THE APPLICATION OF A TIME-ACCURATE, UNSTRUCTURED EULER SOLVER, COUPLED WITH A BOUNDARY LAYER SOLVER FOR THE SOLUTION OF TRANSONIC AEROELASTIC PROBLEMS ON COMPLEX CONFIGURATIONS

S.A.Galpin, BAE SYSTEMS Airbus UK

Keywords: *aeroelastics, complex configurations, Euler, time-accurate, unsteady*

Abstract

Transonic aeroelastic experience on Airbus aircraft has shown that purely linear Computational Fluid Dynamics methods, are lacking sufficient accuracy for high Mach numbers (M>0.7) on high aspect ratio configurations. For these configurations, aeroelastic responses can be highly non-linear, due to transonic aerodynamic effects. In order to predict these non-linear aerodynamic pressures and forces, a time-accurate, unstructured Euler solver, coupled with a 2D boundary layer method, has been developed.

Steady state unstructured Euler solvers have evolved significantly over recent years, and are now used routinely by the Airbus UK for modelling steady state 3D flows around complex geometry. The ability to model complex wing/body/nacelle/pylon configurations, in a time-accurate environment is seen as a significant increase in capability. It is intended that a time-accurate unstructured Euler solver would be used to complement existing linear CFD methods for aeroelastic design work.

1 Introduction

Aeroelastic calculations, such as flutter and Limit Cycle Oscillations (LCO) require three main inputs:

- Aerodynamic data
- Structure stiffness data
- Mass data.

Currently aircraft are designed by developing a high-speed aerodynamic wing shape to fit a required mission, i.e. endurance, payload and performance. Typically, the aeroelastic performance of this high-speed wing shape is then assessed using simple linear theory. Any aeroelastic problems identified by linear theory are then addressed by changing only two of the three possible aeroelastic inputs, the structural stiffness or the mass distribution of the aircraft, Fig.1. Only very rarely is the aerodynamic shape altered for aeroelastic reasons. In addition, the high-speed aerodynamic shape is often 'frozen' well before the aeroelastic design is completed. Redesigning the high-speed aerodynamic shape late in the aircraft design process is very expensive.

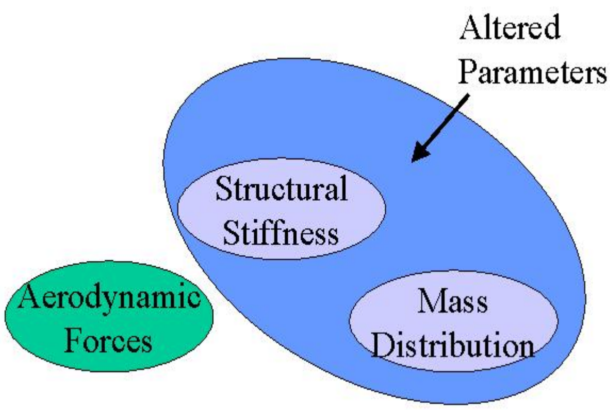

Fig.1. Current Aeroelastic Design Process.

Steady state unstructured Euler solvers have evolved significantly over recent years, and are now used routinely for modelling steady state 3D flows around complex geometry. BAE SYSTEMS has therefore developed a time-accurate, unstructured, viscous-coupled Euler solver that will be used in a transonic aeroelastic environment. However, because of the significant increase in CPU usage, the Euler code will be used in the short term to produce 'correction factors' for linear theory. The Euler code will also be used to look at specific problems not solvable with linear theory i.e. LCO and control buzz. Therefore, the Euler code will supplement the existing linear theory. In the longer term, the Euler code will be coupled with a structural solver (e.g. MSC NASTRAN) to give fully coupled solutions.

The code has reached sufficient maturity to be applied to current complex configurations. For future aircraft designs transonic aeroelastics will be used as a design parameter during the high-speed aerodynamic design process. This will therefore minimise risk during the high-speed wing shape design process, minimising structural and mass distribution reworking for aeroelastic reasons. This will reduce design cycle times, help maintain weight estimates and reduce costly flight test programs by developing designs that are 'mature' at first flight.

