NASA SBIR 2017 Phase I Solicitation

A1.05 Computational Methods & Tools - High Fidelity Mesh and Geometry Tools

Lead Center: LaRC

Participating Center(s): AFRC, GRC

Technology Area: TA15 Aeronautics

During 2012-2014, NASA sponsored a study aimed at determining future directions for Computational Fluid Dynamics (CFD) research that would subsequently enable significant advancements in aeronautics. This study (CFD 2030 Study: A Path to Revolutionary Computational Aerosciences (http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20140003093.pdf), noted many shortcomings in the existing technologies used for conducting high-fidelity simulations, and made specific recommendations for investments necessary to overcome these challenges. Chief among the recommendations was the need for robust higher-order discretization schemes, scalable solvers on high-performance computing (HPC) platforms, and adaptive h-p mesh refinement. It was recognized that the generation of meshes suitable for high-accuracy CFD simulations constitutes a principal bottleneck in the simulation workflow process as it requires significant human intervention. Similarly, providing access to the underlying geometry definition, as needed for both high-order simulations and adaptive gridding, is currently not available and is a further roadblock to the ubiquitous use of these technologies.

In the area of geometry definition, a critical need arises from the fact that laser scans of the aircraft surfaces are often used to generate computer aided design (CAD) files. This often leads to surface geometries with unphysical surface waviness as part of the surface fitting process of the laser scan point clouds resulting in poor CFD meshes and, hence, erroneous CFD solutions. Also, in some cases the CAD geometry is only represented as a tessellated surface while continuous and differentiable surfaces are needed for meshes that would yield accurate solutions and accurate mesh refinement results. A tool that could remove unphysical surface waviness as well as fit tessellated CAD surfaces and output a continuous and watertight spline surface is needed for practical applications.

To enable accurate CFD solutions, proposals are solicited in two areas:

- To develop robust means for generating meshes suitable for high-order accurate flow solvers, that can be demonstrated not to compromise the accuracy of the simulations. The three-dimensional unstructured grid tool developed during this research effort should be capable of creating mixed-element meshes. In conjunction with these meshes, geometry information must be easily accessible in a heterogeneous distributed computing environment through well-defined, yet lightweight, Application Programming Interface (API).
- To develop a CAD tool that could generate high-quality, continuously splined surfaces free of unphysical waviness and tessellated faces.

The new capability will be demonstrated for configurations of interest to NASA aeronautics (http://www.aeronautics.nasa.gov/programs.htm) in terms of accuracy, speed and robustness. The proposers must present a convincing case that the proposed approach has the potential of meeting these metrics.

Phase I research is expected to develop the technology and demonstrate it on relatively simpler configurations, while Phase II will increase the technology readiness level and include demonstration on more complex configurations.

Note: This subtopic is focused on addressing high fidelity meshes and geometry tools as they relate to large scale, complex fluid dynamics simulations. If you are interested in proposing to the broader topic of computational technologies addressing emerging high performance computing hardware, you should NOT propose to this subtopic but instead view subtopic ID# S5.01.