ANALYSIS OF A CLOSE COUPLED NACELLE INSTALLATION USING A PANEL METHOD (VSAERO) AND A MULTIGRID EULER METHOD (MGAERO)

David Lednicer
David Tidd
Analytical Methods, Inc.
Redmond, Washington USA
and
Nigel Birch
Rolls-Royce, plc
Derby, United Kingdom

Abstract

It is the intent of this paper to present a computational analysis of a close coupled turbofan nacelle installation using the VSAERO panel method and the MGAERO multigrid Euler method. The results are compared with wind tunnel data obtained by Rolls-Royce using a turbine powered simulator transport aircraft model. The model was tested during 1982 in the transonic wind tunnel at the Aircraft Research Association located in Bedford. It is shown that a reasonable simulation of the flowfield can be calculated with the panel method at a low transonic Mach number cruise condition, while the multigrid Euler method is required to more accurately calculate the features of the flowfield. Results from adding a viscous correction iteration in both codes are presented.

Introduction

The demand for higher thrust engines to be installed on commercial transport aircraft has led to the development of exceedingly large diameter, high bypass ratio turbofan engines. Early wing mounted installations of high bypass ratio engines have placed the engines on pylons well away from the wing to avoid excessive installation drag penalties. As fan diameters have increased in size, it has proven necessary to position engines closer to the wing in order to both maintain the current ground clearance and to avoid extending the already heavy main landing gear legs. To reduce the risk of unfavorable aerodynamic interference between the engine cowl and the wing, a means is required to assess installations before full-scale hardware is built. Empirical attempts to develop low drag, closely coupled installations have historically indicated a boundary as to how close the engine can be positioned relative to the wing, illustrated in Fig. 1. As larger diameter fans have made it necessary to design installations within this boundary, the development load has fallen onto computational fluid dynamics (CFD) methods to supplement wind tunnel testing.

Successes in designing closely-coupled installations within this boundary using CFD have been reported in the literature (Ref.2 and 3). However, it is necessary for each organization involved with commercial transport propulsion integration to internally assess the computational tools available and gain experience in analyzing this design problem. In 1990, Rolls-Royce initiated a program for the analysis of a close coupled turbofan nacelle installation using the VSAERO panel method. Subsequently, the MGAERO multigrid Euler method has also been used to analyze the geometry. Existing wind tunnel data taken at transonic speeds by Rolls-Royce with a turbine powered simulator (TPS) transport aircraft wind tunnel model was used to assess the applicability of these CFD methods. It should be noted that the VSAERO and MGAERO results were generated before access to the experimental data was obtained.

Experimental Configuration

The transport aircraft model to be analyzed was designed by Rolls-Royce to represent a modern technology, narrow bodied airliner, with two wing mounted high-bypass turbofan engines. This wind tunnel model consisted of a

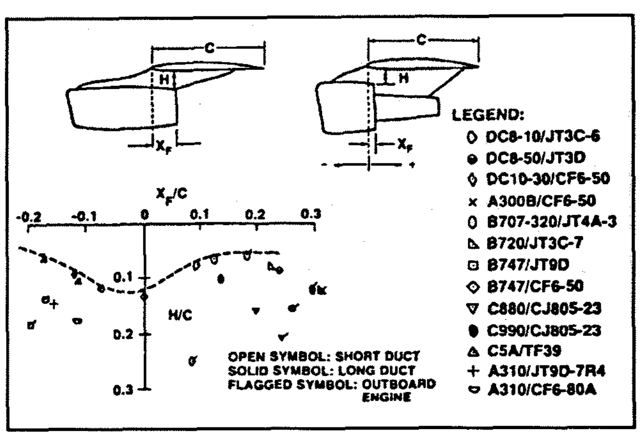

Fig. 1. Transport engine installations designed using wind tunnel test methodology.[1]

representative fuselage, a wing with an undercarriage hinge fairing, three flap track fairings and a pylon mounted engine nacelle. No vertical or horizontal tails were included. The nacelle enclosed a turbine powered simulator (TPS) to simulate the fan flow. Core flow was provided by the exhaust from the TPS drive turbine. A faired nose was used in place of the cockpit cab. Two separate nacelle configurations were tested consisting of a 3/4 length fan cowl, separate flow nacelle and a full length fan cowl, mixed flow nacelle. In addition the full length fan cowl was tested in a position 12 inches further forward of the wing which is the configuration used for the results described here. Computational comparisons with the other two configurations produced similar results. In all cases, the engine and nacelle were mounted pitched nose up 2.4235 degrees, relative to the Wing Reference Plane and toed in 1.5 degrees.

The half model, referred to as the B60, was tested in the transonic wind tunnel at the Aircraft Research Association at Bedford in 1982. The model contained eight rows of wing pressure taps, one row on either side of the pylon and three rows on the engine cowl (Fig. 2). Run conditions used for the computational results are listed in Table 1.

Table 1
Run Conditions

Mach number	.77	.80
Angle of Attack	2.74 deg	2.74 deg
Reynolds number	3.8×10^6/ft	5.3×10^6/ft
Tunnel Total	14.34 psia	14.34 psia
Tunnel Pstatic	9.68 psia	9.40 psia
Tunnel Ttotal	305 K	309 K
TPS Fan Pressure ratio	2.41	2.48
TPS Fan mass flow	2.748 lb/sec	2.733 lb/sec
TPS exhaust mass flow	4.357 lb/sec	4.335 lb/sec
TPS exhaust Ttotal	297 K	299 K

