Nationaal Lucht- en Ruimtevaartlaboratorium
National Aerospace Laboratory NLR

NLR TP 97255

Computation of flow around fighter aircraft

H. Schippers and J.I. van den Berg

DOCUMENT CONTROL SHEET

	ORIGINATOR'S REF. NLR TP 97255 U		SECURITY CLASS. Unclassified

ORIGINATOR
National Aerospace Laboratory NLR, Amsterdam, The Netherlands

TITLE
Computation of flow around fighter aircraft

PRESENTED AT
the Conference on Analysis, Numerics and Applications of Differential and Integral Equations, Stuttgart, October 9-11, 1996.

AUTHORS	DATE	pp	ref
H. Schippers and J.I. van den Berg	970507	15	10

DESCRIPTORS

Aerodynamic configurations	Grid generation (mathematics)
Computational geometry	Multiblock grids
Computational fluid dynamics	Pressure distribution
Euler equations of motion	Structured grids (mathematics)
Fighter aircraft	Transonic flow
Finite volume method	Wind tunnel tests

ABSTRACT
The aim of the present paper is to demonstrate that current Computational Fluid Dynamics (CFD) technology can compute vortical flows around complete fighter aircraft. As a representative test case the flow around an F16-like aircraft is computed at transonic conditions. Pressure distributions over the wing are compared with measured data. The comparison indicates that the complex flow structure above the wing is predicted quite well.

Contents

(**15** pages in total)

Summary

The aim of the present paper is to demonstrate that current Computational Fluid Dynamics (CFD) technology can compute vortical flows around complete fighter aircraft. As a representative test case the flow around an F16-like aircraft is computed at transonic conditions. Pressure distributions over the wing are compared with measured data. The comparison indicates that the complex flow structure above the wing is predicted quite well.

1 Introduction

The flow around fighter aircraft is dominated by vortical wakes as generated at sharp leading edges of strakes and wings. The interaction of vortical wakes with the structure of the aircraft is of paramount interest with regard to aerodynamic matters related to the operational use of fighter aircraft (see e.g. Ref. 7) . These aerodynamic matters can be investigated by flight measurements, windtunnel experiments and Computational Fluid Dynamics (CFD) simulations. Both flight measurements and windtunnel experiments have as disadvantage the high costs due to the instrumentation and preparation of the model. Also beforehand, a limited number of flight conditions has to be specified. In the last decade, much progress has been made in the development and application of advanced CFD methods (see e.g. Refs. 4, 5, 10). Computational costs are reduced significantly and the turn-around times have decreased due to the developments in computer hardware and numerical methods. Some advantages of CFD simulations are: they yield information of the complete flow field and they can be applied to investigate aerodynamic behavior of aircraft at potential unsafe flight conditions.

Advanced CFD methods are based on the numerical solution of the Navier Stokes or the Euler equations. Vortical flows cannot adequately be simulated with mathematical models of lower level of sophistication such as linear potential theory or the nonlinear full-potential equation. During the last years an extensive validation of Euler flow solvers around delta wing configurations and generic fighter configurations (see e.g. Ref. 1, 2) has been carried out. At sharp edges the numerics of the algorithm for solving the time-dependent Euler equations automatically provokes separation and therewith the generation of vorticity at the edge, resulting in the formation of a shear layer which rolls up into a vortex. It has been shown that the convection of the vorticity is captured accurately by an Euler solver. As a consequence, these Euler solvers capture the global features of the vortical flow and predict the correct level of the overall aerodynamic forces and moments for a substantial part of the Mach-alpha plane.
The aim of the present paper is to demonstrate that current CFD technology (based on Euler solvers) can compute vortical flows around complete fighter aircraft. As a representative test case the flow around an F16-like aircraft is computed at transonic conditions. Pressure distributions over the wing are compared with measured data. The comparison indicates that the complex flow structure above the wing is predicted quite well.

