A Summary on the State of Aerospace Computational Fluid Dynamics Technology

Mori Mani[1]*, Andrew J. Dorgan[2]

[1]Aerospace Computational Design Laboratory, Massachusetts Institute of Technology, Cambridge, Massachusetts, USA

[2]The Boeing Company, St. Louis, Missouri, USA

Mini Review

Received: 30-Aug-2023, Manuscript No. JPAP-23-111699; **Editor assigned**: 01-Sep-2023, Pre QC No. JPAP-23-111699 (PQ); **Reviewed**: 15-Sep-2023, QC No. JPAP-23-111699; **Revised**: 22-Sep-2023, Manuscript No. JPAP-23-111699 (R) **Published**: 29-Sep-2023, DOI:10.4172/2320-2459.11.4.002.

***For Correspondence:** Mori Mani, Aerospace Computational Design Laboratory, Massachusetts Institute of Technology, Cambridge, Massachusetts, USA

E-mail: mmani@mit.edu

Citation: Mani M. A Summary on the State of Aerospace Computational Fluid Dynamics Technology. Res Rev J Pure Appl Phys. 2023;11:002.

ABSTRACT

Over the past several decades, computational fluid dynamics has been increasingly used by the aerospace industry for the design and study of new and derivative aircraft. In this review we note the centrality of geometry and importance of turbulence models, higher-order numerical algorithms, and output based mesh adaptation. Challenges in each area are briefly noted. The review concludes with an outlook toward a future in which certification by analysis and model-based design are standard practice.

Keywords: Computational Fluid Dynamics (CFD); Geometry; Dynamics; Algorithms

INTRODUCTION

The aerospace industry continually searches for new technologies to provide better products at reduced cost and time to market. Computational Fluid Dynamics (CFD) is one such pivotal technology that the aerospace industry has invested in, and now witnesses the returns in terms of routine application. The advent of computing technology with sufficient power to solve less simplified forms of the governing equations of fluid dynamics gave rise to numerical methods and modern CFD in the early 1960s. Three-dimensional Euler and Navier-Stokes (N-S) solvers were developed in the 1980s as were codes based on the full potential equation, such as Boeing's TRANAIR code [1]. While TRANAIR is a valuable tool with significant capabilities for lift and drag prediction at cruise conditions and those with small regions of flow separation, it is not a general-purpose flow solver and, thus, other methods have also been developed and employed within Boeing that are suited to analyzing a larger portion of the flight envelope and for treating supersonic and hypersonic flows [2-3].

In the 1990s and 2000s, the aerospace industry regularly applied 3D RANS codes under conditions seen by an aircraft in cruise. CFD was also used heavily in design optimization, risk mitigation, augmentation of wind tunnel data (e.g., including the impact of model distortion or wind tunnel wall interference, providing Reynolds number corrections from model to flight scale), and diagnosing observations from flight tests (e.g., F/A-18E/F asymmetrical lift loss [4].

To minimize grid generation time and accelerate the overall CFD process, Boeing developed and matured unstructured mesh technology during the early 2000s. Mesh mechanics and flow solvers were built, including the finite-volume code BCFD and the finite-element code GGNS [5-7]. The maturation of unstructured mesh technology has fostered automation and significantly reduced the time an engineer spends on mesh generation, thus allowing a single individual to do the work that would have required a whole team of people in the past. The increased speed of grid generation, the automation of certain processes, the development of adjoint-based mesh adaptation and shape optimization, and the expanded access to significant HPC resources have brought CFD into a new era in which it is a critical and essential element of digital design and Model-Based Engineering (MBE).

LITERATURE REVIEW

The importance of CFD to the aerospace industry

CFD plays a critical role in the design and analysis of aerospace vehicles. Used early in the design process, CFD can reduce rework that would otherwise be needed to ensure the vehicle meets its objectives and safety related standards. The first aircraft of Boeing commercial airplanes that benefited from CFD from its inception was the Boeing 777. By the time Boeing 787 was being designed, CFD capabilities had matured enough to be applied to a much broader area of the aircraft design. Today, as CFD technology has continued to mature, it has become a standard tool of the aerospace industry alongside the wind tunnel. CFD has not, and likely will not, replace wind tunnel testing in the foreseeable future, although we have witnessed parity of these complementary methods, which both aim at a common objective: To accurately model the flight aerodynamics of an aerospace vehicle. While CFD can go from geometry to predictions of forces and moments in a matter of hours, it can take months to design and fabricate a wind tunnel model and plan the test; however, once the model is installed in the tunnel and the air is turned on, results can be rapidly collected for database generation. The aircraft design methodologies adopted over the last decade exhibit a specific advantage of computational methods over wind tunnel testing. Shape optimization technology and modern HPC resources have made it possible to digitally design an aircraft (not just components), leading to program-level schedule compression and cost savings, as well as a greatly reduced wind tunnel test campaign.