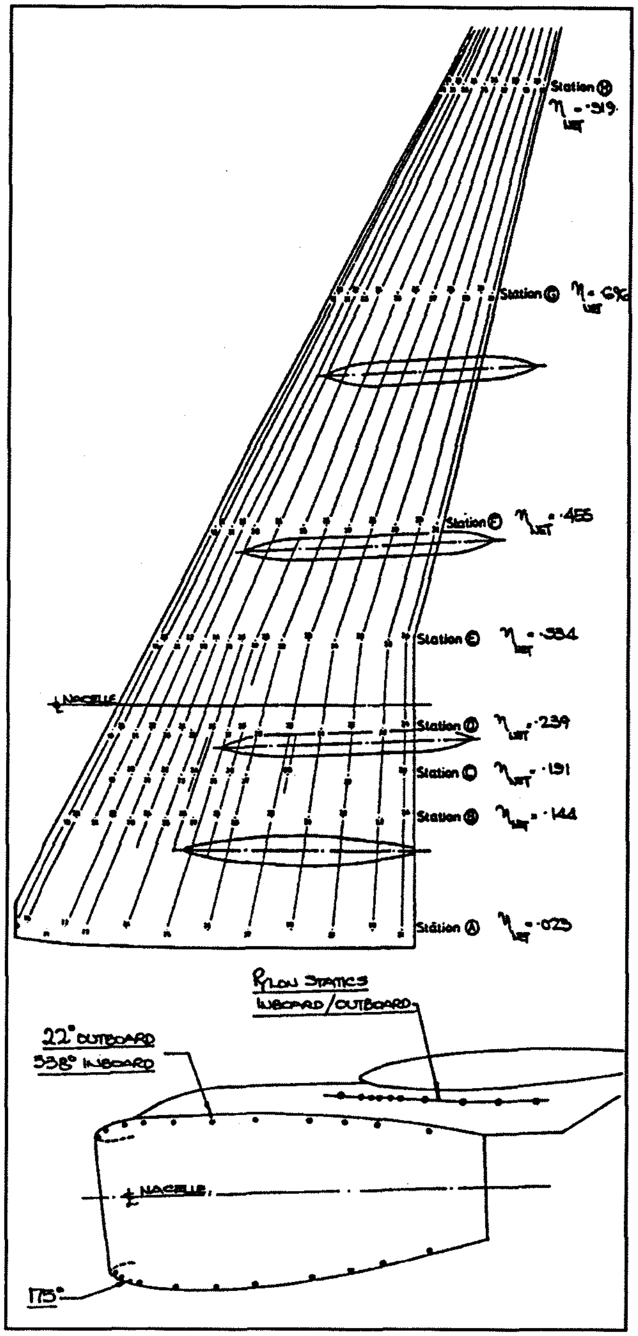

Fig. 2. Experimental surface pressure tap locations.

Description of Analysis Methods

Two different computational fluid dynamics codes, a Linearized Potential method and an Euler method, were used to analyze this geometry. An assessment of the ease of geometry modeling and computational accuracy for this transonic flowfield were goals of the analysis.

The panel method used was the VSAERO (Version E.5) computer code[4] which is used in the aircraft design and analysis environment[5,6] throughout the world. VSAERO is a full-configuration, aerodynamic analysis method that solves the linearized potential flow equations. The basis of the computer program is a surface singularity panel method using quadrilateral panels on which doublet and source singularities are distributed in a piecewise constant form. Effects of wake shape are treated in an iterative wake relaxation procedure, while the effects of viscosity are added in an iterative loop coupling potential flow and integral boundary layer methods. Boundary layer displacement is modeled by transpiration through the body surface in succeeding potential solutions. Compressible flow can be modelled by applying either a Karman-Tsien correction of the incompressible flow or a Prandtl-Glauert linearization to the compressible flow.

Most Euler methods require either a structured, body fitted grid in the flowfield or an unstructured, but still body aligned, flowfield grid. The Euler method used here, MGAERO (Multi Grid **AERO**dynamics)[7] instead uses a multigrid calculation procedure on an equally spaced Cartesian mesh structure to discretize the governing Euler equations for inviscid compressible flow. The use of the Cartesian multigrid technique eliminates the labor-intensive task of building complex body-fitted grids, thus allowing for rapid turnaround of results. The finest grids are only positioned in high curvature regions and the flow areas of interest. A given configuration is treated as a Boolean sum of components that may intersect, thus allowing the geometry to be prepared in a modular fashion. The intersections are treated automatically by the code and need not be previously defined by the user. The geometry of each component is defined using a set of surface panels similar to VSAERO. Surface curvature is computed at each grid to surface intersection point for application of the surface boundary condition. In contrast to most body fitted grid techniques, there is no direct relationship between the surface paneling and the local grid cell structure.

Results from a viscous correction capability that has been recently added to MGAERO are also presented herein. The boundary layer procedure used in MGAERO follows a similar procedure to that used in VSAERO. The technique begins from a converged inviscid solution and uses the lag entrainment method to integrate the boundary layer properties along computed surface streamlines. This provides surface transpiration velocities at all points along the streamline. The viscous correction is then applied to the Euler equations in an iterative procedure whereby the surface boundary condition is modified by the computed transpiration velocities. This requires interpolation of the transpiration velocities from the surface streamlines to the grid/surface intersection points. To date the modelling of strong shock boundary layer interaction has required the use of about 5 viscous/inviscid iterations utilizing 20 multigrid "V" cycles on the finest grid level in each visit to the inviscid solver.

