46th AIAA Aerospace Sciences Meeting and Exhibit
7 - 10 January 2008, Reno, Nevada

AIAA 2008-1017

Parallel Adaptive Mesh Refinement Scheme for Three-Dimensional Turbulent Non-Premixed Combustion

X. Gao* and C. P. T. Groth†

University of Toronto Institute for Aerospace Studies
4925 Dufferin Street, Toronto, Ontario, M3H 5T6, Canada

A parallel adaptive mesh refinement (AMR) algorithm is described for predicting turbulent non-premixed gaseous combusting flows in three space dimensions. The Favre-averaged Navier-Stokes equations governing a reactive mixture of thermally perfect gases, the two transport equations of the k-ω turbulence model, and the time-averaged species transport equations, are all solved using a fully coupled finite-volume formulation on body-fitted multi-block hexahedral mesh. The numerical algorithm adopts a cell-centred upwind finite-volume discretization procedure and uses limited solution reconstruction, approximate Riemann solver based flux functions to determine the inviscid (hyperbolic) flux at cell interfaces. The viscous (elliptic) components of the cell face flux are evaluated by employing a hybrid average gradient-diamond path approach. For the treatment of near-wall turbulence, both low-Reynolds-number and wall-function formulations of the k-ω model are used, with a procedure for automatically switching from one to the other, depending on mesh resolution. A flexible block-based hierarchical octree data structure is used to maintain the connectivity of the solution blocks in the multi-block mesh and facilitate automatic solution-directed mesh adaptation according to physics-based refinement criteria. This AMR approach allows for anisotropic mesh refinement and the block-based data structure readily permits efficient and scalable implementations of the algorithm on multi-processor architectures. Numerical results for turbulent non-premixed methane-air diffusion flames are described to demonstrate the validity and potential of the parallel AMR approach for predicting complex combusting flows.

I. Introduction

With recent advances in computational fluid dynamics (CFD) and numerical methods for combusting flows, as well as advances in high-performance-computing hardware, numerical modelling has become an important powerful and effective tool for the design of advanced combustion systems. The importance of numerical modelling has increased with the increasingly stringent emission legislation imposed by governments worldwide. The latter has made the combustor and engine design process much more challenging. As virtually all practical combustion systems involve turbulent combustion and pollutant and particulate emissions are controlled by the details of the turbulent fuel-air mixing and combustion processes, a detailed understanding of the strong nonlinear interaction between the turbulent flow structure, chemical kinetics, and thermodynamic properties of the reactants and products is required for obtaining improved low-emission combutor designs.

Three primary tools for performing simulations of turbulent combusting flows have emerged: (i) direct numerical simulation (DNS); (ii) large-eddy simulation (LES); (iii) and Reynolds- or Favre-averaged Navier-Stokes (RANS) simulation techniques, each possessing various advantages and disadvantages.[1,2] In DNS, all of the turbulent and chemical length and time scales are fully resolved. For this reason, DNS is a powerful tool for studying turbulent flame structure and turbulence/chemistry interactions in detail. It is mainly reserved for understanding the basic processes of combustion phenomena, such as extinction and re-ignition, flow and

*PhD Candidate, Email: gao@utias.utoronto.ca, Student Member AIAA
†Associate Professor, Email: groth@utias.utoronto.ca, Senior Member AIAA