ADAPTIVE LOCAL GRID REFINEMENT FOR MULTIBLOCK SOLVERS

M. Amato, G. Iaccarino, L. Paparone

CIRA, Italian Aerospace Research Centre
Aerodynamics, Computational Methods Group
Via Maiorise, 81043 Capua (CE), Italy

URL "http://www.cira.it/research/CFD/mcp.html"

Abstract

The benefits on the efficiency of a Navier-Stokes flow solver by the use of an adaptive procedure to locally refine Multiblock grids are shown. The procedure is currently fully automatic in two dimensions and is based on a conservative coupling algorithm between blocks, a flow gradient biased sensor and the mapping of *ghost* topology on the computational grid. The approach is meant to fit the industrial requirements of robustness, small turnaround time, accuracy and efficiency. Two applications have been performed computing the viscous flow around airfoils in different flow conditions in order to show the flexibility and efficiency of the strategy. The intent of the work is to test the applicability of our procedure for typical 2D industrial applications, like the determination of the drag rise or buffet onset of an airfoil, that require a large set of computations and with constrains on the grid, coming from a wide range of flow features, that are quite different along the testing envolope.

Introduction

The application of Computational Fluid Dynamics (CFD) to industrial problems has been reviewed by many investigators, focussing on the role that CFD plays in the aerodynamic design cycle[1]. On the other hand, the research in CFD is addressed to the increase of efficiency of the available algorithms and to the improvement in the physical understanding and, therefore, in the numerical modelling of complex turbulent process[2].

The use of *state-of-art* CFD methods (i.e. Reynolds Averaged Navier-Stokes solvers with advanced differential turbulence models) is always limited due to the involved efforts required for generating suitable grids and the high computational time to obtain the flow solution with sufficient accuracy. Therefore numerical flow simulations are often performed by using computational grids which have to be balanced between fineness for solution accuracy and coarseness for CPU time reduction. In order to overcome these heavy limitations, different strategies have been adopted and coupled, generally, with the multiblock technique.

In this paper we present an adaptive procedure to locally refine the grid (into selected blocks) by using the flow features as driving force. The enforcement of the conservation property and of the nominal accuracy of the numerical scheme between block interfaces are an important characteristic of the present approach. In this way the flow solutions are independent from the internal boundaries generated inside the computational domain. Moreover, the link between flow features (in particular, some selected gradients) and grid refinement yields an effective procedure to accurately capture grid independent solutions in a reasonable CPU time.

In the next section we introduce the numerical method and the adaptive strategy; then, we briefly describe the automatic procedure to perfom aerodynamic analysis of airfoil sections and, finally, we present two sample applications.

The numerical method

The numerical method is based on the multiblock approach. The main features are: the flexibility of grids (discontinuous or locally refined across block interfaces[3]), the multizone approach[4], and the efficiency of the time-marching procedure to steady state. Recently new boundary conditions and an artificial dissipation scheme for the simulation of hypersonic viscous flows have been introduced and the application of the solver to a delta wing with a winglet is presented in Ref. 5.

The proposed adaptive procedure to enrich the grid into selected blocks automatically detects some flow properties (i.e. steep gradients) and generates a locally refined grid. It is divided into three steps:

- the mapping onto the user defined grid on *ad hoc* topology (the so-called *ghost* topology);
- the flow solutions on the coarsest (continuous across interfaces) available grid level;
- the refinement of some blocks and the computation of a new solution.

The last step is repeated until the desired accuracy (or a maximum number of grid cells) is reached.

The starting point is a general user-defined grid, made up of a number of blocks depending on geometry constrains (a simple one-block C-type grid can be generated for airfoil applications). The *ghost* topology is, then, mapped over the existing grid; a collection of standard topologies has been built up for some aeronautical configurations (for the applications presented here a simple 20-blocks decomposition, figure 1, has been used). An automatic mapping procedure to link the *ghost* topology and the user-defined grid has been developed.