<u>VSAERO Analysis Results</u>

Digital data describing the nacelles and the wing, supplied by Rolls-Royce, were used to build the VSAERO model. The blunt trailing edge of the wing was reduced to zero thickness in the course of producing the VSAERO model's wing. This was carried out to simplify the wake modelling in VSAERO and has little effect on the wing aft loading characteristics. The model of the fuselage was constructed from part drawings. A combination of tabulated inspection points provided by Rolls-Royce and data taken from part drawings were used to model the engine pylons. Similarly, digitized points were used to model the landing gear fairing and the flap track fairings. All intersections were calculated to develop a continuous surface paneling along the component junctions. As all cases to be run were at symmetric flight conditions, only a half model was built. A high panel density was used on the model, containing over 4300 surface panels (Fig. 3 and 4). Constant doublet strength, trailing edge type wakes were attached to the wing and fairing trailing edges. A jet wake, with a linear change in doublet strength in the streamwise direction, was attached to the nacelle trailing edge. The change in temperature across the jet wake boundary is not modelled in VSAERO. In total, 3585 wake panels were used. All cases were run on a Silicon Graphics 4D/240 Power Series IRIS workstation with run times averaging 6500 seconds. All visualization of results was done with a three-dimensional graphics postprocessing program, OMNI3D.

Run conditions were supplied by Rolls-Royce from those used in the ARA wind tunnel. The specified normal velocities used to model the flow into and out of the nacelle were calculated using the equations of isentropic flow. As operating conditions for all cases were at transonic Mach numbers (.77 to .84), a means of correcting for compressibility effects was necessary. After considerable study, it was decided to use the Prandtl-Glauert compressibility correction, with flux boundary conditions, using a second-order C_p calculation. A Prandtl-Glauert correction with approximate normal velocity boundary conditions, using a tangent gas law C_p calculation, was also found to give good answers, except in the leading edge suction region.

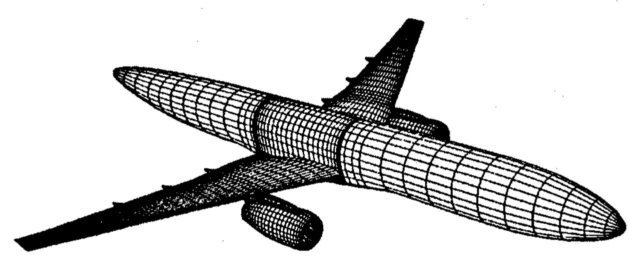

Fig. 3. VSAERO model of Rolls-Royce B60 powered model upper front view.

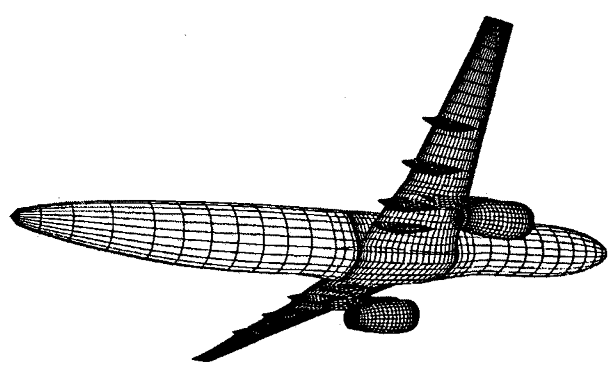

Fig. 4. VSAERO model of Rolls-Royce B60 powered model lower rear view.

To compute the boundary layer effects on the model, approximately 230 streamlines were calculated on the configuration and boundary layer calculations were performed along these paths. All boundary layers were forced to transition at a Re_θ value of 200, to represent mostly turbulent flow. Five viscous/potential iterations and one wake shape relaxation were run for each case.

Comparisons of VSAERO calculated inviscid and viscous results with experimental data for the run at a Mach number of 0.77 are presented in Fig. 5. In all cuts it is can be seen that the predicted viscous effects from VSAERO are small. Moving outboard along the wing, it can be seen that the surface pressures calculated by VSAERO on the wing lower surface are in quite close agreement with those measured experimentally. There is a slight over prediction of the nacelle installation effects on the wing cuts close to the pylon. The upper surface of the wing has regions of sonic flow at this condition, including a shock on the outboard wing, which VSAERO is unable to predict. However, the calculated pressure distribution on the upper surface of the inboard wing compares well with those measured experimentally. A slight overprediction of the leading edge suction is a result of the compressibility correction. Calculated pressures near the nacelle keel and on either side of the nacelle crown line are in excellent agreement with experiment. On the pylon, VSAERO calculated pressures exhibit differences when compared with experimental results. On the outboard side of the pylon, differences arise near the wing leading edge, while inboard there are large differences in the "gully" region, where there is a small region of supersonic flow on the configuration. Here, VSAERO calculates the velocities to be higher than the experimental measurements. The pressure taps in this area are only about half a panel width from the wing so a greater panel discretization in this area may be required.

MGAERO Analysis Results

The geometry descriptions used in constructing the VSAERO model were also used to develop the MGAERO model. A higher panel density was used to represent the surfaces in the MGAERO model which consisted of over 15000 surface panels. Additionally, the surfaces were not trimmed at intersections which made the model setup faster. The wing was extended inside the fuselage, the wing fairings inside the wing and the pylon inside the wing and nacelle. The wing trailing edge was modeled with the finite thickness present on the wind tunnel model.

A Cartesian grid arrangement containing 7 levels of refinement with a total of about 550,000 grid points was developed. The finest grids were concentrated on the wing leading edge with the fuselage and nacelle only containing up to a level 4 and level 5 refinement, respectively. The nacelle exit conditions were modelled using a specified total pressure, total temperature and Mach number at the fan exit. The artificial viscosity used to stabilize the Runge-Kutta iteration in MGAERO can cause a too rapid disintegration of the jet plume. In order to prevent this a sensor is used to detect the plume boundary and use less artificial viscosity in the numerical scheme for this region.

Viscous effects were only computed for the wing component for which a total of 138 surface streamlines were traced. The boundary layer calculations were performed assuming a turbulent boundary layer over the entire length of each streamline. All computations were carried out on a Silicon Graphics Crimson workstation. Run times were about 6 hours for the basic inviscid run and a further 8 hours for 5 viscous iterations. Some code optimization is still required to reduce the CPU requirements for the viscous correction.

