Sixth International Symposium on Turbulence and Shear Flow Phenomena
Seoul, Korea, 22-24 June 2009

COMPUTATION OF FLOW AROUND A NACA 0015 AEROFOIL WITH ZNMF JET CONTROL: POTENTIAL SAVINGS OF AN UNSTRUCTURED MESH?

Juan Uribe, Alistair Revell
School of Mech, Aero & Civil Eng, The Univ of Manchester, Manchester M60 1QD, UK
juan.uribe@manchester.ac.uk, alistair.revell@manchester.ac.uk

Charles Moulinec
STFC Daresbury Lab, Daresbury, Warrington WA4 4AD, UK

Vassili Kitsios[1,2], Andrew Ooi[1], Julio Soria[3]
[1] Walter Bassett Aerodyn Lab,Dept of Mech Eng, Univ. of Melbourne, 3100 Victoria, AUSTRALIA
[2] Lab d'Etudes Aérodynamiques (LEA), Univ de Poitiers, 86036 Poitiers, FRANCE
[3] Lab for Turb Res in Aero & Comb, Dept of Mech & Aero Eng, Monash Univ, 3800 Melbourne, AUSTRALIA

ABSTRACT

The flow around a NACA 0015 aerofoil with a zero-net-mass-flux jets (ZNMF) located at the leading edge of the aerofoil is simulated using Large Eddy Simulation (LES). An incidence angle of 18° is used such that without flow control, the flow exhibits laminar separation from the tip of the aerofoil and an associated loss of lift. The oscillating ZNMF jet is designed to introduce small scale structures into the boundary layer at the mouth of the jet to breakup the larger, more stable low speed structures. These long thin structures are perturbed to the point that they become unstable and turbulent flow ensues, where higher momentum enables the flow to maintain attachment around the tip and suction surface of the aerofoil. This is a complex three dimensional flow which requires fine temporal and spatial resolution and in this paper, LES are undertaken using the Smagorinsky subgrid scale model on both structured and unstructured meshes of around $10M$ and $2M$ points respectively. It is found that while the structured mesh is able to capture correctly this flow control mechanism for some periods of the cycle, there appear to be difficulties, in spite of employing necessary resolution requirements. Several causal factors are identified for ongoing study. The same calculation run on an unstructured mesh of 80% fewer cells appears to be able to maintain the controlled state of the flow previously observed in experiment, but the computation on the unstructured mesh is in fact 'assisted' by the influence of numerical noise generated upstream of the aerofoil at the non-conformal interface of grid cells.

INTRODUCTION

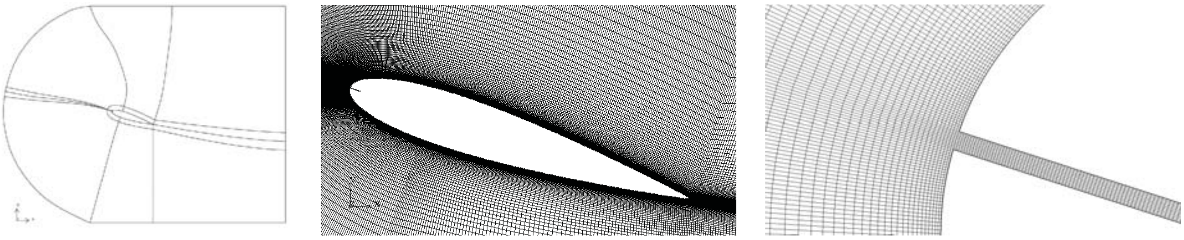
Flow control by means of suction and blowing has been performed on a wide range of flow configurations, where the cyclic increase and decrease of lift and drag is affected via the alteration of the boundary layer, thus minimising the region of separated flow. In the case of aerofoils at high angles of attack, the use of zero-net-mass-flux jets (ZNMF) has been proved to be less expensive than continuous steady jets since for the latter the amplitude required to obtain the same effect is markedly larger (Seiferet et al., 1993). The influence of the control jet depends primarily on the frequency of oscillation of the jet and its location with respect to the point of flow separation (Smith and Glazer, 1998). For angles of attack beyond the post-stall region, this point occurs close to the leading edge. By applying a ZNMF jet in this region momentum is transferred to the system without net mass addition, which results in the formation of vortex pairs that are convected along the aerofoil. The oscillating frequency is set such that the layer of injected vorticity has sufficient time to be convected far enough away from the jet orifice so as to avoid disturbance, creating a constant trail of vortices that promotes the transition to turbulence and induces re-attachment of the boundary layer.