Verification and Validation (V&V)

To use any design tool with confidence one needs assurance of its validity. This of course applies to CFD tools, which notoriously have many user-controlled “Knobs” that can tremendously affect the resulting solution: Thus, the quality of the predictions can be highly dependent on choices by the analyst. This is not meant to suggest that a given technology needs to be perfect; rather, the accuracy bounds need to be understood. Setting these bounds and expectations is the job of V&V exercises. More precise definitions are given by Mani and Dorgan, but verification can be thought of as the process of determining that a particular numerical algorithm has been implemented as intended (i.e., is “Bug Free”), while validation is the process of determining if the numerical algorithm adequately replicates observations in the physical world.

Geometry challenges

One of the most fundamental inputs affecting the CFD simulation is the aircraft geometry. The geometry used for a CFD simulation is generally smooth, defeatured, and watertight. The defeaturing step requires some level of intuition into what features the flow cares about, and it is done to simplify the grid generation process and to accelerate the time required to generate a flow solution. A second challenge comes from the continued development and use of numerical aerodynamic shape-optimization tools. These methods may manipulate the geometry definition directly, but most often they instead manipulate the initial CFD mesh as a surrogate for the true geometry. In this case, the final mesh, which presumably embodies some desirable features, needs to be turned into a form interpretable by the CAD system. Another challenge arises from the increased interest and use of mesh adaptation, and this has heightened awareness of geometric fidelity. In mesh adaptation it is critical to project the adapted grid to the geometry rather than the initial surface mesh to avoid geometric deterioration during mesh refinement.

Mesh adaptation

As CFD utilization has rapidly grown, concerns regarding the dependence of a particular solution on the chosen mesh has similarly increased. CFD simulations of new problems can resort to mesh refinement studies to establish gridding strategies that yield solutions of sufficient accuracy relative to some stated requirement. An example of this approach, and the solution sensitivity that exists due to mesh resolution in various codes, was shown in the AIAA Drag Prediction Workshops and in the First High-Lift Prediction Workshop [8]. Alternately, one can develop automated mesh adaptation strategies that determine where best to place and orient the mesh. Obviously, this is not a trivial effort, though significant progress has been made, and remains an area of active research for both steady and unsteady flows.

Turbulence modeling

In all but the case of Direct Numerical Simulation (DNS), some form of model is required to represent the effect of fluid turbulence on the flow. For Large Eddy Simulation (LES) or Detached Eddy Simulation (DES), a portion of the turbulence is resolved (i.e., captured) both spatially and temporally. In these cases, the turbulence model (or subgrid model) represents the energy contained by structures smaller than some cutoff size. In industry, the most common approaches use the mesh spacing to set the cutoff, and the model is used to provide the effect that turbulent structures smaller than the mesh have on the resolved scales. When the temporal dynamics of the flow is not needed it is more efficient to directly solve for the time average of the flow *via* the RANS equations. This is the most common technique applied in industry today, but it puts a great burden on the turbulence model since every effect of the turbulence on the mean flow is the responsibility of the model. The current states of both the steady state and unsteady modeling approaches are briefly discussed below (for more info see Mani and Dorgan 2023). Steady simulations (Reynolds-averaged Navier–Stokes models)-at Boeing, the workhorse RANS models for design, analysis, and risk mitigation are primarily the SA model (Spalart and Allmaras) and its variations as well as and Menter’s Shear Stress Transport (SST) model (Menter). While all work well for attached flows, and flows with small amounts of separation, they can be inadequate for other situations dominated by large boundary layer separation, such as those seen at high angles of attack. This is unfortunate given the relative computational efficiency of RANS.

In addition to the advancements of the SA model, many other modifications have been studied, but rarely have they provided general improvements. One current and novel idea is the Macroscopic Forcing Method (MFM), which determines the "Closure Operators" that govern the mean-field mixing of a given flow field [9].

Unsteady simulations

Unsteady flow simulation is also essential for industrial applications, not just for treating regimes where RANS is insufficiently accurate but also for those cases where the time history or dynamics of the flow is of interest or for cases in which there is moving geometry. Within Boeing, this task in years past has fallen to DES where the near-wall flow (i.e., Boundary Layer) is treated with RANS while the off-body, separated flow is treated with an LES model [10]. Aside from the original DES model, which is based on the SA model, others based on SST have been conceived and applied [11,12]. While all these methods have been incredibly valuable there are some deficiencies in them as well. An approach that has shown promise in this regard (and that has been recently developed and tested for industrial application) is Wall-Modeled LES (WMLES). WMLES is a hybrid approach (LES is coupled with a wall model, similar to the well-known wall function treatment used in some RANS codes) and pushes the LES region into the boundary layer. There has been a substantial amount of research on equilibrium and nonequilibrium near-wall modeling for WMLES over the past decade [13-15]. Significant progress has been made, and the results are encouraging, but to make WMLES a routinely used tool for the aerospace industry, we need continued validation on different cases that illustrate mesh-independent predictions of the quantities of interest and studies that define where and when WMLES is the desirable modeling approach [16-19].