The MGAERO computed and experimental values for the surface pressure distribution at a Mach number of 0.77 are shown in Figure 6 and may be compared with the VSAERO computations in Figure 5. The agreement on the wing lower surface is similar to that obtained from VSAERO with the installation effects slightly overpredicted again. The advantage gained from the newly developed viscous correction in MGAERO is clearly apparent in the results on the wing upper surface. The inviscid solution shows an overprediction of the suction pressure over the entire wing upper surface with the viscous results in much closer agreement. The shock position and strength in the outer wing is also much more closely predicted by the viscous results. However, it appears that the shock is still not quite far enough forward which may be as a result of the artificial viscosity combining with the shock/boundary layer interaction characteristics. On the inboard wing a very small shock appears to be present on the upper surface which has been smeared by the MGAERO calculation. The agreement on the nacelle and pylon cuts are also similar to those obtained with VSAERO although it is clear that some additional field mesh density is required to pick up the suction around the nacelle leading edge where only a level 5 density was used. It is expected that improvements in the pylon will be made by adding in viscous effects to this component. Work on this is currently underway.

The pressures computed by MGAERO and measured experimentally for a slightly higher Mach number of 0.8 are presented in Figure 7. The inviscid results from MGAERO show similar agreement to the FLITE3D results presented previously in Ref. 8 for this flight condition. In contrast to MGAERO, FLITE3D is an unstructured Euler code. The main difference in the two run conditions presented here is that the shock position on the outboard wing is significantly further aft in the higher Mach number case. The MGAERO viscous calculation results show

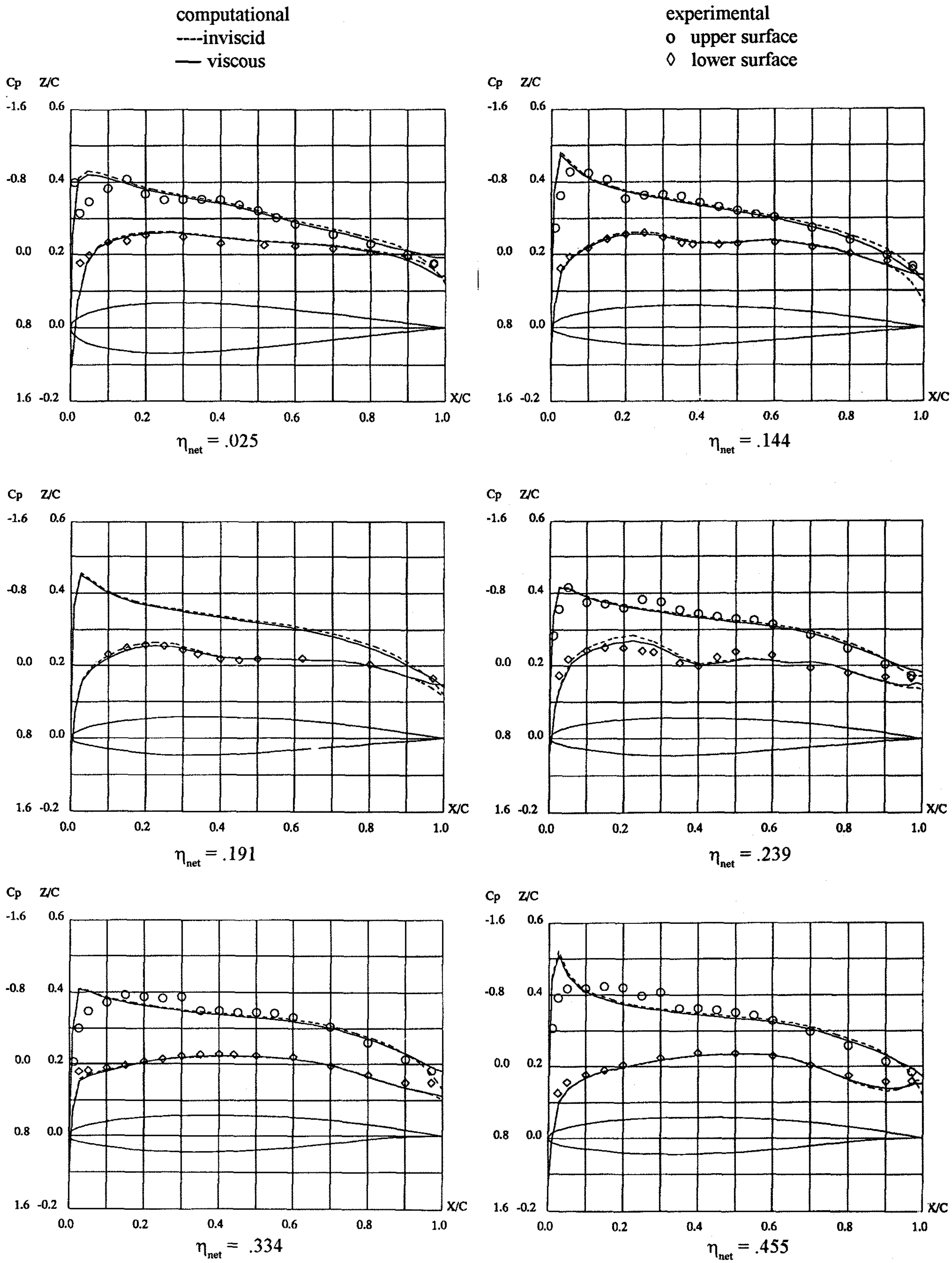

Fig. 5. Comparison of VSAERO and Experimental Results, M = .77, Alpha = 2.74 deg.