The case studied here is the flow around a NACA 0015 aerofoil at an angle of incidence of $\alpha = 18°$ with an oscillatory zero-net-mass-flux jet at the leading edge, as studied experimentally by Tuck and Soria (2004). The numerical simulation of these flows presents a formidable challenge, due to the complex time-dependent interaction of the jet and the boundary layer. The disparity of scale between the flow structures close to the jet and those induced by the separated boundary layer entails high spatial and temporal resolution requirements. The inherent necessity to simulate the influence of such small structures effectively rules out turbulence modelling approaches based on the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations, and thus a LES is deemed requisite.

Previous computations of the same configuration without control (i.e. without the jet) have been performed by Kitsios et al. (2006), who employed a block structured mesh of 6×10^6 cells and these results, together with more recent results awaitting publication, are shown to be in good agreement with the reference experimental data. With the resulting numerical data sets, a stability analysis has been undertaken to determine the effect of small perturbations on the transition to turbulence Kitsios et al. (2007).

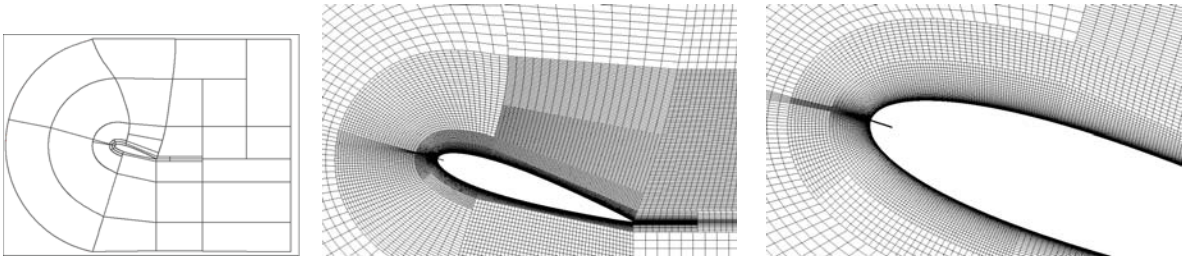
Objectives

The aim of the current paper is to present a highly resolved LES of the controlled case and to make use of the experimental data to compare and assess the predictions obtained with both structured and unstructured meshes. The use of unstructured meshes is an attractive alternative since the total number of cells can be reduced dramatically by limiting mesh refinement to the specific regions of interest. However, the use of this type of meshing with LES remains contentious, since the use of non-conformal interfaces has often been observed to introduce numerical oscillations (Lau-



(a) Block structured (b) Close up of the aerofoil (c) Close up of the jet

Figure 1: Structured mesh of 9.4M cells



(a) Non-conformal blocks (b) Close up of the aerofoil (c) Close up of the jet

Figure 2: Unstructured mesh of 2M cells

rence (2006); Afgan et al. (2007); Benhamadouche et al. (2005)). Moulinec et al. (2005) showed that the use of tetrahedral cells without any advanced gradient reconstruction, does not conserve kinetic energy and illustrated the advantages of using polyhedral cells.

This paper will therefore attempt to provide some insight into two main areas. Primarily, a predictive assessment of LES for what is undoubtedly a very challenging case and secondly an answer to the question posed in the title; whether or not it is feasible to run an LES of this case on an unstructured grid. This would be a valuable option given the spatial and temporal requirements of these computations. While it is hoped that LES may eventually be employed to closely examine the complex flow interactions in flow control with a view to optimising and future design of aerodynamic control systems, the first step is to identify a suitable computational platform upon which one is confidently able to conduct these numerical tests.