The outlined code has been evaluated using several configurations. These range from simple 3D configurations, such as the LANN (**L**ockheed-Georgia, **A**irforce Flight Dynamics Laboratory, **N**ASA-Langley and **N**LR**)** wing [1], Fig. 2, through to complex 3D wing/body configurations with nacelles and pylons Figs.3 & 4. Unsteady transonic aerodynamic effects of propeller driven aircraft have also been studied, Fig.5.

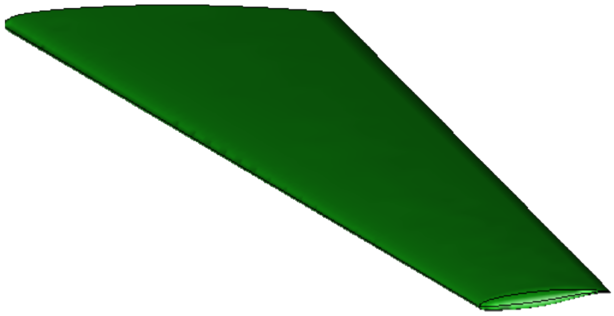

Fig.2. LANN Wing.

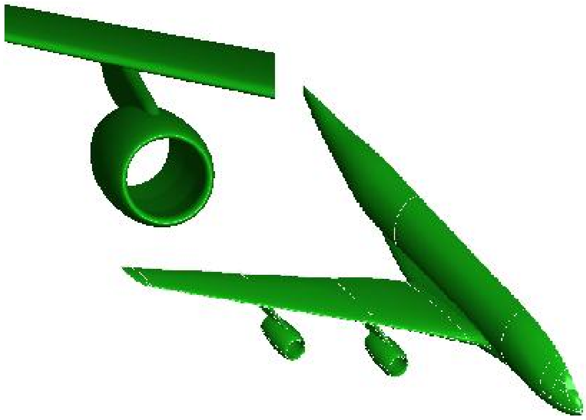

Fig.3. Wing Body Nacelle Pylon Configuration.

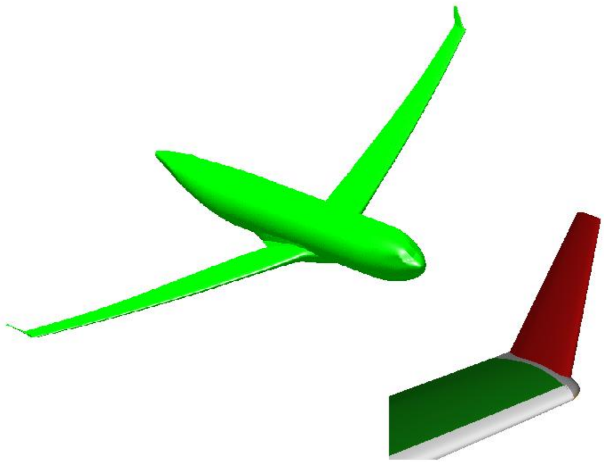

Fig.4. Wing Body Winglet Configuration.

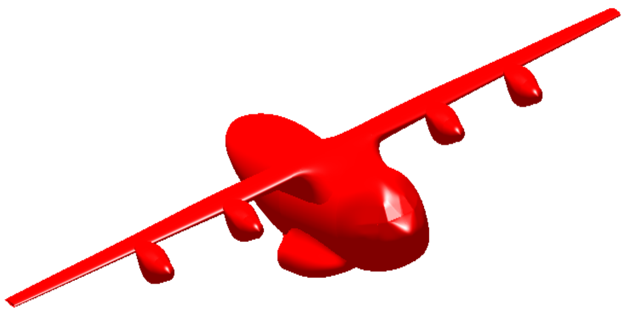

Fig.5. Wing, Body, Nacelle and Propeller Configuration.

