent flows, clearly pointing to the need for eddy resolving simulations, for example, WMLES (wall-modeled LES) or hybrid RANS/LES. In addition, as the gas turbine engine designs continue to evolve to seek more performance improvements, blade loadings, pressures, and temperatures are being pushed far beyond today's state-of-the-art products, resulting in reduced operability margins. To accurately predict these operability limits even higher fidelity representation of components will be needed, and engine transient simulations requiring long integration times will need to be conducted.

To enable routine, rapid, and accurate modeling of the flow through an entire engine, a phased and building block approach is proposed to help establish advances in CFD algorithms, physics models, and computing infrastructure (Figure 1). Figure 1 outlines a phased approach defining intermediate milestones for various swim lanes at the module or component level. In order to demonstrate improvements in CFD accuracy, high quality experimental data will be required to benchmark against. Ideally, Common Research Models (CRM) for each of the swim-lanes would be defined, designed, developed and executed with collaboration from industry, academia and government to provide experimental data with sufficient fidelity to demonstrate improvements in CFD models and algorithms. The final objective of the grand challenge is to conduct a full engine simulation from model build to processed results within one week, in a manner that has sufficient geometric fidelity and physics accuracy to reduce/complement engine testing and meet performance, operability, emissions, and durability accuracy metrics.

The advances in modsim (modeling and simulation) will result in significant time and cost benefits at the component level as well as the full engine level through elimination or significant reduction in testing. The ability to accurately explore the design space will result in more optimized designs that meet stringent emissions regulations in addition to delivering significant reductions in SFC (specific fuel consumption) with enormous savings in operating costs. The ability of CFD models to predict engine performance and operability during transients will eventually enable aircraft certification by analysis. It is estimated that all these benefits will add up to billions of dollars (U.S.) of savings per year to the aviation industry. Similar estimates of cost savings due to advances in modsim have been made by other authors as well [8].

The paper is organized as follows. The propulsion systems components of compressor, combustor, and turbine are described in Sections II, III, and IV, respectively. Each section is designed to provide current status, opportunities, gaps and challenges, and testing and validation needs. Integration and secondary flows are discussed in Section V, followed by the full engine simulation in Section VI. Conclusions are summarized in section 7. A multi-stage high-pressure compressor rig is presented in the Appendix as an example of a common research model for validation experiments.

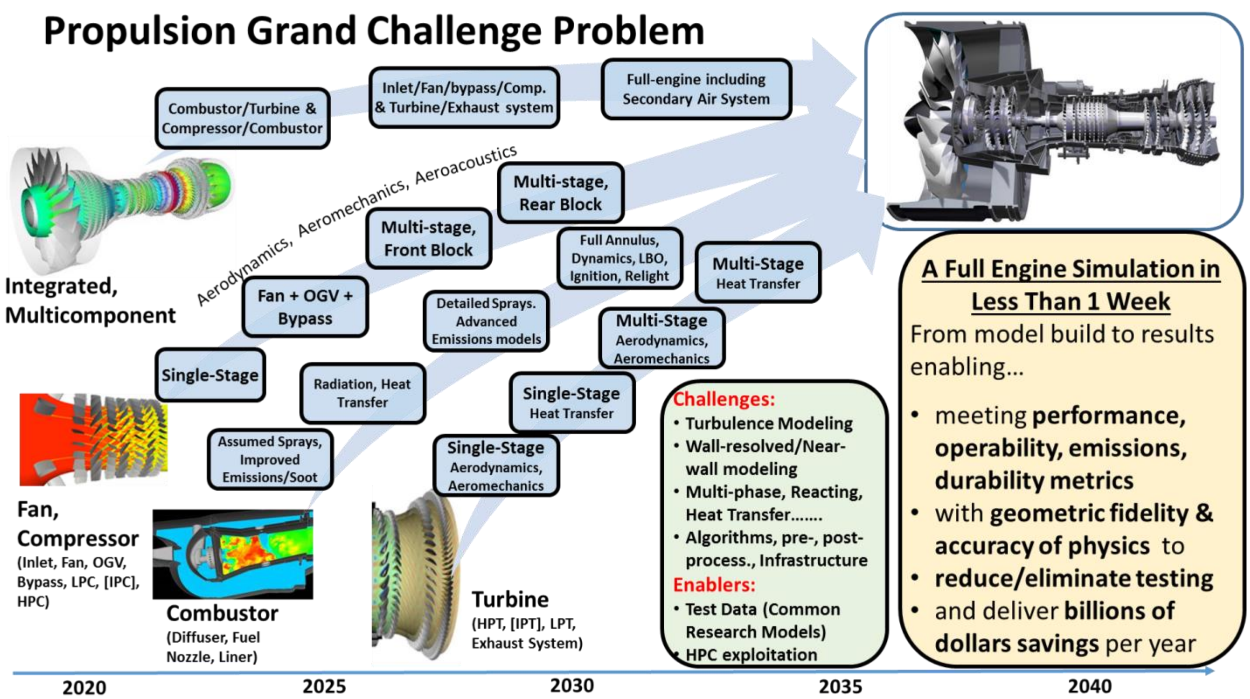

Fig. 1 Aircraft propulsion grand challenge problem.

II. Compression System

The compression systems typically comprises of a fan, a low-pressure, and a high-pressure compressor. Its role in the engine is twofold. The fan bypass stream provides the bulk of the engine thrust with high propulsive efficiency. The core stream efficiently compresses the air for the combustor, while maintaining good operability over a range of conditions. Each of the modules encounters specific modeling challenges.

As a general rule the references in this section as well as the rest of the paper are meant to be illustrative examples and are by no means exhaustive or imply preferences.

A. Current Status

Fan and compressor present many challenges for the state-of-the-art CFD methodology. These include complex geometry including leakage flows, tip-gaps, seals, etc.; multiple frames of reference to handle rotating components; compressible flow associated challenges including shock boundary layer interaction, shock/shock interaction, shock/tip vortex interaction, and wide range of Mach numbers, which can present numerical challenges. Turbulence physics challenges include laminar-to-turbulent transition, wake formation, and evolution, secondary flows. There are also inter-row interactions between rotating blades and stationary vanes, which also include various forms of fluid-structure interactions. These challenges are further exacerbated by the need to design the fan and the compressor for a wide range of conditions to ensure performance, operability, and structural integrity of the design. CFD predictions at off-design conditions are usually substantially less accurate than those conducted for the compressor design point. Accurate and routine predictions of the stability margin of a compressor across the entire map (i.e., across multiple compressor rotating speeds) is still out of reach for today's state-of-the-art CFD methodology, as applied in propulsion industry. Reliable predictions of flutter margin for fans is another outstanding challenge, as is the routine prediction of vibratory stress throughout the compression system. Finally, predicting fan performance and operability in distortion, such as generated by crosswind conditions, still require further improvements in CFD methodology.

Future vehicles will be requiring smaller, higher power density core engines to improve thermal efficiency, enable higher bypass ratio turbofans for higher propulsive efficiency, and to provide power and propulsion for hybrid electric/gas turbine propulsion systems [1]. Propulsion/airframe integration for such novel vehicles will present new challenges for fan designers, stemming from more complex interactions throughout the fan stream – inlet distortion being only the most obvious one. Full wheel (full 360-degree) unsteady models have already been routinely applied for fan stream CFD, but further improvements are needed to capture all the geometrical and physical complexity.

At the same time, the HPC (high-pressure compressor) (some designs have an intermediate pressure and a high pressure sections) represents the heart of the core engine and if it does not provide the performance, operability, and flow splits through the engine and the combustor, high-pressure and low-pressure turbine (some designs have an intermediate pressure turbine as well) all suffer and cannot make-up for the deficiencies in the HPC. The HPC of a single aisle aircraft with >100 passengers will require an HPC with approximately 10 stages (10 rotating blade rows and 10 stationary stator blade rows) plus an inlet and exit guide vane. Furthermore, the blade count in each blade row of an HPC is typically different and often a prime number for aeromechanic stability. Therefore, very costly full wheel HPC models containing thousands of blade/vane passages may be required to get accurate results for URANS (unsteady RANS) simulations.

Given the complexity of the compression system modules, most new designs are derivatives from existing designs. For example, the compressor maps of existing HPC's are validated with experimental measurements and the design tools and procedures are updated to better estimate the losses throughout the system. For next generation HPC's these compressor maps are scaled via proprietary methodologies to generate conceptual designs. A sequence of design, pre-test analysis, build, test, post-test analysis, design tools/model updates, followed by another cycle starting with design is utilized to close on the final design. Component maps are critical to this process and are used to perform trades across all of the engine components as well as for propulsion airframe integration. In a HPC that typically has 7-10 stages with variable geometry and various bleeds for the vehicle environmental control services, combustor and turbine cooling, and engine operability; this process is fairly complicated and time consuming. Each OEM may make different design trades in terms of number of HPC stages (constrained by blade loading), stage reaction, blade loading, pressure ratio, flow range, shrouded versus unshrouded stators, variable versus fixed stator, stator cavity design, stator cavity seal design, casing treatments, bleed configurations, etc.

State-of-the-art CFD in 2020 is routinely applied for prediction of fan and compressor performance maps. CFD methodology typically relies on RANS modeling that can be applied using steady simulations with reduced order models such as mixing planes at inter-row interfaces, or using unsteady simulations. URANS simulations often require simplification of the geometry, since resolving all the blade passages in a compressor using the full-wheel approach would be cost prohibitive, if applied to compute all the points of a compressor performance map. For fans, which typically contain two orders less blade/vane passages than, for example, high-pressure compressor, full wheel URANS is substantially less costly and has become more established, especially when considering flutter and impact of distortion. In general, RANS modeling itself is usually limited to eddy-viscosity models.

