VISCOUS COUPLING TECHNIQUES USING UNSTRUCTURED AND MULTIBLOCK MESHES

Dr. J. Szmelter
British Aerospace Airbus Limited - P.O. Box 77 - Bristol BS99 7AR, U.K.

Abstract

A new technique which allows for the viscous coupling of Euler solvers based on irregular i.e. Multiblock and Unstructured Meshes has been proposed and implemented. The method allows for modelling of viscous flows with limited separation over complex civil aircraft configurations. Within the limitations of the assumptions of boundary layer theory, the method has several advantages by comparison with standard explicit Navier-Stokes solvers. In particular it avoids the problems associated with the generation of the highly stretched meshes required by such solvers for complex configurations in the region of the boundary layer. The generation of highly stretched meshes is particularly difficult for tetrahedral or triangular elements used in Unstructured Meshes. Since the larger size of elements is admissible, the method is efficient by comparison with explicit Navier-Stokes solvers. The penalty for the incorporation of viscous effects is on average of the order of 15% with respect to an Euler solution. The modelling of turbulent flows achieved using the lag-entrainment boundary layer method is satisfactory for wing design applications and in practice is more accurate than standard turbulence models used in Navier-Stokes solvers.

Introduction

The work reported in this paper has been specifically directed towards civil aircraft applications, for which the numerical modelling of complex configurations and complex flows, combined with the requirement for accurate modelling of viscous effects pose an important class of problems.

In this paper a viscous-inviscid interaction method is proposed which can overcome some of the problems characteristic to other methods. Its application is limited by the well known assumptions of the boundary layer approximations and the method in general cannot be used for the calculation of large, viscous dominated regions of separation for which the full Navier-Stokes equations have to be solved. However, it provides a valuable solution to many aerodynamic problems, particularly within the civil wing design process, for which cruise drag reduction and buffet margins are critical and therefore the ability to model accurately transonic flows is essential.

Methods based on viscous-inviscid interaction between Full Potential or Euler solvers and boundary layer codes are very well established for simple (i.e. wing alone or wing body) configurations. Indeed, the effectiveness of such Euler methods codes has been significantly enhanced by the use of viscous coupled techniques. Today these single block based suites of codes are extensively and routinely used by the wing designers.

In this paper an extension of such methods to allow for the calculation of flows around geometrically complex configurations is proposed. The method uses a combination of an Euler solver based on irregular (i.e. Unstructured or Multiblock) mesh techniques coupled to a boundary layer code.

The Multiblock and Unstructured Meshes techniques are complementary. Multiblock has achieved a significant advancement in aerodynamic design methodology, however experience with practical applications has highlighted both advantages and shortcomings in the technique. In practice the Multiblock meshes are used for moderately complex configurations such as wing, body, pylons and nacelles assemblies. Mesh generation from scratch, when a new topology needs to be created, is usually long winded and requires specialist training. For new, more complex, detailed geometries for which Multiblock mesh generation reaches its limits of application, the use of Unstructured Meshes is more appropriate. The flexibility and generality offered by unstructured tetrahedral and triangular meshes to model geometrically very complex configurations have been well recognised, particularly as the method offers relatively short lead times to generate new meshes. However the successful application of Unstructured Meshes techniques is still largely limited to the solution of the Euler equations since the generation of highly stretched, good quality unstructured meshes suitable for the solution of the three dimensional Navier-Stokes equations for complex geometries is still in its pioneering stages. Moreover, associated flow codes do not readily provide solutions within the range of acceptable costs and accuracy needed for civil aircraft design. Although useful calculations can be performed for problems for which wall functions can be safely used, the accuracy offered by the turbulence models commonly used with the Navier-Stokes flow solvers requires further research.

Viscous-inviscid interaction methods offer a valuable engineering tool suitable for specific classes of problems. Most importantly, for fast turn around in the design process, the computing times required by solutions of the viscous coupled Euler methods are very competitive by comparison with the Navier-Stokes codes.

Numerical Procedure and Results

The methodologies presented in this paper form a further extension of the work reported earlier by the author [1,2,3]. The general viscous-inviscid interaction procedure using irregular meshes can be summarised by the following steps:

1. Euler Solution using an 'inviscid mesh'
2. Interpolation of the flow values from the 'inviscid mesh' into the 'viscous mesh'
3. Coupling procedure
4. Solution of the boundary layer equations using the 'viscous mesh'
5. Interpolation of the flow values from the 'viscous mesh' into the 'inviscid mesh'.
6. Return to step 1.

To maintain the generality of the procedure it is worthwhile to organise the components of steps 1 to 5 as separate modules which can be easily exchanged to allow the use of different types of flow solver individually sympathetic to the problem under consideration. Thus the Euler solvers may be selected to operate using either Multiblock or Unstructured Meshes. Moreover different solvers such as explicit, implicit, multigrid, etc. can be implemented. Coupling procedures may also be exchanged, with the option of, for example, choosing between semi-inverse and quasi-simultaneous coupling techniques. Different boundary layer flow solvers such as two-dimensional integral methods or three dimensional differential methods can be applied.

While the above mentioned modules provide a standard set of routines for many existing solvers, the important part of the presented method, which ensures its generality and flexibility, lies in the introduction of the concept of non-overlapping 'inviscid' and 'viscous' meshes and in the development of sufficiently accurate interpolation routines.