FLOW DETAILS

The flow over an NACA 0015 with a ZNMF jet has been experimentally investigated by Tuck and Soria (2004), where a wide range of parameters were studied. The experimental Particle Image Velocimetry (PIV) data is available for comparison with the LES predictions. The configuration computed here corresponds to the greatest lift enhancement obtained with an angle of attack of $\alpha = 18°$ and a Reynolds number based on the chord of the aerofoil, c, the molecular viscosity ν and free stream velocity, U_∞ of $Re = U_\infty c/\nu = 3 \times 10^4$.

The ZNMF jet at the leading edge is characterised by a velocity ratio between the r.m.s. velocity of the jet $u_{j,rms}$ and the free stream velocity of $VR = u_{j,rms}/U_\infty = 0.67$. The time-dependent jet velocity is prescribed by $u_j(t) = \sqrt{2}u_{j,rms}\sin(2\pi f t)$, where f is the jet frequency and t is the time.

The other non-dimensional parameters necessary to describe the jet inlet are the non-dimensional forcing frequency $F^+ = fc/U_\infty = 1.3$ and the momentum blowing coefficient $c_\mu = 2\frac{h}{c}\,(u_{j,rms}/U_\infty)^2 = 1.38{\times}10^{-3}$, where $h = 1.5{\times}10^{-3}c$ is the height of the jet inlet channel. The length of the channel is $l = 3 \times 10^{-2}c$. In the experiment, the aerofoil was manufactured with a two dimensional slot spanning the entire leading edge in a domain that was $5c$ in the span wise direction (Tuck and Soria, 2004).

COMPUTATIONAL DETAILS

Compared to the LES of the uncontrolled case by Kitsios et al. (2006), the present calculations require a substantially finer mesh resolution in the region around the leading edge of the aerofoil where the ZNMF jet is located. In the first instance, a block structured mesh of similar topology to that used by Kitsios et al. (2006) is used, with modifications to the leading edge region as shown in Figure 1 to capture the influence of the jet.

The computational domain has the origin at the leading edge and the boundaries are located at $-3 < X < 8$, $-3 < Y < 3$ and $0 < Z < 0.25$, where $X = x/c$ is the stream wise direction, $Z = z/c$ the span wise and $Y = y/c$ is the third direction. Although the length, l_z, on the span wise direction is small, similar lengths have been used (See You and Moin (2008) where $l_z = 0.2c$ was used.)

Mesh setup

Two meshes have been generated, one block-structured and one unstructured using non-conformal interfaces, so called 'hanging nodes'. The maximum non-dimensional wall distance is $y^+ < 1.2$ and the meshes have been designed to have values of $x^+ < 20$ and $z^+ < 10$ around the surface of the aerofoil. Precursor calculations were run using URANS models in order to ascertain the necessary geometrical sizing to achieve this resolution and effort has been taken to generate mesh lines which fall perpendicular to the surface of the aerofoil, so as to minimise error in wall normal and parallel gradient calculation. The structured mesh has 128 cells on the span wise direction, 900 around the aerofoil and 80 normal to the surface of the aerofoil for a total of 9.4×10^6. The maximum viscosity ratio obtained during the calculation was $\nu/\nu_t < 6.48$, where ν_t is the turbulent viscosity.

For the unstructured mesh the exact same resolution has been maintained in a thin region around the aerofoil itself, while non-conformal interfaces have been introduced to give mesh size reductions in regions away from the surface. The unstructured mesh was designed to maintain the level of accuracy obtained in the structured mesh, which may be considered as the baseline case for comparison. This study thus investigates the proportion by which a structured mesh might be reduced; in contrast to an alternative strategy where one might attempt to focus on higher resolution in key areas of the domain whilst reducing less important areas and thus obtaining a lesser overall reduction in mesh size. The mesh was coarsened to reach a target of 2.5×10^6 cells. The refinement was carried out in all 3 directions by decomposing the domain in 42 blocks as seen in Figure 2. The blocks were designed so that the cells in each block have an aspect ratio lower than 2 and a cell volume less than one tenth of the turbulent length scale predicted by prior 2D URANS simulations.

