Reacting Flows and Vortices

Enterprise
Rolls-Royce

Coordinator
Anne Bourlioux
Department of mathematics and statistics
Université de Montréal

Reference
An article on this topic is available on the problems page and can be downloaded.

Abstract

Computational Fluid Dynamics [CFD] is widely used in industry to solve fluid flow problems. These problems can range from fairly simple flows through building ventilation systems to very complex, multiphase, reacting flows in gas turbine engines. The limitations encountered in using CFD are still related to computing power. Even though we have made significant progress in the last two decades, today's computers are still not fast enough to solve completely industrial reacting flows. Many simplifications and averaging computations must be made in order for meaningful solutions to be obtained within a reasonable time.

In reacting flows, the main simplification involves the chemical processes. Even for the simplest methane-air flame, the actual chemistry may involve dozens of chemical species and hundreds of reactions. The rate equations governing these reactions are very stiff ordinary differential equations whose solution would require very small numerical time steps; these steps are too small to be practical.

One promising approach in modelling reacting flows is to uncouple the chemistry from the flow equations. In this way a larger time step could be used to capture the larger-scale fluid dynamics and the chemical processes could then be solved separately using specialized software. If one uses this approach, the challenge is to make sure that the chemistry solver has all of the fluid dynamic information that it requires. Of particular interest are the regions in the flow where large and small vortical structures are formed. Once the regions have been properly identified, important thermodynamic data from them (such as temperatures and residence times) can be obtained and transmitted to the chemistry solver.

This project will be focused on the development of a methodology that may be used to obtain such quantitative data from a CFD solution for a gas turbine combustor. Starting from the article "An objective definition of a

vortex", by G. Haller (J. Fluid Mech. (2005), Vol. 525, pp. 1-26), we hope to develop a method that will be used on typical CFD flow field results.