2 Outline of CFD simulation system

For the computation of flow around complex aircraft geometries NLR has developed a 3D simulation system, which is based on a multizone approach. The functionalities of this system and some civil industrial applications have been described in Ref. 3. The flow domain around the aircraft is divided into multiple non-overlapping blocks. This multizone approach allows the generation of structered computational grids around aircraft as well as the application of different types of flow equations in different zones of the flow domain. In the NLR flow simulation system the thin-layer Navier-Stokes equations (completed with a turbulence model), and the Euler equations are used. In this paper only results of the Euler flow solver are presented. The main features of this system are: 1. Geometrical modelling, 2. Flow domain decomposition, 3. Grid generation, 4. Flow solver, and 5. Graphical data visualization procedures.

Geometrical modelling

The decomposition of the flow domain and the subsequent grid generation assume the availability of a CAD model. Mostly, these CAD models are also used in the product definition, development, and manufacturing process. Usually they are not designed for CFD applications. The available CAD model of the F-16 like aircraft is described by Bézier patches. It has features, which pose some problems in mesh generation such as overlapping and intersecting surfaces as well as gaps between surfaces. The geometry-modelling code ICEM-CFD is used to customize the CAD model in order to make the aircraft geometry suitable for aerodynamic computations.

Domain decomposition

During interactive sessions on graphic workstations the flow field around the aircraft is divided into blocks, without overlaps or gaps. The topology of each block must be equivalent with the topology of a cube, although some degenerations are allowed. Block faces are not necessarily connected one-to-one, but “partial block boundary interfacing” is allowed which means that any part of one block face may be connected to any compatible part of another block-face. To this end the concept of compound faces was introduced (see e.g. Ref. 8). Compound faces are advantageous because local changes to the topology can be made, without affecting the block decomposition of the complete flow domain. When these compound faces are used, the number of blocks around a complex geometry like a fighter aircraft can bc controlled.
The topology of the computational grid around the F-16 like aircraft is a HH-type topology, H-type in both chordwise and spanwise direction. The computational space around the half model of the aircraft is subdivided into 41 blocks. The outer boundary in normal direction is located $3L_{ref}$

units from the aircraft, where the unit L_{ref} equals the length of the aircraft. In spanwise direction the outer boundary is positioned $2L_{ref}$ units from the vertical plane of symmetry, while in both up- and downstream direction it is also positioned $2L_{ref}$ units from the aircraft.

Grid-Generation

The next step is the generation of a structured grid in each block of the computational domain. The way the domain decomposition is carried out strongly influences the quality of the grid in terms of the ability to achieve high grid resolution there where necessary, without introducing grid points there where the flow gradients are small. The concept of multi-blocked structured grids has the advantage that the grid can be tuned near complex parts of the aircraft and that the density of the grid-cells can be increased to provide enough grid points for the flow solution. Figure 1 depicts the grid on the aerodynamic surface of the aircraft. Elliptic techniques (see Ref. 9) have been applied in order to achieve smooth highly orthogonal grids. The total number of cells in the half space around the F-16 like aircraft is 1,161,216. In chordwise direction 148 cells run from the apex to the outlet of the configuration; 32 cells run in spanwise direction along the fuselage; in the direction normal to the configuration 32 cells are positioned. On the upper surface of the wing 36 cells run in spanwise direction and 40 in chordwise direction. Figure 2 depicts the grid in the vertical plane of symmetry.

Solving the flow equations

The NLR computational method to solve the unsteady Euler equations is based on Jameson's finite-volume method (see Ref. 6), i.e. it is a time-explicit, spatially cell-centered central-difference scheme. Fourth order artificial dissipation terms are added to the discretized equations to provide stability, i.e. to prevent decoupling of the solution at odd-even number of grid points. Second order artificial dissipation is added to prevent oscillations near shocks. To obtain a steady-state solution the time-integration is performed by a 4-stage Runge-Kutta scheme. Multigrid, enthalpy damping, implicit residual averaging and local time-stepping are used to accelerate convergence to a steady-state solution.

At internal boundaries between block-faces care is taken to enable accommodation of slope-discontinuous gridlines as well as discontinuities in cell size across the interface. It is assumed that the flow remains symmetric with respect to the vertical plane of symmetry. At the solid wall the normal component of the velocity vanishes. The boundary conditions at the outer boundary are based on Riemann invariants.

