Technical Evaluation Report

Jürgen Seidel, PhD
Department of Aeronautics, US Air Force Academy
USA

jurgen.seidel@afacademy.af.edu

Keywords: Workshop, Computational Fluid Dynamics, Design, Analysis, Industry

ABSTRACT

The NATO AVT-366 workshop on the "Use of Computational Fluid Dynamics for Design and Analysis: Bridging the Gap Between Industry and Developers" was held virtually on 16-19 May 2022. All sessions were unclassified. The workshop audience included participants from NATO countries.

At the workshop, 16 papers were presented by industry (7), academia (5), and government research institutions (4). Each day, a keynote presentation anchored the theme for the following presentation and discussions. The presentations were arranged such that industry perspectives were presented on the first day. On the second day, advances to integrating computational fluid dynamics (CFD) in the design process and extensions of use cases were presented. The implications of the presentations from day one and two were discussed. Day 3 provided an overview of the drivers of CFD technology development and presentations of concepts to integrate computational tools into the design environment. The final day concluded the workshop with a presentation of an integration concept and the technical evaluation presentation.

This technical evaluation report summarizes the core ideas and concepts presented during this workshop. It also provides and overview of the discussions and concludes with a summary and recommendation.

1.0 INTRODUCTION

The NATO AVT-366 workshop on the "Use of Computational Fluid Dynamics for Design and Analysis: Bridging the Gap Between Industry and Developers" was designed to explore the application and adoption of computational fluid dynamics (CFD) methods in industry and to identify current requirements and capability gaps. The aim of the workshop was to capture current practices and determine a way forward to allow wider adoption of CFD in industry, especially early in the design cycle of an air or maritime vehicle. Presentations from industry, academia, and government research institutions highlighted current practices and developments. The discussions focused on identifying current bottlenecks in industry CFD use, which served as cornerstones for proposals of a way forward. The workshop was relevant and timely considering the current stage of CFD use in industry and ongoing work in both the USA and Europe to develop simulation frameworks that will allow users access to multi-fidelity simulation tools in an integrated toolchain to maximize the utilization of all data sources, such as high-fidelity and low-fidelity simulations, experiments, ground tests, and flight tests, for improved and less time consuming vehicle design.

2.0 DAY 1 (MONDAY, 16 MAY 2022)

Day 1 of the workshop focused on presentations from industry participants. All papers provided an overview of the use of CFD in the industry setting and highlighted the obstacles encountered by industry to employ CFD at a larger scale in the design process. The conclusions where discussed during the discussion session at the end of Day 2.

2.1 Keynote 1: Winkler and Heller: Virtualization of Military Aircraft Design by Means of CFD

The authors made the point that industry requires computational tools that are fast, robust, ideally allow for variable fidelity, and provide fast solutions. All these points were reiterated in other papers from industry participants. The paper presented an overview of current use of CFD in aircraft design but also highlighted areas where significant improvements to CFD methods are required to effectively utilize CFD in the design process. Examples included high angle of attack flow fields (with an emphasis of aircraft stability), intake aerodynamics, weapons bay cavity flows, and flow separation. In many of these areas, only unsteady high fidelity CFD solutions provide acceptable answers, but with the associated large resource requirements in terms of computing infrastructure and run time. The authors contrasted the CFD capabilities with current wind tunnel data acquisition methods and highlighted the ability to generate large, accurate aerodynamic databases in a relatively short time once models are built. However, experimental methods that provide detailed surface or flow volume data are not adequately developed for routine data generation. The authors gave a range of CFD coverage for the most relevant design disciplines and concluded that in performance and load analysis CFD is quite well advanced whereas stability and control, aeroelasticity, and aeroacoustics lag behind.

The authors pointed out a number of current bottlenecks preventing a broader use of CFD methods in aircraft design, leading with turbulence modelling and uncertainty quantification. In addition, they acknowledged the need for automation of the CFD process and the lack of integration of CFD in the design toolchain, e.g. in the MDAO process. Furthermore, flow prediction accuracy with current RANS turbulence models is largely dependent on the exact case under investigation but flow solver settings do not transfer between configurations. Improvements in flow prediction can be achieved using scale-resolving simulation approaches, but due to the associated computational effort and compute requirements application of such tools is still out of scope.

