JURNAL KEJURUTERAAN AWAM (JOURNAL OF CIVIL ENGINEERING) Vol. 11 No. 2, 1998

A Compressible Turbulent Flow Solver For Complex 3D Configurations

M.T. Manzari and **K. Morgan**
Institute of Numerical Methods in Engineering
Faculty of Civil Engineering
Universiti of Wales, Swansea, SA2 8PP, UK

ABSTRACT

A numerical procedure is presented for simulating three dimensional turbulent flow problems. The mass-averaged Navier-Stokes equations are solved together with the low-Reynolds k - ω two-equation turbulence model. The standard Galerkin approach is used for spatial discretisation. Stabilisation and discontinuity capturing is achieved by the addition of an appropriate diffusion. An explicit multistage time stepping scheme is used to advance the solution in time to steady state. The study of realistic problems involving complex geometries can be achieved by using parallel computers. The results of a simulation involving transonic turbulent flow about a complete aircraft are presented.

NOMENCLATURE

C^j_{IIs}	weight coefficient for edge II_s in direction x_j
D_f	weight coefficient for face f
D_k, D_ω	dissipation of k & ω defined by Eq. (6)
D_{kj}	diffusion of k in direction x_j defined by Eq. (5)
$D_{\omega j}$	diffusion of ω in direction x_j defined by Eq. (5)
e, E	internal and total energy
F^j	inviscid flux in direction x_j
G^j	viscous flux in direction x_j
k	turbulence kinetic energy

l_I	number of boundary faces connected to node I
m_I	number of edges connected to node I
M_L	lumped mass matrix
p	pressure
P_k, P_ω	production of k & ω defined by Eq. (6)
Pr	Prandtl number
Pr_t	turbulent Prandtl number
q_i	heat flux in direction x_I
Re	Reynolds number
S	source term
S_{ij}	strain tensor defined as $(u_{i,j} + u_{j,i})/2$
t	time
T	temperature
u_i	velocity in direction x_i
U	vector of unknowns
x_i	i - th coordinate

Greek symbols

α, α^*, β, β^*, σ_k, σ_w	turbulence model closure constants
γ	specific heat ratio
δ_{ij}	Kronecker delta
μ	molecular viscosity
μ_t	turbulent viscosity
ρ	density
ω	turbulence specific dissipation
T_{ij}	stress tensor defined by Eq. (3)
T_{ij}	turbulent stress defined by Eq. (3)

Superscripts

-	prescribed normal boundary flux
n	normal flux
t	turbulent

INTRODUCTION

The simulation of inviscid compressible flow over realistic aerodynamic configurations may now be routinely accomplished using an unstructured mesh approach [1,2,3]. It has been demonstrated that accurate inviscid flow simulations require relatively small meshes and that calculations can be performed, in a reasonable time, on a wide variety of computer platforms. However, the simulation of viscous compressible flows over complete aircraft remains a challenging task. The presence of the viscous dominated regions in the flow domain, requires the use of highly stretched elements close to solid boundaries and this impacts on the performance of the solution algorithm. In this work, we employ an unstructured grid finite element approach which has already been tested for several configurations in both the laminar and turbulent flow regimes [4, 5]. Turbulent flow is simulated using the low-Reynolds k – w turbulence model [6]. The generation of an appropriate unstructured grid over a general three dimensional body, is a non-trivial task. Here, a capability is employed in which layers of highly anisotropic tetrahedral are generated near solid surfaces by an advancing normal method. The remainder of the grid is completed by the generation of isotropic tetrahedra using a Delaunay approach [7]. Accuracy considerations will require the use of very large grids when realistic configurations are analysed and a parallel implementation of the solution algorithm is used to reduce the elapsed time requirements of the approach. The result of a simulation of transonic turbulent flow over B60 aircraft is included.