Predictions typically agree well with the data at design point, and they are usually less accurate at off-design conditions, especially for part power conditions (i.e., at speeds lower than the design speed). Accurate prediction of stall margin and the assessment of design changes aimed at enhancing the stability margin (such as casing treatment, for example) remain a big challenge [9]. CFD is typically not relied on in itself, but rather has to be used in conjunction with other information such as reduced order models, design experience, historical rules, and the like.

B. Opportunities

In recent years there has been a trend towards reducing some of the assumptions in the CFD methodology to try to improve the predictions of flow in fans and compressors. Increases in computing power are enabling more routine unsteady simulations, including full-wheel simulations, in particular for fans. These advances are also enabling an inclusion of much greater geometrical detail in the simulations, especially when it comes to end-wall features, bleeds, and leakages.

Eddy resolving simulation approaches, such as hybrid RANS/LES, have been assessed for improved prediction of end-wall features in smaller compressor configurations (i.e., containing less blade rows or stages) [10]. They have also been successfully applied for fans in distortion, providing improved prediction of turbulent mixing. In parallel, full wheel URANS simulations for full compressor configurations are now being enabled by development of new solvers that can harness the advances in computer hardware, such as GPUs [11].

All these advancements in computing power and modeling will have a beneficial impact on CFD predictions of the entire compression system, for performance, operability, and structural integrity. Some of the specific areas where the benefits of improved predictions will be most significant in the near term include fan performance and operability in distortion; compressor operability at higher blade loadings and relatively larger clearances (a feature inherent to high power density cores); better quantification of the impact of specific end-wall geometrical features on performance and operability enabling improved designs of such features.

In the context of full engine simulations, it is important to note the velocity profiles exiting the compressor and through the pre-diffuser in the combustor can have a significant influence on the flow distribution through the combustor liner and in turn on the performance of the combustor. Hence an accurate simulation of the compressor is the first step in the successful (accurate) simulation of the whole engine

C. Gaps and Challenges

Accurately predicting laminar-to-turbulent transition on fan blades, especially in the presence of manufacturing and assembly uncertainties, as well as inlet flow distortion, remains one of the critical challenges for the application of CFD methodology to the fan rotor and exit guide vane. While wall-resolved LES may be a good approach to better understand the associated flow physics, more practical methods will be needed, marrying RANS-based transition models, stability analysis tools, and eddy resolving simulation.

Other challenges concerning the fan center around fan flutter, fan noise, and high-fidelity bypass stream assessment. Interactions of the fan blade wakes with the fan exit guide vanes remains critical for noise predictions and further improvements are needed – eddy resolving simulation will play a critical role. Finally, high fidelity predictions of the flow throughout the entire bypass stream may unlock new design opportunities for new trades.

As elsewhere in turbomachinery components of the engine, CFD predictions of the mixing of wakes and distortion in multi-stage environment remain critical. It is well understood that RANS tends to under predict such mixing, thus resulting in more coherent wakes and distortion patterns. This is sometimes confounded by insufficient grid resolution, which in turn overly diffuses such features, resulting in a wide range of seemingly contradicting results. Eddy resolving simulations provide a path to improved prediction of turbulent mixing [12]. However, their efficient implementation for multi-stage configurations still needs to be developed – how best to develop grids to support resolving turbulent eddies where it matters without unnecessarily dramatically increasing the overall size of computational grids remains an area of active research.

Eddy resolving simulation and higher grid resolution will also help improve the prediction of the physics of end-wall features, such as corner roll-ups/separation and tip clearance flows. RANS typically over predicts the strength of such features, corner separation tends to be too large and the tip clearance vortex tends to persist too far downstream of the rotor blade. Similar to wake mixing, localized eddy resolving simulation may be sufficient to improve the predictions, however, careful consideration needs to be given to grid generation and targeted grid refinement.

Rapid and automated inclusion of small geometrical features at the end-walls will also benefit from improved gridding methodologies, whether they are based on unstructured meshes with adaptive mesh refinement, or multi-block structured grids with some sort of embedded localized refinement using oversetting techniques. Incorporating trenches, rubs, end-wall gaps, platform gaps, leakages, cavities and seals, and bleed ports into the CFD model will provide a better geometrical representation of the compression system, and will enable improved predictions, in particular at off-design conditions.

Finally, it is important to highlight that full-wheel unsteady simulations for HPCs (and LPCs), even as URANS, are still very large and expensive to be routinely executed in the design cycle. Such multi-stage configuration contain 1000s of blade passages, which would require CFD runs with 1000s of CPUs per run, and executing that is still far from being routine at OEMs. Enhancing high-performance computing infrastructure, both in terms of accessible hardware and software scalability, remains a necessary step for advancing CFD predictions of compression systems.

D. Testing and Validation Needs

Validating CFD for compression systems remains a challenge in itself – high-quality, detailed data is needed to assess the performance of CFD models. However, in order to get detailed information about the flow-field, more non-intrusive instrumentation needs to be used to minimize the impact of the instrumentation on the performance of the component itself. This is especially true for high-pressure compressor multi-stage experiments, where small changes in blockage can have a huge impact. Overall and per stage performance is usually reported, but the inter-stage profiles are relatively coarse and are not available throughout the machine, at all stations. Total pressure and temperature measurement are often reported at a handful of discrete locations and need to be processed very carefully so that can be interpreted against the results from CFD simulations. This is going to be even more challenging for small core compressors. At the same time, very little has been reported on evolution of turbulence levels in high-pressure compressors, and how that impacts the mixing. Finally, special care has to be given to measuring clearances, gaps, and any leakages in the system, since it is well understood that these have a large impact on operability, and to validate CFD model predictions accurate information about geometrical features and boundary conditions needs to be provided.

Highly instrumented, high-fidelity experimental rigs and experiments are expensive. An efficient approach is to establish common research configurations via consortia of OEMs (original equipment manufacturers), universities, and government labs to provide high-quality data made available widely for model validation. The Appendix provides further elaboration of such a proposal as an example that can be followed for other components' and multi-component experiments.

III. Combustor Module

The combustor module sits between the compressor and turbine. Its main components include the pre-diffuser, the main diffuser (typically a dump diffuser), the inner and outer annuli, the combustor liner (or flametube), and the FSNs (fuel spray nozzles). Typically, the combustor liner in modern gas turbines is a full annular liner with multiple FSNs equally distributed circumferentially. The combustor module is what supplies energy to the entire gas turbine. Fuel is burnt at high and nearly constant pressure in the module. The flow within the combustor is typically in the low Mach number regime (M<0.3). The primary heat addition and the maximum temperature of the working fluid (air) occur within the combustor. Military gas turbine engines, particularly for combat aircraft, will have afterburners and re-heat systems downstream of the turbine. While the discussion in this section focuses on the main combustor, similar considerations apply to afterburners as well except the flow in them ranges from subsonic to supersonic regimes. We will not discuss afterburners specifically although it would be within the scope of full-engine simulations.

A. Current Status

The simulation of the combustor module provides unique challenges not only due to chemical reactions and heat addition, but also because of interactions of multiple physical process occurring and multiple physical and time-scales in the combustor flow field [13]. The combustor flow field has a range of length and time scales generated by the geometry. The fuel preparation—fuel droplet formation, primary and secondary break-up, evaporation and transport of droplets—represent another range of scales. Mixing of fuel vapor with the airflow is a critical process to simulate/model. Combustion of the fuel-air mixture involves chemical reactions involving hundreds of chemical species and thousands of reactions that have a wide range of time scales. In general, the key reactions for heat release occur on time scales much smaller than flow timescales. An inevitable aspect of combustion of hydrocarbon fuels, especially liquid hydrocarbon fuels, is the accompanying production of CO_2 (carbon dioxide) considered a greenhouse gas, and pollutant emissions such as NOx (oxides of nitrogen), CO (carbon monoxide), soot, and other NvPM (non-volatile particulate matter). The formation and destruction/oxidation of these occur on different timescales and follow complex processes that need to be modeled. Other important physics that the simulations need to include are the radiation from the soot and high temperature gas as well as the convective heat transfer to the walls of the combustor liner that determine the wall temperatures and durability of the combustor liners.

Current simulations of gas turbine combustors, especially for the design of practical combustors by OEM's, typically employ a combination of RANS, URANS, and LES (large eddy simulations) to suit the purpose for which the simulations are being performed while balancing the need for fast turnaround times of these simulations. Typically low-Mach pressure-based numerical methods with extension for compressible flow (for acoustics and simulations through the turbine inlet and afterburners) are employed. They are more naturally suited for the low-Mach flow in the combustor and computationally efficient especially for RANS and URANS. However with more use of LES, the use of density-based high-Mach methods with preconditioning to handle low-Mach flows is quite prevalent though the computational time requirements for density-based methods still tend to be higher. The simulations typically encompass the full combustion system domain from compressor (outlet guide vane, or prediffuser) exit to turbine inlet (inlet guide vane), to avoid having to specify boundary conditions to the many inlets to the combustor liner although liner-only or liner and annuli simulations are frequently carried out. Generally most of the important geometric features are captured in the CFD model, though some simplifications are made. The simulations typically employ one of a variety of combustion models such as flamelet [14, 15] with detailed chemistry, eddy break-up [16] (less in use these days), eddy dissipation concept [17] with reduced chemical kinetics with corresponding ways of modeling the very important interactions between turbulence and chemistry. Lagrangian spray modeling (e.g., Ref [18], with empirical based primary [19] and secondary break-up models [20] with simplified and tractable evaporation models [21, 22] are used. The fuel spray boundary conditions for drop size distributions and velocities are typically specified and empirical based. NOx production is typically dominated by the temperature of the mixture (thermal NOx) and is modeled by the Zeldovich mechanism [23] and soot is typically modeled by relatively simple two equation models (e.g., Ref. [24]).