Another bottleneck identified by the authors is the issue of uncertainty quantification (UQ), which is a driving requirement for aircraft certification. Current CFD solution show a large dependence of the results on solver settings and turbulence model selection; obtaining a quantitative and repeatable uncertainty for variations in geometry or flight condition is not possible with current RANS models.

The authors also raised the point of how to exploit the massive amounts of data generated by current CFD solutions. As mesh densities increase and scale resolving simulations are being conducted, it becomes increasingly difficult to analyse and correlate the resulting data and flow features and their effect on aerodynamic predictions.

The presentation concluded with four areas of opportunity for CFD in the design process: i) the ‘numerical wind tunnel’, when utilized in early project phases, allows for better informed decisions which potentially avoid costly fixes at later stages; ii) certification by analysis, which requires increased CFD fidelity and reduced solution uncertainty, which in turn will allow regulators to become familiarize themselves and gain confidence in the methods, and iii) use CFD to gain a better understanding of complex flow fields, leading to new insights that, likely indirectly, will flow into new aircraft designs. The authors also showed that current CFD validation studies lack the geometric complexity of real aircraft and questioned if more realistic geometries are needed to improve CFD model development.

2.2 Paper 1: Leppard: Current Use and Future Requirements of Computational Fluid Dynamics – A BAE Systems Perspective

The paper introduced the use of CFD tools in use at BAE Systems in the design process of their platforms. It pointed at the fact that during a product's life cycle, there are areas where improved CFD tools would provide significant benefits. The key concept for analysis was the simple equation that effectiveness=quality×acceptance, where 'quality' refers to numerical accuracy and physical realism and 'acceptance' encompasses usability, affordability, and timeliness of solution. The author pointed out a number of areas where CFD is currently being used at BAE Systems. Key opportunities were identified in improving the prediction of separated flows and using CFD for the prediction of stability and control characteristics. The author, like the authors of the keynote, recognized the possible prediction improvements when using scale resolving simulations but pointed out that these methods are not currently feasible at full vehicle scale. An area of particular interest was the affordable and accurate prediction of separated flows where current RANS based CFD methods fall short. In the area of stability and control predictions, CFD is hampered by a lack of confidence in the solutions and prediction uncertainty, especially in the computation of Reynolds number effects, dynamic derivatives, and control surface effects. The author showed examples from AVT-301 and AVT-316 to illustrate the shortcomings of current simulations. The authors provided a unique perspective on the subject area as they identified obstacles to the development of better models. The main challenges were noted as the lack of commonly available datasets for incremental validation of models and the slow adoption of new simulation technology by industry. As a remedy, direct collaboration between industry and academia was suggested.

2.3 Paper 2: Smith and McWaters: Aerodynamic Database Requirements for the Detailed Design of Tactical Aircraft: Implications for the Expanded Application of CFD

The paper introduced a historic view of the development time of recent fighter aircraft which showed that have significantly increased over the last six decades. This was contrasted with the requirement that development cycle needs to be cut to about five years (Dr. Will Roper, US Undersecretary of Defense, 2019). The paper continued to lay out the development challenges in today's fighter aircraft. These include the wide range of flight conditions and manoeuvres, complex control surfaces, and weapon carriage. From a fluid dynamic perspective, a number of phenomena were highlighted as key challenges: shock-boundary layer interaction, vortex flows and bursting, 3D boundary layers, serpentine inlets, and weapons bay acoustics. Assuming these flow fields can be accurately computed, the paper went on to estimate the computational cost of replacing current wind tunnel campaigns with CFD simulations during the design of an aircraft. While the details of the assumptions could be questioned, the analysis showed that replacing all wind tunnel entries with CFD simulations would incur approximately twice the cost of the wind tunnel testing. As the previous papers, the authors conclude that the validation of computational methods and significantly improved computational efficiency is necessary to allow CFD to 'buy itself into a program'. However, the authors also pointed to a number of design areas where CFD is advantageous: transonic forces and moments, store loads, air data, store separation, and complex flow interactions such as jet effects. The paper concludes that fighter aircraft design presents significant challenges because of the large flight envelope and the limited validation of CFD methods for relevant configurations. Improvements need to be made in the efficiency and accuracy of turbulent scale-resolving simulations for the prediction of separated flows.