Numerical setup

The flow computations are carried out using the standard Smagorinsky (1963) model implemented in the finite volume solver *Code_Saturne* (Archambeau et al., 2004).

The time step has been fixed to $\Delta t/t^* = 5 \times 10^{-5}$ where $t^* = c/U_\infty$ and the maximum CFL number was 1.2. An iterative method for the gradient reconstruction has been used. This method improves accuracy over the traditional Gauss approximation but requires more computational effort. The computations have been carried out using a second order scheme in time and space and using algebraic multigrid solver for the pressure resolution.

These calculations have been performed on SFTC Daresbury Laboratory Blue Gene/P and on UK national facility HECToR (http://www.hector.ac.uk). The computation of the structured grid takes about 6 to 8 seconds per iteration on 2048 cores (VN mode = 4 cores per node) of BlueGene/P. The simulation was started without the jet (i.e. uncontrolled) for a time of $t/t^* = 10$ and then the jet was activated and run for a time of $t/t^* = 40$. For the case of the unstructured mesh, the time per iteration was 4 to 5 seconds on 512 cores of HECToR. The simulation had been initialised from the structured results but it has only been run for 10 non-dimensional seconds, since large problems were encountered trying to achieve convergence due to the numerical errors introduced by the non-conformal interfaces.

STRUCTURED MESH RESULTS

Calculations were initialised using results from 2D calculations using an URANS model for the uncontrolled flow. The LES were also initially run for the uncontrolled case, which in addition to aiding convergence, provided some results for verification against the uncontrolled experimental work as well as earlier numerical work by Kitsios et al. (2006), although long-time averaged were not performed as this work focuses on the controlled case.

The long time averaged was carried out for 10 non-dimensional seconds for the structured mesh. For the unstructured mesh no-log term averaged is presented due to the convergence problems described above. The results were also averaged on the span wise direction, which has 128 cells in the near-wall region for both meshes.

Preliminary observations

Figures 3 to 5 show a comparison between the flow field from the experiment of Tuck (2004) and the present results from the numerical simulation on the structured mesh. Figures 3 and 4 display instantaneous flow for the uncontrolled and the controlled flows respectively; experimental flow visualisation was undertaken using coloured dye injection and numerical results show contours of the velocity magnitude. In both cases the qualitative agreement is good; the size and shape of regions of separated flow are similar and boundary layer thickness in the controlled case is comparable. These are instantaneous snap shots and the size of the separated region is observed in the numerical simulation to shrink and grow over the cycle of the flow. Figure 5 compares the long-time averaged flow streamlines for the controlled case and indicates a significant disagreement between the numerical and experimental results. Moving around the leading edge and along the upper surface, the long-time averaged flow is initially comparable but then the numerical results exhibit a strong recirculation region that is not observed in the experiment; the experiment reports a single isolated recirculation region from around $0.2 < X < 0.4$ whereas the LES reports a combined recirculation region extending from $0.1 < X < 0.85$. Upon closer inspection the LES separated flow region is composed of two parts; the first recirculation occurring at roughly the same location as shown in the experiment and the second region, from $0.5 < X < 0.85$, is much stronger and dominates the upper surface flow. In order to examine further this feature, the cyclic nature of this flow is first described in more detail.

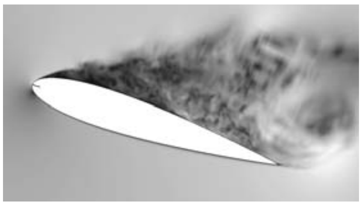
Observations over a period of the flow

Close examination of the experimental flow reveals a complex periodic process whereby large vortical structures are generated at the leading edge and convected downstream along the upper surface of the aerofoil. Tuck (2004) observes that there are generally two vortices present on the upper surface at any one time; throughout the cycle these structures are seen to remain in close proximity to the surface which is a further indication of attached flow in the controlled case. The time taken for each vortex to travel the length of the chord was observed to be approximately equal to twice the period of the control jet itself.

