

High-Order Methods in NASA's Next Generation of Computational Fluid Dynamics Tools

David M. Schuster, PhD.
NASA Technical Fellow for Aerosciences
NASA Engineering and Safety Center
NASA Langley Research Center
Hampton, Virginia
USA

ABSTRACT

The missions of the National Aeronautics and Space Administration (NASA) routinely produce unique requirements and challenges for development and application of Computational Fluid Dynamics (CFD) methods. NASA presently embodies four distinct Mission Directorates: Aeronautics Research, Exploration Systems, Science, and Space Operations. These missions generate requirements for systems that operate in a wide variety of environments. They range from the high-speed flight of aerodynamically optimized vehicles operating in the earth's atmosphere to spacecraft designed for missions that don't favor aerodynamic optimization, some operating in the atmosphere of planets and planetary moons such as Mars and Venus or Saturn's moon Titan. Systems supporting these vehicles, such as rocket and jet propulsion, reaction control systems, fluid and thermal transfer systems, etc. can also generate their own unique set of flow phenomena that challenge today's CFD methodology.

Through the NASA Engineering and Safety Center (NESC), NASA annually conducts state-of-the-discipline assessments in fifteen distinct engineering disciplines. These assessments are performed by the NASA Technical Fellows that lead Technical Discipline Teams (TDT) of recognized experts in these fifteen areas. In the Aerosciences discipline, three topics have been identified as the top challenges for the discipline: aero-plume interaction prediction, unsteady separated flows, and aerothermodynamic prediction. These challenge areas are defined by the Agency's high-risk projects and problems on which the NESC is requested to perform independent testing, analysis, and assessments. When viewed as a whole, these tests, analyses, and assessments provide a clear view of the recurring technical challenges facing Agency engineers and researchers and can be used to guide future research and technology development.

The present state-of-the-art in the application of CFD at NASA is the use of Reynolds-Averaged Navier-Stokes (RANS) solvers, primarily executed in a steady-state mode of operation. In isolated cases, Unsteady RANS (URANS) solvers have been employed when steady RANS solutions produce poorly converging or oscillating results or in cases, such as aeroelastic analysis, which require unsteady aerodynamic simulation. For most traditional external and internal aerodynamic flows, structured overset grids or unstructured grids are employed to minimize geometric modeling and grid generation times. Grid adaptation, primarily as a series of coarse-grain intermediate processing steps is also seeing use on particularly complex flow problems and configurations. In the case of aerothermodynamic flows, engineers have been forced to continue to employ structured grid techniques as the present unstructured grid methodology has proven inadequate in the prediction of surface heating. In the area

of aero-plume interaction modeling, two-gas, frozen chemistry simulation is generally the state-of-the-art, with some production solvers capable of predicting flows with only a single gas component.

Prediction of flows falling into the afore-mentioned top Aerosciences technical challenges have severely stressed the present state-of-the-art in CFD prediction and for some problems, such as unsteady separated flows and aero-plume interaction cases, engineers have begun employing Large Eddy Simulation (LES) and Hybrid RANS/LES techniques. In some isolated aero-propulsion interaction cases, chemically reacting flow simulations have been applied. These methods are highly evolutionary and engineers have little experience in their application, so they cannot be heavily relied upon in today's application environment. Therefore, this leads one to muse over which numerical technologies will be included in the CFD tools that will be employed 30 years in the future.

This presentation will describe specific technical problems that have stressed NASA's traditional CFD methods to their breaking point and will link these issues to the Agency's top Aerosciences technical challenges. The discussion will then shift to the characteristics of future CFD solvers that will be required to attack these challenges and how these characteristics differ from the present state-of-the-art. High-order methods certainly appear to have a place in the development of future CFD tools and some of the physical characteristics of our most challenging problems suggest that high-order methods are the only way to effectively solve them. But there are some relatively severe implementation issues that face these methods, particularly in the area of general applicability and robust operation as an engineering tool. Desired characteristics of next-generation CFD solvers will be discussed and the author's view of which emerging numerical technologies might be employed to address these attributes will also be presented.