2.4 Paper 3: Shaw et al.: Some Applications of Computational Aerodynamics to Support Guided Weapon Design and Development

This paper presented a change in application of CFD from combat aircraft to missiles. That said, the conclusions were the same as stated above: CFD simulations need to become better, faster, and cheaper to be useful in the design cycle. After a historic perspective of the use of CFD during the design cycle, showing that CFD analyses took longer than a design iteration (Era 1) in the early days of CFD but can now be accomplished multiple times within an iteration (Era 3), the paper continued to highlight some of the critical areas in missile aerodynamics where CFD predictions are deficient. The paper echoes the sentiments highlighted in the previous articles that improved RANS turbulence models are required to capture the relevant flow physics on a generic missile. It acknowledged the improvements possible with scale-resolving simulations but argued that using this simulation methodology puts CFD back in Era 1 because the simulations take substantially longer than current RANS methods. Therefore, the authors identified a clear need for a renewed focus on expanding the applicability of RANS models, requiring an open approach to model development with tight collaboration between industry and research and development partners. In addition, a need for high-quality validation and calibration experiments was identified. The paper concluded that for many applications current RANS methods are sufficient but scale resolving methods will have to be used when necessary. They also observed a tension between industry and the research and development community as industry needs to improve the performance and applicability of current modelling approaches with the concomitant time and resource requirements, whereas research and development seems to focus on models that better capture the flow physics with limited regard to the significant increase in expense of time and cost.

2.5 Paper 4: Rasthofer et al.: A Hybrid Continuum-Kinetic Approach for High Altitude Rocket Exhaust Plume Simulation This paper presented a change

The paper discussed the simulation of rocket exhaust plumes at high altitudes. To date, the majority of computational studies focus on low-altitude flights, where continuum approaches based on the Navier-Stokes equations can be used. In contrast, at high altitudes, kinetic approaches such as the Direct Simulation Monte Carlo (DSMC) method have to be used due to the rarefied gas flows. Because of the vast length scale differences in rocket exhaust plume flows, hybrid approaches, combining continuum methods for the flow in the vicinity of the nozzle exit with kinetic methods for the surrounding rarefied gas flow, provide a viable computational method.

The paper developed such an approach and stated appropriate transfer conditions on the interface, providing consistency in the thermo-chemical models. The authors showed the successful application of the method to a reacting rocket exhaust plume simulation at an altitude of 114 km. The paper served as a reminder that CFD simulations are not limited to continuum environments, although the vast majority of simulations are performed in that space.

2.6 Paper 5: Yuan et al.: CFD-aided Military Ship Design and Helicopter Operation

This paper addressed the application of CFD methods to the investigation of the flow over ships with a particular focus on the operation of helicopters in the ship airwake. The airwake behind the superstructure of ships is highly turbulent and presents a significant challenge for helicopter pilots, especially when added to the random six degree of freedom motion of the ship itself. The authors demonstrated the application of high fidelity CFD methods for this problem and noted that CFD is a critical ingredient for this investigation as simulations can be used to determine operational boundaries safely without having to contend with weather conditions. As in previous papers, the author highlighted the need for increased accuracy, reliability, speed, and affordability to use the full potential CFD simulations can offer. Of particular concern in ship CFD simulations is that in incompressible flow found in ship simulations, the computing time increases quadratically with the number of grid points as opposed to linearly for simulations at higher Mach numbers. This significantly increases the computationally cost of these simulations. The other unique characteristic of ship simulations, when compared to aircraft simulations, is the large size of the ship body, resulting in Reynolds numbers on the order of 10^8. Such high Reynolds numbers and geometry scales exacerbate the problem of CFD applications as scale-resolving simulation methods become extremely costly. The goal of these simulations is to provide flow field data for helicopter flight simulators. The simulation results showed that significant detail of the geometry are required to capture the wake flow characteristics. In addition, long time histories need to be computed to provide an adequate dataset and reliable time averages. With these two conditions met, the authors achieved very good comparison with the experimental data. However, the investigation showed that computational efficiency is highly geometry and mesh dependent. The addition of ship motion (heave, pitch, and roll) significantly increased compute cost as the motion is orders of magnitude slower than the frequencies in the airwake. The paper provided an illustrative example of the complexity and multitude of consideration necessary to optimize a simulation strategy for these flow fields.

3.0 DAY 2 (TUESDAY, 17 MAY 2022)

Day 2 of the workshop presented a transition from the industry perspectives to the presentation of concepts of CFD aided design and optimization frameworks. The papers and presentations are summarized below, followed by a synthesis of the discussion at the end of the day.