The well established single block viscous coupled codes developed for simple geometries, such as wing alone or wing-body, use the same surface mesh for the solution of both Euler and boundary layer equations. In the case of complex configurations, the generation of a Multiblock mesh is likely to introduce singular points on the surface, thus making it unsuitable for use in the calculation of the boundary layer. For some simpler cases a special Multiblock mesh can be devised which avoids such singular points, however apart from introducing additional difficulties, the resulting meshes are likely to be of poor quality, especially for the reliable performance of boundary layer routines. Of course for Euler solvers based on Unstructured Meshes the generation of a separate regular 'viscous mesh' is compulsory. Furthermore care must be taken in identifying a 'good viscous mesh'. The attributes of such mesh depend primary on the specifics of the boundary layer method which is used in this procedure.

In our implementation, initially a suite of codes was developed for the computation of two dimensional steady-state transonic viscous flows. Proof of concept was first illustrated on the basis of two dimensional unstructured meshes. The unstructured meshes suites used in this work originated at the University College of Swansea. The two dimensional unstructured meshes are generated using the Advancing Front technique and the Euler solver is based on the Runge-Kutta time stepping scheme. The basic space discretisation is accomplished by employing a finite element, cell vertex, finite volume equivalent scheme [4]. The semi-inverse viscous-inviscid interaction technique [5], shown in Fig. 1, has been introduced together with a two-dimensional integral boundary layer method.

The integral laminar/turbulent boundary layer method, pioneered at DRA, uses either a prescribed pressure distribution (direct mode) or the predicted growth in displacement thickness (inverse mode) to drive the boundary layer equations. The approach is based on the 'lag-entrainment' method [6,7] which has been further developed by DRA to incorporate curvature effects on turbulence structure [8], compressibility effects [9], low Reynolds Number correction, normal shear stress terms [10] as well as wake calculation [11]. The laminar portion of the boundary layer is calculated by the Thwaites compressible method, with transition predicted by the Granville correlation criterion. With forced transition the laminar separation bubble is calculated by Horton's method.

For applications to realistic civil aircraft configurations it is important that the theoretical design/prediction tools are able not only to calculate well behaved attached laminar and turbulent flows but also to handle 'off design' behaviour such as : shock induced and trailing edge boundary layer separation and interaction between shock induced and trailing edge separation. This capability was demonstrated first by DRA [12] , by coupling the lag-entrainment code to a Full Potential method and later by BAe [13] - the first in Europe to couple the lag-entrainment method to an Euler method. Le Balleur has extended the semi-inverse coupling method to deal with massively separated flows [14].

To illustrate the potential capability of the two dimensional method, preliminary calculations were performed for the two well known RAE2822 test cases:

Case 3: $M = 0.600$ $\alpha = 2.57^0$ $Re = 6.3 \times 10^6$
Case 6: $M = 0.725$ $\alpha = 2.92^0$ $Re = 6.5 \times 10^6$

The numerical viscous results are compared with experiment [15] in Fig. 2. The quality of the solution is good and it illustrates the importance of proper trailing edge and wake treatment.

The two dimensional test cases have been obtained on specially generated unstructured meshes for which the wake sheet has been treated as a boundary line defined between the trailing edge point and the downstream far field boundary. Special boundary conditions have been introduced to ensure continuity of flow across the upper and lower wake regions. This approach, although common in Multiblock methods, is limiting the generality of the technique especially in three dimensions. The alternative concept of wake treatment using non-overlapping 'Euler' and 'boundary layer' meshes for two dimensional cases is illustrated in Fig. 3 where the computational points used for the boundary layer calculation lie on the wake line which is independent of triangular elements.

The method has been further generalised for three dimensional flows and is valid for Euler solvers based on 'irregular meshes' i.e. for both Multiblock and Unstructured Meshes. For such codes two separate non-overalpping meshes are applied: one is a standard tetrahedral or multiblock mesh used for the Euler solution, the other is used for the calculation of the boundary layer and consists of a quadrilateral surface mesh. An example of an unstructured Euler mesh for a typical civil aircraft wing configuration is shown in Fig. 4. Examples of 'viscous' meshes for wing and wake sheet surfaces are shown in Fig. 5 and Fig. 6, they illustrate the fact that several approaches can be taken for the boundary layer calculation, depending on the type of geometrical complexities on the wing.

The Multiblock Euler flow solver was developed from the basic Jameson finite volume, cell centered, multistage Runge-Kutta scheme. It also utilises block amalgamation and multigrid techniques. Currently either the semi-inverse or the quasi-simultaneous coupling technique can be used optionally to couple the inviscid Multiblock flow solver to the same two-dimensional integral boundary layer code adopted for two-dimensional solvers.

An interpolation routine needs to be used to interpolate values from the inviscid solution to the 'viscous mesh' points. The values of Cp in the case of semi-inverse coupling and the velocity components in the case of quasi-simultaneous coupling are interpolated from quadrilateral multiblock mesh points on the surface of the wing and on the wake, at present the viscous coupling is introduced for the wing only. A linear, weighted function consistent with the flow solver is used for interpolation. As illustrated in Fig. 7, more than one or none of the boundary layer 'viscous mesh' points may be present in a multiblock surface element and the values used in the interpolation are taken only from the nodes of the multiblock element to which the boundary layer point belongs. Such approach allows the choice to ignore 'holes' resulting from the presence of underwing pylons as shown in Fig. 5 and to use a viscous mesh instead, as in Fig. 6. It also allows for future extension of the method for application to other than wing components of an aircraft.

This concept of non-overlapping meshes is general in its approach and is also valid for the viscous coupling to Unstructured Meshes, as shown in Fig. 8 and as discussed further on in this paper.