GOVERNING EQUATIONS

The system of governing equations for a compressible turbulent flow comprises the mass-averaged equations of mass, momentum and energy. In the context of two-equation turbulence modelling, the system is closed by introducing transport equations for the turbulence kinetic energy *k* and the turbulence specific dissipation ω. The complete system of governing equations is written in the non-dimensional conservative form

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}^j}{\partial x_j} - \frac{\partial \mathbf{G}^j}{\partial x_j} = \mathbf{S} \tag{1}$$

where the summation convention is employed and

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho E \\ \rho k \\ \rho \omega \end{bmatrix} \quad \mathbf{F}^j = \begin{bmatrix} \rho u_j \\ \rho u_1 u_j + p\delta_{1j} \\ \rho u_2 u_j + p\delta_{2j} \\ \rho u_3 u_j + p\delta_{3j} \\ (\rho E + p) u_j \\ \rho k u_j \\ \rho \omega u_j \end{bmatrix} \quad \mathbf{G}^j = \begin{bmatrix} 0 \\ \tau_{1j} \\ \tau_{2j} \\ \tau_{3j} \\ u_i \tau_{ij} - q_j + D_{kj} \\ D_{kj} \\ D_{\omega j} \end{bmatrix} \tag{2}$$

and S = [0 0 0 0 0 P_k - D_k P_ω - $D_w]^T$ for i and j = 1, 2, 3. Here all of the variables are time-averaged mean values and the dimensionless form employed is based upon the density, velocity and molecular viscosity of the free stream and a characteristic length of the problem. In the above equations, t denotes time, x_i, the coordinate relative to a Cartesian coordinate system $Ox_1x_2x_3$, u_i the velocity in direction x_i, ρ density, p the pressure and δ_{ij} the Kronecker delta. The total energy per unit mass is defined as $E = e + u_iu_i/2 + k$, where e is the mass-averaged specific heat ratio γ. obeying the equation of state $p = \rho\,(\gamma-1)T/\gamma$, where T is the temperature. The stress tensor, T_{ij}, represents the effect of both molecular and eddy viscosities and, for a Newtonian fluid, may be written as

$$\begin{aligned}\tau_{ij} &= \frac{2\mu}{Re}\left(S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) + \tau_{ij}^t \quad \text{with} \\ \tau_{ij}^t &= \frac{2\mu_t}{Re}\left(S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) - \frac{2}{3}\rho k\delta_{ij}\end{aligned} \tag{3}$$

Here $S_{ij} = (u_{i,j} + u_{j,i})/2$ and μ_t is the turbulence viscosity defined as $\alpha^* \rho kRe/\omega$. In addition, the heat flux in direction x_j is modelled as

$$q_j = -(\frac{1}{Re})\left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_j} \tag{4}$$

and

$$\begin{aligned}D_{kj} &= \tfrac{1}{Re}\left(\mu + \tfrac{\mu_t}{\sigma_k}\right)\tfrac{\partial k}{\partial x_j} \\ D_{\omega j} &= \tfrac{1}{Re}\left(\mu + \tfrac{\mu_t}{\sigma_\omega}\right)\tfrac{\partial \omega}{\partial x_j}\end{aligned} \tag{5}$$

The production and dissipation of k and ω are modelled as

$$\begin{aligned}P_k &= \tau_{ij}^t\frac{\partial u_i}{\partial x_j} & P_\omega &= \frac{\alpha\omega}{k}P_k \\ D_k &= \beta^*\rho\omega k & D_\omega &= \beta\rho\omega^2\end{aligned} \tag{6}$$

In the above equations, $\beta = 3/40$, $\beta^* = 0.09$, $\alpha^* = 5/9$, $\alpha^* = 1$, $\sigma_k = \sigma_\omega = 2$ are closure constants.