3.1 Keynote 2: Martins: CFD-based Aircraft Design Optimization

The authors presented a multi-disciplinary optimization (MDO) method that includes CFD flow solutions with a focus on aircraft design. The design framework is built on open-source tools communicating using a python framework. For a large number of design variables, the authors argue that gradient-based methods are essential due to the number of function evaluations required in the process. During the optimization process, the author showed non-viable solution that require an extremely robust CFD solver.

For optimization, the geometry is wrapped in parametrization curves with control points and geometry changes are seamlessly translated to the surface mesh using python tools. Another open-source tool is used to translate the surface deformations to the volume mesh for flow solution. The final step is the adjoint computation which is the most efficient method for large numbers of design variables. To highlight results of the optimization suite, the author showed wing optimizations for the NASA CRM wing for different flight conditions and constraints.

The author then explained the extension of the optimization framework to aeroelastic problems, where the adjoint method was extended to include the structural design parameters. The whole aeroelastic optimization is managed in the same python framework as the aerodynamic optimization outlined above.

The underlying philosophy for the approach to develop this framework was highlighted as the interaction between theory, implementation, and application, and the author argued that the application of a tool has to be an integral part of the development process to achieve maximum impact.

3.2 Paper 6: Taylor: The Pivotal Roles of Model Validation Maturity in Advancing the Capabilities and Adoption of CFD

The author discussed the role of validation maturity after asserting that industrial users need to know the accuracy of a simulation output and that this information should be available before a model is used. This notion was contrasted with the observation from the ASME Validation and Verification standard that such an a priori assessment of accuracy is only valid for a particular quantity at validation conditions. Outside of the validation condition, engineering judgement has to be used to assess model accuracy.

From this viewpoint, the paper discussed the components of a model validation activity and pointed out that model validation includes model accuracy quantification and model validation assessment, where metrics are based on cumulative repositories. In particular, when maturing the validation process, data provides a quantifiable basis for progressive refinement and, in the end, establishes a predictive capability of the model with accuracy quantification. The author also pointed out that for a model there is always a developer and a user perspective, the first seeking to improve the model as new data becomes available while the latter considers the model complete and ready for use. In order to provide the best validation, the author suggested a validation dialog that allows the user to better understand the model and, in turn, provide better insight and feedback. To develop this insight, numerical experimentation, presented using an example from AVT-316, was shown to provide significant value as e.g. the turbulence model is changed for the simulation of the flow for a given geometry and flow conditions. The author concluded with a recommendation to update current guidance for CFD validation and noted the opportunity to comprehensively include model validation in digital transform programs.

3.3 Paper 7: Stern et al.: Digital Design: The Way Forward

This paper discussed digital design methods as applied to ship design. It presented an overview of historic and current design methods and concluded this review with the assertion that current frameworks still limit the design space and remain very time consuming and expensive. The paper then pointed to recent studies, tests, and demonstration projects that are based on a simulation based design (SBD) framework and argued that current digital twin concepts supersede this approach.

With this background, the paper laid out a new initiative aimed at demonstrating the efficacy of a multi-fidelity, multidisciplinary simulation-based digital design process. The approach includes many components, starting with design requirement definitions, the inclusion of advanced physics, and concepts to handle big and sparse datasets obtained from experiments, low- and high-fidelity computational tools, operational data, and reduced order models. Further, machine learning (ML) and artificial intelligence (AI) methods will be included in the digital design process and the MDO component. The initiative also includes design assessment and decision making tools and culminates in virtual performance trials to assess designs.

Some examples were provided for a number of the components of the approach. The paper then showed details on how ML and AI is used to achieve dimensionality reduction of the parameter space and the development of surrogate models and describes the MDO approach. An example was provided that demonstrates how the digital design process works for a grillage panel, providing a significant weight reduction under strength constraints. The argument is made that ML can not only be used to obtain an optimal solution but also to understand how to improve the optimization process. The paper concluded stating that the greatest benefits will be achieved at the confluence of design, computational tool, and MDO, aided by AI and ML.