After the boundary layer calculation is completed, a similar linear interpolation function is used to pass information about transpiration velocity values between the rectangular 'viscous mesh' and the centers of multiblock surface cells. Such set of interpolation routines is only valid for use with the two-dimensional integral boundary layer and needs to be extended when the three dimensional differential boundary layer is introduced. A fast searching algorithm is used to identify points used in the interpolation process between the two meshes. Both interpolation and searching routines are similar to those used in context of multigrid when non-overlapping meshes are employed. The quality of the solution is sensitive to the distribution of points on the 'viscous' quadrilateral mesh with respect to the Multiblock 'Euler' mesh. Therefore, in order to ensure freedom of easy surface 'viscous mesh' generation for complex configurations, a highly flexible surface panelling mesh generator had to be developed.

The method has been extensively tested using quantitative comparisons with results obtained using overlapping viscous-inviscid solutions, Navier-Stokes results and experiments. An example of results obtained by the method for typical civil aircraft configuration is presented in Figs. 9a and 9b which show surface inviscid meshes, corresponding viscous meshes for the wing as well as surface Cp contour plots.

The work using three dimensional unstructured meshes is still in its initial stage of development, however, promising preliminary results have been obtained. Again, as it was the case for the Multiblock technique, the concept of non-overlapping meshes is introduced. For the inviscid solution, the method uses an Unstructured Meshes Euler suite which originated at University College of Swansea. Surface mesh generation is performed by the Advancing Front technique, volume mesh generation uses Delaunay Triangulation. A finite element Runge-Kutta Euler flow solver with TVD type of artificial viscosity is applied. The same modules for semi-inverse coupling and two-dimensional integral boundary layer as for the multiblock version of the code are being used. Also the same flexible panelling surface mesh generator for the 'viscous' boundary layer meshes is employed.

Wake treatment has not yet been implemented in the current version of the code, therefore the interpolation routines are active for the wing only. The Cp values are passed from the inviscid solution to the boundary layer computational points as shown previously in Fig. 8. In the case of unstructured meshes the linear interpolation takes into account Cp values calculated in the four nodes of the tetrahedra whose face forms a surface mesh. Of course a computational point from the boundary layer 'viscous' meshes will always lie in the area of this boundary face. However, similarly as for the multiblock meshes, the methods admits more than one or none of the boundary layer points to be present within one tetrahedral element.

After the boundary layer calculation is completed, the transpiration velocity is interpolated using a linear function, from the regular, rectangular viscous mesh into every node of the tetrahedral finite element (cell vertex) mesh, which lies on the surface of the wing. The use of the Euler equations with modified boundary condition incorporating transpiration velocity coming from the boundary layer solution allows for the next viscous- inviscid interaction cycle to be performed.

The use of the fast searching algorithm to identify points employed in the interpolation process between two meshes is even more important in Unstructured Meshes than in Multiblock. Current experience indicates that viscous meshes, which would be traditionally used for wing-body single block Euler viscous coupled methods can be used with adequate accuracy also for wings with typical geometrical complexities for civil aircraft configurations. However further research is needed to investigate both for Multiblock and for Unstructured Meshes what is a 'good' viscous mesh. For the accurate calculation of shocks waves and adequate prediction of separation incorporation of the wake treatment and special treatment of the trailing edge region are essential. For three dimensional unstructured meshes, the concept of using non-overlapping meshes in the sense illustrated in Fig. 3 for two-dimensional problems is planned.

The application of viscous-inviscid interaction techniques using Unstructured Meshes has an additional bonus since it allows the achievement of good quality solutions without the need for the generation of highly stretched tetrahedral elements in the vicinity of the wall. Such elements are necessary for the solution of the full Navier-Stokes. equations. The proposed viscous-inviscid interaction method uses much larger elements than it would be required by Navier-Stokes solvers. Consequently, for explicit flow solvers, this allows much larger time steps to be used. The method is efficient, since the penalty for the incorporation of viscous effects is on average of the order of 15% with respect to corresponding Euler solutions.

Conclusions

A new methodology has been proposed which demonstrates the feasibility of combining irregular, Multiblock and Unstructured Meshes Euler solvers with boundary layer methods to develop a technique capable of predicting viscous flows for application to wing design process. The described method is general and allows to take advantage of many already existing codes. Moreover, any improvement in flow prediction using Euler solvers such as multigrid or parallel computing will be also beneficial in the context of the proposed viscous-inviscid interaction technique. The method is also applicable to the simulation of unsteady flows and is capable of taking advantage of adaptive mesh techniques. Also, the efficiency with which viscous solutions are obtained shows its good potential for use in the generally expensive design optimisation codes.

Acknowledgements

The author would like to thank Mr . John Welch (BAe Airbus Ltd.) for his help in the preparation of this paper.

References

1. J. Szmelter and A. Pagano, 'Computational Techniques for Unsteady Aerodynamics.' Proceedings of the 19-th ICAS Congress/AIAA Aircraft System Conference, Anaheim 1994.

2. J. Szmelter and A. Pagano, 'Viscous Flow Modelling using Unstructured Meshes for Aeronautical Applications.', Lecture Notes in Physics 453 ed S.M. Deshpande et al., Springer-Verlag, 1994.

3. J. Szmelter and A. Pagano, 'Viscous-Inviscid Interaction Techniques using Unstructured Meshes.', ed. K.W. Morton et al., Clarendon Press, Oxford 1995.