Initial / Boundary Conditions

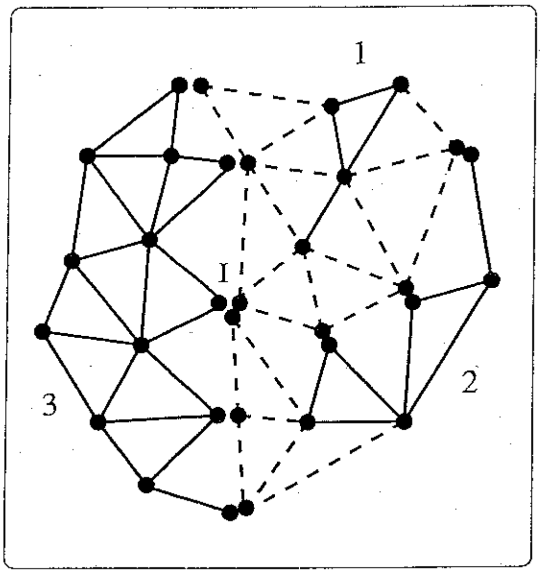
Suppose that the solution is required in a spatial domain, Ω , which is bounded by a closed surface, T, with unit outward normal vector $\boldsymbol{n} = (n_1, n_2, n_3)$. The correct specification of a problem governed by equation (1) will then require the definition of an initial condition and appropriate boundary conditions. For the initial condition, it will be assumed that free stream values are imposed everywhere in Ω at some time $t = t_0$. At a wall boundary, the no slip condition $u_i = 0$ is imposed. If the wall is assumed to be isothermal, the temperature of the wall will also be prescribed, while a zero heat flux condition is imposed if the wall is assumed to be adiabatic. The turbulence kinetic energy, k, is set to zero at a wall while ω is required to demonstrate the correct asymptotic behaviour as the wall is approached. At a far field boundary, the conditions which have to be imposed will depend upon the local nature of the flow.

SOLUTION ALGORITHM

A weak variational formulation of the problem is adopted as the starting point for the development of an approximate solution procedure. The region Ω is discretised into an unstructured assembly of tetrahedral elements and a Galerkin weighted residual method is employed [8]. A piecewise linear finite element approximation is sought, with the integrals appearing in the Galerkin statement being evaluated using an edge-based data structure [2]. The result, at node I of the mesh, is the equation

$$\begin{aligned}
&\left[\mathrm{M}_L \frac{d\mathrm{U}}{dt}\right]_I = [\mathrm{M}_L]_I \mathrm{S}_I + \\
&-\sum_{s=1}^{m_I} \frac{C^j_{II_s}}{2}\{(\mathrm{F}^j_I + \mathrm{F}^j_{I_s}) - (\mathrm{G}^j_I + \mathrm{G}^j_{I_s})\} + \\
&(\sum_{f=1}^{\ell_I} D_f\{(6\bar{\mathrm{F}}^n_I + 2\bar{\mathrm{F}}^n_{If_1} + \bar{\mathrm{F}}^n_{If_2}) - \\
&(6\bar{\mathrm{G}}^n_I + 2\bar{\mathrm{G}}^n_{If_1} + \bar{\mathrm{G}}^n_{If_2})\})_I
\end{aligned} \tag{7}$$

where the summations extend over the m_I edges, and the l_I boundary faces connected to node I. The term $\{\ldots\}$ is only non-zero if node I is on the



Sketch 1 Domain decomposition; Interior edges (solid lines) and Interface edges (dashed lines)

boundary, the overbar presents a prescribed normal boundary flux, $C^{j}_{II_s}$ and D_f are the weight functions associated with edge II_s and face f, respectively, and ML is the lumped mass matrix.

This formulation represents a central difference type of approximation and, in order to produce a practical scheme, a consistent numerical flux is substituted for the actual inviscid flux. This is accomplished by using a flux function designed to produce an unstructured grid implementation of the artificial dissipation scheme of Jameson et al. [1]. This flux function consists of a blend of stabilisation and discontinuity capturing operators, with a pressure sensor controlling the magnitude of the discontinuity capturing term. The solution is advanced by an explicit three-stage Runge-Kutta shceme.