Figure 6 displays a time history trace of stream wise velocity at two locations near to the upper surface of the aerofoil, one near to the leading edge and one near to the mid-chord location, as well as the cyclic ZNMF jet velocity for reference. Figure 7 shows a series of instantaneous snapshots of flow streamlines and isocontours of the Q criteria, respectively. These snapshots correspond to the four time locations as identified in Figure 6. The formation of the upper surface vortices is part of a continuous cycle driven by the jet interaction at the leading edge for which the explanation is offered here in the following paragraphs.

The uncontrolled flow is characterised by an early separation from a laminar boundary layer. The aim of the jet is to de-stabilise this boundary layer so that it may transition to a turbulent state. In doing so, the boundary layer gains momentum close to the surface and is able to sustain attached flow for a longer distance around the aerofoil, which is picked up by the near wall velocity trace as a rapid acceleration; this is observed in Figure 6 at two occasions

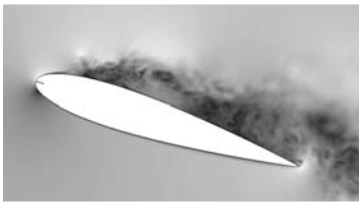
(a) Expt (coloured die streaks)



(b) LES (structured grid, velocity magnitude)

Figure 3: Uncontrolled case : instantaneous snapshot of flow

(a) Expt (coloured die streaks)



(b) LES (structured grid, velocity magnitude)

Figure 4: Controlled case : instantaneous snapshot of flow

(a) Expt (coloured die streaks)

(b) LES (structured grid)

Figure 5: Controlled case : long-time averaged flow

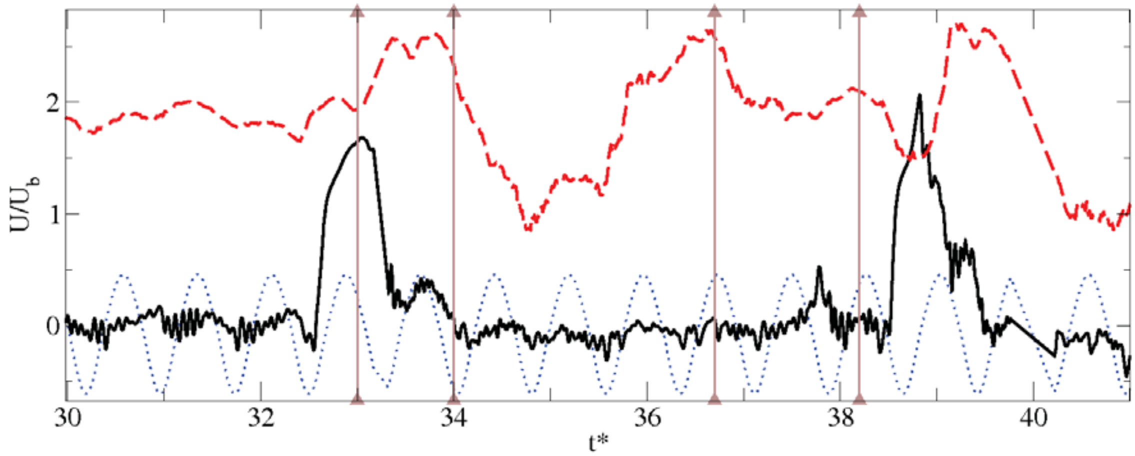


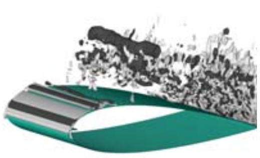
Figure 6: Time history of stream wise component of velocity at two locations; (*solid line*): close to leading edge $(X_1, Y_2 = 0.1, 0.05)$; (*dashed line*) close to mid-chord $(X_2, Y_2 = 0.5, -0.06)$; (*dotted line*): ZNMF jet velocity

