Dimitri J. Mavriplis
ICASE
Nasa Langley Research Center
Hampton, VA

Abstract:

Unstructured Mesh Related Issues in Computational Fluid Dynamics (CFD) – Based Analysis and Design

The use of unstructured meshes for computational fluid dynamics problems has gained widespread acceptance over the last decade with the emergence of fast and robust grid generation packages and the continuous improvement of CFD flow solvers. This talk will look beyond the step of initial mesh generation for CFD analysis problems, towards other mesh related issues which are involved in current and future large-scale simulations based on parallel computing architectures. These include techniques such as adaptive meshing and dynamic load balancing, as well as fast parallel unstructured multigrid solvers. Examples of implementations of these methods and their use in parallel steady-state applications will be given. For unsteady flow simulations, techniques for moving meshes as well as overlapping unstructured meshes will be discussed and their implications for parallel computing will be addressed. Additional mesh related issues which arise in the context of design optimization problems, such as the requirement to obtain grid sensitivities with respect to design variables will also be discussed. Finally, the use of higher order methods (higher than order 2) in CFD could dramatically affect the requirements of future mesh generation schemes, and the implications of this trend will be examined.