Once the multiblock grid and the coarsest solution are available the refinement takes place. The sensitivity of the flow gradient to the flow features[6] is used as adaptivity criterion. It is worth to note that the starting (coarse) grid must posses a minimum level of definition such that the main flow features are represented.

Considering the gradient (∇) of a generic flow variable (f), block B is refined if:

$$max(\nabla f)_B \geq \epsilon \cdot \frac{1}{NB} \cdot \sum_{B=1}^{NB} (max(\nabla f)_B)$$

where ϵ is a tolerance input parameter, and NB is the total number of blocks. The gradient is evaluated by using differences between cell values, that is in the index space.

It is clear that in blocks where the flow is essentially smooth the sensor has a low value that corresponds to a sufficient grid resolution, and therefore the block is not refined. Viceversa, where the gradients are steep the sensor has a high value and the block will be refined.

For inviscid calculations, in absence of contact discontinuity, the pressure has been used as driving force to refine the grid. For viscous flows the pressure is not suited because it does not detect boundary layers and wakes. An analysis of the influence of different sensors has been carried out previously[7]; therefore, we use a weighted average of the total pressure and velocity as flow gradient.

The evaluation of the level of accuracy reached during the adaptation procedure is an important topic to assess the effectiveness of the method itself. Standard measures of grid-indipendence are the jumps of the aerodynamics coefficients from a grid-level to the refined one. This procedure, used for example in the work of Dannenhoffer[6], requires the definition of a maximum tolerance level for each coefficient (lift and drag for example). The analysis of the computed flow field by means of the refining sensor allows to employ an alternative stategy to halt the solution procedure. By enriching the grid, the flow gradient is captured with an increasing accuracy (number of grid cells) and, therefore, the sensor decreases its overall value except across shocks. After a certain number of refining steps the sensor is approximatively constant across the blocks or in each block with respect to the previous step and then the convergence has been reached.

The automatic procedure for airfoil flows

In figure 2 a sketch of the general automatic procedure to compute the viscous flow around an airfoil is reported. Given the geometry of the profile, a one-block C-type coarse grid ($\approx$ 100.000 grid cells) is easily builted in an automatic way by using a biharmonic grid generator. By this way the clustering of the grid points near the solid wall can be obtained in order to correctly capture the steep gradients in the boundary layer and the traling edge region. At this time, the *ghost* topology is needed to create the multiblock domain decomposition.

When an initial solution (over this coarse grid) is obtained the adaptation procedure takes place. The closed loop in figure 2 includes: the evaluation of the flow gradients and the selection of the blocks to be refined; the generation of the refined grid into the selected blocks; the computation of the new solution (over the actual grid); the evaluation of the convergence of this procedure towards the desired accuracy.

This procedure has been used to compute viscous flows around airfoils with the aim of evaluating some aerodynamics characteristic, such as the lift-drag curve, or the drag divergence for transonic regimes. In these case, a large number of computations are needed by changing the flow conditions (Mach number and the incidence).

A large amount of CPU can be generally saved by starting a computation from one with close free stream conditions. For high incidence flows or supercritical Mach numbers this approach is not always succesful. By employing an automatic refinement procedure the need of a good starting solution on the fine grid is not present anymore.

Applications

In the following two applications of the described methods are presented (two available fine grids have been used in order to test the accuracy of the solution with respect to a reference one):

- RAE 2822 Airfoil: transonic flow conditions ($M_\infty = 0.73$, $Re = 6.5^6$, $\alpha = 2.78^o$), one-block grid courtesy of DRA.
- FX 63-137 Airfoil: subsonic flow conditions ($M_\infty = 0.15$, $Re = 1.0^6$, $\alpha = 12^o$), one-block grid courtesy of DASA.