4. O. Hassan, E.J. Probert, K. Morgan, and J. Peraire, 'Domain Decomposition Combined with Adaptive Remeshing for Problems of Transient Compressible Flow', Proceedings 13th International Conference on Numerical Methods in Fluid Dynamics, Rome 1992.

5. J. C. Le Balleur, 'Calcul par Couplage Fort des Ecoulements Visqueux Transsoniques incluant Sillages et Decollemants. Profils d'Aile Portant', La Recherche Aerospatiale, May-June 1981.

6. J.E. Green, D.J. Weeks and J.W.F. Brooman, 'Predicton of Turbulent Boundary Layers and Wakes in Compressible Flow', RAE technical report TR-2231, 1973.

7. J.E. Green, 'Application of Head's Entrainment Method to the prediction of Turbulent Boundary Layers and Wakes in Compressible Flow', RAE Technical Report TR72079, 1972.

8. P. Bradshaw, 'The Analogy between Streamline Curvature and Buoyancy in Turbulent Shear Flow', J. Fluid Mech, Vol 31, part 4 pp. 753-778, 1968.

9. J.E. Green, 'The prediction of turbulent boundary layer development in compressible flow.', J. Fluid Mech, Vol 31, part 4 pp. 753-778, 1968.

10. P.R. Ashill, DRA Bedford - Private Communication

11. H.B. Squire, A.D. Young, 'The calculation of the profile drag of aerofoils.', A.R.C.R. & M. 1838, 1937.

12. B.R. Williams, 'The prediction of separated flows using a viscous-inviscid interaction method.', RAE Tech Memo Aero 2010, 1984.

13. A. Pagano, 'A Viscous Coupled Euler Method.', Presentation at the AGCFM Workshop on Separated Flows, RAE Farnborough, 1985.

14. J.C. Le Balleur, 'Viscous-inviscid Calculation on High Lift Separated Compressible Flow over Airfoils and Wings', Proceedings AGARD Cp-415, 1992.

15. P.H. Cook, Mc Donald, M.C.P. Firmin, 'Aerofoil RAE2822: Pressure Distribution and Boundary Layer and Wake Measurements', AGARD AR 138, Paper A6, 1979.