3 Results

Figures 3 and 4 present the upper surface pressure distribution on the wing for two lift coefficients at a free-stream Mach number of M_∞ = 0.90. For the low lift coefficient (Fig. 3) the isobars tend to cluster just at the leading edge. This is attributed to a shock developing at the leading edge. With increasing lift the shock sweep angle increases and the shock moves from the leading edge of the wing to a more downstream position. This can clearly be seen in the pressure distribution for the high lift coefficient (Fig. 4). For both lift coefficients a rear shock just upstream of the trailing edge develops in order to fulfil1 the Kutta condition. Inspection of figure 4 reveals that for c_L = 0.5890 a complex shock system has developed above the upper surface of the wing. The forward and rearward shock merge into a so called X-shock system.

Figures 5 and 6 present a comparison of the computed and measured surface pressure distribution on several wing sections at constant semi-span at M_∞ = 0.90 for two lift coefficients. These figures indicate that the complex flow structure above the wing is predicted quite well. Near the leading edge of the wing both the computed and measured data show a similar shock induced pressure increase. As can be observed from figure 6, for the high lift condition ($c_L = 0.5890$), the position of the forward shock corresponds equally well with the measured data. In the computational results there is a slight difference in the detailed pressure distrbution downstream of the rear shock location. This difference is attributed to the fact that viscous effects are not modelled in the Euler solver,

4 References

1. J.I. VAN DEN BERG, H.W.M. HOEIJMAKERS, H.A. SYTSMA: Numerical Investigation into High-Angle-Of-Attack Leading-Edge Vortex Flow. AIAA Paper 92-2600, (1992).
2. J.I. VAN DEN BERG, H.W.M. HOEIJMAKERS, F. BRANDSMA: Numerical Investigation into Vortical Flow about a Delta-Wing Configuration up to Incidences where Vortex Breakdown occurs in Experiment. AIAA Paper 94-0621, (1994).
3. J.W. BOERSTOEL, A. KASSIES, J.C. KOK, S.P. SPEKREIJSE: ENFLOW, a full-functionality system of CFD codes for industrial Euler/Navier Stokes flow computations. NLR TP 96286, (1996).
4. J. FLORES, S.G. REZNICK, T. HOLST, K. GUNDY: Transonic Navier-Stokes Solutions for a Fighter-Like Configuration. Journal of Aircraft 25, 875-881, October 1988.
5. S. HEISS, A. EBERLE, L. FORNASIER, W. PAUL: Application of the Euler Method EUFLEX to a Fighter-Type Airplane Configuration at Transonic Speed. AIAA Paper 92-2620, (1992).
6. A. JAMESON, W. SCHMIDT, E. TURKEL: Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Scheme. AIAA Paper S 1- 1259, (1981).
7. Y.M. RIZK, G.P. GURUSWAMY: Numerical Investigation of Tail Buffet on F-IS Aircraft. AIAA Paper 92-2673, (1992).
8. S.P. SPEKREIJSE, J.W. BOERSTOEL, P.L. VITAGLIANO: New Concepts for Multi-block Grid Generation for Flow Domains around Complex Aerodynamic Configurations. NLR TP 9 1046, (1991).
9. S.P. SPEKREIJSE, J.W. BOERSTOEL. J.L. KUYVENHOVEN. M.J. VAN DER MAREL: Surface grid Generation for Multi-Block Structured Grids. NLR TP 92267, (1992).
10. T.J. WELTERLEN, J.A. CATT: Evaluating F-16 Nozzle Drag Using Computational Fluid Dynamics. AIAA Paper 94-0022, (1994).