The first test case has been selected because it presents the typical flow features of airfoils flying at transonic conditions: shock wave/boundary-layer interaction, separation bubble near the traling edge. The adapted grid is reported in figure 3 while the Mach contours in figure 4. With respect to a fine continuos grid an overall save of $\approx 30\%$ grid cells has been achieved. In terms of CPU time the saving is $\approx 35\%$.

The comparison of the adapted pressure distribution on the body with the reference solution over the globally fine grid (figure 5) shows the goodness of this approach even in presence of strong discontinuity across the internal boundaries. In a previous work, [7] a particular attention was paid to the evaluation of the skin friction coefficient over locally refined grids. In figure 6, the skin friction is reported for both the adapted and reference solutions: the agreement is very satisfactory.

In terms of global aerodynamics coefficient the results are very promising; the loss in accuracy is $\approx 1\%$ in term of lift and is ≈ 10 counts for the drag.

In figures 7-10 some velocity profiles are reported with the aim of showing that the adapted solution describes the same physics of the reference solution but saving CPU time.

The second test case is related to the analysis of the *maximum lift* of an airfoil. This is one of the most interesting problems in which the answers from a Navier-Stokes solver could be very useful. Even in this case, where a strong separation bubble occurs, the refining strategy works quite well.

In figure 11 the adapted grid is reported while in figure 12 the Mach contours are shown. It is worth to note the differences between the two adapted grids represented in figure 3 and 11. In the transonic case a very fine grid is used on the upper surface of the airfoil (from leading edge to 90% of the chord) and in the wake. In the subsonic case the refinement is very strong in the leading edge and in the wake including the region (on the lower surface) of high curvature of the airfoil.

The distribution of the pressure and skin friction on the wall is reported in figure 13 and 14 together with the reference solution. It is clearly evident the agreement between the adapted solution and the refence one in terms of Cp distribution. The analysis of the skin friction distribution outlines that the extension of the recirculating zone is closely predicted and the friction drag (the area below the curves plotted in figure 14) is quite similar even if locally there are some differences.

The lift coeffficient is predicted within 0.5%, while the total drag is overexstimated by 10%. The friction drag is computed with 10 counts difference compared to the reference value.

In this test case the accuracy of the viscous dominated variables is strongly affected by errors due to the simple algebraic turbulence model that has been used (Baldwin-Lomax model). Moreover, this model is very sensitive to the grid resolution in reverse flow regions, see velocity profiles in figures 15, 16 and 17; therefore better results should be obtained by using a different model.

Conclusions

An adaptive local grid refinement procedure has been presented and applied to compute viscous flows around airfoils.

The aim of the work is to test the proposed procedure with respect to industrial requirements of robustness, small turnaround time, and efficiency when applied to compute viscous flows around airfoils along their complete working envelope.

The approach is based on a conservative coupling algorithm between blocks, a flow gradient biased sensor, and the mapping of a *ghost* topology on the computational grid. The capability of retaining the nominal accuracy of a fine reference grid using adaptive local grid refinement has been shown.

References

1) S.Rill, K.Becker, 'Computational Fluid Dynamics in Airplane Design - An Industrial Point of View',**Numerical Methods for Fluid Dynamics V**, pp. 73-93, Clarendon Press, Oxford 1995.

2) J.L. Thomas, 'A Perspective of Computational Fluid Dynamics Research at NASA', **Numerical Methods for Fluid Dynamics V**, pp. 19-35, Clarendon Press, Oxford 1995.

3) B.Kassies, R.Tognaccini, 'Boundary Conditions for Euler Equations at Internal Block Faces of

Multiblock Domains Using Local Grid Refinement', AIAA Paper 90-1590.

4) M.Amato, L.Paparone, 'On the Application of Multizone Modelling and LGR Technique for High Lift Airfoil Analysis', Proceedings of the Second European Computational Fluid Dynamics Conference (ECCOMAS) 94, 5-8 September 1994, Stuttgart, Germany.