Despite many limitations and missing many details of the physicochemical processes in the modeling, the current simulations are pretty good at predicting some of the basic performance characteristics of the combustor [25]. Generally, depending on how accurately the boundary conditions can be specified, the all-important flow-splits through the various features of the combustor, heat release, and the exit temperature traverse in terms of circumferentially averaged radial profiles and 2D temperature contours are well predicted with a reasonably high level of confidence. The trends of NOx with both operating conditions and changes to geometry, both qualitatively (i.e., relative levels) and quantitatively, are usually well captured. The trends for CO are also well captured qualitatively though good quantitative predictions can be obtained depending on the complexity of the chemistry used. Current status of soot predictions is generally not very reliable to make design decisions though they can provide some limited guidance on trends. Thermal modeling for combustor walls is done by applying boundary conditions from CFD (radiation, near-wall temperatures, and heat transfer coefficients) to a separate finite-element based thermal model, though conjugate heat transfer approaches are being pursued. Key operability metrics include LBO (lean blow-out), ignition and altitude relight, and combustion stability/noise. Current methods have shown to be able to predict LBO [26] but it still not clear which combination of models can reliably predict LBO for design use. Methods for predicting ignition and altitude relight, typically based on calculating ignition probabilities, have been developed [27, 28] but they need to be further exercised. Methods for calculating combustion noise employ the concept of flame transfer functions either experimentally derived or through targeted simulations [29] that are used in simplified acoustic models [30]. A full unsteady simulation (using say LES) of the full annular combustor would be required to fully resolve the combustion instabilities and acoustics in the combustor. This is currently technically possible [31], but cannot be done routinely in a design cycle due the turnaround time and computational resources needed.

B. Opportunities and Benefits

Even with limitations of the current models to predict all the quantities of interest in a combustor, the routine use of combustion CFD in combustor design has resulted in demonstrably reduced number of rig tests over the past two decades. It has also enabled greater exploration of the design space to lead to more optimized designs. Continued improvements in the accuracy and speed of combustion CFD will result in further reductions of tests and savings of hundreds of millions of dollars in combustor testing across all the OEMs.

Regulatory requirements imposed by international bodies such as the ICAO's (International Civil Aviation Organization's) CAEP (Committee on Aviation and Environmental Protection) and the EASA (European Union Aviation Safety Agency) are getting progressively stringent on pollutant emissions including NvPM. Since the efforts to reduce one or more of these emissions in the combustor tend to increase one or more of the other emissions or adversely affect operability, it is becoming increasingly difficult to arrive at optimum combustor design that maintains its high efficiency and meets other performance requirements while also meets all the emissions regulations using traditional empirical design rules. Improvements in the accuracy and speed of CFD simulations to predict all the emissions and other operating characteristics hold the key to meeting these requirements in a cost-effective manner.

In the context of full engine simulations, it is important to note that boundary conditions for the turbine will come from the combustor simulation and the accuracy of combustor simulations has a significant impact on the accuracy of the turbine simulations. The temperature profile and pattern, not only in the bulk of the flow but especially the temperature near the end-walls (hub and tip) as well as the turbulence and unsteadiness from the combustor will have a major impact on the accuracy of turbine simulations and prediction of overall engine performance.

Enablement of fast construction of the geometry and grids or methods employing grid-less techniques or automatic generation of grids and adaptive mesh refinements will greatly reduce the burden on the users and the turnaround time of computations.

C. Gaps and Challenges

Limitations of current turbulence models are well known, especially for swirling and recirculating flows in combustors. Additionally, flows in the combustors exhibit unsteadiness due to large scale structures and possible precessing vortex cores and unsteadiness in the inlet flows. It is expected that LES, that overcomes many of these issues, will be more routinely employed in the design cycle with increasing computational speeds and resources. LES still has similar modeling issues as RANS such as sub-grid models for turbulence chemistry interactions, scalar mixing, and interactions of spray droplets with the flow. Models exist for these phenomena but are mostly extensions of the models used in RANS. Further developments of these models that exploit the advantages of and are more suitable for, LES are being developed.

Many of the current limitations and gaps in the modeling liquid sprays, emissions, heat transfer, and operability have been identified in the Current Status portion of Section III. Computationally tractable models for spray formation (including fuel filming, primary, and secondary breakup) need to replace the current practice of empirically specifying spray boundary conditions. Methods such as VoF (Volume of Fluid), CLS (Conserved Level-Set) [32], SPH (Smooth Particle Hydrodynamics) [33], and other hybrid methods are being developed and demonstrated. Though they are very expensive computationally, it is hoped that a priori simulations can be performed to derive boundary conditions for the actual combustor simulations and/or to derive general phenomenological or statistic based break up models, which can be used in combination with RANS and LES. Efficient parallelization of the Lagrangian spray tracking is another area for improvement. Turbulent combustion models such as Conditional Moment Closure [34] Artificially Thickened Flame [35] are being used and/or further developed. Turbulence-chemistry interactions are fundamentally and exactly handled by PDF (probability density function) methods [36] and have been around for a long time, but are not in routine use since they are still computational very expensive. A more tractable PDF-based approach is the stochastic fields method [37] and it is likely to be more routinely used with increasing availability of computer resources. Soot modeling is especially challenging since a lot of soot (two to three orders of magnitude more soot than what exits the combustor) is produced in the primary zone in an intermittent fashion, which is a challenge to model, but most of it is oxidized in the dilution zone downstream of the primary zone. It is very important to get the difference between these two large numbers to get an accurate prediction of combustor exit soot concentrations. This is an area of active research and several variations of advanced soot models are being worked on [38, 39]. As mentioned previously the tradeoffs between operability and emissions need to be accurately modeled to design optimal combustors with reduced testing. Heat loads to the combustor walls increase significantly with increased operating pressure and aggressive cycles that modern day gas turbines are continuously moving towards. Accurate thermal modeling to determine combustor wall temperatures is crucial to determining the durability and life of the combustor.

In addition to the advances in the modeling of the physicochemical processes, it is equally important to make advances in the numerical algorithms and approaches to exploit exascale and beyond computing platforms. Given the computational effort that will be needed for this full-engine simulation grand challenge problem, this aspect may be the bottleneck that may need to be addressed aggressively.

D. Test and Validation Needs

As with any model development, validation data are very important, especially at conditions relevant for the particular practical device. Given that gas turbine combustors operate at very high pressures and temperatures (the highest within the engine) the experiments should be conducted at the relevant operating conditions to the extent possible. This is especially true for certain quantities such as soot, radiation, spray-formation and evaporation (potentially under supercritical conditions), combustion instabilities, etc., involving processes that are highly dependent on operating conditions. Several such efforts are underway with pressures above atmospheric, though not yet close to the actual operating conditions. Altitude relight experiments need to be conducted at sub-atmospheric pressures and temperatures.

Although testing of a practical gas turbine combustor is much less expensive than say a compressor or turbine test, the data obtained, though valuable for overall validation, are usually at the exit of the combustor and do not provide vital information about processes going on in the primary zone and vicinity of fuel injectors. It is usually a problem of access, particularly optical access needed for advanced diagnostics, for experiments to be conducted at high pressures and temperatures. It is important to construct experiments that can be conducted at high pressures and temperatures that have the needed optical access. These experiments will be complex, time-consuming and expensive. The idea of common research models and efforts driven by consortia of OEMs, universities, and government laboratories would be the optimal way forward. For some operability data we may need to rely on OEM data on actual combustors.

IV. Turbine Module

The turbine module sits downstream of the combustor and upstream of the exhaust nozzle. The role of the turbine is to extract work from the flow to drive the fan and compressor via a shaft along the centerline of the engine. The turbine is typically broken into 2-3 multi-stage modules across the HPT and LPT (high pressure turbine and low pressure turbine, respectively) flow path, each has unique challenges due to Reynolds, Mach and temperature range. While the HPT is typically dominated by high temperature flows, hot streaks, shock – boundary layer interactions and significant secondary flows, the LPT is more sensitive to variations in Reynolds number, flow transition and separation, and typically has more stages.

A. Current Status

The high level design objectives of the turbine is to maximize efficiency and life while minimizing weight subject to manufacturing and cost constraints to meet system level requirements. CFD can play a role to evaluate the trade study to meet system level requirements. The design space can be large consisting of 100s of design parameters thus necessitating the use of steady state RANS early in the design phase transitioning to more accurate and more computationally intense transient URANS later in the design phase.

Flow unsteadiness in turbomachinery is both stochastic, i.e., turbulence driven, and deterministic, i.e., stator-rotor periodic interaction driven. The interaction of the two frequency ranges poses a formidable task both in case one decides to model or to resolve this process. Design iterations are generally accomplished by modeling all scales of turbulence with RANS or URANS, the latter being used when capturing the large-scale unsteady interaction between blade rows is deemed necessary. The design process requires modeling of 100s of blades and many complex geometric features that necessitate the use of a fast modeling approach such as RANS. While largely successful and fast, these methods suffer from accuracy issues in the presence of complex flow physics. Traditional two-equation RANS turbulence models used in HPT airfoil design have known deficiencies in predicting such complex flows, as highlighted, among others by Denton [40] in his summary of limitations of current approaches in predicting turbomachinery flows. Traditional turbulent flow modeling techniques based on the linear Boussinesq approximation of turbulent Reynolds stresses and available models for laminar to turbulent transition are limited in applicability and often fail to correctly predict the limits where flow separates and transitions. The inability to adequately predict these effects using existing CFD tools forces designs to be conservative, leading, for example, to a need for additional cooling flow in S1V and S1B designs, which reduces cycle efficiency. The inherent uncertainty in current prediction methods is therefore limiting possible performance improvements. Alternatively, resolving the turbulent unsteady flow field in gas turbine through DNS (Direct Numerical Simulation) or LES (Large Eddy Simulation) has been shown to be very accurate, but significantly more computationally expensive [41–43] even with simplified geometries

Boundary conditions represent one of the fundamental challenges in turbomachinery design. In the case of the high-pressure turbine it is essential to capture the gas temperature profile exiting the combustor in order to accurately and efficiently design the cooling strategy. Uncertainties in the temperature profile in both radial (profile and pattern factor) and circumferential (hot streak) may result in margin in cooling, which can adversely impact performance. Furthermore, the level of unsteadiness and turbulence entering the S1V presents a significant challenge.