2.1 CFD Method.

BAE SYSTEMS have developed an in-house unstructured, time-accurate, Euler solver, FLITE3D, which uses a tetrahedral mesh to solve the Euler equations via an explicit multistage Runge-Kutta procedure. The code is cell vertex based, with a low-dissipation Pseudo-Laplacian TVD type numerical dissipation scheme. Several techniques are used to improve convergence including an agglomeration multi-grid process.

It is perhaps simplest to think of a viscous-coupled, time-accurate solution as a collection of viscous-coupled solutions, each at a given time-step throughout a prescribed oscillation. The user inputs a simple harmonic oscillation displacement (a sine wave) and assigns this to a surface or collection of surfaces. The motion is then represented by a time-varying transpiration velocity. This transpiration velocity is applied to the oscillating surface in a series of equal time-steps. This process can simulate, for example, a wing moving in pitch, heave, yaw or a combination of all three. Time-accuracy of the Euler equations is achieved by using the dual-time method of Jameson [2].

Considerable effort has been directed towards reducing the user time and CPU time for the preparation of model geometry and meshes used for a FLITE3D solution. For a typical complex wing, body, pylon and nacelle configuration, the CFD model generation would take 4-5 hours of user time. The mesh generation would take around 2 hours of CPU time on a single node of a HP-V class machine, for a 5 million cell case. This makes FLITE3D an extremely important tool in a production environment.

3.1 Results

3.1.1 LANN Wing

Comparisons to experimental results have been encouraging especially for the coupled boundary layer solutions. In addition, the code has been benchmarked to other comparable CFD codes, again with favourable results.

Fig.6 shows a simple steady state comparison between experiment and a viscous-coupled version of the code.

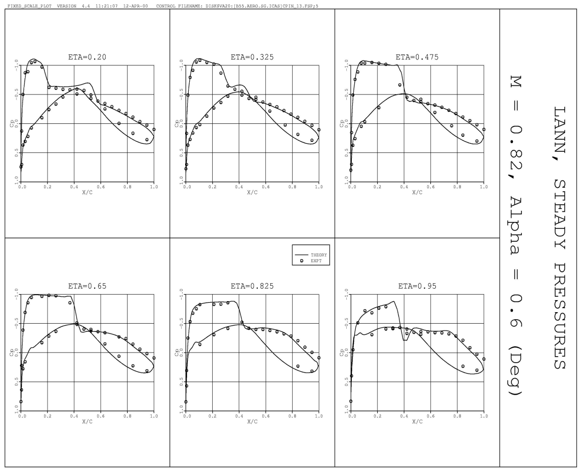

Fig.6. Steady State Pressures.

Fig.7 shows a comparison between experiment and both inviscid and viscid versions of the code, for a simple pitch oscillation on the LANN wing. The real (in-phase) and imaginary (out-of-phase) parts of the pressure are plotted for six stations on the outboard wing. The inviscid results are fair in terms of rooftop and aft loading predictions. However, the shock strength and location are not well resolved, as would be expected from an inviscid solver. The viscous effects are significant at constant incidence, reducing shock strength and moving the predicted shock location by around 10% x/c.

The implementation of a boundary layer solver to an unstructured, time-accurate, Euler solver is not a trivial problem, and adds a CPU overhead of around 30%. The coupling of a boundary layer method does however produce extremely good predictions.

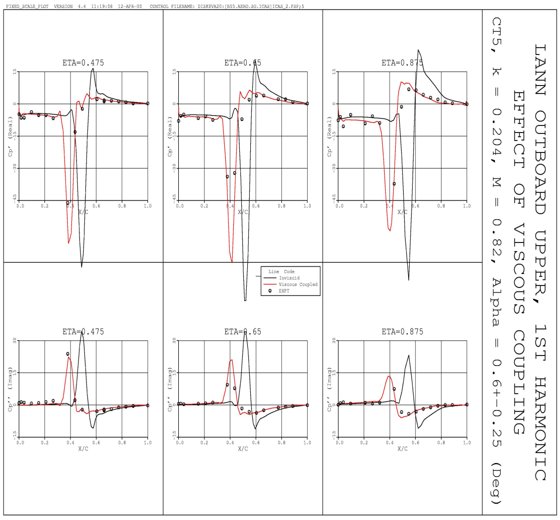

Fig.7. Real and Imaginary Inviscid/Viscous Pressures.