5) M.Amato,G.Iaccarino,M.Marini, '1st US-Europe High Speed Flow Field Database Workshop - ELAC Delta Wing Problem',6-9 November 1995, Houston, Texas (USA).

6) J.Dannenhoffer, 'Grid Adaptation for Complex 2D Transonic Flows', Sc.D. Thesis, MIT, Cambridge, Massachusetts (USA), 1987.

7) M. Amato, G. Iaccarino, L. Paparone, 'Automatic Local Grid Refinement for Multiblock Solvers', Proceedings of the 5th International Conference on Numerical Grid Generation 96, 1-5 April 1996, Starkville, Mississippi (USA).

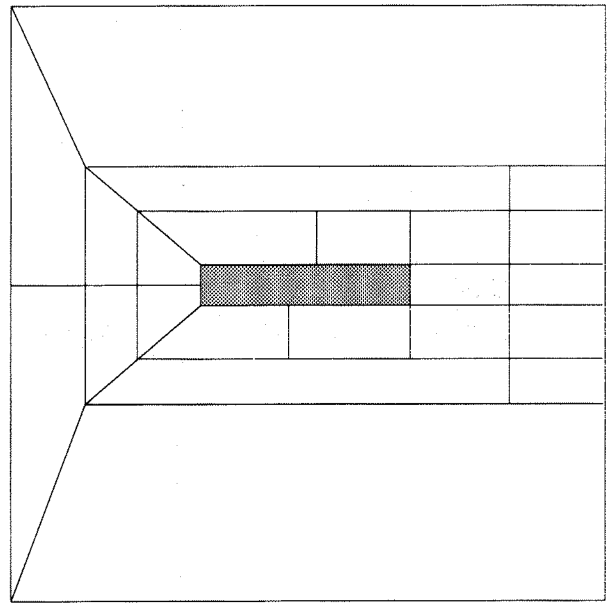

Fig. 1: *Ghost* Topology

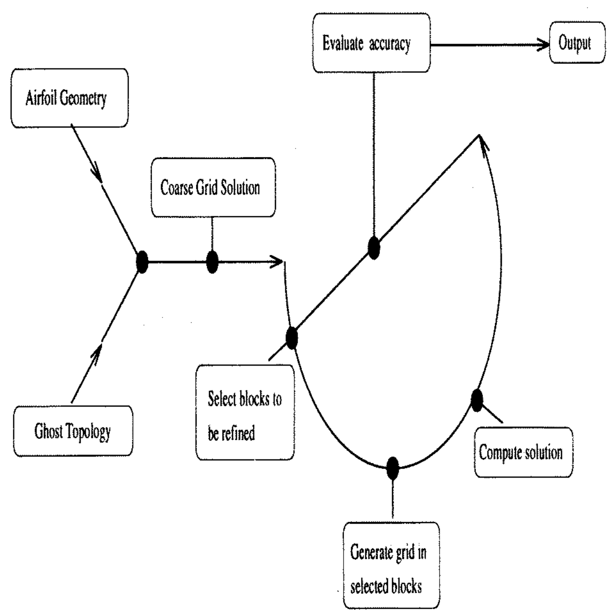

Fig. 2: Adaptation Algorithm

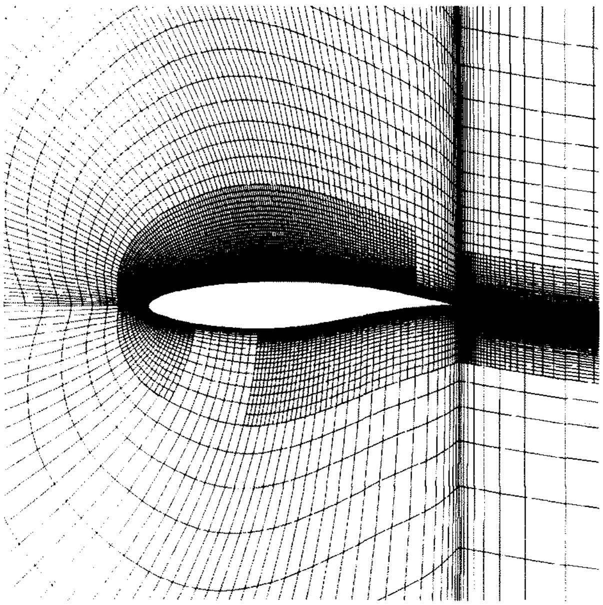

Fig. 3: RAE 2822 Adapted Grid

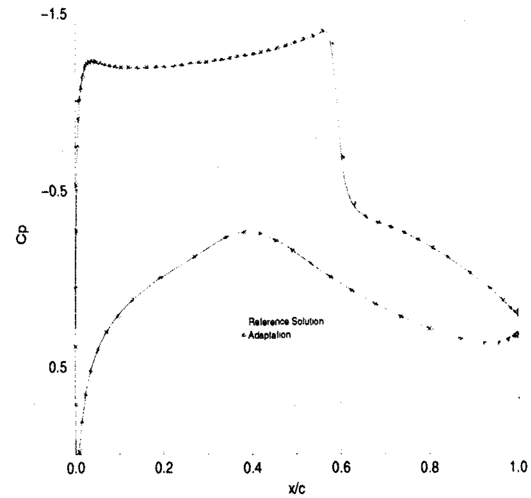