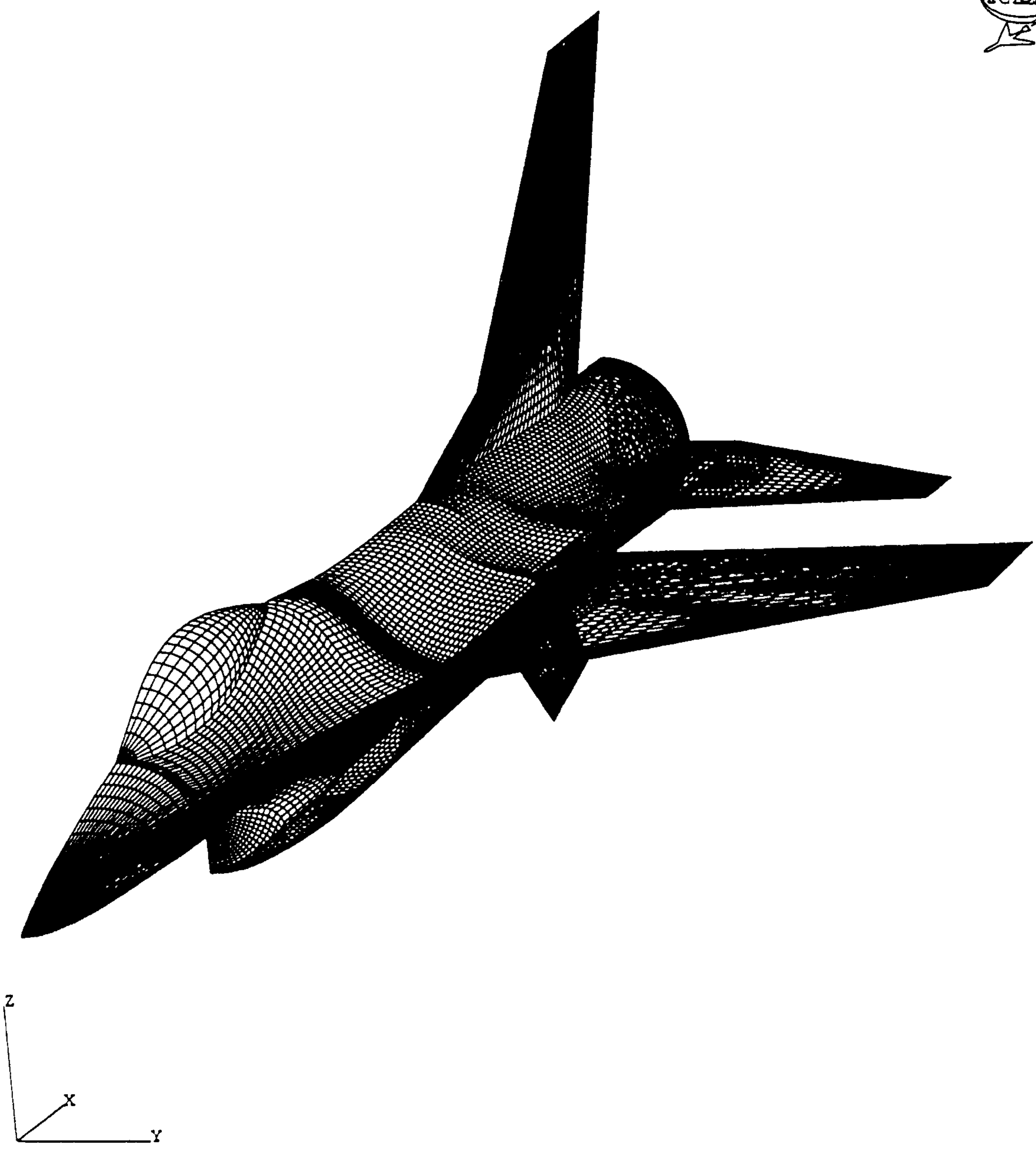

Fig. 1   Grid on aerodynamic surface

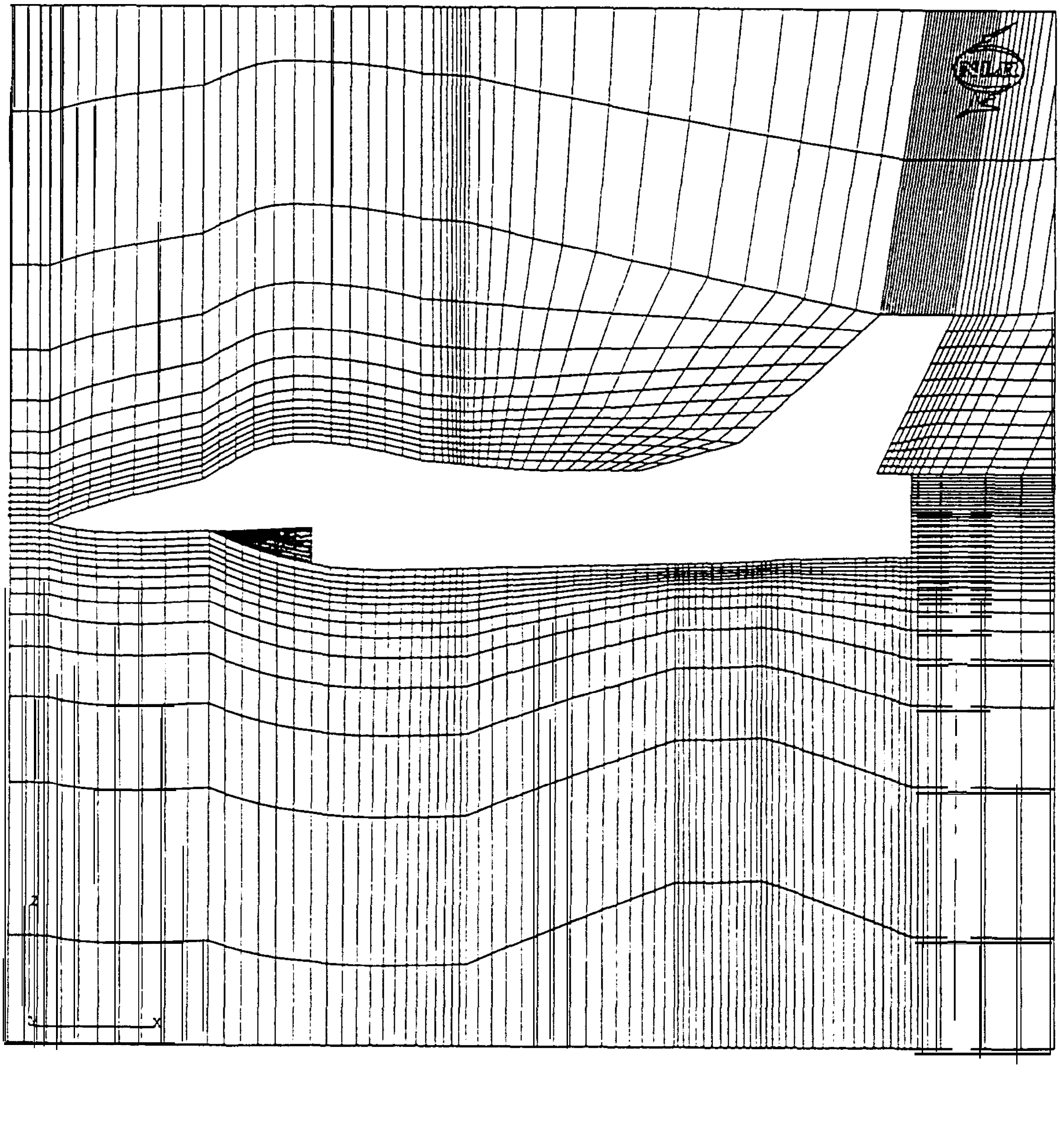