3.4 Paper 8: CFD Model of an AUV inspired by the Cownose Ray

The paper discussed the use of CFD to design underwater swimming systems with performance that resembles the manoeuvrability and efficiency of fish locomotion. The focus was on developing a bioinspired AUV design, modelling the swimming dynamics of rays because of the efficiency of their propulsion. Computationally capturing the motion of the Cownose Ray during forward motion, the simulations used the kinematics to investigate the effect of variation in frequency and amplitude of the motion. The investigation found the typical reverse von Karman Street downstream of the vehicle and concluded that the efficiency of the motion of the Cownose Ray is extremely high.

3.5 Discussion

This first round of discussion was attended by 42 workshop participants. The focus was on the lessons learned from the input of the industry presentations. As noted in the description of the individual papers, there was an overwhelming consensus that accuracy, fidelity, and cost are the drivers for increased CFD adoption by industry. The question if CFD will ever replace experiments was raised but the consensus was that both experimental and computational data provide value at the design stage. It was noted that model scale testing also has flaws, especially in the maritime arena where real world Reynolds numbers cannot be reached at model scale. Based on the flow features described in the presentations that pose problems for current simulation methods, the argument was made that combat aircraft design poses much more stringent requirements for simulations than transport aircraft because of flow field complexity and the size of the flight envelope. To alleviate the problem of uncertainty quantification, calibrated CFD methods were mentioned and those are easier to apply to transport aircraft than to combat aircraft.

Another consideration mentioned during the discussion was the goal of the simulations and the required fidelity. In particular, flight control development was discussed and it was pointed out that CFD is quite good with benign flow fields, mostly associated with low angles of attack. Once the nonlinear effects become important, current RANS modelling approaches are insufficient but scale-resolving methods are generally considered prohibitively expensive and do not necessarily improve flow predictions. However, CFD does have advantages over experimentation when complex flow scenarios, e.g. jet effects or control surface deflections, need to be investigated. In addition, determining dynamic derivatives is much more readily possible in CFD than in experiments, which was suggested as a focus area for CFD.

Another topic of discussion was the types of flows CFD development should target. Phrased differently, the question was which area of CFD application promises the biggest gains for industry. It was reiterated that the nonlinear effects at higher angles of attack proof to be a big hurdle. In order to improve CFD modelling, it was suggested to move away from standard test cases toward more complex and realistic geometries.

The discussion led to the observation that multi-disciplinary problems are an area where CFD simulations can have a big impact as many of these problems are very hard to study experimentally. It was also noted that while simulations can take a long time, wind tunnel testing also requires significant time, especially when model building, instrumentation, and test setup are considered. In addition, most of the wind tunnels are getting old and are not being updated, so the question was asked if the significance of wind tunnel testing will be decline simple due to the unavailability of the required resources.

It was iterated multiple times that affordability of CFD simulations is the biggest hurdle faced by industry, with a close second the modelling uncertainty in modern air and maritime vehicle flow fields. Finally, the difficulties of current models to reliably compute flows with significant separation were repeatedly noted.

4.0 DAY 3 (WEDNESDAY, 18 MAY 2022)

Day 3 continued the presentations of methods to integrate CFD into the design process.

4.1 Keynote 3: Mavriplis: Drivers for Industrial CFD: CFD Vison 2030, Grand Challenges, and Certification by Analysis

The authors started by presenting an outline of activities in the USA to provide a framework for accelerating CFD adoption in industry and improving CFD technology. The first, the CFD Vision 2030 study by NASA, was reviewed in terms of its goals and findings. Some of the critical problem identified include i) declining investment in basic research and development for simulation based analysis and design, ii) current inability to reliably predict turbulent flows and flow separation, iii) challenges of managing the vast amount of data generated by current and future simulations. The study placed an emphasis on physics-based predictive modelling, uncertainty quantification and error analysis, process automation, including integration of multi-disciplinary optimization, and efficient utilization of exascale computing platforms. The study produced the CFD Vision 2030 roadmap and provided a set of recommendations for NASA to help alleviate the problems identified above. This effort has led to the Grand Challenge Problems as benchmarks for the required step changes in engineering design capability.

The second study presented was Boeing's 20-year vision for virtual flight and engine testing, in particular the certification by analysis (CbA) portion. CbA is desirable because of its efficiency and the possibility to uncover design problems before flight test. In addition, analysis could lead to better designs and an acceleration of the product development. The challenges are the requirement that calculation based methods have to proof equal accuracy to experiments or flight tests. As pointed out earlier, complex flows are challenging for current simulation methods. Furthermore, CbA requires many full airplane and engine simulations, challenging computing and data storage resources. The study provided a roadmap for improving modelling capabilities and identified short term, mid term, and long term goals for simulation capabilities required in the CbA process. Key enabling technologies were identified as i) scale resolving simulation methods, ii) combustion/heat transfer modelling, iii) grid generation, iv) multidisciplinary integration, and v) solution confidence tools.