(a) at $t^* = 33.0$

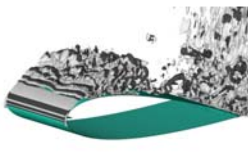
(b) at $t^* = 34.4$

(c) at $t^* = 36.7$

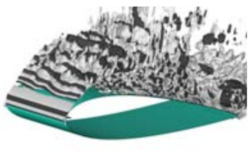
(d) at $t^* = 38.2$



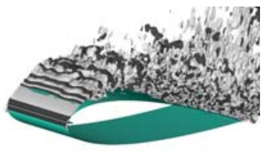
(e) at $t^* = 33.0$



(f) at $t^* = 34.4$



(g) at $t^* = 36.7$



(h) at $t^* = 38.2$

Figure 7: Instantaneous flow streamlines (*top*) and iso-surfaces of Q (*bottom*) from structured LES

($t/t^* = 33$ and 39). It is conjectured that the scale of this acceleration is overpredicted by the current results, perhaps due to inadequate sub-grid scale modelling.

This brief period of flow attachment directly precedes the formation of one of the characteristic upper surface vortices at the leading edge, from where it is convected, or rolled, down the aerofoil. In the numerical results these vortices are seen to be generated at a much larger interval; close to 8 times the period of the control jet cycle rather than around two times the jet cycle as observed in the experimental work.

The size of these structures, influenced by the frequency and amplitude of the jet, is critical to maintaining a continuously attached flow. If they become too large, the flow will be unable to reattach downstream of the vortex, leading to a larger secondary recirculation, as observed in the present numerical results; see Figure 7. The large acceleration observed at $t/t^* = 33$ and the large structures that persist at the trailing edge, are most likely coupled. The vacation of the structure from the trailing edge is likely to create a low pressure region which feeds the acceleration at the leading edge and so, a larger trailing edge recirculation onsets a larger acceleration, as observed in the computation.

Given that there are two jet cycles for each upper surface vortex travel, the first jet cycle leads to the formation of a vortex, and the second cycle provides further momentum to move this structure along the upper surface. This alternating pattern of re-attachment and vortex formation, where in places the flow has almost re-laminarised, provides a difficult challenge for simulation.

Computational shortfalls

The computational results with the structured mesh initially reproduce a similar flow behaviour to that observed experimentally, with a small-sized structure generated at the leading edge. However, the vortices are observed to slow down towards the trailing edge and linger for longer periods before being shed into the wake. Eventually these structures appear to grow in radius and cause a large separated region at the trailing edge. Since this cycle has a period much larger than the experiment, the long-time averaged flow from the computation is significantly different (as seen from Figure 5); the long term presence of a large vortex on the upper surface (although not as large as the uncontrolled case) effectively makes the effect of the jet diminish.

It is not immediately obvious why the numerical solution on the structured mesh fails to maintain the controlled state observed in the experiment, but there are several possible factors which may contribute to a greater or lesser extent, which are discussed in the following paragraphs with a view to shaping further research on this case, some parts of which are already under way.

Spatial resolution. Although current best practice guidelines for Large Eddy Simulation are somewhat arbitrary, these have been applied in the present work. However, tests are under way with a spatial resolution of half that used in the original structured mesh to examine the influence of this factor, especially near the jet exit.

Sub Grid Scale modelling. As generally reported, the standard LES subgrid scale (SGS) model, the Smagorinsky model erroneously calculates the value of sub grid turbulent viscosity to be permanently proportional to velocity strain rates, and is thus intrinsically unsuitable for simulation of laminar flow; it will always act to increase viscosity beyond what it should be. If the flow is overly diffusive in regions where turbulence might otherwise destabilise the flow beyond a critical point, then it is likely that this would prevent the boundary layer from becoming fully turbulent, or at least reduce the near wall momentum required to transport the vortex downstream. In general, the best practice guidelines recommend that SGS viscosity does not exceed $10\times$ the level of molecular viscosity, and in this computation it does not exceed 7. However, it may be that such general guidelines are not applicable to this flow and as such, additional calculations are under way to examine the impact of employing an alternative SGS model, on the same mesh. For this purpose, the WALE scheme (Nicoud and Ducros, 1999) was selected given reports of its particular suitability for similar cases.