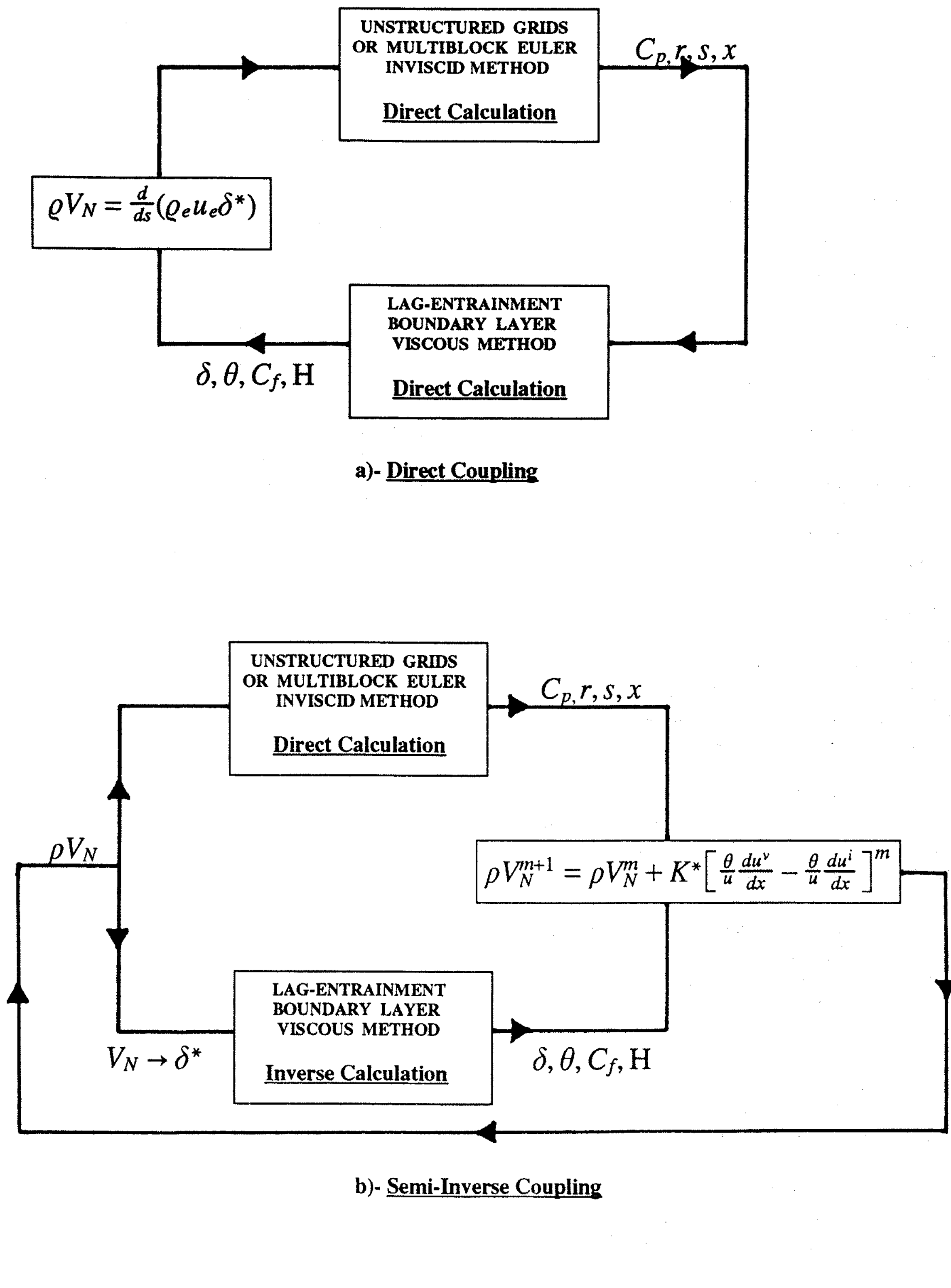


Fig. 1. Viscous-Inviscid Coupling Techniques

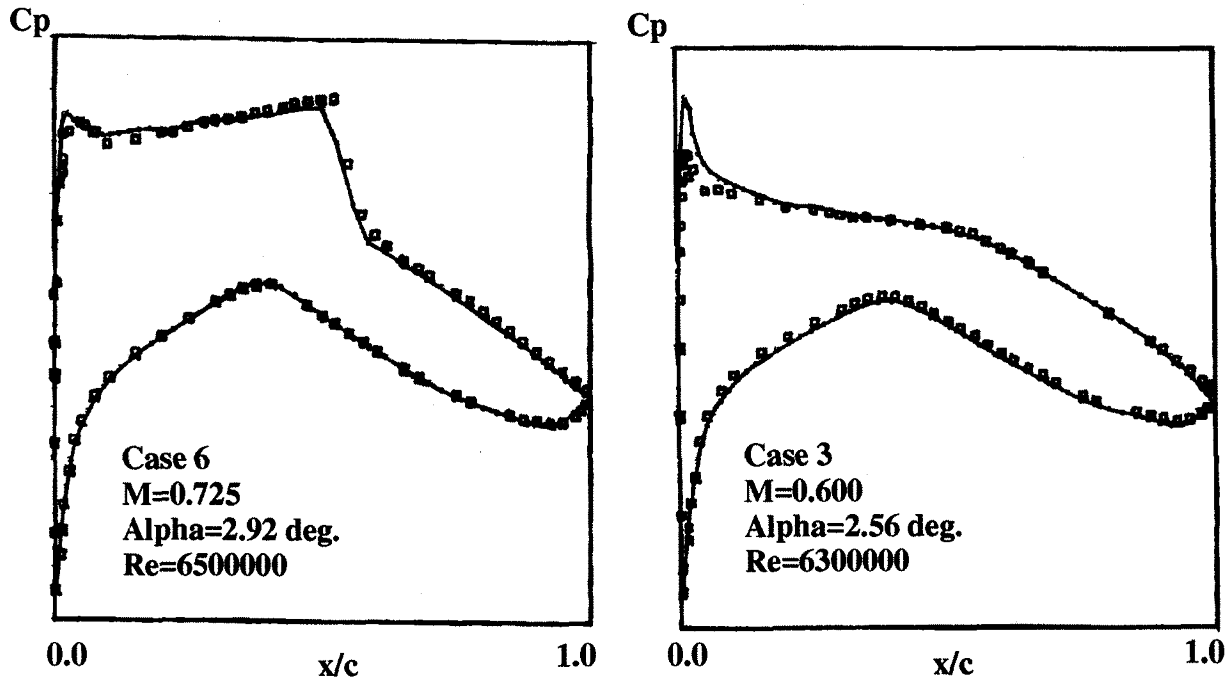


Fig. 2. RAE2822 Aerofoil : Comparison of Pressure Distributions
——— Calculation ☐ Experiment