The authors stated that the AIAA CFD2030 Integration Committee has been formed to provide a community to leverage and integrate enabling technologies and communicate with other stakeholders. This committee has provided an updated CFD2030 roadmap and defined a number of Grand Challenge projects aligned with the NASA CFD Vision 2030 study.

In conclusion, the authors identified five technology focus areas and noted that the presented roadmaps from CFD2030, CbA, and the Grand Challenge roadmap are complementary. In addition, the authors pointed out that while the roadmaps address technical challenges, there are logistical and organizational challenges that need input from governments, academia, industry, software developers, and regulators to overcome.

4.2 Paper 9: Hirdaris: Hydroelasticity of Ships: Recent experiences with RANS CFD models

This paper gave a short overview of the history of fluid-structure interaction (FSI) in ship design and modelling. It continued by looking at the various modelling approaches in FSI simulations ordered by complexity and computational effort. The performance of current FSI models was shown for two model problems and the results indicate that two-way FSI coupling is essential to capturing experimental results correctly, although there is a sensitivity to the discretization method of the structure. The paper concluded that RANS CFD is possible for small scale model problems but may be problematic for full scale simulations. The author also pointed out the need for modelling the uncertainties.

4.3 Paper 10: Gueyffier et al.: SoNICS: A Novel Software Framework for Aerodynamics Simulations

This paper discussed the development of a new CFD software generation that leverages the experience gained from existing software to improve accuracy, robustness, and speed of the solver while also increasing flexibility and performance for the end user. The new solver was designed to include the Lattice Boltzmann method and to make use of modern heterogeneous computer hardware technology. The approach represents software as a graph which builds a user code by assembling generic operators. The current status includes cell- and node-centered codes for both CPUs and GPUs, achieving a speedup of 14 times over the current elsA code and a speedup of 100 times on a GPU over a CPU. The authors concluded that the new SoNICS framework is better aligned with user needs, has a larger feature scope than the current elsA code, and allows for new capabilities needed by current code users.

4.4 Paper 11: Morton: Bridging the Gap Between High Fidelity Multi-Disciplinary Simulations and the Design of Military Systems with Physics Based Surrogates

The paper presented a new way of looking at the use of computational tools in the design process. The underlying idea is that of a 'Decision Software Pyramid', which starts with data obtained from systems, flight, and ground tests that are consolidated in Data Driven Analytics (DDA) by machine learning (ML) or artificial intelligence (AI). In parallel, Physics Based Analytics (PBA) are developed based on CFD, computational structural dynamics (CSD), and other high fidelity computational models. Based on DDA and PBA, Digital Surrogates (DS) can be built that allow Decision Support Applications (DSA) to be utilized by engineers and operators.

A key point was that while large quantities of data exist for current systems, it is far too time consuming to interrogate that data when a decision needs to be made. In contrast, through DDA and PBA, a well-constructed DS can provide such information in real time. The challenge was identified as developing surrogates that retain the accuracy and fidelity of the underlying DDA or PBA information but are able to be executed exceedingly fast.

The paper provided an example of the implications of such a software tool in a vehicle design cycle. In the early design stages, a DS could be used to down-select configurations. As additional details of a design become available, they can be included in PBA to update the surrogate. Such a surrogate would allow for design optimization even in an MDO setting. The paper concluded that by injecting PBAs into traditionally used DDA allows for shorter development times and reduced system cost.

4.5 Paper 12: Ham: Addressing Challenges to Industrial Adoption of high-fidelity CFD: a Small Software Company Perspective

This paper presented the perspective of a small company to the challenge of industry adoption of high-fidelity CFD. The introduction listed many of the technical challenges mentioned above, especially the technical readiness of current models as well as time and resource constraints, but also emphasized management and bureaucratic hurdles.

To address issues of accuracy of numerical solutions, the paper noted that increasing formal accuracy is not enough to obtain solutions that match experimental data; instead, minimizing numerical dissipation is required. The paper also pointed out that to improve time constraints in the CFD process chain, all processes must be considered to improve the time to solution. For the resource constraint CFD solution, leveraging current compute technology, in particular, GPU-accelerated nodes, was noted as a critical step and performance metrics were shown. However, the paper noted that it is also important to improve parallel processing for mesh generation and mentions image based post-processing.

