NASA SBIR 2020 Phase I Solicitation

A1.05 Computational Tools and Methods

Lead Center: LaRC

Participating Center(s): ARC, GRC

Technology Area: TA15 Aeronautics

Scope Description

Computational Fluid Dynamics (CFD) plays an important role in the design and development of a vast array of aerospace vehicles, from commercial transports to space systems. With the ever-increasing computational power, usage of higher fidelity, fast CFD tools and processes will significantly improve the aerodynamic performance of airframe and propulsion systems, as well as greatly reduce non-recurring costs associated with ground-based and flight testing. Historically, the growth of CFD accuracy has allowed NASA and other organizations, including commercial companies, to reduce wind tunnel and single-engine component tests. Going forward, increased CFD fidelity for complete vehicle or engine configurations holds the promise of significantly reducing development costs by enabling certification by analysis. Confidence in fast, accurate CFD allows engineers to reach out of their existing design space and accelerate technology maturation schedules. NASA’s CFD Vision 2030 Study (https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20140003093.pdf) highlighted the many shortcomings in the existing computational technologies used for conducting high-fidelity simulations and made specific recommendations for investments necessary to overcome these challenges.

One area of research is scale-resolving numerical simulations, which are playing an increasing role in CFD analysis of new and existing aerodynamic configurations at off-design conditions. It is well-known that traditional steady-state Reynolds Averaged Navier-Stokes (RANS) analysis performs poorly in separated boundary layers and shear layers. Time-accurate Wall-Modeled Large Eddy Simulation (WMLES) and hybrid RANS-LES have demonstrated increased accuracy for a subset of these flows where large scale fluctuations are computationally resolved while the near-wall small scale fluctuations are modeled. Since RANS can accurately compute attached flow regions, it is desirable from a computational cost perspective to initiate scale resolving simulations just upstream of the separated flow regions. However, unsteady disturbances must be added to kick-start scale resolving simulations but an accurate and robust approach to accomplish that is lacking. The goal of this solicitation is to overcome this deficiency. One approach is to insert synthetic turbulent eddies at the start of the scale-resolving flow domain to effectively “trip” the flow. Several methods have been reported in the literature to generate these turbulent fluctuations, but these are not general enough to apply to realistic aircraft configurations, do not evolve into resolved physical turbulent structures in a reasonable amount of time/space, or cause large acoustic fluctuations rendering them inapplicable to aero-acoustic analysis.

An ideal turbulence generator for embedded scale-resolving simulations targeting hybrid RANS-LES and wall-modeled LES would satisfy the following criteria:

- Easy to implement/apply to general aircraft/rockets configurations locally embedded within a larger CFD
Use existing upstream RANS data (e.g., using the Spalart-Allmaras turbulence model), such as velocity profile and estimated Reynolds stresses, and little to nothing else in terms of user parameters.

Develop into realistic turbulence under 10-15 boundary layer thickness from the plane (or volume) where it is applied (based on first order statistics and two-point correlations).

Require little to no change to an existing scale-resolving flow solver independent of grid paradigm (unstructured, structured overset, or Cartesian).

Properly handle the inner region of hybrid RANS-LES and WMLES simulations leading to fast skin friction recovery within 10-15 boundary layer thickness.

Create negligible acoustic fluctuations (i.e., smaller than magnitude of attached wall-bounded turbulence).

What is being solicited is a “plug and play” software module that could be easily inserted in an independent CFD solver (e.g., a NASA code) to provide necessary input for scale resolving simulations. The awardee will demonstrate the software tool for carefully selected relevant test cases, before delivering it to NASA.

References

https://www.nasa.gov/aeroresearch/programs/aavp

https://www.nasa.gov/aeroresearch/programs/tacp

Expected TRL or TRL range at completion of the project: 3 to 6

Desired Deliverables of Phase II

Analysis, Software, Research

Desired Deliverables Description

The deliverable will be a software tool that could be used in conjunction with computational fluid dynamic solvers to perform scale resolving simulations that are relevant to NASA missions, particularly the Aeronautics Research Mission Directorate (ARMD) where this capability is needed for flow control applications, aircraft maximum lift prediction and certification by analysis. The awardee will demonstrate the developed computational tool on relevant aerodynamic configurations before delivering it to NASA.

State of the Art and Critical Gaps

NASA’s CFD Vision 2030 Study identified several impediments in computational technologies and this solicitation addresses one of those related to application of scale resolving simulations needed for expanding the scope of application of CFD across the aircraft flight envelope, particularly in the prediction of maximum lift.

Relevance / Science Traceability

Various programs and projects of NASA missions use computational fluid dynamics for advanced aircraft concepts, launch vehicle design and planetary entry vehicles. The developed technology will enable design decisions by ARMD and Human Exploration and Operations Mission Directorate (HEOMD).