Hybrid simulation and wall modelling of flows separating from highly-swept wing

N. Li, M.A. Leschziner
Sponsors: EPSRC, BAE Systems

Sweep is a feature of all modern wings. Relatively low sweep is used predominantly at moderate speeds, while large sweep is appropriate to high-speed flight. In both cases, a high angle of attack provokes separation, often from the relatively sharp leading edge. The nature of the separation process and the flow structure depend sensitively on the sweep: at low sweep, closed recirculation zones tend to form, while at high sweep separation is ‘open’ and characterized by streamwise oriented vortices. In the extreme case of a highly-swept, delta-type plan, strong vortices emanate from the leading edge, which are initially non-turbulent over most of their radial extent and then may undergo a vortex breakdown with consequent high levels of turbulence. The objective of the present study was to explore the potential and limitations of LES and LES-RANS hybrid methods for predicting separated flow from swept wings. This work is complemented by experimental measurements at University of Manchester.

The computational approach taken is based on the adaptation of a multi-block, body-fitted finite-volume LES method, combining second-order spatial discretisation with a fractional-step time-marching scheme and a multigrid solution of the pressure-Poisson equation. For the geometry of Fig. 1, at the experimental Reynolds number of 210,000 (based on wing-root chord), a fully wall-resolving simulation was estimated to require about 300 million nodes within a block-structured H-topology method. This grid density could not be accommodated with the resources provided, quite apart from the fact that industrial constraints would not permit such grids. The finest grid that could be used by Imperial College contained around 23.6 million nodes. With this grid, the wall-nearest nodes were determined to lie within a universal wall distance y^+<22 over virtually the entire surface of the wing. Hence, a RANS-type method had to be developed and applied the near-wall region to be coupled to the outer-field LES. This was done partly within this project and partly within a contemporaneous EU-Framework-6 project.

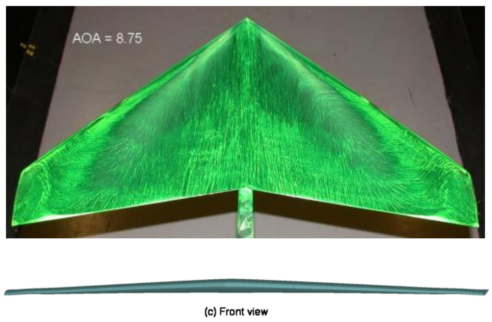

Fig. 1: Geometry of the simulated wing

Two LES-RANS schemes were developed at Imperial College, taking advantage of a previously developed general multi-block LES scheme with dynamic subgrid-scale modelling. These are documented in Temmerman et al (2004) and Tessicini et al (2006, 2007). The one used for the wing geometry is a zonal method involving the superposition of a fine near-wall RANS grid on top of the background LES grid. Fig.

2 provides a graphical representation of the method, in particular the manner in which the two zones are coupled by transfer of numerical information. Successful applications of this method range from simple channel flow to complex three-dimensional flows separating from a hill-shaped obstruction in a duct. Within the RANS layer, parabolized RANS equations are solved, with pressure and velocity at its upper boundary extracted from the LES. The RANS solution results in a wall shear stress that is fed back to the LES as a boundary condition.

In addition to the development of the scheme and its testing by reference to simple shear flow in a channel, the scheme was extensively tested in a physically-complex flow other than that around the wing, but closely related to it in terms of the physical and computational challenges involved. That geometry, a hill-shaped obstacle in a duct, is shown in Fig. 3. This particular configuration, at a Reynolds number of 130000, based on the hill height, was chosen primarily because of the availability of highly detailed LDA and HWA data generated at Virginia Tech (Byun and Simpson (2005)) specifically to assist the validation of computational schemes for 3d turbulent, separated flow. A key advantages offered by the geometry is that its predicted gross flow features depend sensitively on the accuracy with which separation is predicted. The separation is highly three-dimensional and involves the formation of two focal points on the leeward side of the hill from which vortices emanate and evolve within a complex wake. Simulations were performed for a sequence of meshes, ranging from 1.5 million nodes to 10 million nodes, the finest being nearly wall resolving. The objective of this sequence was to examine the efficacy of the zonal RANS-LES schemes, both by comparison with experimental data and by reference to highly resolved pure LES. Results from this study are reported principally in Tessicini et al (2007), demonstrating in particular the advantages of the zonal scheme in coarse-grid simulations. The near-wall physics of this flow are very complex, and ongoing wall-resolving simulations with grids of 37 million nodes (Li and Leschziner (2007)) have revealed, in particular, highly intricate reverse-flow behaviour within the buffer region of the near-wall layer. A disadvantage of the flow is, however, that the inflow contains an influential boundary layer for which elaborate precursor simulations needed to be performed in order to prescribe the unsteady properties in this layer, as part of the prescription of the inlet boundary conditions.