Fig. 5: RAE 2822 Cp Distribution

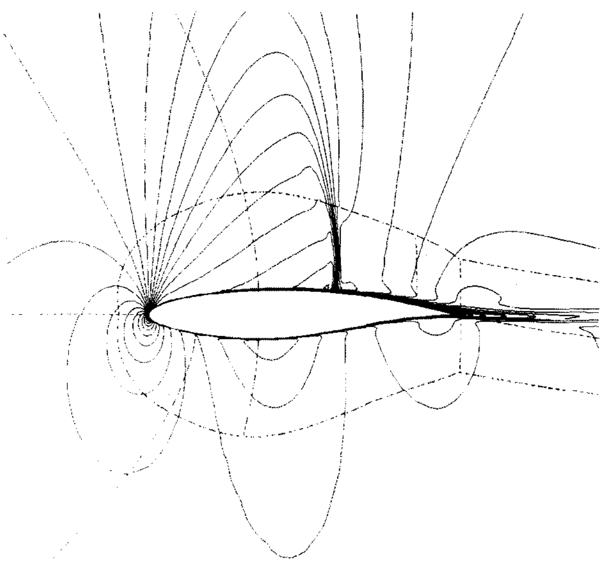

Fig. 4: RAE 2822 Iso-Mach Lines

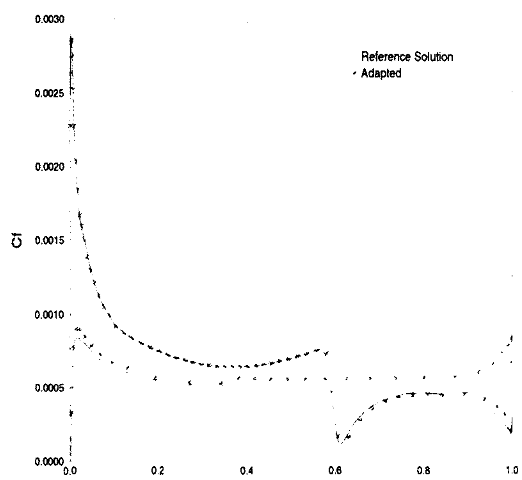

Fig. 6: RAE 2822 Cf Distribution

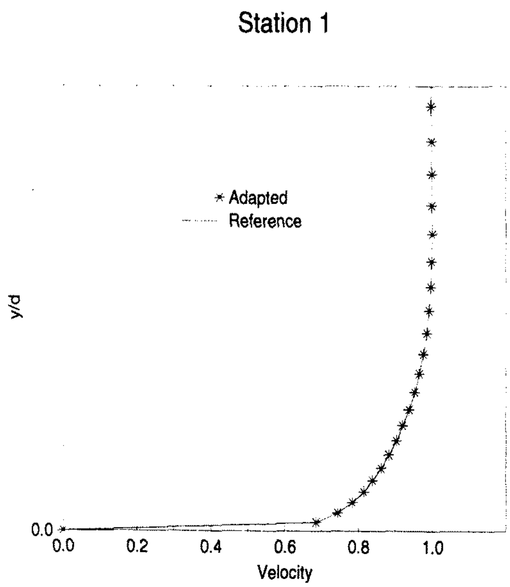

Fig. 7: RAE 2822 Velocity Profile

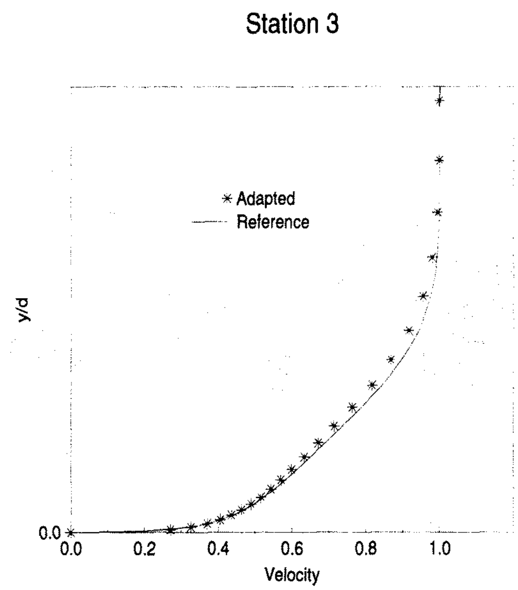

Fig. 9: RAE 2822 Velocity Profile