Fig. 2 Grid in plane of symmetry

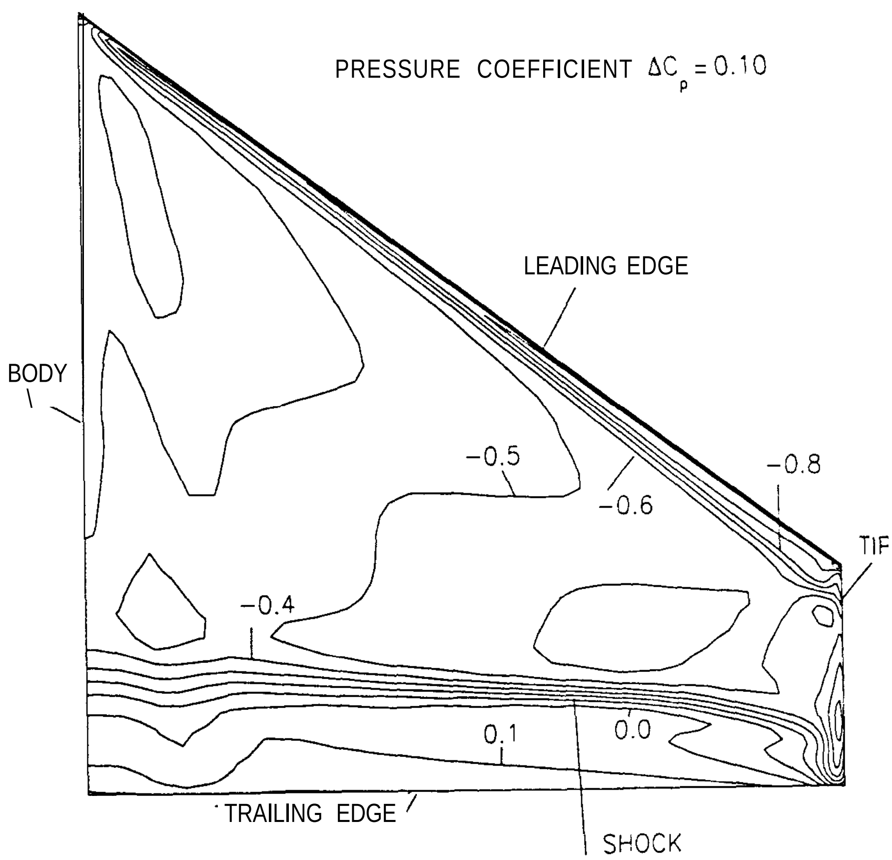

Fig. 3 *Wing pressure distribution at $M_\infty = 0.90$, lift condition $c_L = 0.3852$*