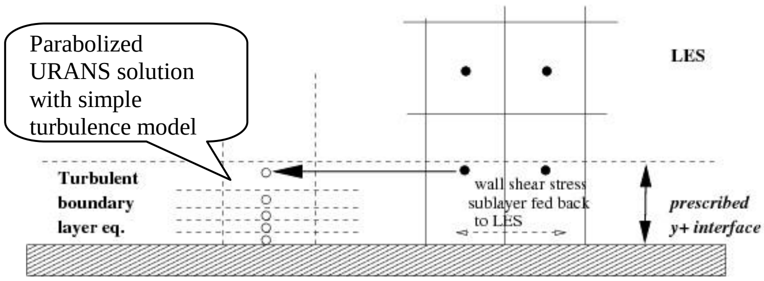

Fig. 2: Zonal RANS-LES methodology

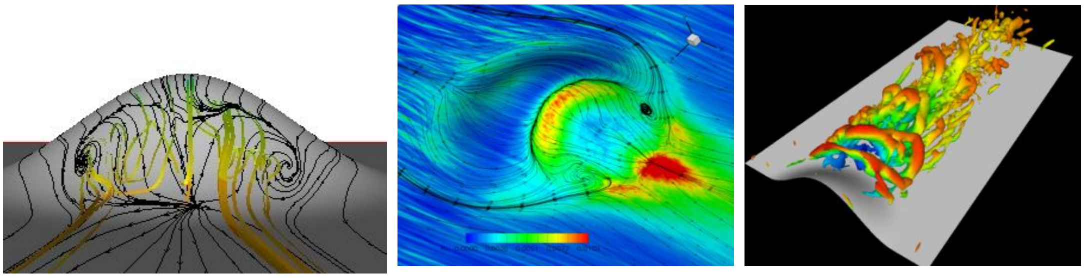

Fig. 3: Hill-shaped obstruction in a duct simulated in addition to the wing in Fig. 1.

For the wing geometry shown in Fig. 1, a substantial effort was made to develop grids that optimally map the geometry, in general, and the highly curved leading edges, in particular, subject to stringent constraints on grid-expansion and cell aspect ratios. This was done in close interaction with the University of Cranfield, and the end result was the selection of two different topologies, both evolved from a prolonged series of preliminary simulations. The grid eventually developed at ICL is the H-topology multi-block configuration, shown in Fig. 4.

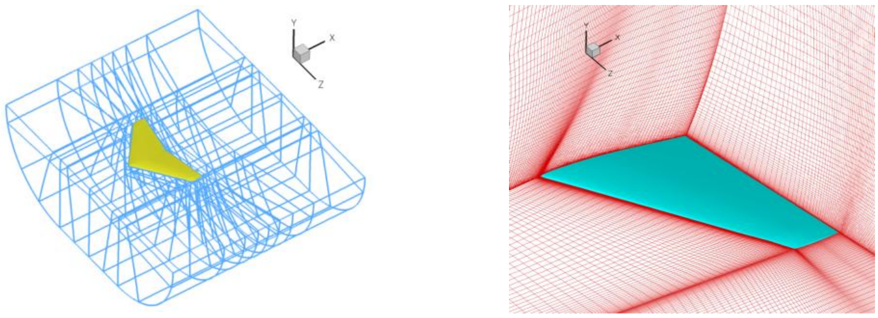

Fig. 4: Multiblock mesh with 23.6 million nodes and 256 blocks (l.h.s.) and mesh details (r.h.s.)