It is noted that the shock strength and its location are critical for a transonic aeroelastic calculation, especially across the outer wing where the largest structural displacements occur and the shocks are strong. Therefore, the added CPU expense of the boundary layer solver is justified. All the remaining solutions in this paper are produced using a viscous-coupled solver.

3.1.2 LANN Wing, Optimised Meshes

During previous unpublished Airbus UK work, considerable effort was applied to develop an ‘optimised’ mesh philosophy. In brief, it has been found that the standard FLITE3D mesh definition used for steady state calculations can be drastically reduced for time-accurate calculations. For example, the mesh density around the wing trailing edge can be cut back heavily, since there is normally no requirement to extract drag from a time-accurate solution. This therefore reduces the size of the volume mesh. The majority of the leading edge suction can be found by careful tailoring of the leading edge local mesh density. The leading edge mesh needs to be reasonably fine to capture the suction peak. The mesh distribution across the rest of the wing surface needs to be blended, to capture the shock well. Fig. 8 shows the difference between the wing surface meshes for the LANN wing tip.

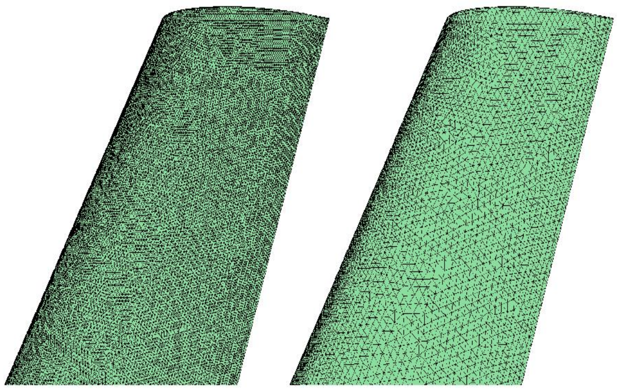

Fig.8. Standard and Optimised Surface Meshes.

This optimised mesh philosophy was found to produce good quality time-accurate, inviscid and viscous-coupled solutions, whilst reducing run times, Fig. 9. The optimised mesh flow solution quality is best judged by the pressure distributions and the integrated local forces and moments. However, there is a small difference in predicted shock strength and location. The optimised mesh has the effect of weakening the shock and moving it slightly forwards.

Since the optimised mesh is to be used primarily for time-accurate calculations, this level of accuracy is thought to be sufficient for the vast majority of calculations. Compared to the linear methods currently used for aeroelastic calculations (even for transonic cases), the optimised mesh solution represents a major step forward in solution accuracy.

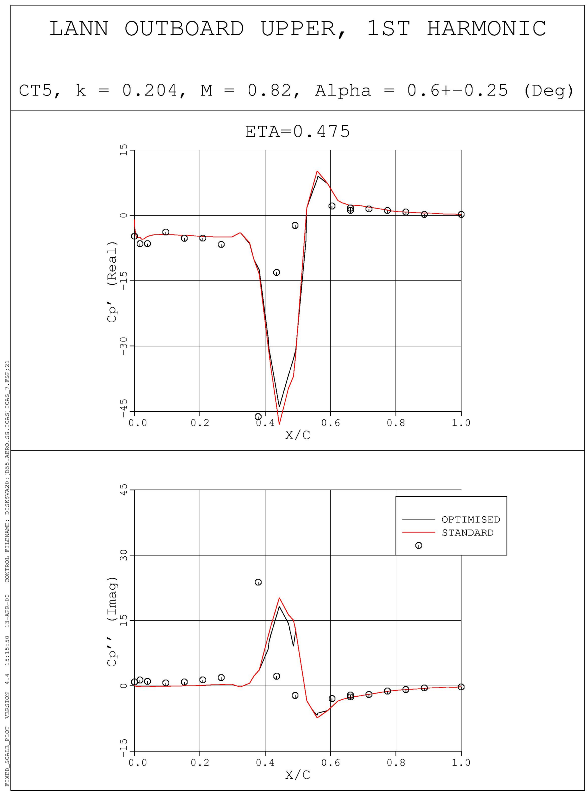

Fig.9. Real and Imaginary Wing Pressures.