Challenges and outlook

CFD has become a mature technology that industry routinely relies upon as a standard tool. We have witnessed the successful numerical/digital design of aerospace vehicles in recent years, replacing (or at least supplementing) the experimental techniques of the past. Additionally, CFD is sometimes relied upon as the sole source of data for certain aspects of databases in the aerospace industry, including ground effects, throttle-dependent effects, and dynamic derivatives. Airframe loads have been developed using CFD and so have airframe and jet noise estimates. However, none of this is to say that all wind tunnel testing has ever been, will ever be, or should be replaced with computational approaches. Rather, we expect the two to continue to be complementary data sources in the creation of models of a vehicle's expected flight behavior.

While a great deal of progress has been made toward this state, the remaining challenges will require continued focus and investment in the years to come. Developing algorithms and refactoring codes to take advantage of emerging accelerator-based HPC paradigms (e.g., GPUs) are the current focus of several CFD development teams.

CONCLUSION

As these accelerated codes become commonplace, they will revolutionize how CFD can be used in design and certification processes. Steady-state calculations for building aerodynamic databases will be completed in days rather than weeks. Time-accurate simulations of highly separated flows, jet noise, inlet dynamic pressure distortion, etc. will become routine and conducted for many flight conditions, as compared to the relatively sparse treatment that can be afforded today. Utilizing accelerated HPC will also allow high-fidelity CFD to affordably fit into multidisciplinary analysis and optimization processes that today rely on lower-order models for flow physics. With this jump in computing power, new bottlenecks will be realized in grid generation, post processing, and analyzing and archiving data. Advances must also be made in those areas at a commensurate rate in order to capture the full potential of CFD as HPC resources improve.

REFERENCES

1. Johnson FT, et al. Thirty years of development and application of CFD at Boeing commercial airplanes, Seattle. Comput Fluids. 2005;34:1115-1151.
2. Bush R. A three dimensional zonal navier-stokes code for subsonic through hypersonic propulsion flowfields. AIAA J. 2012;1988-2830.
3. Buning P, et al. Numerical simulation of the integrated space shuttle vehicle in ascent. AIAA J. 2012;1988-4359.
4. Stookesberry D. CFD modeling of F/A-18E/F abrupt wing stall-a discussion of lessons learned. AIAA Pap. 2012;2001-2662.
5. Mani M, et al. A perspective on the state of aerospace computational fluid dynamics technology. Annu Rev Fluid Mech. 2023;55:431-457.
6. Mauery T, et al. A guide for aircraft certification by analysis. AIAA J. 2021;32:1598-1605.
7. Kamenetskiy DS, et al. Numerical evidence of multiple solutions for the Reynolds-averaged Navier–Stokes equations. AIAA J. 2014;52:1686-1698.
8. Rumsey CL, et al. Summary of the first AIAA CFD high-lift prediction workshop. J Aircr. 2011;48:2068-2079.
9. Mani A, et al. Macroscopic forcing method: A tool for turbulence modeling and analysis of closures. Phys Rev Fluids. 2021;6:054607.
10. Spalart PR, et al. A one-equation turbulence model for aerodynamic flows. AIAA Pap. 1992;6:439.
11. Strelets M. Detached eddy simulation of massively separated flows. AIAA Pap. 2001;879.
12. Winkler CM, et al. Refinement of a two-equation hybrid RANS/LES model in BCFD AIAA Pap. 2013;2013-3079.
13. Larsson J, et al. Wall-modeling in large eddy simulation: Length scales, grid resolution and accuracy. Annu Res Brief. 2010;2020:39-46.
14. Park GI, et al. An improved dynamic non-equilibrium wall-model for large eddy simulation. Phys Fluids. 2014;26:015108.
15. Bose ST, et al. Wall-modeled large-eddy simulation for complex turbulent flows. Annu Rev Fluid Mech. 2018;50:535-561.
16. Goc KA, et al. Wall-modeled large eddy simulation of an aircraft in landing configuration. AIAA J. 2020;2020-3002.
17. Lozano-Durán A, et al. Performance of wall-modeled LES for external aerodynamics. NASA. 2021
18. Lozano-Durán A, et al. Machine learning building-block-flow wall model for large-eddy simulation. J Fluid Mech. 2023.
19. Slotnick JP, et al. CFD Vision 2030 study: A path to revolutionary computational aerosciences. 2014;1992-439.