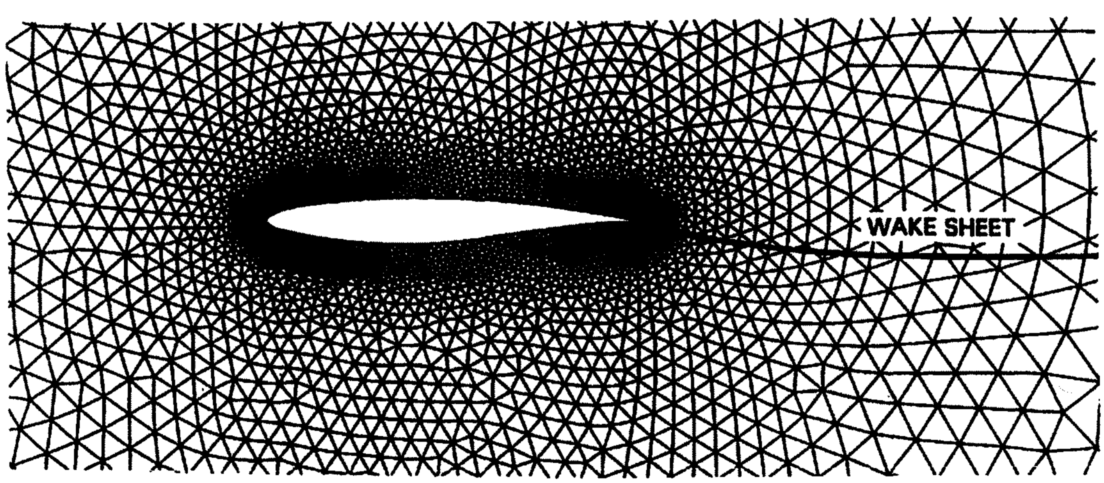


Fig. 3. Wake treatment for non-overlapping meshes

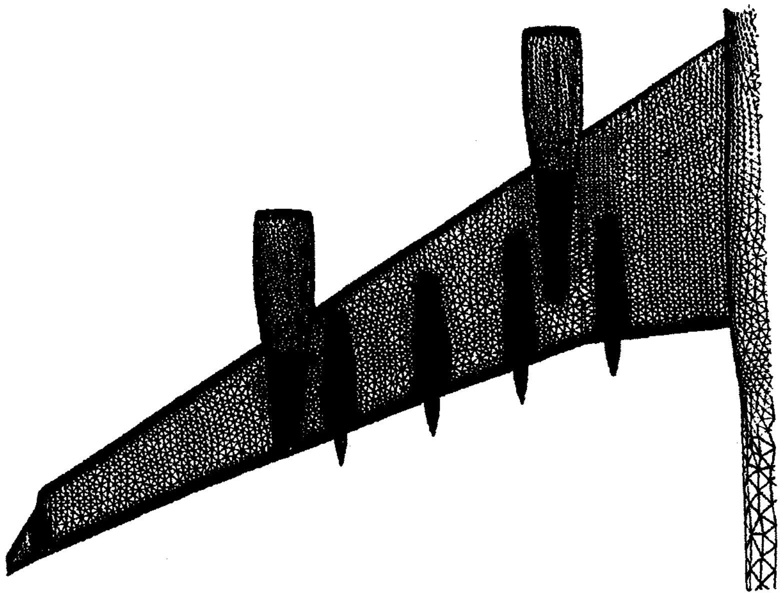


Fig. 4. Example of surface mesh suitable for Euler solutions showing typical geometrical complexities of civil aircraft wings

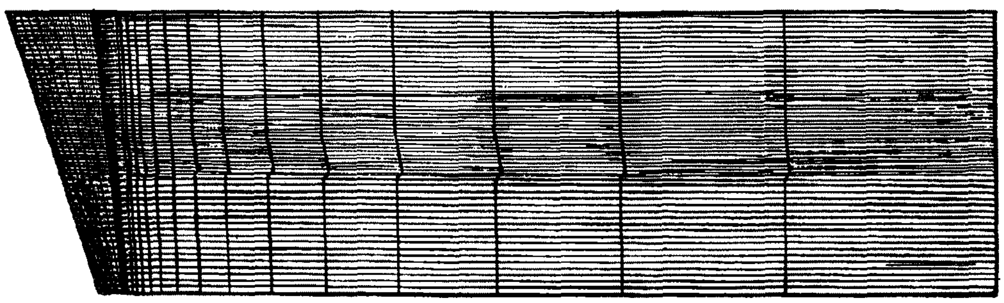


Fig. 5. Wing + wake "viscous" mesh which can be used in the presence of pylons and winglets