In general, it was found that the run time required by an optimised volume mesh was around 30% that of a standard volume mesh.

Table 1 shows the CPU times and mesh sizes for the two mesh philosophies, for both the LANN wing and a typical wing, body, pylon and nacelle configuration.

Configuration	Mesh Type	Volume Mesh Size (Millions)	Flow Solver Run Times (Hours)
Wing Only (LANN)	Standard	1.35	40
Wing Only (LANN)	Optimised	0.41	12
Wing, body, nacelle and pylon	Standard	6	175
Wing, body, nacelle and pylon	Optimised	2	50

Table 1. Run times on a HP V-class for Viscous-Coupled calculations.

This level of time (and therefore cost) saving is critical for the efficient solution of time-accurate problems, in a real production environment. In this time context, the slight reduction in accuracy (shock location and strength) between the two mesh philosophies is justifiable.

3.2 Complex Configurations.

3.2.1 Wing Body Configuration.

Time-accurate calculations involve a significant CPU expense. In order to reduce the CPU expense it has become standard practice to reduce the geometrical complexity of the model, in many cases by simply removing the body. Removing the body tends to reduce mesh size and improve convergence.

It is therefore important to determine the effect of the body on the unsteady wing pressures and loads. Fig. 10 shows that for a typical high aspect ratio configuration the body has a significant effect on the unsteady pressures across the majority of the wing span, including the tip region. As noted before, for a high quality transonic aeroelastic solution predicting the outboard shock strength and location are critical.

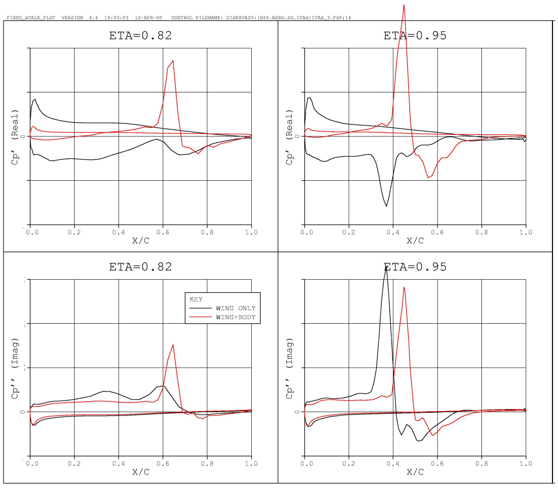

Fig.10. Real and Imaginary Outboard Wing Pressures.

Fig. 11 shows unsteady wing spanwise loads. Again, the body has a significant effect on the unsteady performance of the wing.

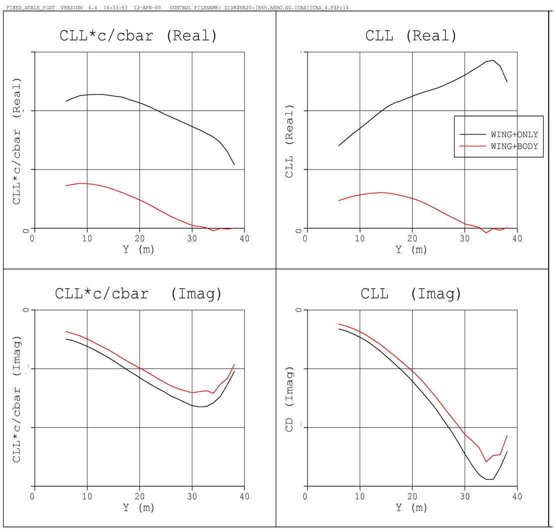

Fig.11. Real and Imaginary Spanwise Loads.