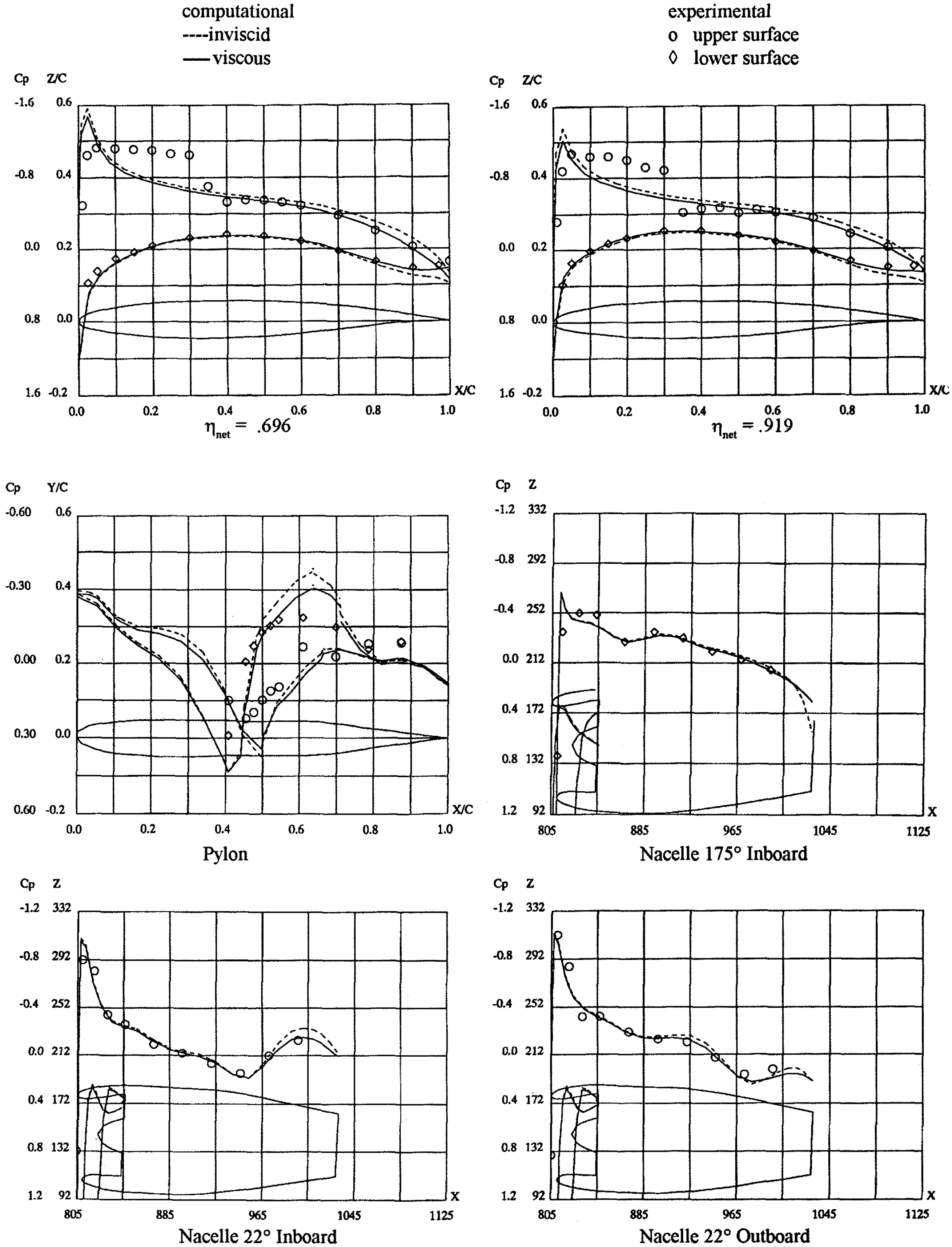

Fig. 5. (continued)