Span wise extent. This flow is inherently three dimensional and the current domain extends a span wise distance of $0.25c$ only. For LES calculations of the uncontrolled case this was assumed to be adequate to allow structured to develop. In general, if a domain is of insufficient span wise extent, one would expect structures in the plane perpendicular to this direction to be amplified as a consequence of a lack of span wise instabilities, which would otherwise weaken the coherence of the structure via momentum diffusion and decreased vorticity.

The cyclic flow control process is delicately balanced and coupled and as shown in this computation, once one part of this process is disrupted the flow pattern is altered. As such this study becomes a useful means of assessing and hopefully identifying the critical factor in the successful computation of this flow.

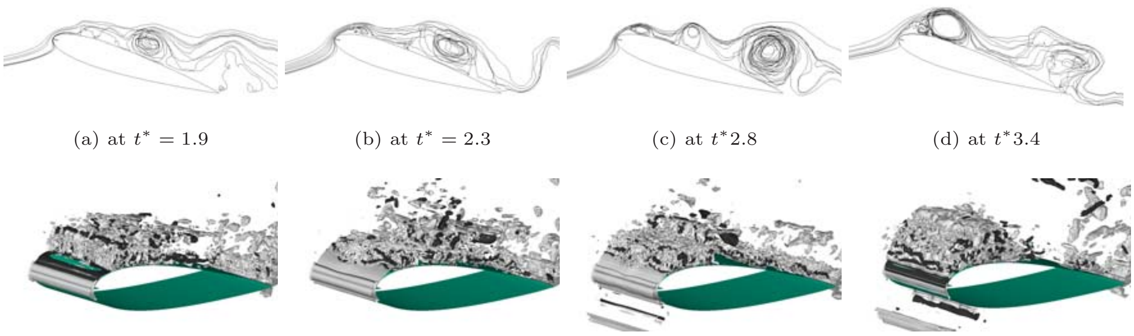
UNSTRUCTURED MESH RESULTS

The calculation on the unstructured mesh was initialised by interpolation from the results of the structured mesh. The instantaneous results appear to indicate to show that the laminar layer near the leading edge breaks quicker than in the structured mesh. Initially, this appears to represent a situation more similar to the experimental observations than obtained in the previous work using the structured grid, but upon closer inspection it becomes clear that this observation is incorrect.

Inspection of the evolution of the flow, (from Figure 8) shows that the the non-conformal interfaces in the mesh introduce numerical oscillations that effectively generate 'noise', equivalent to introducing fluctuations or synthetic turbulence. Inaccurate gradient reconstruction at the interface between blocks leads to inaccuracies in the flow field. As a result, patterns which can be considered as turbulence are observed below the aerofoil, where there clearly should not be any such flow structures (Figures 8(g) and 8(h)).

A further sign that there is a different flow structure in this computation is obtained when one considers that the vortices roll down from the leading edge at a similar period compared to the jet period, which indicates that they are not linked to the attachment-formation-push-attachment cycle described in the previous section. Instead it is likely that the instabilities generated from the non-conformal interface are increasing the level of turbulence in the flow directly upstream of the leading edge, which permanently destabilises the flow separation and effectively acts as a permanent control jet.

Finally, it is also possible that the grid interfaces are ac-



(a) at $t^* = 1.9$ (b) at $t^* = 2.3$ (c) at $t^*2.8$ (d) at $t^*3.4$

(e) at $t^* = 1.9$ (f) at $t^* = 2.3$ (g) at $t^* = 2.8$ (h) at t^*=3.4

Figure 8: Instantaneous flow streamlines (*top*) and iso-surfaces of Q (*bottom*) from unstructured LES

tively constraining the flow pattern; since the grid coarsens rapidly away from the surface there is a numerical preference to prevent larger structure from persisting. It is possible that the errors introduced by the non-conformal interfaces could be reduced by avoiding a large refinement between them but this would increase the number of blocks necessary to gain advantage from this method. The blocks were designed to follow the pattern from the turbulent length scale which grows rapidly away from the wall where a refinement is needed to capture the effects of molecular viscosity. But even if the error is reduced, it would be very difficult to measure.