In conclusion, removing the body significantly alters the unsteady wing pressures and loads. Therefore, whilst the CPU time of the calculation has been reduced, the overall quality of the flow solution obtained limits the usefulness of such a model simplification. In steady state wing design it is standard practice to design wings in the presence of body. The same practice should be adopted for unsteady, aeroelastic wing design.

3.2.3 Pylon and Nacelle Installations.

Modern high aspect ratio transport aircraft have highly integrated wing, pylon and nacelle designs. Engine integration is critical to the overall performance of the aircraft. It is no longer acceptable to design a ‘clean’ wing and simply attach the pylon and nacelle at the end of the design. For modern steady state design, the wing, pylon and nacelle must be designed as one complete aerodynamic assembly. Fig. 12 shows the effect of pylon and nacelle on the steady-state wing pressures, the pylon is between stations 0.62 & 0.68 semispan.

Inboard of the pylon the wing lower surface experiences an overspeed due to the pylon. The wing upper surface shows an underspeed in the pylon region. The upper wing shock system is significantly altered. The presence of pylon and nacelle are therefore important for steady state flow solutions.

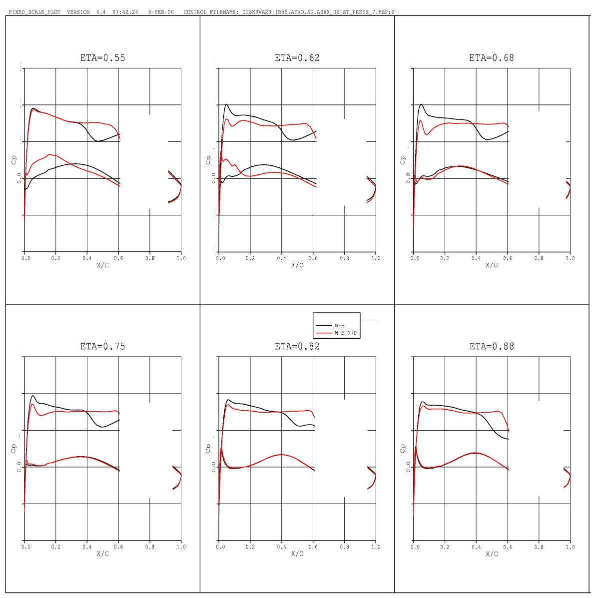

Fig.12. Steady State Outboard Wing Pressures.

(N.B. Shock pressures deliberately removed.)

Being able to calculate the aerodynamic influence of pylons and nacelles on the integrated wing performance for a dynamic calculation is a major project deliverable. Fig. 13 shows real and imaginary 1st harmonic pressures with and without pylon and nacelles for two wing sections. Note that the real pressures show that the presence of the nacelles has a significant effect. The nacelles change the shock strength and performance of the wing leading edge. Being able to produce high quality flow solutions of this nature will enable the wing, pylon and nacelle aerodynamic design to include an aeroelastic design element. This will reduce design cycle times, help maintain weight estimates and reduce costly flight test programs by developing designs that are 'mature' at first flight.

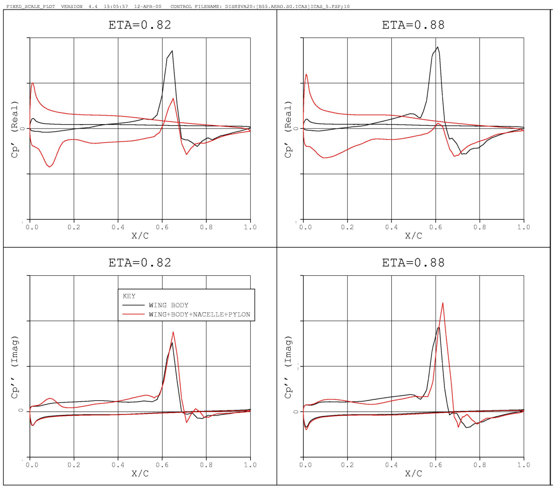

Fig.13. Real and Imaginary Outboard wing pressures.