PARALLEL SOLVER

A parallel implementation of the flow solver is achieved by using the single program multiple data concept, in conjunction with the use of standard PVM or MPI routines for message passing. No attempt has been made to optimise the coding for performance on any particular computer platform. The starting point for using a parallel solver is to decompose the mesh for the original computational domain into a number of smaller meshes of sub-domains, as illustrated in Sketch 1. For the example to be presented here, this is achieved be employing the recursive spectral bisection software of Simon [9]. Following this decomposition, the nodal points and the mesh edges are locally renumbered within each sub-domain. The governing equations are then solved on the individual sub-domains using the standard algorithm, with the sub-domain solutions being patched together to produce the solution of the original problem. In the edge based flow solver, the main data structure takes the form of a pointer form edges to points. In the main computation, consisting of loops over edges in the mesh, information from points is gathered to edges and edge information is scattered and added to points. In the parallel implementation, edges in the mesh are owned by only one domain and are not duplicated. Nodal points are owned by one domain, but are duplicated at sub-domain boundaries, creating a halo of dummy points, to enforce data locality. At the start of each time step, the interface nodes obtain contributions from the interface edges. These partially updated interface nodal contributions are then broadcast to the corresponding interior nodes in the neighbouring sub domains. A loop over the interior edges is followed by the receiving of the interface node contributions and the subsequent updating of all interior nodal values. The sending of the updated values back to the interface nodes completes a time step of the procedure. The process is implemented in such a way that it attempts to allow computation and communication to take place concurrently.

RESULTS

To demonstrate the performance of the solution algorithm, transonic turbulent flow over a model of B60 aircraft is considered. This configuration consists of wing, fuselage, pylon and nacelle. The free stream conditions are defined by a Mach of 0.8 and an angle of attack 2.738 degrees, while the Reynolds number is 10×10^6 based on the mean aerodynamic chord. The generated surface grid is shown in Figure 1 and contains 66,796 triangular elements on the body and the symmetry plane. In the viscous mesh generation process, 18 layers of the elements in the first layer being 5×10^{-5} the mean aerodynamic chord. The complete volume grid consists of 3,478,261 tetrahedral elements and 587,008 nodal points. The simulation was performed on a CRAY T3D, using 64 processors, and required 25 x 10-6 seconds per time step per nodal point. The computed distribution of the pressure contours, after 25,000 time steps, on the wing is shown in Figure 1. It is seen that a complex shock pattern is formed on the upper surface of the wing. This adds to the complexity of the flow domain by inducing shock-shock and shock-boundary layer interactions. It is therefore crucial to predict the position and strength of the shock accurately. Two main factors which control the accuracy of results are the level of numerical dissipation and the mesh spacing size. In this problem, the numerical dissipation was kept to a minimum and a reasonably fine surface mesh was used. However, the resolution of the shock can be further improved by refining the size of elements near the leading edge of the wing and/or using adaptivity methods to refine the shock locally. In both cases, a considerable increase in the number of elements is expected which results in even more expensive computations. The detail of mesh, Mach, pressure and density contours for a plane cut through the engine are shown in Figures 2 and 3. These figures show the complexity of the flow about the engine and the significance of the engine outflow. The figures also show the shock waves on the upper surface of the wing and the high pressure waves in front of the engine. In Figure 4, surface pressure coefficients for two different sections of the wing are compared with the experimental data. It is seen that the computed pressures for the lower surface are in good agreement with the experimental data. The computed pressures for the upper surface are also of the correct level and in some parts coincide with the data from experiment. The shock wave on the upper surface of the wing is also well represented and confirms the suitability of the proposed method.

CONCLUSIONS

It has been demonstrated that turbulent compressible transonic flow about a complete aircraft can be simulated by using a parallel unstructured grid approach. A finite element based algorithm has been presented for the simulation of three dimensional compressible turbulent flows on unstructured grid approach. A finite element based algorithm has been presented for the simulation of three dimensional compressible turbulent flows on unstructured tetrahedral grids. Turbulence is modelled with the two-equation $k-\omega$ model and the resulting numerical scheme is implemented in terms of an edge based data representation of the mesh. The results obtained for a model aircraft show that simulations with desirable accuracy can be conducted for industrial configurations. The running time requirements need to be reduced before the approach can be routinely employed in the design environment.

