



NASA STTR 2009 Phase I Solicitation

T8 Computational Fluid Dynamics (CFD) Mesh Creation

NASA's work in advanced aeronautics and space vehicle development relies on Computational Fluid Dynamics (CFD) codes such as FUN3D, which numerically solve equations of fluid motion over a discrete mesh of points in three dimensions. Extensive CFD modeling is required for a wide range of NASA missions, which include subsonic commercial aircraft, rotorcraft, supersonic and hypersonic vehicles, and planetary exploration vehicles.

Subtopics

T8.01 Computational Fluid Dynamics Mesh Creation

Lead Center: LaRC

A critical step in CFD modeling is mesh creation. A judicious placement of mesh points is required to optimize computing efficiency while maintaining a specified level of discretization accuracy. This placement is further constrained by the need to capture disparate characteristic length scales and flow feature orientations. A result of these constraints is that many mesh elements formed by connecting points can have very high aspect ratio -- $O(10,000:1)$ or more. Rapid generation of such a mesh and its subsequent adaptation to better resolve the problem physics and reduce discretization errors are critical to the application of CFD to complex real world problems of interest. While current meshing methods, those using advancing front/layer, and/or Delaunay algorithms, have been successfully applied to complex problems, additional research and development is needed in the area of mesh generation to reduce human involvement and increase robustness.

Proposals are sought, resulting in the development and improvement of software packages for high-aspect ratio, three-dimensional meshing and re-meshing. The mesher must accommodate cell aspect ratio requests of at least 10,000:1 even in the presence of a curved metric tensor field to enable high Reynolds number finite-volume Computational Fluid Dynamics applications. In regions of high anisotropy, mesh cells should be layers of semi-structured hexahedra or triangular prisms to allow non-dissipative capture of bow shocks, boundary layers, free shear layers, wakes, contact surfaces, and so forth. Furthermore, to provide uncertainty estimates for the computational results, the mesher should enable mesh adaptation, whereby an existing mesh is adapted to improve the solution based on the problem physics and/or a solution error estimate.