A preliminary study using 7.5 million nodes was first conducted and then followed by detailed studies on a much finer mesh that contains 23.6 million cells (384x192x320). The mesh has an especially high density around the leading edge, trailing edge and wing tips, and the maximum cell aspect-ratio of the finer mesh is about 70. The mesh was partitioned into 256 equal-sized blocks for efficient parallel computing, as shown in the left-hand-side plot of Fig. 4. The simulations were performed on HPCx (using IBM POWER5 processors), UK's leading computing facility. One typical run of the fine-grid simulation costs about 30,000 CPU hours.

Results of the study have been published in Li and Leschziner (2007[1,2]). One visualisation and two quantitative comparisons between predicted and measured streamwise and spanwise velocity profiles at 50% of wing-span are shown in Fig. 5. Comparisons reported in the referenced papers demonstrate a computed flow topology that is very similar to that observed in the experiment. Detailed comparisons of the velocity fields also show good agreement in the middle portion of the wing. However, towards the wing tip, agreement deteriorates. In particular, the computation failed to resolve a secondary separation region on the suction side provoked by fluid moving from the pressure side onto the suction side at the tip. This is attributed, primarily, to insufficient resolution in the wing-tip area. The overall conclusion is thus affordable RANS-LES schemes are appropriate and promising, but require very high resolution, not necessarily linked to wall processes, in near-tip regions in which the major outer flow features are exceptionally complex.

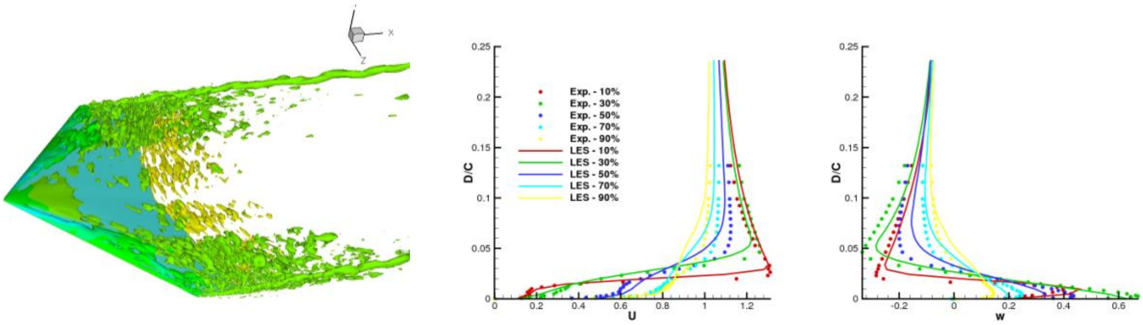

Fig. 5: Vortical structures, visualised by the "Q-criterion" (l.h.s.) and velocity profiles at 50% of wing-span (r.h.s.)

References

Byun, G. and Simpson, R.L. (2005), Structure of three-dimensional separated flow on an axisymmetric bump, AIAA Paper 2005-0113.

Li, N. and Leschziner M.A. (2007), Large-eddy simulation of flow over a swept wing with approximate near-wall modeling, AIAA-2007-1118.

Li, N. and Leschziner M.A. (2007), Large-eddy simulation of flow over a swept wing with approximate near-wall modelling, The Aeronautical Journal, 111, pp. 689-698.

Li, N. and Leschziner, M.A. (2007), Highly-resolved Large Eddy Simulation of separation from a three-dimensional hill, Proc. 11th European Turbulence Conference, Porto, June 2007, pp. 407-409.

Temmerman, L., Hadziabdic, M., Leschziner, M.A., and Hanjalic, K., (2004), A hybrid two-layer URANS-LES approach for large eddy simulation at high Reynolds numbers, International Journal of Heat and Fluid Flow, vol. 26 (2), pp. 173-190.

Tessicini, F., Temmerman, L. and Leschziner, M.A. (2006), Approximate near-wall treatments based on zonal and hybrid RANS-LES methods for LES at high Reynolds numbers, International Journal of Heat and Fluid Flow, vol. 27 (5), pp. 789-799.

Tessicini, F., Li, N. and Leschziner, M.A. (2007), Large-eddy simulation of three-dimensional flow around a hill-shaped obstruction with a zonal near-wall approximation, International Journal of Heat and Fluid Flow, vol. 28 (5), pp. 894-908.