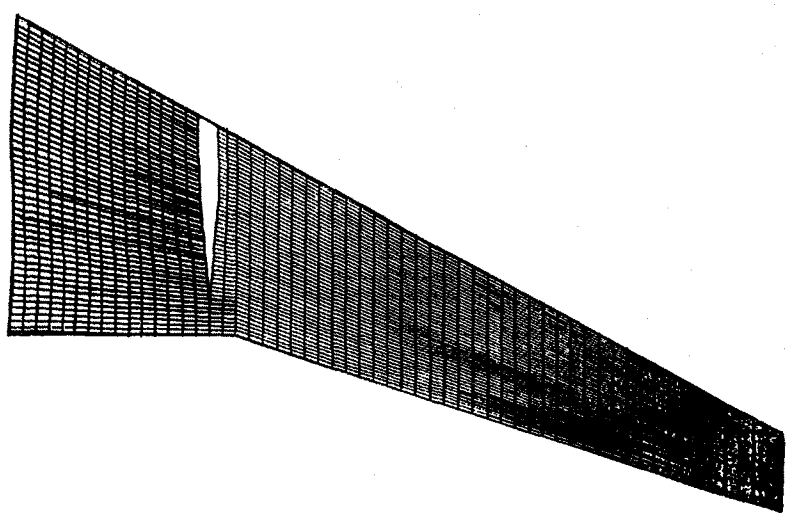


Fig. 6. Alternative "viscous" mesh taking into account the presence of a pylon

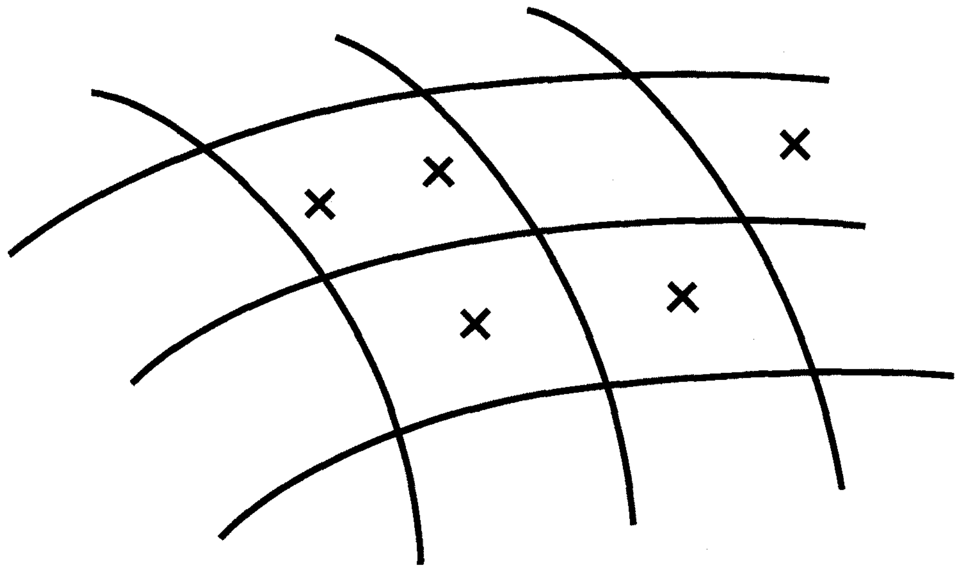


Fig. 7. Non-overlapping Multiblock and Boundary Layer meshes

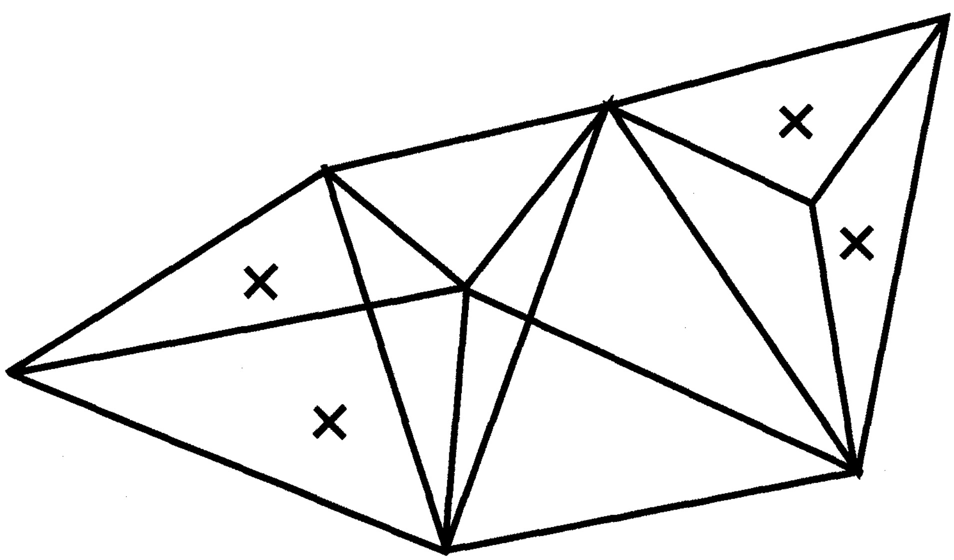
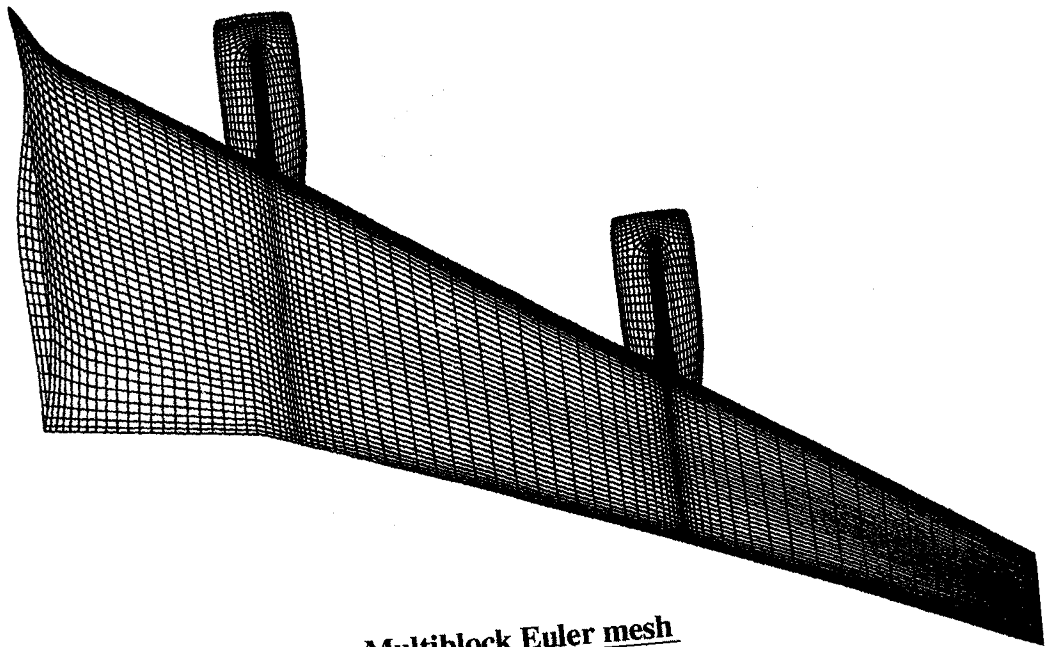
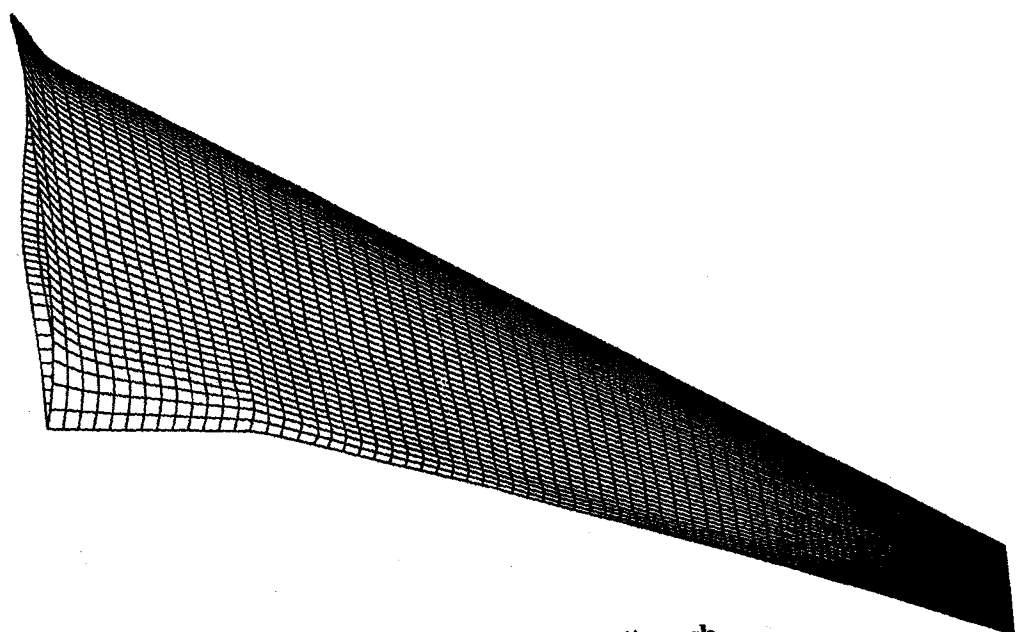