The paper also included examples of funding streams for small businesses and notes that work force development can be challenging and industry acceptance largely relies on someone at the OEM to champion software usage and application. In conclusion, the paper emphasized the barriers to large scale CFD adoption in industry listed above and includes the risk-averse culture at large OEMs as a bottleneck. In addition, the important role of governments for small business funding was acknowledged.

4.6 Paper 13: Stradtner et al.: Multi-fidelity Aerodynamic Data Set Generation for Early Aircraft Design Phases

The paper introduced a data set generation workflow motivated by the realization that enabling inclusion of CFD in the early design phases holds significant promise to design future fighter aircraft. The approach is characterized as a multi-fidelity aerodynamic data set generation workflow. It encompasses the complete design workflow from geometry modelling to mesh generation, flow solution, and includes surrogate modelling to make use of high fidelity and low fidelity flow solution tools. Current tools are coupled using an integrated environment that is designed to achieve comprehensive automation. The underlying geometry definition is based on the CPACS standard which allows for tailored geometry representations for specific tasks.

The flow solver library includes low-fidelity models such as empirical methods and lifting line solution as well as high-fidelity CFD methods. The approach uses design of experiment (DoE) methods for sampling and hierarchical kriging methods to compute flow solutions throughout the flight envelope.

The paper shows the example of the design of the DLR-Future Fighter Demonstrator (DLR-FFD) in two scenarios. The first application is to increase dataset fidelity over a range of angles of attack using targeted CFD simulations to improve the prediction of the lift-, drag-, and moment coefficients obtained from low-fidelity methods. The second example is a design iteration, which showed a strong pitch-up for a range of angles of attack. This phenomenon is well captured using the multi-fidelity model.

The paper concluded that multi-fidelity modelling is a good way to include CFD simulations in the early stages of aircraft design. Relatively few high-fidelity datasets are needed to update aerodynamic characteristics throughout the parameter space to verify design decisions. The authors noted that comprehensive automation and parametric modelling is important for the integration of multi-fidelity tools and for the development of multi-fidelity surrogate models.

5.0 DAY 4 (THURSDAY, 19 MAY 2022)

The last day of the workshop included the final keynote presentation and the second part of the discussion to include all presentation at the workshop, concluding with suggestions for a way forward.

5.1 Keynote 4: Couailler and Görtz: CODA: A European Perspective for a Next-Generation CFD, Analysis and Design Platform

The authors presented a collaboration for a next-generation aircraft design platform. The paper stated the current reliance on wind tunnel and flight tests for aerodynamic analysis and certification and the desire to apply CFD earlier in a design as well as in the certification process, as was also mentioned in keynote 1. The key bottlenecks stated in the paper echoed the observations made by other authors throughout the workshop, namely turbulence modelling for complex flows, including scale-resolving simulations, uncertainty quantification, robustness, and speed, but included the need for automated flow field analysis, variable fidelity and accuracy approaches, the seamless integration of CFD in MDAO tool chains, and a requirement for a synergistic approach to integrate simulation data with wind tunnel data. The motivation to develop a new code was drawn from the CFD Vision 2030 Study, noting the need to be able to operate efficiently on rapidly changing HPC architectures and the need to utilize multiple layers of parallelism and the vast number of GPUs available on today's systems. In addition, the paper identified algorithmic opportunities including discontinuous Galerkin methods and new developments of preconditioners, multigrid algorithms, and implicit schemes that allow for increased robustness, which leads to solution accuracy and performance improvements. The paper also recognized the need to move beyond monolithic CFD codes to allow for modern software engineering approaches and to simplify integration in MDA frameworks.

The new software approach is based on a number of existing codes and ongoing fundamental algorithm research, and key software libraries for pre- and post-processing and grid manipulation. The architecture relies on a Python control layer that allows plugins to handle individual tasks and exchange of data between those plugins. The paper continues with a description of a number of the currently available specialized libraries for a sparse linear solver, mesh technologies. The overarching software design philosophy is characterized by a plug-and-play and mix-and-match approach. Another aspect is the idea of hiding the compute system details within an 'HPC layer' which allows for simplified porting to new architectures. Overall, the approach envisions different levels of programmers and users interacting with the code and these users having different needs for access to different parts of the codes.