Depending on the stage of design, boundary conditions between stator and rotor may be required and can be handled with either mixing plane (steady state) or sliding mesh (transient). The former enables fast simulation at the expense of accuracy. The later provides for enhanced representation of unsteady physics at the expense of speed. Furthermore, the exit of the high pressure turbine requires a boundary condition that brings in additional modeling challenges. A full engine analysis will require integration with the combustor and downstream components thus eliminating these uncertainties.

Again, depending on the stage of design the periodic nature of the high pressure turbine may be exploited with periodic boundary conditions since full wheel simulation are computationally taxing. Since the vane and blade counts are not typically integral, modeling the periodicity may be required, which again balances speed and accuracy. For a full engine simulation clearly the full 360 of the entire module will be required.

Boundary conditions in the radial direction for design present yet another challenge. The turbine consists of Axial turbines consists of alternating rows of stationary (stator comprised of vanes) and rotating (rotor consisting of blades) have gaps in the hub and casing surfaces. These are typically purged from bypass compressor air and this flow can substantially impact both aerodynamics and heat transfer. In order to reduce boundary condition uncertainty additional geometric fidelity can be included (adding the full wheelspace geometry, for example) but this comes at the expense of computational cost and a balance is required. A full engine simulation will overcome these potential challenges.

B. Opportunities and Benefits

Simulation fidelity and accuracy requirements for turbine design continue to evolve as engine technology evolves and there is increasing demand for faster and more accurate solutions to evaluate trade studies sooner in the design process. The past decade has seen a gradual shift towards transient CFD in later stages of design as software and hardware technology has improved. It has been widely established that URANS CFD can provide good directional guidance on aerothermal design space of turbines, but the fidelity on design timeframes is still lacking to displace physical tests. In general 4 keys attributes are required to position CFD to offset physical testing (1) robust and efficient model build times for real geometry, (2) improved physics algorithms, (3) improved numerical algorithms, and (4) improved fidelity of boundary conditions in the axial, radial, and circumferential directions. In order to displace physical testing accuracy and efficiency targets need to be identified and simulation workflows need to be sufficiently validated over a broad design space to realize credibility. Once done, physical testing can be offset, which will have tremendous impact on cost and development times.

C. Gaps and Challenges

Modern HPT vanes and blades are highly three-dimensional with strong secondary flows and tip vortices. Various turbine components involve complex geometric features such as small tip clearances, purge cavities, end wall contouring, film cooling and trailing edge cooling holes and turbulators. These components need to be optimized for two competing aero-thermal performance metrics: heat transfer and pressure drop. Accurate modeling of these features is critical to correctly predict the complex flow field in the turbine [44]. In addition, these geometries can slowly vary over time. Modeling of cold to hot transformation is critical to enable design and determine proper operating conditions. In addition, thermal and mechanical stresses over time can have a first order impact on clearances and leakage flows. Service degradation after many cycles impacts geometry change over time. For example, High Pressure turbine surface roughness can vary widely [45]. Another important consideration in terms of geometry is manufacturing variation that can result in significant part-to-part and engine-to-engine variation. Furthermore, non-uniform thermal barrier coatings on turbine airfoils presents another challenge. The modeling approach used for design, needs to not only accurately capture these complex geometric features, but also account for the uncertainty and variation in them.

Depending on the conditions a turbine blade row is operating under, shock boundary layer interactions could be an important physical phenomenon that impacts the aero performance of the airfoil. The shock represents a strong and localized adverse pressure gradient that can lead to thickening of the boundary layer and sometimes separation and hence an increased aerodynamic loss. The SBLI interaction region of strong flow unsteadiness, often across a wide range of frequencies, can lead to structural vibrations, and in extreme cases, structural fatigue as well. It has been widely established that RANS can predict the physics to a certain degree of accuracy but significant improvements are needed.

Laminar to turbulent transition is another factor that needs to be accurately modeled. These components operate in harsh conditions that can lead to surface roughness, which could strongly influence the boundary layer transition behavior. For flows with low freestream turbulence intensities, transition starts with linear instabilities, such as Tollmien–Schlichting waves, and break down to turbulence, also called as natural transition. Turbines however operate under higher turbulence intensities and transition can occur via various mechanisms – diffusion of turbulence into the laminar boundary layer (bypass transition) or separation-induced transition that may occur in the detached shear layer. It is important to be able to accurately model these various transition phenomenon, as well as surface roughness.

Another important physical flow feature is the generation and evolution of wakes. Both thermal and momentum wakes. The trailing edge wakes generated by each blade row mixes as it propagates downstream, but the frame change between rows can potentially amplify this mixing. Early blade rows with heavy film cooling generate deep thermal wakes. The thermal wake mixing is important to capture because the thermal wake is a significant driver of S1B temperatures due to the Kerrebrock-Mikolajczak effect. Inadequate modeling of the wake decay impacts not only loss calculations but also the distribution of the temperature field entering the S1B domain (critical for determining cooling-hole placement).

In a gas turbine, the temperature of the gas entering the HPT is significantly higher than the melting temperature of the HPT components, so relatively cool air has to be incorporated into the design .Ability to correctly understand and model heat transfer physics is critical to turbine design. In addition to the effects of thermal wakes, the S1B is subjected to purge flow from the upstream wheel cavity to ensure that hot gasses do not enter the wheelspace separating the moving and stationary parts of the HPT and cause significant damage. Accurate modeling of the mixing rate and the migration path of the purge flow is important. Heat transfer physics is also important in HPT components that often make use of surface film cooling along the airfoil and endwall surfaces. For example, a large number of film cooling holes are present on the S1V, that provide a thin protective layer of film along the surface of the components. Each film hole will see complex flow interaction as the main gas flow path mixes with the cooling flow. In addition, internal cooling via internal convective flow surfaces for the augmentation of heat transfer has become common through turbulators and pinbanks in cast airfoils. These surface enhancement methods continue to play a large role in today’s turbine cooling designs and modern designs have evolved in geometric complexity that require significant computational resources to model accurately.

D. Testing and Validation Needs

As model build, physics models, numerical algorithms continue to evolve to meet accuracy and efficiency requirements it will be essential to demonstrate the advances in a quantifiable way. The design space for the turbine is extremely large consisting of thousands of design variables. Steady RANS and URANS will continue to be the workhorse of design for decades to come. However, in order to offset physical testing, it will be essential to validate the design with advanced simulation in late stage design to realize the opportunities described earlier. Physical testing is required with a high degree of precision over a broad and representative design space. This includes, but is not limited to, spatial and temporal measurements of boundary conditions in the axial, radial, and circumferential direction, which serve as inputs to CFD models; spatial and temporal measurements of temperatures, pressures, and velocities required to establish key design metrics including durability and performance.

V. Integration and Secondary Flows

A pioneering effort towards a fully coupled front-to-back simulation of a gas turbine engine was conducted under the Stanford ASCI program in the early 2000’s [4]. The team made extensive progress towards coupling of engine components, with interfaces between different solvers located between compressor and combustor, and combustor and turbine, owing to the fact that turbomachinery was analyzed using a compressible RANS solver, and the combustor was analyzed using a low-Mach number pressure-based LES solver. In the course of the program, CHIMPS, a sophisticated tool for coupling of CFD solvers was developed [5]. As a culmination of the program, PW6000 engine was simulated at a design point, albeit with relatively limited interactions between components.

It’s important to note that, even though the grids used in those simulations were relatively coarse when considered from today’s perspective, the simulations still required 1000s of CPUs, and were at that time well out of reach for most OEMs. They were simply too large to be routinely conducted as a part of design process, which may, in part, explain why these methods were immediately not absorbed at the OEMs that supported this program.

Finally, these full engine simulations also exposed some limitations of the approach that was taken in this particular program. LES solver in the combustor captured a lot of dynamics and details of turbulence mixing, but the adjacent components – compressor and turbine – were solved using RANS solvers, with URANS effectively filtering out most of that physics. And even though special methods were developed to exchange information between RANS and LES solvers [46], it was immediately clear that the approach is suboptimal and is resulting in a striking loss of information. This was particularly true for combustor/turbine coupling. In addition, coupling of compressible and incompressible solvers posed a particular challenge for capturing transient effects. In terms of interactions with the secondary flow path, the effects such as compressor leakages and bleeds, and combustor and turbine cooling, were all captured through relatively simplistic boundary conditions.

A. Current Status

Despite these pioneering efforts towards component integration at Stanford, the focus at OEMs in the last 15 years remained mostly on improving the fidelity of component simulations, with the information exchange between components effectively being captured through boundary conditions. This exchange could take different levels of fidelity, from only exchanging averaged values, to exchanging time-averaged profiles, 2D flow variable distributions and, ultimately, unsteady boundary conditions.

In part, this trend was driven by the need of component designers to validate the performance of individual components at a wide range of conditions, along the engine operating line, as well as for excursions away from the opline (operating line). While it was demonstrated in the ASCI program that the component coupling could be handled for design conditions, analyzing other conditions was likely to add complexity, and remained unexplored.

Only more recently [47–53], there has been renewed focus on component coupling, in particular when it comes to inlet/compression system coupling, as well as combustor/turbine coupling. A more widespread use of eddy resolving simulation in turbomachinery components opened up new possibilities for higher-fidelity component coupling.

In terms of secondary flows, the integration of elements of secondary flow path with the main gas path has typically lagged behind improvements to the modeling of main gas path. This was mainly driven by the geometrical complexity of the secondary flow path components, whether in the compression system or in the hot section. In addition, these flows are often at a much lower Mach number, and time-scales, than, say, flow through rotating turbomachinery, making different types of CFD solvers optimal for each of those two domains. As a result, most of the time, the CFD analysis of the secondary flow path is conducted as a standalone simulation, and the interaction with the main gas path handled through simplified boundary conditions. Occasionally, this is done in an iterative fashion.