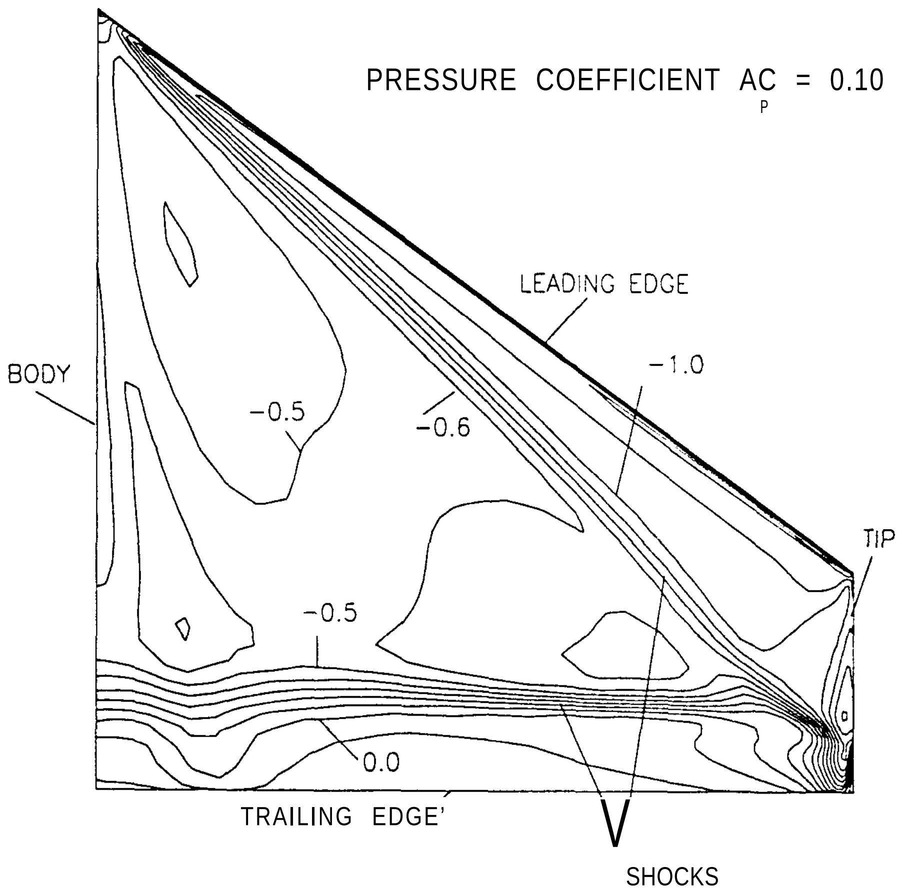

Fig. 4 Wing pressure distribution at $M_\infty = 0.90$, lift condition $c_L = 0.5890$

