MTCA'09

ABSTRACTS

Modelling "Steady" and Unsteady Aerodynamic Flows Using High-Order Methods

Dimitris Drikakis

Cranfield University

The talk presents the development and implementation of high-resolution and high-order methods for aerodynamic flows featuring vortex generation, transition and turbulence separation and shock/boundary-layer interaction. The focus of the presentation is on implicit large eddy simulation (ILES), however, case studies in the framework of the classical RANS and DES approaches will also be discussed. Moreover, the implementation of these methods in conjunction with unstructured grids will be presented. The case studies will include flows around swept wings, high-lift devices, subsonic/transonic wings with and without winglets, transonic cavities, and helicopter flows.

Tailored Unstructured Meshes for Efficient Simulations

Oubay Hassan and Kenneth Morgan

Swansea University

The advantages of unstructured mesh methods are well known and centre, mainly, on the observation that they provide a powerful tool for the rapid discretisation of domains of complex shape. A standard method for generating appropriate meshes for viscous flows is to begin by constructing stretched structured layers adjacent to solid surfaces. The remainder of the computational domain is then discretised in a consistent fashion, using an automatic unstructured mesh generation procedure. The isotropic Delaunay method, with automatic point creation, provides the fastest method of generating high quality unstructured meshes. However, the outer layer of the elements generated by the advancing layers method often includes stretched element faces and these can cause significant boundary recovery problems when the mesh is presented to a standard Delaunay generator. The remedy, for problems of this type, is to employ an alternative Delaunay triangulation, with a modified in-circle criterion, and the effectiveness of this approach will be demonstrated.

The simulation of compressible viscous flow over a high speed ground vehicle will then be considered. Problems such as engine boundary conditions, the intake design and the modelling of the interaction between a rotating wheel and the ground will be addressed and the results of full vehicle calculations will be presented.

Practical viscous flow simulations, undertaken with classical low order solution techniques, typically require the use of a significant number of points in the boundary layer regions, if the desired level of accuracy is to be reached. Normally this requires that about 70% of the grid points are used to resolve the viscous dominated regions. High order discretisation techniques offer the promise of reducing the number of points necessary to resolve these regions and an approach designed to lead to a high order local extension to a standard solution algorithm will be presented. The method is readily implemented and uses the structure in the mesh generated by the advancing layers method to produce a higher order discretisation in the direction normal to the solid surfaces. The results of some initial studies will be included to demonstrate the numerical performance of the proposed approach.

Co-volume methods, which are staggered in both time and space, exhibit a high degree of computationally efficiency, in terms of both CPU and memory requirements compared to, for example, a finite element method. Despite the fact that real progress has been achieved in unstructured mesh generation methods since late 80s, co-volume schemes have not generally proved to be effective for simulations involving domains of complex shape. This is due to the difficulties encountered when attempting to generate high quality meshes, satisfying the mesh requirements necessary for co-volume methods. A newly developed approach for generating two-dimensional and three-dimensional meshes satisfying these criteria and produces a high quality triangulation enabled the triangulate a number of domains of practical interest. The generalized Yee's scheme for the solution of the electromagnetic scattering is used to demonstrate the effectiveness of the co-volume method on practical industrial geometries when appropriate meshes are generated.

Algorithm Issues Associated with Extending CFD Applicability to the Full Flight Envelope