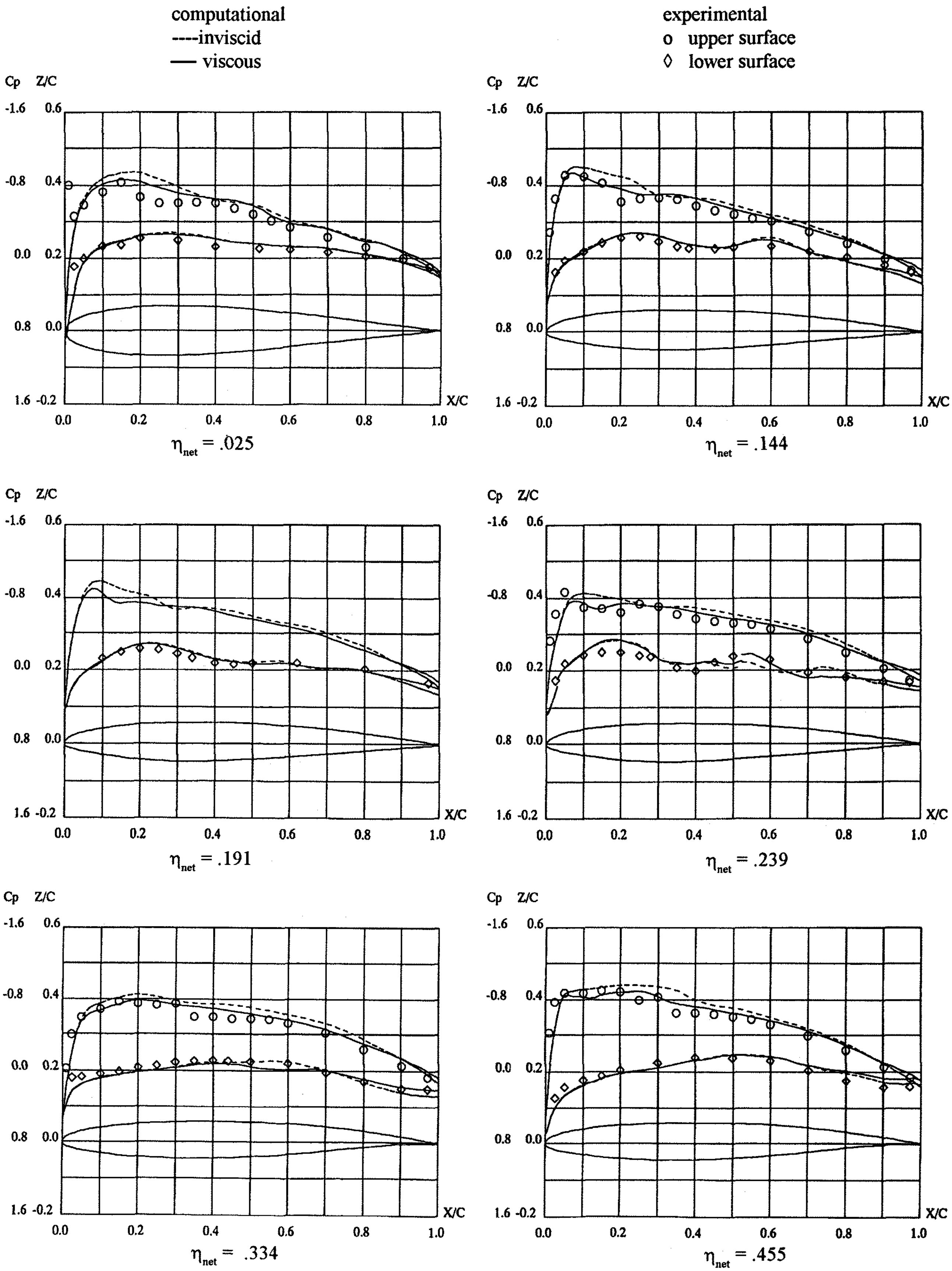

Fig. 6. Comparison of MGAERO and Experimental Results, M = .77, Alpha = 2.74 deg.

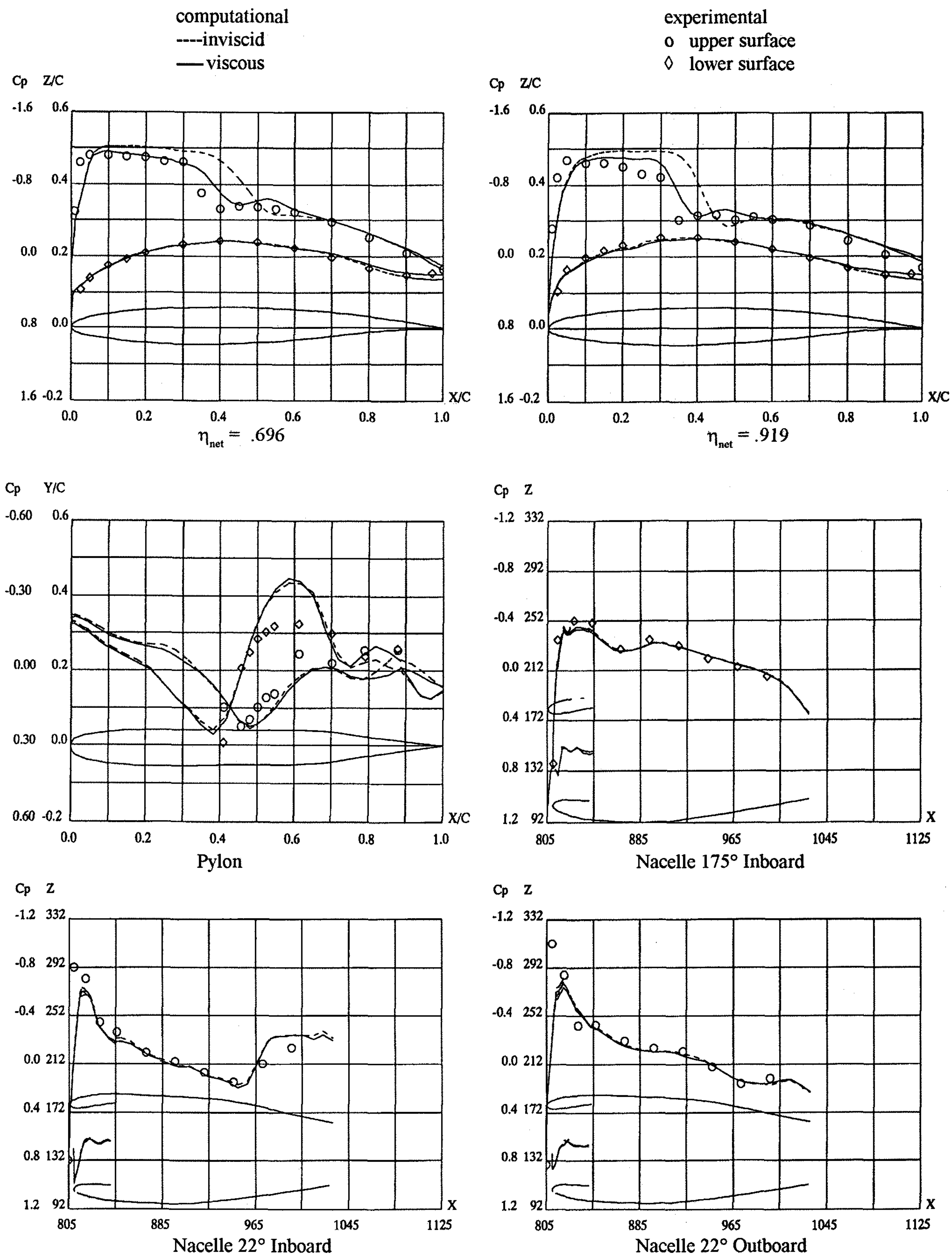

Fig. 6. (continued)