Fig. 14 shows the wing 1st harmonic spanwise loads and moments, with and without nacelles and pylons present. It is clear that the nacelles and pylons have a significant effect on the predicted spanwise loads and moments. Large discontinuities are predicted in the unsteady 1st harmonic spanwise loads at the pylon locations. Note that the pylons and nacelles influence the unsteady spanwise loads across the entire span, from root to tip, not just at the pylon locations.

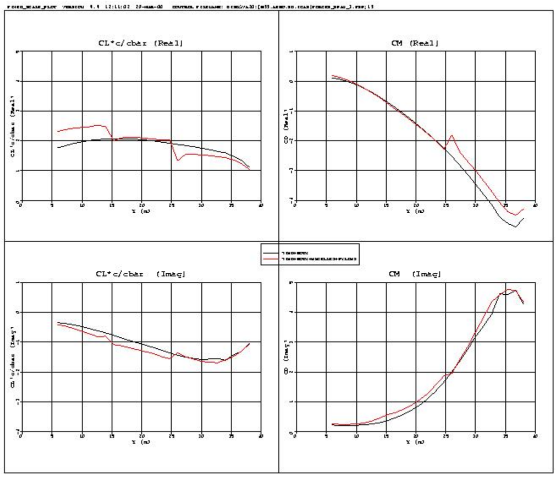

Fig.14. Real and Imaginary Spanwise Loads.

4.1 Conclusions.

BAE SYSTEMS has developed a time-accurate, unstructured, viscous-coupled Euler solver that will be used in a production environment for the solution of transonic aeroelastic problems. This solver has been applied to a variety of complex configurations.

The main conclusions drawn are:-

- Viscous effects for time-accurate solutions are important, reducing shock strength and moving the predicted shock location by around 10% x/c, for a given incidence. The implementation of a boundary layer solver adds a CPU overhead of around 30%. It is noted that the shock strength and its location are critical for a transonic aeroelastic calculation. Therefore, the added CPU expense of the boundary layer solver is justified.

- In order to reduce flow solution times an 'optimised' mesh philosophy has been developed. This optimised mesh philosophy was found to produce good quality time-accurate, viscous-coupled solutions, whilst reducing run times. However, the optimised mesh has the effect of weakening the shock and moving it slightly forwards.

For good quality time-accurate predictions, the complexity of the geometry must be well modelled.

- The body has been shown to have a significant effect on the unsteady pressures and span-wise loads across the majority of the wing span, including the tip region.

- The nacelles and pylons have a significant effect on the predicted spanwise loads and moments. Large discontinuities are predicted in the unsteady 1st harmonic spanwise loads at the pylon locations.

The code has reached sufficient maturity to be applied to current complex configurations. For future aircraft designs transonic aeroelastics will be used as a design parameter during the high-speed aerodynamic design process. This will therefore minimise risk during the high-speed wing shape design process, by identifying non-linear features, such as LCO and control buzz at an early stage in the design process. The transonic aerodynamic data from the time-accurate Euler code will therefore reduce structural and mass distribution reworking for aeroelastic reasons. This will help to maintain weight estimates and reduce costly flight test programs by developing designs that are 'mature' at first flight.

However, because of the significant increase in CPU usage, the Euler code will be used in the short term to produce 'correction factors' for linear theory and to look at specific problems not solvable with linear theory. i.e. the Euler code will supplement the existing linear theory.

5.1 References.

[1] Haase W, Chaput E, Elsholz E, Leschziner M and Muller U. *ECARP, Validation of CFD Codes and Assessment of Turbulence Models.* 1st edition, Vieweg, 1997.

[2] Jameson, A. AIAA Paper, Time Dependent Calculations Using Multigrid wing Applications to Unsteady Flows Past Airfoils and Wings, 91 – 1596, June 1991.