B. Opportunities and Benefits

As mentioned, in recent years there has been an increased use of eddy resolving simulation approaches in turbomachinery, which has been particularly true for turbines [50]. This has resulted in a renewed interest in high-fidelity coupled simulations of combustor and turbine that could potentially open up a path towards a more integrated design of these two components, and unlock new design trades that the current practice of component-centric design is preventing. The methodologies for coupling of combustor and turbomachinery solvers have evolved and improved (Ref. [51–53]), and the use of eddy simulation throughout the computational domain would maintain the high-fidelity representation of flow physics across both components. High fidelity predictions of temperature profile attenuation through the turbine and better understanding of the impact on durability and performance are now within reach [50].

At the same time, the integration of the rear of the high pressure compressor with the combustor prediffuser has also attracted considerable interest recently. Integrated simulations of the compressor and the combustor would allow a better assessment of potential design trades, when it comes to compactness, length and weight of the engine, as well as in support of novel combustor architectures. Advances in eddy resolving simulation, and the increases in available computational resources, now enable an inclusion of a greater geometrical detail and higher-fidelity predictions of the flow physics in such simulations.

Another element of the engine where integration has played a more significant role in recent years concerns the integration of the inlet and the compression system. While coupled inlet/fan simulations have been around for a while, in particular to assess the impact of flow distortion on fan performance, operability and structural integrity, in recent years there is even more interest in incorporating the other elements of the compression system. The starting point is the inclusion of the LPC (low pressure compressor) in the inlet/fan computations, enabling a more thorough assessment of environmental effects on the compression system. A further step, concerns the addition of the intermediate pressure (IPC) compressor, in some designs, and HPC (high pressure compressor) to such integrated simulations, which then enables a more systematic studies of LPC/IPC/HPC (or spool) matching and spool interactions, which is currently only considered when an engine comes to a test phase. In other words, integrated simulations of the entire compression system would unlock additional design trades between different elements of the compression system during the design cycle. The main barrier for such simulations, assuming they are RANS based, has long been the computational cost. As mentioned earlier, full wheel simulations for the entire compression system would contain 1000s of blade/vane passages and would require up to 10,000 of CPUs to be computed efficiently. If eddy resolving simulation is used in a targeted, localized fashion to capture certain elements of the flow physics, the computational cost would go even higher.

Finally, it is important to consider the integration with the elements of the secondary flow path. Incorporation of the bleed ports in the low and high-pressure compressor using higher fidelity modeling is an obvious first step. Better understanding of the flow interactions with those ports will have enormous benefits for performance, operability (including the impact of environmental effects) and structural integrity of the compression system. As mentioned before, the integration of these secondary flow path elements may require different solvers, different gridding methodologies, and the application of code coupling tools.

C. Gaps and Challenges

As discussed in other sections, targeted, selective application of eddy resolving simulation in turbomachinery will be a key enabler for high-fidelity coupled component simulations. While applying hybrid RANS/LES or WMLES for turbine simulations has become more widespread, these methods have yet to be applied more broadly throughout the compression system. It is likely that for full-wheel simulations of the entire compression system the application of eddy resolving simulation will be done in a selective fashion, focusing on capturing a particular aspect of flow physics (e.g., mixing of distortion, wakes, end-wall features such as tip clearance flows and corner roll-ups).

For successful coupled combustor/turbine simulations, high-fidelity representation of cooling, both for the combustor and for the turbine, is of essence. This will require automated meshing/gridding and the ability to control the resolution in the vicinity of cooling holes. Some form of eddy resolving simulation will likely have to be used to better predict cooling mixing, than what can typically be achieved with RANS modeling. Similarly, for successful coupled compressor/combustor simulations, high-fidelity simulations of diffusing flows will be of essence. Correctly predicting flow separation in the combustor prediffuser under various forms of distortion coming out of the rear of the high pressure compressor is going to require eddy-resolving simulations, with validated best practices for spatial and temporal resolution.

As elaborated before, the rapid and automated inclusion of small geometrical features at the end-walls will also benefit from improved gridding methodologies, whether they are based on unstructured meshes with adaptive mesh refinement, or multi-block structured grids with some sort of embedded localized refinement using oversetting techniques. Incorporating trenches, rubs, end-wall gaps, platform gaps, leakages, cavities and seals, bleed ports, cooling holes and slots, into the CFD model will provide a better geometrical representation of the components, and will enable improved predictions, in particular at off-design conditions.

While representing the secondary flow path elements with boundary conditions may be satisfactory for design operating conditions, it is very likely that at off-design conditions it will be difficult to correctly represent these effects through simple boundary conditions, and a more integrated approach will have to be taken, further increasing the complexity and the cost of the simulations.

Finally, it is important to highlight the cost of such coupled simulations. The cost of coupled component simulations, such as combustor/turbine or compressor/combustor is more manageable. For only a sector simulations, such computations could be conducted with 1000 to 2000 CPUs in a relatively short period of time – within about a week currently. However, if one is to consider even a sector-based, not full-wheel, front-to-back engine simulation, that applies eddy resolving simulation (e.g., hybrid RANS/LES and/or WMLES) throughout the combustor and turbine, and in targeted regions of the compression system, a typical simulation would contain up to 5 billion grid points, and require up to 5000s of CPUs for a two weeks to compute a relatively small integration time of 100 milliseconds, with a total cost of the order of 10^6 CPU hours, as shown in87]. In order to expand these simulations to a full-wheel front-to-back engine configuration at a similar fidelity, it was estimated that the computational grid may require 15 to 20 billion of computational cells, and 30,000 to 50,000 CPUs. For 100 milliseconds of integration time, 2 weeks of running would be required, with the computational cost at the order of 10^7 CPU hours. This already illustrates the computational expense of integration. Note that these simulations did not include the entire bypass stream, nor did they include the details of the secondary flow path (e.g., leakage and bleed paths in compressors and cooling passages in turbines). Now, consider the computational resources that would be needed to compute multiple operating points along the engine opline and excursions from it, as well as any transient simulations where the integration time is measures in seconds. It is very likely that those will remain out of reach until major advancements in GPU computing (or other hardware platforms) become widely available.

D. Testing and Validation Needs

Validating coupled component simulations represents an important challenge in itself. Engine components are usually tested in isolation, where more detailed data is acquired, and then they are assembled in the full engine for engine tests, where very little detailed data can be acquired. In other words, controlled experiments in combustor/turbine, or compressor/ combustor or LPC/HPC rigs are difficult and rare. And yet, only such experiments would provide a good platform for validating integrated component simulations. It is very likely that government agencies will need to develop rigs to provide such validation platforms.

Similarly, controlled experiments for secondary flowpath, and especially the secondary flow path integrated with the main gas path are relatively rare. Probably the approach towards providing useful data will be to tie these to coupled component experiments and provide for additional data acquisition to characterize the secondary flows as much as possible.

VI. Full Engine Simulation

The proposed Propulsion Grand Challenge is to conduct a full engine simulation for one operating point from model build to processed results within 1 week, in a manner that has sufficient geometric fidelity and physics accuracy to reduce/complement engine testing and meet performance, operability, emissions, and durability accuracy metrics. It is proposed to approach these simulations in a multi-tiered fashion, where the complexity and the cost of simulations would depend on the technical objective of each simulation. For example, to assess performance at design point, or any other discrete operating point, a sector approach may suffice. For operability related questions, full-wheel simulations will be a necessity. Details of bypass stream may or may not be required depending on the objective. The same would be true for secondary flows, such as leakage and bleed flow paths in compressor and turbine cooling flowpaths – for durability related questions, including these effects at sufficient fidelity will be critical. Finally, for some purposes, long-term time integration is required. Table 1 presents some computational time estimates based on specs for a standard CPU cluster in 2020 for these tiers, assuming for example WMLES or hybrid RANS/LES, with geometric features important to the operation of the engine for performance and durability including the needed, but not limited to, combustor and turbine cooling features and secondary flow seals, which are typically small in scale and rich in complexity. Indeed the grid sizes and the computational requirements also depend on the level of accuracy for the various quantities that we need as part of the objective. It is reasonable to expect that computational resources and speeds will increase significantly in the next two decades and algorithms will be developed to exploit those platforms, making especially the last two tiers in the table more feasible. It is also possible that advances in technologies like artificial intelligence, machine learning and low-order models will serve to significantly reduce the resource requirements listed in the table.

Table 1. Full engine simulations cost estimates

Type of full engine simulation	Grid Size #Control Volumes	Integration Time	Computational time Total core-hours
Main gas path (sector)	$O(10^9)$	O(100 ms)	$O(10^6)$
Main gas path (full annulus)	$O(10^{10})$	O(100 ms)	$O(10^7)$
Main gas path + bypass stream	$O(10^{10}\text{-}10^{11})$	O(100 ms)	$O(10^7\text{-}10^8)$
Including secondary flow path	$O(10^{11}\text{-}10^{12})$	O(100 ms)	$O(10^8\text{-}10^9)$
Long time integration for transients	$O(10^{11}\text{-}10^{12})$	O(10 s)	$O(10^{10}\text{-}10^{11})$

The availability of such simulations would deliver billions of dollars of savings per year to the aviation industry. They would enable better understanding of component and spool interactions, and provide a platform for more rapid and efficient component and spool matching. At off-design conditions, they would enable a better assessment of engine operability, structural integrity and durability than it is currently achievable in isolated component simulations. They would provide a platform for the analysis of the engine performance through transients, which is currently only analyzed via testing. More generally, full engine simulations would complement engine testing in multiple ways – they would provide a capability to conduct a true engine pre-test CFD, they could be used to help with test data interpretation, and so on. Finally, they would also provide a platform to study propulsion/airframe integration and the impact of environmental effects on the engine. In addition, this platform would enable better understanding of engine performance in operation, and the potential risks of hardware deterioration.