The paper then showed the status of a number of the pieces of the code, including the high-order unstructured solver, an overset grid approach, an immersed boundary method, local resolution refinement, multi-grid performance improvements, and applications to various geometries with the achieved performance improvements. In conclusion, the paper provided a perspective with a list of current activities for improving the performance and scope of the software suite and a roadmap for future developments.

5.2 Discussion

The second part of the discussion focused on how to overcome the identified gaps in applying CFD methods in aircraft design. Discussion participants largely agreed that better validation cases are necessary to improve current models. The discussion included a review of the Grand Challenge projects identified by the CFD 2030 study and a gap was identified as modern combat aircraft are not included in the challenge cases. While it was noted that the list can be appended, it was also pointed out that NATO STO would be the proper venue for such a project. There was some discussion about the complexity of the challenge problem and if certain smaller problems that address the bottlenecks identified during the workshop would be beneficial. In addition, it was pointed out that NATO STO workshop results were often archived rather than utilized to improve on test cases and results through multiple consecutive and refined research task groups. It was also suggested to scrutinize recent RTG reports to identify critical problems and possible solution. The example of AVT-301 was given where flow separation was identified as a problem but sufficiently well characterized comparison datasets are no available. With regard to validation experiments, the idea of a common research model for combat aircraft was presented and it was suggested to form an exploratory team to mine RTG data and to identify characteristics of a relevant research geometry. Because of the large flight envelope of relevant aircraft, an experimental dataset spanning relevant Reynolds and Mach numbers was suggested. To get some statistical information for uncertainty quantification, careful instructions will have to be provided for simulations of such a geometry. As for Reynolds number effects on combat aircraft, ET-225 was highlighted. While many agreed on this approach, the workshop organizers re-focused the discussion on the topic of 'bridging the gap between industry and developers.' The question was posed as to which geometries and solution methods were most relevant to industry. The procedures of the American Institute of Aeronautics and

Astronautics High Lift Prediction Workshop were presented as a possible solution since there are technical focus groups to determine which technologies are beneficial to address a particular problem. This could be adopted for future efforts to predict the aerodynamics of combat aircraft. There was also a suggestion to identify areas where CFD can be used right now vs. what areas need further method development.

6.0 CONCLUSION

The workshop brought together partners from industry, academia, and government research institutions. Over a four day period, the current state-of-the-art of CFD utilization in the military aviation and maritime industries was presented. In addition, a number of ongoing development efforts were shown that highlighted ways to better integrate CFD in the early vehicle design stages with some papers presenting applications of design frameworks to a variety of problems.

It was quite clear from the industry presentations that there is indeed a gap between current CFD method developments and industry needs. Developers are striving for improved predictions of complex flow fields, but bottlenecks identified by the papers from industry focused on solution fidelity, reliability, accuracy, timeliness, and cost. The importance of the latter two issues were viewed as not being sufficiently addressed by current developments.

Presentations by government institutions and industry-government partnerships, developing novel design frameworks combining new algorithms with a multi-fidelity simulation approach and surrogate models for aircraft aerodynamics, aimed at addressing the industry requirements within multi-fidelity frameworks that allow for targeted utilization of relevant simulation methods for a particular portion of the flight envelope. In my opinion, these approaches hold great promise to include high-fidelity simulations in the design process when such simulations are required.

However, there are still topics in predicting fundamental flow physics that remain open. In particular, prediction of the flow over modern combat aircraft at high angles of attack, and the associated separated flow and vortex development, is an area where more research is needed. While scale-resolving simulation methods present significant promise for these flows, they were shown to not necessarily provide improved results. In addition, such simulations increase the computational time and cost significantly when compared to current RANS methods. It remains to be seen if RANS models can be improved to include the relevant flow physics or if features of these flow are so geometry dependent that turbulence models cannot capture the phenomena.

While this research is ongoing, I believe that the highly integrated multi-fidelity approaches leading to the development of surrogate models are the most practical way forward, especially when the models are continually updated as new data becomes available. It is, however, a formidable challenge to develop and maintain these frameworks and to integrate them into the design toolchain of aircraft designers. The approaches presented in this workshop showed many similarities but differed in the details. The flexibility afforded by the modular design is crucial to allow for rapid adoption of new technologies and techniques at all stages of the design cycle.