SUMMARY

The complex interaction of a ZNMF jet at the leading edge of a NACA 0015 aerofoil has been simulated using LES and results have been obtained on both structured and unstructured meshes. It is observed that the experimentally observed flow mechanism is simulated but encounters problems associated with the relaminarisation of the characteristic rolling vortex that occurs on the suction surface and a larger structure persists at the trailing edge which leads to a different dynamic on the upper surface.

Acknowledgements

This work was partly supported by the UK Engineering and Physical Sciences Research Council (EPSRC) under Grant No. EP/D053994/1. Additional support was provided by EPSRC under the auspices of Collaborative Computational Project 12. Part of this work was carried out under the HPC-EUROPA project (RII3-CT-2003-506079), with the support of the European Community - Research Infrastructure Action under the FP6 "Structuring the European Research Area" Program.

REFERENCES

I. Afgan, C. Moulinec, R. Prosser, and D. Laurence. Large eddy simulation of turbulent flow for wall mounted cantilever cylinders of aspect ratio 6 and 10. *International Journal of Heat and Fluid Flow*, 28(4):561–574, 2007.

F. Archambeau, N. Mechitoua, and M. Sakiz. A finite volume method for the computation of turbulent incompressible flows - industrial applications. *International Journal on Finite Volumes*, 1(1), 2004.

S. Benhamadouche, J. Uribe, N. Jarrin, and D. Laurence. Large eddy simulation of a symmetric bump on structured and unstructured grids, comparison with RANS and T-RANS models. In *Turbulence and Shear Flow Phenomena 4*, pages 325–330, 2005.

V. Kitsios, B. Kotapati, R. Mittal, A. Ooi, J. Soria, and D. You. Numerical simulation of lift enhancement on a NACA0015 airfoil using ZNMF jets. *Center for Turbulence Reasearch,Proceedings of the summer school program*, pages 457–468, 2006.

V. Kitsios, A. Ooi, and J. Soria. Spatio-temporal stability analysis of the seprated flow past a NACA 0015 airfoil with ZNMF jet control. In *Sixteenth Australasian Fluid Mechanics Conference*, 2007.

D. Laurence. Large eddy simulation with unstructured finite volumes. In B.J. Geurts E. Lamballais, R. Friedrich and O. Métais, editors, *Direct and Large-Eddy Simulation VI*, volume 10 of *ERCOFTAC*, pages 27–38, 2006.

C. Moulinec, S. Benhamadouche, D. Laurence, and M. Perić. LES in a u-bend pipe meshed by polyhedrl cells. In W. Rodi and M. Mulas, editors, *Engineering Turbulence Modelling and Experiments 6*, pages 237–246, 2005.

F. Nicoud and F. Ducros. Subgrid-sacle stress modelling based on the square of the velocity gradient tensor. *Flow, turbulence and combustion*, 62:183–200, 1999.

A. Seiferet, T. Bachar, M. Shepshelovich, and I. Wygnanski. Oscillatory blowing: A tool to delay boundary layer separation. *AIAA Journal*, 31:2052–2060, 1993.

J. Smagorinsky. General circulation experiments with the primitive equations: I the basic equations. *Monthly Weather Review*, 91:99–164, 1963.

B. Smith and A. Glazer. The formation and evolution of synthetic jets. *Physics of Fluids*, 10:2281–2297, 1998.

A. Tuck. *Active Flow control of a NACA 0015 airfoil using a ZNMF jet.* PhD thesis, Monash University, 2004.

A. Tuck and J. Soria. Active flow control over a NACA 00115 airfoil using a ZNMF jet. In *Fifteenth Australasian Fluid Mechanics Conference*, 2004.

D. You and P. Moin. Active control of flow separation over an airfoil using synthetic jets. *Journal of Fluids and structures*, 24:1349–1357, 2008.