VII. Conclusions

Computational Fluid Dynamics is an important tool in the development process of aircraft engines. CFD is routinely executed in design trade studies at the component and system level focusing on meeting system targets that help enable engine targets: meeting regulatory requirements for noise and emissions, maximizing fuel efficiency and durability while minimizing cost, weight and time to market. While CFD is used routinely in the design process, physical testing is still required at nearly all phases of design for all modules including the compression system, combustor, turbine and secondary flows. This is primarily due to the inability to accurately simulate a complex design space on timeframes that are acceptable for engine program development. Scale resolved approaches have considerably closed the gap on accuracy, but the turnaround time is still not acceptable.

Further developments of physics models, algorithms, model build processes, post-processing capabilities and hardware infrastructure are required to position CFD to be able to offset physical testing at the component and module level. In this paper a roadmap has been summarized on the existing gaps for each module and the benefits that closures of the gaps would bring. As simulation accuracy and performance improve, confidence in component and system analysis and design will increase to a point where physical testing can be reduced. Furthermore, as the vision for fast and accurate simulations for the entire CFD ecosystem unfolds it will enable the ultimate goal of simulating flow through the entire engine for a mission scenario that could serve as a digital twin serving as a foundation to realize the ultimate goal of certification by analysis. By establishing a Grand Challenge Problem as a sequence of Challenge Problems the community will benefit from advances along the way towards realization of the Grand Challenge problem stated in this paper.

It is estimated that all of the benefits outlined in the previous two paragraphs will add up to billions of dollars (U.S.) of savings per year to the aviation industry. However, it is stressed that in order to realize the goals and the benefits of the grand challenge problem a targeted and sustained investment from funding agencies promoting coordinated research into the enabling technologies is of paramount importance.

Appendix – Path to an HPC Common Model for CFD Code Enhancement and Validation

As stated in the compressor section, the HPC is critical to establishing the performance and operability of the engine. In addition, the HPC supplies the air flow for the core engine to drive the fan, cooling flow to the hot section components, and bleed air for the aircraft environmental control system. Furthermore, SOA CFD tools are not sufficient to accurately and quickly analyze a 7-10 stage HPC over its entire operating range. For these reasons an HPC common model is necessary to aid in the advancement of HPC computational speed and accuracy. In this section the objectives are to 1) review open databases and CFD assessment exercises for a single stage of an HPC to illustrate the necessity for validation data, and 2) identify key compressor design considerations and design parameters to develop a common HPC research model for this grand challenge problem.

A. NASA Single Stage compressor Data bases and Code Assessment exercises

In the 1970s and 1980s NASA produced a number of compressor datasets that have been used by the turbomachinery community as a basis for the validation and development of turbomachinery analysis tools, including the growing field of CFD codes. LDV (Laser Doppler Velocimetry) was customized to measure the axial and tangential velocity inside the rotating passages of transonic compressors. The transonic fan NASA Rotor 67 was the first major dataset acquired with a single-channel LDV, which captured the shock and wake structure in an isolated transonic fan [54-57]. Subsequently, NASA Stage 67 (Rotor 67 + Stator 67) was the first dataset that captured the unsteady fan rotor/stator blade row interactions with the same single-channel LDV system [58, 59]. Additional experimental test cases produced by GRC include the NASA Stage 35 [60], which incorporates a full compressor stage. In addition, NASA built a 5-ft (1.524 m) diameter centrifugal compressor in which to make detailed probe and optical measurements for code validation - results are summarized in [61]. Centrifugal compressor scaling studies [62] and code validation datasets [63] were used to improve centrifugal compressor CFD codes and the resulting designs.

Perhaps the most comprehensive and widely utilized of the NASA datasets is the NASA Rotor 37 data set, having been the basis for the 1994 ASME/IGTI Blind CFD code assessment exercise. This was a period when CFD RANS/URANS codes had limited access to data depicting the flow field in the rotating passages of high speed (compressible) turbomachinery to use for code validation and model development. Building upon NASA's experience acquiring data in NASA fan rotor 67 with a single velocity component system, NASA designed and developed the first laser anemometer measurement system (early 1980s) that acquired axial and tangential velocities simultaneously inside the rotating passages of a transonic compressor. NASA Rotor 37 has an extensive set of aerodynamic probes and LDV data across the rotor operating range from maximum flow to near-stall conditions at 60% speed (fully subsonic), 80% speed (transonic), and 100% design rotor speed (fully supersonic in the rotor frame of reference). Detailed laser anemometer measurements were acquired at high flow, design flow, and near stall flow conditions at streamwise and radial planes. Gas velocity measurements acquired with this laser-based non-intrusive optical instrumentation were used to study the flow physics associated with the shock / rotor tip clearance flow, the shock / boundary layer interactions within the rotor, and wake decay characteristics downstream of the rotor. Each of the eight unique CFD codes utilized various grid types, grid topologies, and turbulence models that represented the state of the art in CFD methods at that time. A summary of the ASME blind test case aerodynamic performance results is presented in Figure A.1, which compares the NASA Rotor 37 experimental and CFD results of overall performance at 100% design speed as well as the radial distribution of pressure ratio, temperature ratio, and efficiency. In summary, many of the CFD code results did not match the experimental data, not only in describing details such as the tip clearance flow, shock structure and rotor wake decay, but also in overall compressor performance and radial distributions of the performance parameters. It was quite surprising to the CFD community at that time that the methods could not capture the performance of a compressor rotor blade row. The data are best summarized in [64–66] and the blade design and geometry are provided by Reid and Moore [67].

Following the ASME Blind CFD code assessment study, The Advisory Group for Aerospace Research and Development also used the NASA Rotor 37 benchmark data set to compare results from a large number of Navier-Stokes CFD codes [68]. These test case activities highlighted the large range of results produced by the various codes, some of which is attributable to how the codes were employed in addition to the underlying code algorithms and methods. These discrepancies between the CFD and experimental results have led to significant improvements in CFD mesh generation, turbulence model implementation, and tip clearance modeling. By all means not an all-inclusive list, but References [69–81] highlight the lessons learned and key findings related to the sensitivity of the NASA Rotor 37 solution to the grid, turbulence model, clearance model, and modeling of the gap between the rotor and stationary flow path for various CFD. To this day, these data are used as a CFD validation tool and are referenced by presenters annually at the ASME and AIAA conferences.

Furthermore, NASA conducted a turbomachinery CFD code assessment of its internal codes and/or codes initiated and sponsored using government funding in 2008. These codes were validated against NASA rotor 37 and NASA stage 35 test data and the results were reported at the 2009 AIAA Aerospace Sciences Meeting in References [82–86]. The results indicated strong agreement among the codes for predicting the compressor speed-line and stall margin. However, to better predict the radial distribution of the compressor performance parameters required using advanced modeling techniques such as unsteady Reynolds-averaged Navier-Stokes equations, large eddy simulation, and accurate resolution of the geometry to capture the impact of secondary flow features on the performance.

In summary, all of the datasets reported in this section were using single stage designs from the early 70s. Multi-stage compressor design has matured significantly since the 70's with advanced three-dimensional designs, higher blade loadings, and better efficiencies. It is time to re-evaluate the design and analysis capability by providing a common research model with the design features that challenge current and future compressor designs.

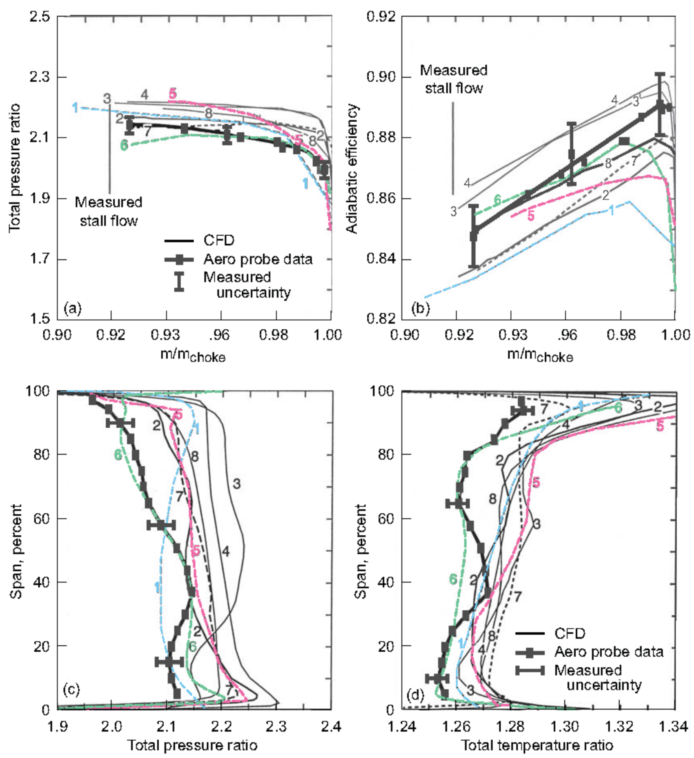

Fig. A.1 NASA Rotor 37 American Society of Mechanical Engineering blind test case results (1994). CFD is computational fluid dynamics, m is the mass flow and mchoke is the choking mass flow rate (from Ref. [64]).

B. HPC Common Model Design Considerations

There are many design considerations for a multistage compressor and a key challenge is to define a common multistage compressor that would be relevant, but not reveal any proprietary information from any given company. The common design must contain design considerations that address the relevant physics that prohibit the companies from designing the compressor in a single design, build, test cycle. A critical design parameter is to determine where in the compressor design trade space between efficiency and blade loading is most suitable for one's application. In general, the higher the blade loading blade loading (represented as the change in enthalpy divided by the square of the rotor tip rotational speed), the more difficult it is to achieve high efficiency. Similarly, lower blade loading results in more stages, longer lengths, and higher weight.