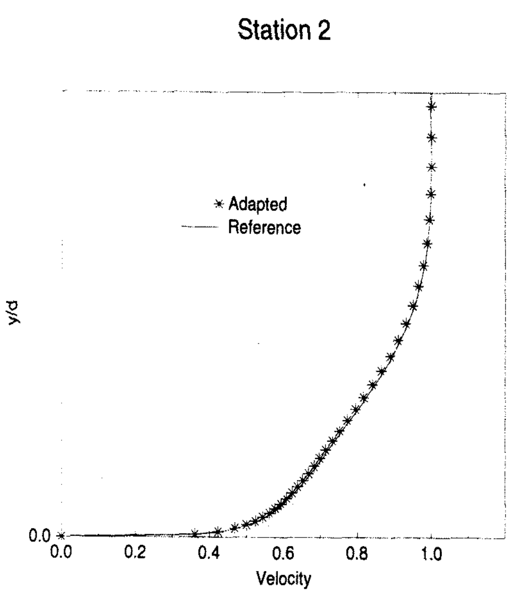

Fig. 8: RAE 2822 Velocity Profile

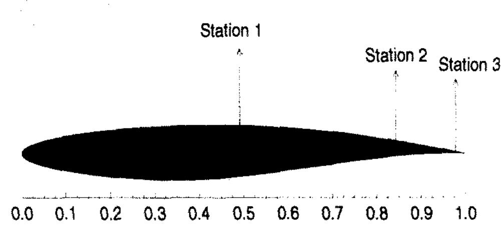

Fig. 10: RAE 2822 Airfoil

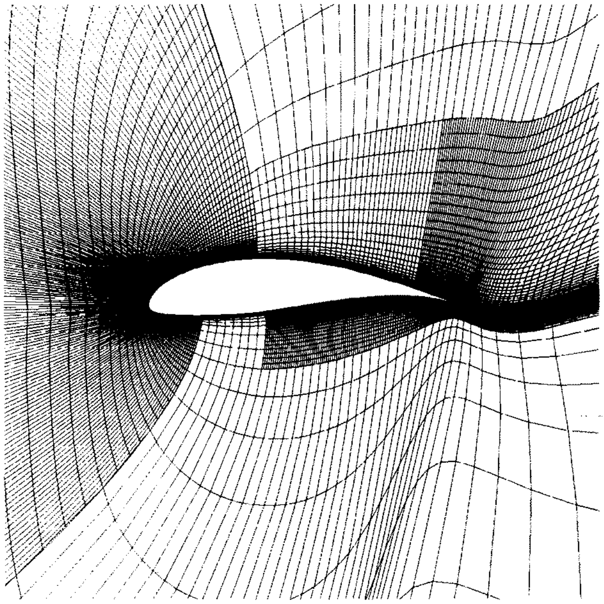

Fig. 11: FX 63-137 Adapted Grid

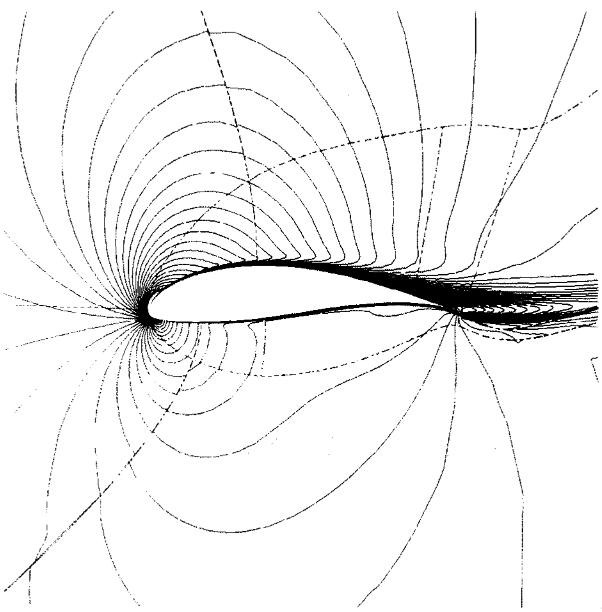

Fig. 12: FX 63-137 Iso-Mach Lines

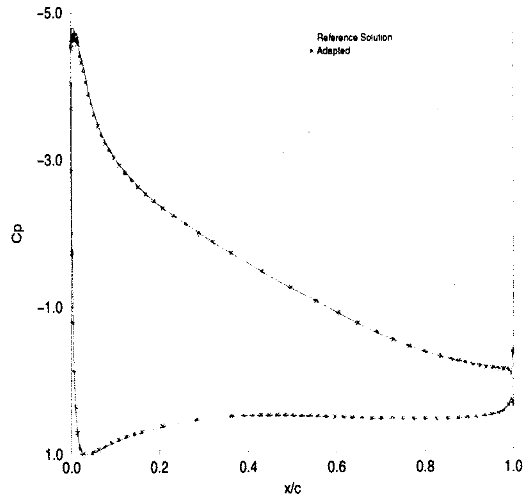

Fig. 13: FX 63-137 Cp Distribution