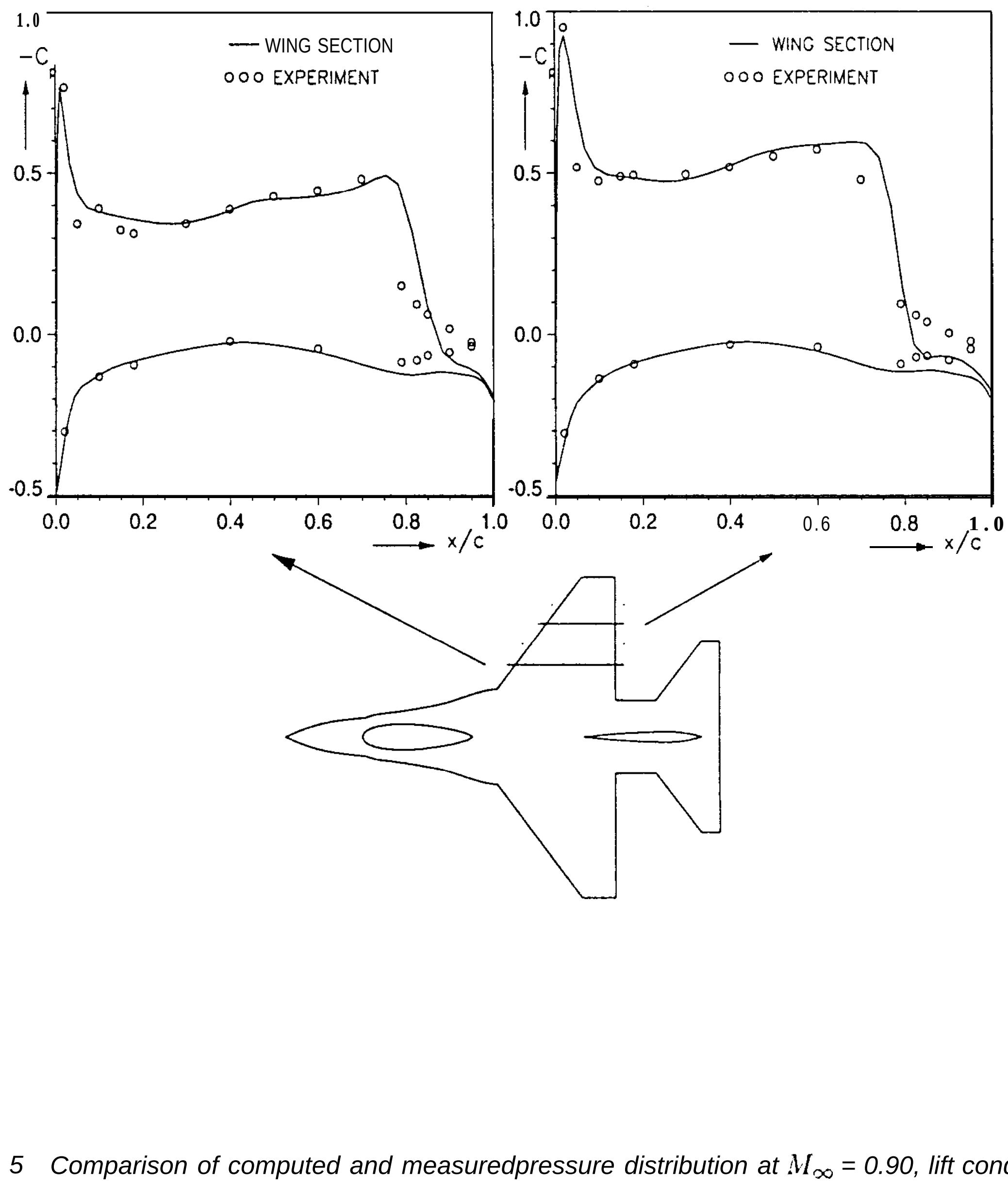

Fig. 5 Comparison of computed and measuredpressure distribution at M_∞ = 0.90, lift condition c_L = 0.3852