Furthermore, in order to obtain detailed test validation data and keep costs reasonable, it is suggested to focus on either the front-block compressor design or the rear-block design of the HPC. This would allow many existing facilities in academia, industry, and government to provide experimental data. A subset of key design considerations for either front or rear block compressors are: number of stages (constrained by blade loading), stage reaction, blade loading, pressure ratio, flow range, shrouded versus unshrouded stators , variable versus fixed stator, stator cavity design, stator cavity seal design, casing treatments, and of course instrumentation requirements. In addition, there are a multitude of compressor blade design parameters such as tip speeds, leading edge radius ratio, aspect ratio, solidity, tip clearances, specific flow, stage pressure ratio, efficiency, and stall margins.

In conclusion, the major challenge is to reach agreement with the turbine engine community to define a relevant compressor design. To this end, the community must agree on a set of design parameters, their definition, and define a range of values that would be of interest. Since each gas turbine engine manufactures will likely have a different design approach that is highly proprietary to them, it is suggested that an unbiased, government team be utilized to interact with the community independently to acquire the information and compile an integrated list of design parameters. The government is uniquely positioned to serve in this role as they routinely work with the turbine industry on proprietary design challenges.

References

[1] Slotnick, J., Khodadoust, A., Alanso, J., Darmofal, D., Gropp, W., Lurie, E., and Marviplis, D., CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences, NASA/CR–2014-218178.

[2] Welstead, J., and Felder, J., "Conceptual Design of a Single-Aisle Turboelectric Commercial Transport with Fuselage Boundary Layer Ingestion," 54th AIAA Aerospace Sciences Meeting, San Diego, CA. 2016.

[3] Jones, S.M., Haller,W.J., and Tong, M.T., An N+3 Technology Level Reference Propulsion System, NASA/TM—2017-219501.

[4] Medic, G., You, D., Kalitzin, G., Herrmann, M., Ham, F., van der Weide, E., Pitsch, H., and Alonso, J., "Integrated computations of an entire jet engine", 52nd ASME Turbo EXPO Conference, Montreal, Canada, May, 2007, GT2007-27094.

[5] Turner, M.G, "Lessons Learned from the GE90 3-D Full Engine Simulations", 48th AIAA Aerospace Sciences Meeting, January 2010, AIAA 2010-1606.

[6] Schluter, J.U., Wu, X., Kim, S., Alonso, J.J., and Pitsch, H., "Integrated RANS LES computations of turbomachinery components : Generic compressor / diffuser", Center for Turbulence Research Annual Research Briefs 2003, Stanford University.

[7] Alonso, J., Hahn, S., Ham, F., Herrmann, M., Iaccarino, G., Kalitzin, G., LeGresley, P., Mattsson, K., Medic, G., Moin, P., Pitsch, H., Schluter, J., Svard, M., van der Weide, E., You, D., and Wu, X., (2006), "CHIMPS: A High-Performance Scalable Module for Multi-Physics Simulations", 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Sacramento, CA, July, 2006, AIAA-2006-5274.

[8] Kraft, E.M., "After 40 Years Why Hasn't the Computer Replaced the Wind Tunnel?", ITEA Journal 2010; 31: 329–346

[9] Moreno, J., Dodds, J., Stapelfeldt, S. C., and Vahdati, M., "Deficiencies in the SA Turbulence model for the prediction of the stability boundary in highly loaded compressors", ASME Journal of Turbomachinery, DOI: https://doi.org/10.1115/1.4047784

[10] Xia, G., Medic, G., and Praisner T., (2018), Hybrid RANS/LES of corner separation in a linear compressor cascade, ASME J. Turbomachinery 140(8), 081004 (Jul 26, 2018), doi: 10.1115/1.4040113

[11] Bob Ni, ADS CFD, private communication.

[12] Tucker, P. G., Wang, Z., "Eddy Resolving Strategies in Turbomachinery and Peripheral Components", J. Turbomachinery. pp. 1-57, https://doi.org/10.1115/1.4048697

[13] Peters, N., "Multiscale combustion and turbulence", Proceedings of the Combustion Institute, Volume 32, Issue 1, pp. 1-25, 2009

[14] Peters, N., Turbulent Combustion, Cambridge University Press 2000.

[15] Van Oijen, J.A., Donini, A., Bastiaans, R.J.M., ten Thije Boonkkamp, H.H.M., de Goey, L.P.H., “State-of-the-art in premixed combustion modeling using flamelet generated manifolds”. Progress in Energy and Combustion Science. Vol. 57, 2016, page 30-74
[16] Spalding, D. B., “Mixing and chemical reaction in steady confined turbulent flames”, Thirteenth symposium (international) on combustion, The combustion institute, pp. 649–657, 1971
[17] Magnussen, B. F., and Hjertager, B. H., “On mathematical models of turbulent combustion with special emphasis on soot formation and combustion”, In 16th Symposium (International) on Combustion. 16(1): 719-729, 1976
[18] Gosman, A. D., and loannides, E., 1983. Aspects of computer simulation of liquid-fueled combustors. Journal of Energy, 7(6), pp. 482-490.
[19] Gepperth, S., Müller, A., Koch, R., Bauer, H.J., “Ligament and droplet characteristics in prefilming airblast atomization”, International Conference on Liquid Atomization and Spray Systems (ICLASS), Heidelberg, Germany, 2012
[20] Schmehl, R., Maier, G., Wittig, S., “CFD Analysis of Fuel Atomization, Secondary Droplet Breakup and Spray Dispersion in the Premix Duct of a LPP Combustor”, ICLASS 2000: 8th International Conference on Liquid Atomization and Spray Systems, Pasadena, CA, USA, 16-20 July 2000
[21] Chin, J., and Lefebvre, A., “The role of the heat-up period in fuel drop evaporation”, 21st Aerospace Sciences Meeting, 1983, AIAA 1983-68
[22] Frossling, N., “ber die Verdunstung Fallender Tropfen,” Gerlands Beitrge zur Geophysik, Vol. 52, 1938, pp.170-215.
[23] Turns. S.R., “Understanding NOx formation in nonpremixed flames: Experiments and modeling”, Progress in Energy and Combustion Science. Vol. 21, 1995, page 361-385
[24] Leung, K.M., Lindstedt, R.P., and Jones, W.P., “A simplified reaction mechanism for soot formation”, Combustion and Flame, vol. 87, pp. 289-305, 1991.
[25] Anand, M.S., Eggels, R., Staufer, M., Zedda, M., and Zhu, J., “An advanced unstructured-grid finite-volume design system for gas turbine combustion analysis,” GTINDIA2013-3537, Proceedings of ASME 2013 Gas Turbine India Conference, Bangalore, 2013.
[26] Anand, M.S., et al., CFD Modeling of Lean Blowout and Ignition Fuel Sensitivity, Book Chapter in Fuel Effects on Operability of Aircraft Gas Turbine Combustors, Eds. M.B. Colket and J.S. Heyne, AIAA Book Series, To be published, 2021
[27] Tang, Y., Hassanaly, M., Raman, V., Sforzo, B., Seitzman, J., “Probabilistic modeling of forced ignition of alternative jet fuels”, Proceedings of the Combustion Institute, 2020. https://doi.org/10.1016/j.proci.2020.06.309
[28] Neophytou, A., Richardson, E., and Mastorakos, E., “Spark ignition of turbulent recirculating non-premixed gas and spray flames: A model for predicting ignition probability”, Combust. Flame, Vol. 159, No. 4, 2012, pp. 1503–1522.
[29] Zhu, M., Dowling, A.P., and Bray, K.N.C., 2001, “Flame transfer function calculations for combustion oscillations”, ASME Paper 2001-GT-0374.
[30] Dowling, A.P., and Stow, S.R., Acoustic analysis of gas turbine combustors. In T. C. Lieuwen and V. Yang (eds.), Combustion Instabilities in Gas Turbine Engines: Operational Experience, Fundamental Mechanisms, and Modeling, AIAA, New York, 2005.
[31] Livebardon, T., Moreau, S., Poinsot, T., and Bouty, E., Numerical investigation of combustion noise generation in a full annular combustion chamber, AIAA paper 2015-2971,
[32] Herrmann, M., “A balanced force refined level set grid method for two-phase flows on unstructured flow solver grids”, Journal of Computational Physics 227 (4), 2674-2706, 2008
[33] Monaghan, J.J., “Smoothed Particle Hydrodynamics and Its Diverse Applications”. Annual Review of Fluid Mechanics. Vol.44. 2012. pages 323-346
[34] Triantafyllidis, A., Mastorakos, E., and Eggels, R.L.G.M., 2009 “Large eddy simulations of forced ignition of a non-premixed bluff-body methane flame
[35] Colin, O., Ducros, F., Veynante, D., Poinsot, T., A thickened flame model for large eddy simulations of turbulent premixed combustion, Physics of Fluids 12, 1843, 2000.
[36] Pope, S.B., “PDF methods for turbulent reactive flows”, Progress in Energy and Combustion Science 11 (2), 119-192, 1985
[37] Mustata, R., Valino, L., Jimenez, C., Jones, W.P., Bondi, S., “A probability density function Eulerian Monte Carlo field method for large eddy simulations: Application to a turbulent piloted methane/air diffusion flame (Sandia D)”. Combustion and Flame. Vol.145. 2006, pages 88-104.
[38] Rigopoulos, S., Modelling of Soot Aerosol Dynamics in Turbulent Flow, Flow, Turbulence and Combustion 103 (2019) 565-604

[39] Koo, H., Eggels R.E., and Anand, M.S., "Validation of a Moment-Based Soot Model on Benchmark and Production Aircraft Engine Combustors", AIAA Propulsion and Energy Forum, August, 2019, AIAA 2019-4206.

[40] Denton, J. D., "Some Limitations of Turbomachinery CFD," ASME paper GT2010-22540.