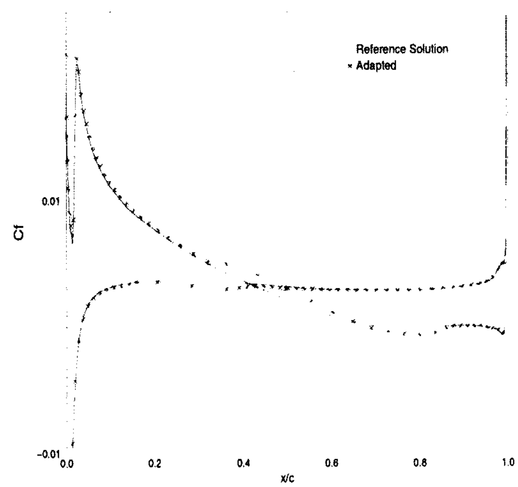

Fig. 14: FX 63-137 Cf Distribution

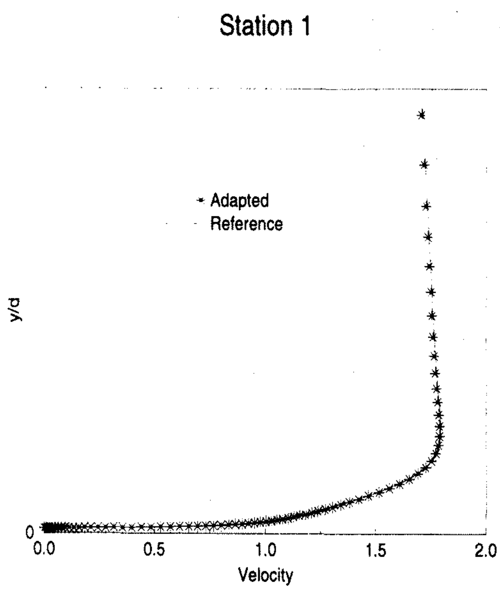

Fig. 15: FX 63-137 Velocity Profile

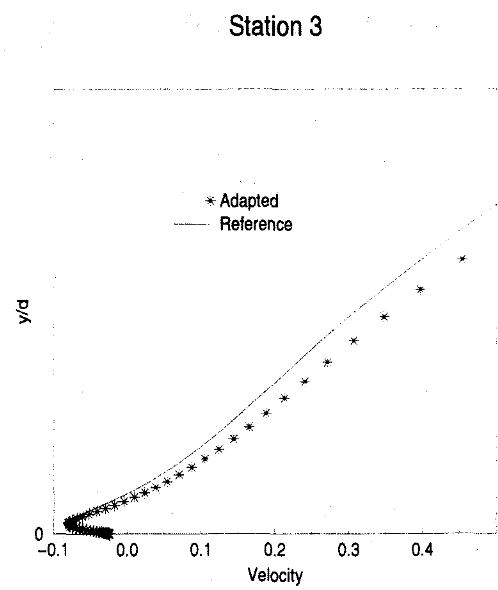

Fig. 17: FX 63-137 Velocity Profile

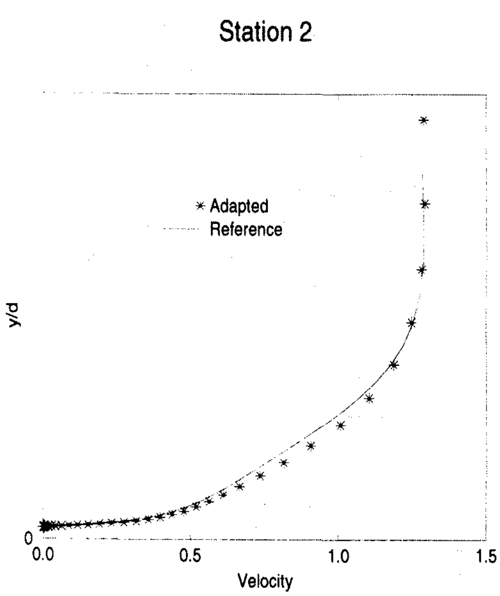

Fig. 16: FX 63-137 Velocity Profile

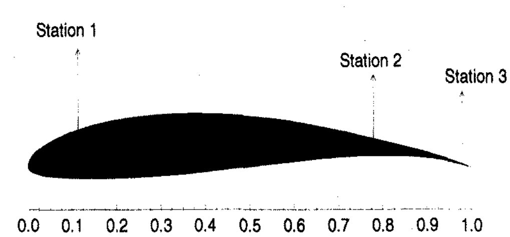

Fig. 18: FX 63-137 Airfoil