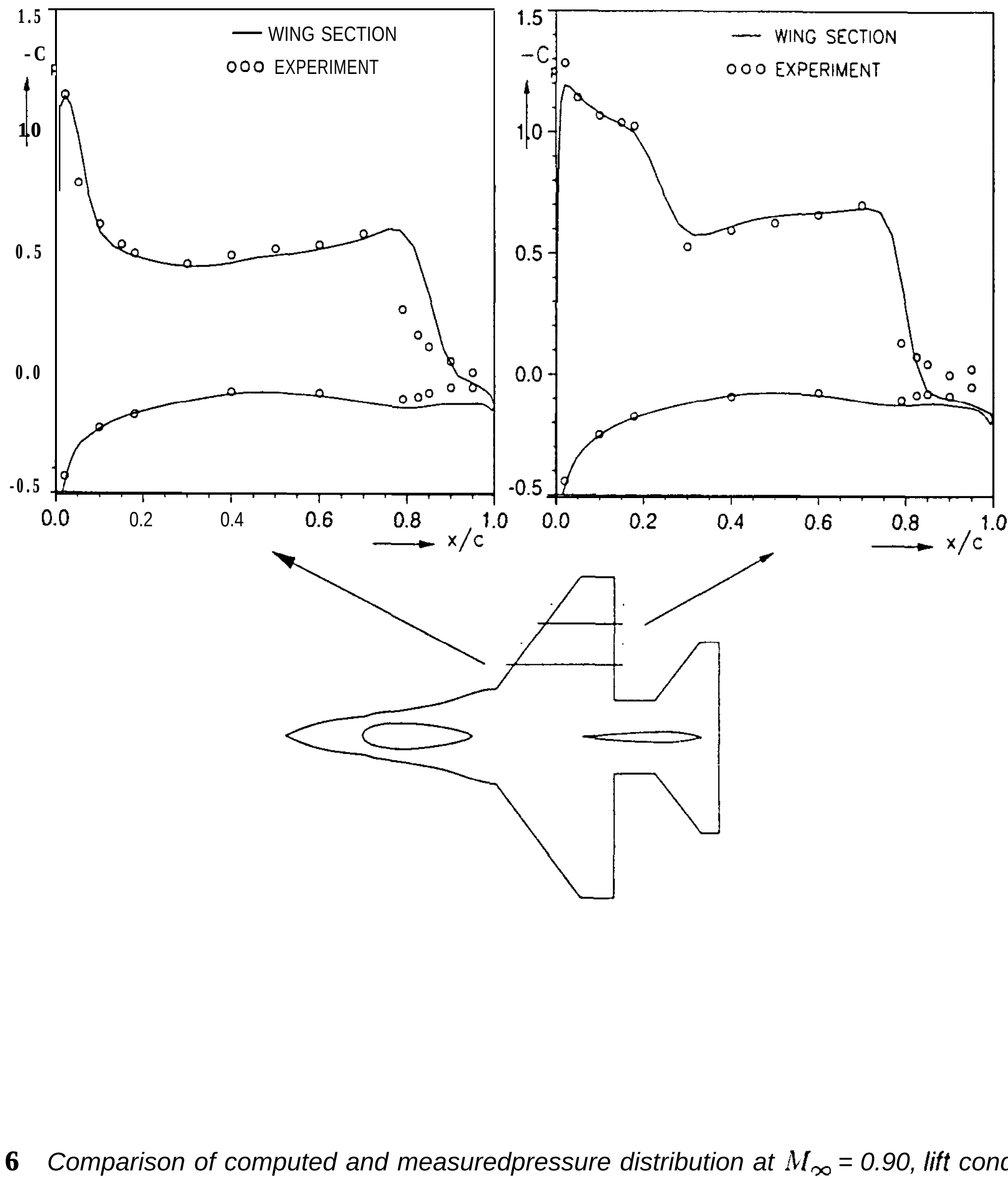

Fig. **6** *Comparison of computed and measuredpressure distribution at* M_∞ *= 0.90, lift condition* c_L *= 0.5890*