Fig. 8. Non-overlapping Unstructured and Boundary Layer meshes



Multiblock Euler mesh



Boundary Layer "viscous" mesh

Fig. 9a. Surface meshes for a typical civil aircraft configuration

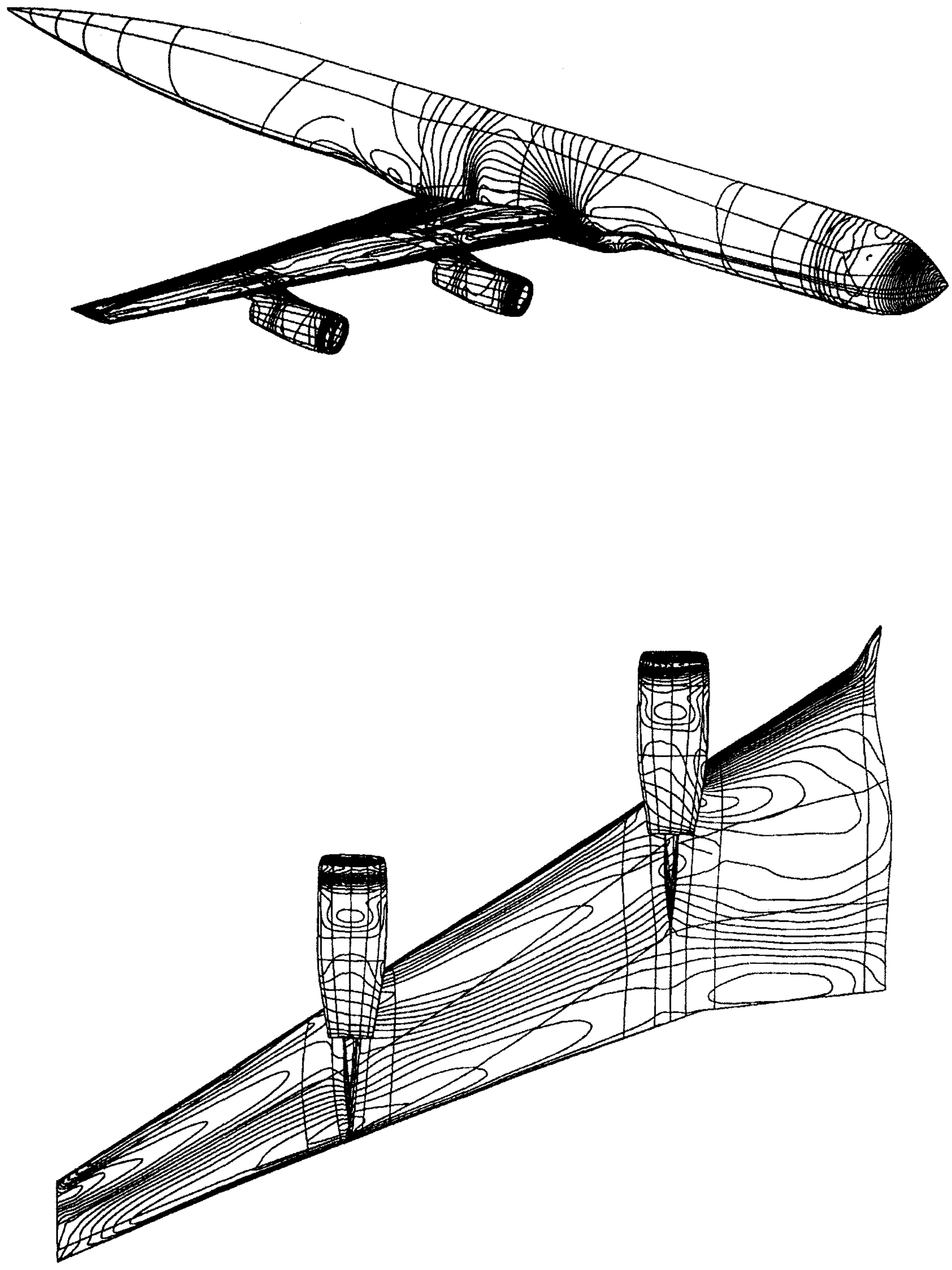


Fig. 9b. Calculated Cp surface contours using Viscous Coupled Euler Multiblock for a typical civil aircraft configuration