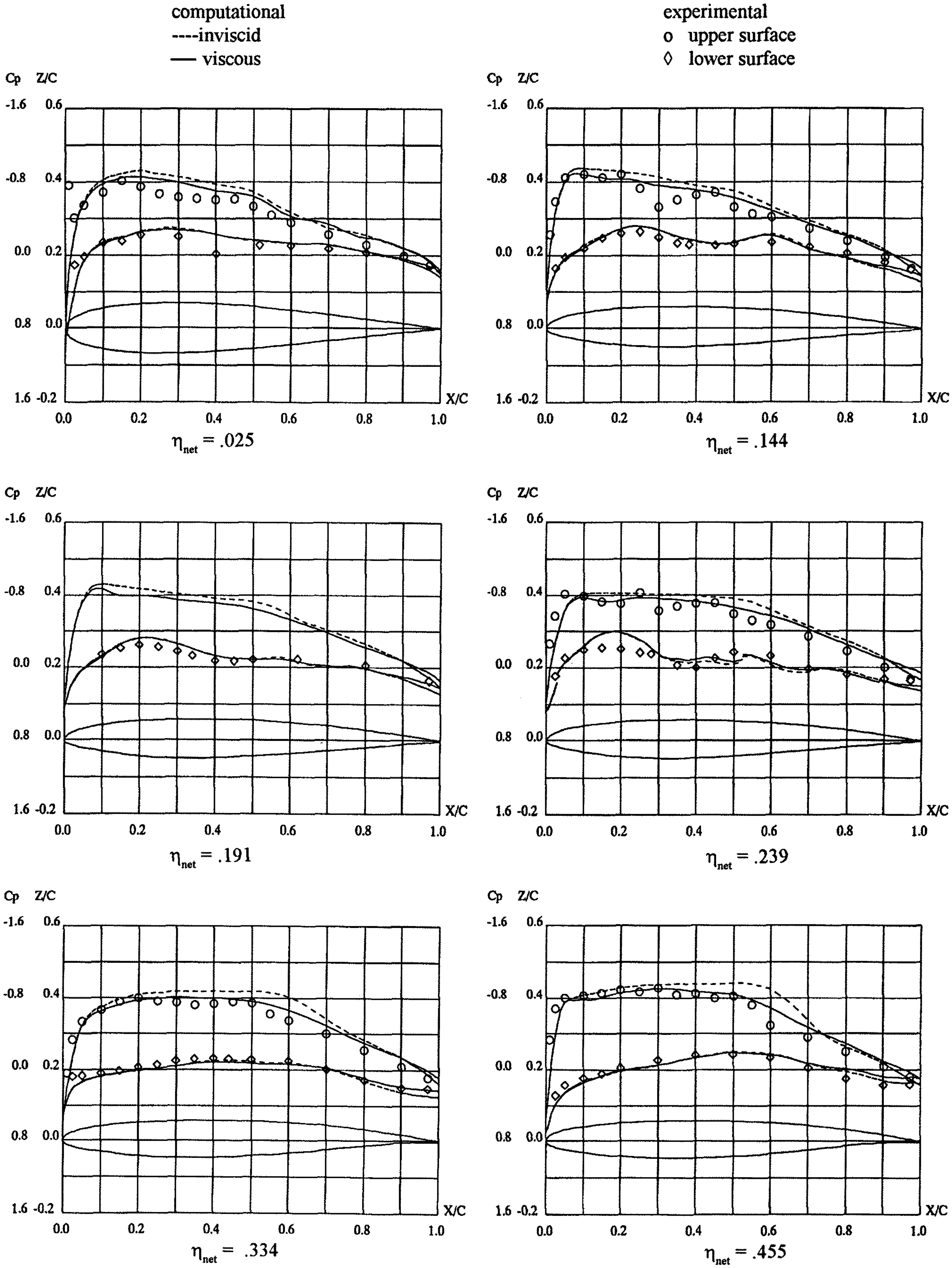

Fig. 7. Comparison of MGAERO and Experimental Results, M = .80, Alpha = 2.74 deg.