Forrester Johnson

The Boeing Company

Over the last two decades CFD has gradually joined the wind tunnel and flight test as a primary flow analysis tool of aerodynamic designers. CFD is now acknowledged to provide substantial value and has even created a paradigm shift in vehicle design, analysis, and support processes. Experience to date has shown that CFD has had its most favorable impact on the aerodynamic design of the high-speed cruise configuration of a transport. This success has raised expectations among aerodynamicists that CFD can become a routine tool for loads analysis, stability and control analysis, and high-lift design processes. In fact, there is considerable thought that it is now feasible to populate the large databases involved with results from Navier-Stokes codes. However, the flight conditions of interest are frequently characterized by complex flows, particularly large regions of separated flows. For example, such flows are encountered on transports at low speed with deployed high-lift devices, at structural design load conditions, or when transports are subjected to in-flight upsets that expose them to speed and/or angle of attack conditions outside the envelope of normal flight. Solutions involving complex flows often involve complex geometries as well, e.g., vortex generators, chines, spoilers, landing gear, etc. Often 30 to 200 million grid points and enormous numbers of iterations are now required to obtain reasonable accuracy. Moreover, grid and even residual convergence are rarely achieved, sometimes leading to considerable confusion and rework downstream. In addition, it remains to be seen whether current turbulence models can handle flows that have considerable smooth surface separation. In one form or another, these problems were also characteristic of the early days of cruise analysis and design CFD codes. They were ultimately overcome by the development of robust discretizations with improved accuracy, powerful numerical solvers, adaptive grid refinement algorithms, non-equilibrium turbulence models, commodity computing and years of user experience. This talk discusses requirements for full flight envelope CFD codes as well as the need for algorithm breakthroughs and possible avenues of research.

Reliability of High-Order Approximations in Aerodynamic Computations on Coarse Meshes

Natalia Petrovskaya

University of Birmingham

A modern toolkit for computational aerodynamics problems appears as a collection of numerical methods where several major techniques are supported by a number of minor algorithms. One class of computational problems that clearly exposes the enormous complexity of a modern aerodynamic solver is computations on adaptive grids where thorough validation of every algorithmic piece of a CFD code used in solution-adaptation iterations is required.

For many numerical algorithms exploited in CFD computations their reliability is based on error estimates that depend on a grid cell size. Those asymptotic error estimates may not work on coarse meshes generated at the initial stage of a solution grid adaptation procedure. A numerical solution on coarse meshes is an essentially discrete function which properties can be significantly different from its continuous counterpart. Thus error control on coarse meshes becomes a challenging problem that should be addressed when a solution grid adaptation procedure is considered.

One part of the problem is that algorithms exploited in a CFD code are often validated independently and separately from each other. As a result, a mesh considered as a fine mesh for one numerical method can still be a coarse mesh for another algorithm if both of them are used together in a CFD solver. Meanwhile, a failure to control the error of a particular numerical method on a coarse mesh can make a negative impact on the entire solver at the next solution-adaptation iteration and can finally result in a divergent numerical solution. Thus in our talk we discuss difficulties associated with CFD computations on coarse meshes in a solution grid adaptation procedure. Several examples related to specific computational algorithms will be considered.

Mesh Movement, Adaptation and Feature Alignment

Ning Qin

University of Sheffield

Some recent developments in meshing techniques for computational aerodynamics will be presented, including an efficient mesh movement method, a buffer layer approach for connecting multi-block structured mesh and effects towards feature alignment effective mesh adaption. The interaction of mesh topology with flow solution accuracy, convergence and robustness will also be discussed.

Bridging the Gaps: H to P Efficiently and CG to DG Transparently

Spencer Sherwin

Imperial College

Hybridization through Lagrange multipliers combined with a Schur complement procedure (often called static condensation in the context of continuous Galerkin linear elasticity computations) has in various forms been advocated in the mathematical and engineering literature as a means of accomplishing domain decomposition, of obtaining increased accuracy and convergence results, and of algorithm optimisation. Recent work on the hybridization of mixed methods, and in particular of the discontinuous Galerkin (DG) method, holds the promise of capitalising on the three aforementioned properties; in particular, of generating a numerical scheme that is discontinuous in both the primary and flux variables, is locally conservative, and is computationally competitive with traditional continuous Galerkin (CG) approaches.

In this presentation we present both implementation and optimisation strategies for the Hybridized Discontinuous Galerkin (HDG) method applied to two-dimensional elliptic operators. We implement our HDG approach within a spectral/hp element framework so that comparisons can be done between HDG and the traditional CG approach. We demonstrate that the HDG approach generates a global system for the Lagrange multipliers that although larger in rank than the traditional static condensation system in CG, has significantly smaller bandwidth at moderate polynomial orders. We show that if one does not consider set-up costs, above approximately fourth-degree polynomial expansions on triangles and quadrilaterals the HDG method can be made to be faster than the CG approach, making it attractive for time-dependent problems. Interestingly the break even point of fourth-degree polynomials is also the break even point at which global operations in CG formulations have been observed to become more expensive than elemental operations.