[41] Michelassi, V., Chen, L., Pichler, R. and Sandberg, R. D., "Compressible Direct Numerical Simulation of Low-Pressure Turbines: Part II –Effect of Inflow Disturbances," ASME Journal of Turbomachinery, July, Vol. 137, 2015.

[42] Pichler, R., Sandberg, R., Laskowski, G., and Michelassi, V., (2017) "High-fidelity simulations of a linear HPT vane cascade subject to varying inlet turbulence", ASME GT2017-63079.

[43] Wheeler, A.P.S., Sandberg, R.D., Sandham, N.D., Michelassi, V., and Laskowski, G., (2016) "Direct numerical simulations of a high-pressure turbine vane", ASME Journal of Turbomachinery vol 138(7).

[44] Laskowski, G.M., Kopriva, J., Michelassi, V., Shankaran, S., Paliath, U., Bhaskaran, R., Wang, Q., Talnikar, C., Wang, Z.J., Jia, F., (2016) "Future directions of high fidelity CFD for aerothermal turbomachinery analysis and design", AIAA 2016-3322.

[45] Bons, J.P., Taylor, R.P., McClain, S.T. and Rivir, R.B., (2001) "The many faces of turbine surface roughness", ASME Journal of Turbomachinery, vol 123:4.

[46] Medic, G., You, D. and Kalitzin, G., (2007), "On RANS/LES Coupling for Integrated Computations of Jet Engines", 52nd ASME Turbo EXPO conference, Montreal, Canada, May, 2007, GT2007-27096.

[47] Vadlamani, N. R., Cao, T., Watson, R., Tucker, P. G., "Toward Future Installations: Mutual Interactions of Short Intakes with Modern High Bypass Fans", J. Turbomachinery. August 2019, 141(8): 081013, DOI: https://doi.org/10.1115/1.4044080

[48] Duchaine, F., Dombard, J., Gicquel, L.Y.M. and Koupper, C.. "Integrated Large Eddy Simulation of Combustor and Turbine Interactions: Effect of Turbine Stage Inlet Condition", ASME Paper, GT2017-63473, 2017.

[49] Bacci, T., Becchi, R., Picchi, A. and Facchini, B., Adiabatic Effectiveness on High Pressure Turbine Nozzle Guide Vanes Under Realistic Swirling Conditions. ASME Paper, GT2018-76637, 2018.

[50] Xia, G., Kalitzin, G., Lee, J., Medic, G., and Sharma, O.P., Hybrid RANS/LES simulation of combustor/turbine interactions, 65th ASME Turbo EXPO, London, UK, June, 2020, ASME Paper GT2020-14873.

[51] Nastase, C., Tristanto, I., Anand, M.S., A file-based approach to multi-component coupled simulations in gas turbine engines, AIAA Propulsion and Energy Forum, August, 2019, AIAA 2019-3831.

[52] Adoua, R., Page, G., Tristanto, I., and Bailey, R., "Integrated RANS Simulations of Compressor-Combustor-Turbine Interactions", 30th International Conference on Parallel Computational Fluid Dynamics, Parallel CFD2018, Indianapolis

[53] Miki, K., Wey, C.T., and Moder, J., Computational study of modeling fully-coupled combustor-turbine interactions by the open national combustion code (OpenNCC), AIAA Propulsion and Energy 2020 Forum, AIAA 2020-3689.

[54] Strazisar, A.J., 1985. "Investigation of flow phenomena in a transonic fan using laser anemometry". J. Eng. Power—T. ASME 107:427-436.

[55] Hathaway, M.D., Gertz, J.B., Epstein, A.H., and Strazisar, A.J., 1986. "Rotor wake characteristics of a transonic axial flow fan", AIAA J. 24:1802–1810.

[56] Wood, J.R., Strazisar, A.J., and Simonyi, P.S., 1987. Shock structure measured in a transonic fan using laser anemometry. AGARD–CP–401.

[57] Strazisar, A.J., Wood, J.R., Hathaway, M.D., and Suder, K.L., 1989. Laser anemometer measurements in a transonic axial-flow fan rotor. NASA TP–2879.

[58] Suder, K.L., Okiishi, T.H., Hathaway, M.D., Strazisar, A.J., and Adamczyk, J.J., 1987. Measurements of the unsteady flow field within the stator row of a transonic axial-flow fan: Part I—measurement and analysis technique. ASME Paper 87–GT–226.

[59] Hathaway, M.D., Okiishi, T.H., Suder, K.L., Strazisar, A.J., and Adamczyk, J.J., 1987. Measurements of the unsteady flow field within the stator row of a transonic axial-flow fan: Part II— results and discussion. ASME Paper 87–GT–227.

[60] Van Zante, D.E., Adamczyk, J.J., Strazisar, A.J., and Okiishi, T.H., 2002. Wake recovery performance benefit in a high-speed axial compressor. J. Turbomach. 124:275–284.

[61] Hathaway, M.D., Chriss, R.M., Wood, J.R., and Strazisar, A.J., 1993. Experimental and computational investigation of the NASA low-speed centrifugal compressor flow field. J. Turbomach., 115:527–542.

[62] Skoch, G.J., and Moore, R.D., 1987. Performance of two 10-lb/sec centrifugal compressors with different blade and shroud thicknesses operating over a range of Reynolds numbers. NASA TM–100115 (AVSCOM–TR–87–C–21 and AIAA–87–1745).

[63] Skoch, G.J., Prahst, P.S., Wernet, M.P., Wood, J.R., and Strazisar, A.J., 1997. Laser anemometer measurements of the flow field in a 4:1 pressure ratio centrifugal impeller. NASA TM–107541 (ARL–TR–1448 and ASME Paper 97–GT–342).

[64] Suder, K.L., 1996. “Experimental Investigation of the Flow Field in a Transonic Axial Flow Compressor with Respect to the Development of Blockage and Loss.” Ph.D. Dissertation Case Western Reserve University, also NASA TM107310, August 1996.
[65] Suder, K.L., and Celestina, M.L., 1996. “Tip Clearance Vortex / Shock Interactions in a Highly Loaded Transonic Compressor Rotor.” J. Turbomach, Vol. 118, no. 2, 1996, pp. 218-229.
[66] Suder, K.L., 1998. “Blockage Development in A Transonic, Axial Flow Compressor”, J. Turbomach., Vol. 120, No. 3, 1998, pp.465-476.
[67] Reid, L., and Moore, R.D., 1978. Design and overall performance of four highly loaded, high speed inlet stages for an advanced, high-pressure-ratio core compressor. NASA TP–1337.
[68] Dunham, J., 1998. AGARD WG26 report - summary of 13 different simulations of Rotor 37 AGARD Advisory Report AR-355, May 1998 CFD Validation for Propulsion System Components. Also summarized in ASME 98-GT-050.
[69] Denton, J., 1996. “Lessons Learned from Rotor 37. Presented at the Third International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows (ICIASF), Beijing, China, September 1-6, 1996.
[70] Dalbert and Wiss, 1995 “Numerical Transonic Flowfield Predictions for NASA Compressor Rotor 37,” ASME 95-GT-326.
[71] Arima, T.A., Sonoda, T., and Masatoshi, S.A., 1997. Numerical Investigation of Transonic Axial Compressor Rotor Flow Using a Low Reynolds Number k-epsilon Turbulence Model (R37 and R67) ASME 97-GT-082.
[72] Arima, T., Masatoshi, S, and Yamaguchi, Y., 1998. The Flow Field in the Tip Clearance Region of an Axial Compressor Rotor.
[73] Arima, T., Sonoda, T., Shirotori, M., and Yamaguchi, Y., 1998. Computation of Subsonic and Transonic Compressor Rotor Flow Taking Account of Reynolds Stress Anisotropy. ASME 98-GT-423.
[74] Calvert, W,J., A Synthesis of NASA Compressor Rotor 37 Using a 3D CFD Code Proceedings of “Verification of Design Methods by Test and Analysis,” Royal Aeronautical Society, www.raes.org.uk.
[75] Chima, R., 1996. A K-omega Turbulence Model for Quasi-Three-Dimensional Turbomachinery Flows, NASA TM10705, AIAA Reno Meeting, 1996.
[76] Chima, R., 1998. Calculation of Tip Clearance Effects in a Transonic Compressor Rotor, ASME J of Turbo, Vol. 120, p. 131-140, 1998.
[77] Shabbir, A., 1996. Assessment of Three Turbulence Models in a Compressor Rotor 96-GT-198.
[78] Hah and Loellbach. 1997. Development of Hub Corner Stall and Its Influence on the Performance of Axial Compressor Blade Rows, ASME 97-GT-042 (to be published in ASME J. of Turbomachinery).
[79] Shabbir, A., 1997. The Effect of Hub Leakage Flow on Two High Speed Axial Compressor Rotors, ASME 97-GT-346 (to be published in ASME J. of Turbomachinery).
[80] Arnone, A., 1997. Grid Dependency Study for the NASA Rotor 37 Compressor Blade ASME 97-GT-384.
[81] Calvert, J., 1997. Evaluation of a 3D Viscous Code for Turbomachinery Flows, ASME 97-GT-078.
[82] Celestina, M. L., and Mulac, R. A. 2009. Assessment of stage 35 With APNASA. AIAA 2009–1057.
[83] Herrick, G.P., Hathaway, M. D., and Chen, J. 2009. Unsteady full annulus simulations of a transonic axial compressor stage. AIAA 2009–1059.
[84] Ameri, A.A., 2009. NASA ROTOR 37 CFD CODE validation: Glenn-HT code. AIAA 2009–1060.
[85] Hah, C., 2009. Large eddy simulation of transonic flow field in NASA Rotor 37. AIAA 2009–1061.
[86] Chima, R.V., 2009. SWIFT code assessment for two similar transonic compressors. AIAA 2009–1058.
[87] Estimates presented at RTRC Full Engine Simulation Workshop (August 11-13, 2020).