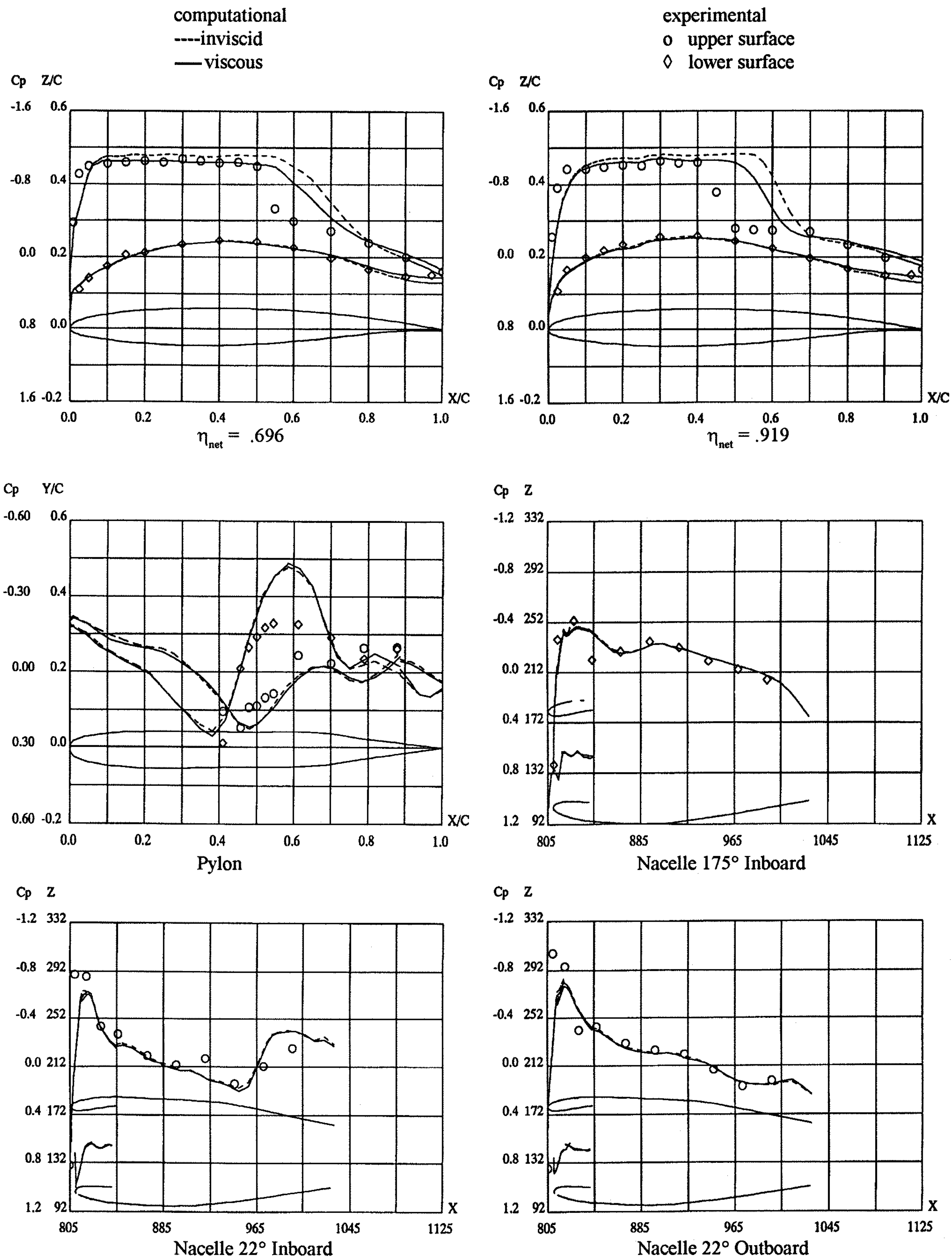

Fig. 7. (continued)

features similar to the lower Mach number case when compared with experiment. However, the shock position has not moved sufficiently forward from the inviscid calculation. From the two sets of results is appears that the shock/boundary layer interaction is greater at the higher Mach number. MGAERO has predicted a smaller shock movement between the viscous and inviscid solutions for the higher Mach number compared to the lower Mach number results.

Conclusions

Results from the computational analysis of a transport configuration with a close-coupled turbofan nacelle have been presented. Comparisons of the results with experimental data has produced several conclusions.

1. VSAERO does a satisfactory job of calculating the subsonic areas of the flowfield for this configuration. If more accuracy is desired in calculating details of the supercritical flowfield, a Full Potential, Euler or Navier-Stokes code must be used.

2. Compressibility effects are best accounted for by the use of the Prandtl-Glauert compressibility correction within VSAERO.

3. The MGAERO multigrid Euler method provides the means necessary to accurately model the entire flowfield for this configuration.

4. The boundary layer calculation capabilities present in MGAERO are an important part of calculating an accurate shock position. Further work is required to investigate shock boundary layer interactions and to add viscous effects on all parts of the configuration.

5. The Cartesian multigrid structure as implemented in MGAERO enables rapid geometry and numerical model preparation for computational analysis in a design environment.

Acknowledgements

The authors wish to thank Rolls-Royce for supporting this work and to thank MOD/RAE for support of the experimental work and for permission to publish the data. Thanks are also due to Ian Gilchrist of Analytical Methods, Inc. and Geoff Pugh and Elaine Marshall of Rolls-Royce for their help with this project. The results and opinions expressed in this paper are those of the authors alone; they do not necessarily reflect the views of Rolls-Royce.

References

1. Rubbert, P.E. and Tinoco, E.N., "Impact of Computational Methods on Aircraft Design," AIAA paper 83-2060, presented at the AIAA Atmospheric Flight Mechanics Conference, August 15-17, 1983, Gatlinburg, Tennessee.

2. Tinoco, E.N. and Chen, A.W., "CFD Applications to Engine/Airframe Integration", Numerical Methods in Engine-Airframe Integration, AIAA Progress in Astronautics and Aeronautics, Vol. 102, pp. 219- 255.

3. Greff, E., Becker, K., Karwin, M. and Rill, S., "Integration of High Bypass Ratio Engines on Modern Transonic Wings for Regional Aircraft", Aeronautical Journal, January 1993, pp. 1-13.

4. Maskew, B., "Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods," AIAA paper 81-0252, presented at the 19th Aerospace Sciences Meeting, St. Louis, Missouri, January 12-15, 1981.

5. Pfeiffer, Neal J. and Zickuhr, Tom D., "Validation of Computational Aerodynamics Applied to General Aviation Configurations," AIAA paper 89-2169, presented at the 7th Applied Aerodynamics Conference, Seattle, Washington, July 31-August 2, 1989.

6. Narramore, J.C., "Application of Computational Fluid Dynamics Methods to the Component Aerodynamic Design of the V-22 Osprey Tilt Rotor Vehicle," Applied Computational Aerodynamics, AIAA Progress in Astronautics and Aeronautics, Vol. 125, pp. 637- 678.

7. Tidd, D.M., Strash, D.J., Epstein, B., Luntz, A., Nachshon, A. and Rubin, T., "Application of an Efficient 3-D Multigrid Euler Method (MGAERO) to Complete Aircraft Configurations," AIAA paper 91-3236, presented at the 9th Applied Aerodynamics Conference, Baltimore, Maryland, September 23-25, 1991.

8. Peiro, J., Peraire, J., Morgan, K., Hassan, O. and Birch, N., "The Numerical Simulation of Flow About Installed Aero Engine Nacelle Using a Finite Element Euler Solver On Unstructured Meshes," Aeronautical Journal, June/July 1992, pp. 224-232.