ACKNOWLEDGMENT

The authors gratefully acknowledge the support of Biritish Aerospace AIRBUS ltd. Access to the CRAY T3D at the Edinburgh Parallel Computing Centre was provided by the UK Engineering and Physical Sciences Research Council under Research Grant GR/K42264.

REFERENCES

[1] Jameson, A., Schmidt, W., and Turkel, E., Nemerical simulation of the Euler equations by the finite volume method using Runge-Kutta time stepping schemes, *AIAA paper 81-1259*, 1981.

[2] Peraire, J., Peiro, J., and Morgan, K., A 3-D finite element multigrid solver for the Euler equations, *AIAA paper 92-0449*, 1992.

[3] Peraire, J., Peiro, J., and Morgan, K., Finite element multigrid solution of Euler flows past installed aero-engines, *Comp. Mech.*, Vol. 11, pages 433-451, 1993.

[4] Manzari, M.T., Morgan, K., and Hassan, O., Compressible turbulent flow computations on unstructured grids, In Hafez, M., editor, *Computational Fluid Dynamics Review*, John Wiley and Sons, 1996.

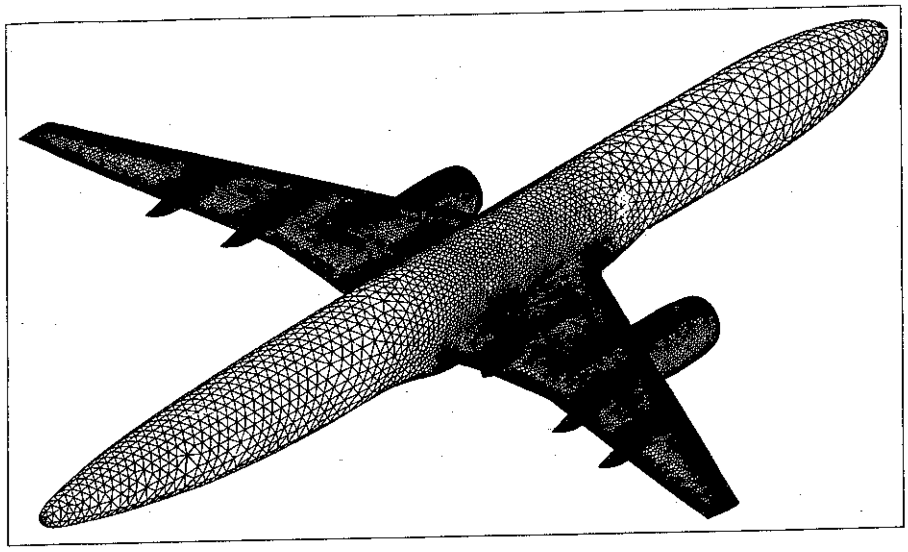
[5] Manzari, M.T., Morgan, K., and Hassan, O., Transonic flow computations using two-equation turbulence models, *International Journal of Numerical Methods in Fluids*, 1998, Submitted.

[6] Wilcox, D.C., Reassessment of the scale determining equation for advanced turbulence models, *AIAA J.*, Vol. 26, No. 11, pages 1299-1310, 1988.

[7] Hassan, O., Probert, E.J., Morgan, K., and Peraire, J., Mesh generation and adaptivity for the solution of compressible viscous high speed flows, *International Journal for Numerical Methods in Engineering*, Vol. 38, pages 1123-1148, 1995.

[8] Manzari, M.T., An Unstructured grid finite element algorithm for compressible turbulent flow computations, *PhD Thesis*, University of Wales Swansea, 1996.

[9] Simon, H.D., Partitioning of unstructured problems for parallel processing, *Computing Systems in Egineering*, Vol.2, pages 135-148, 1991.



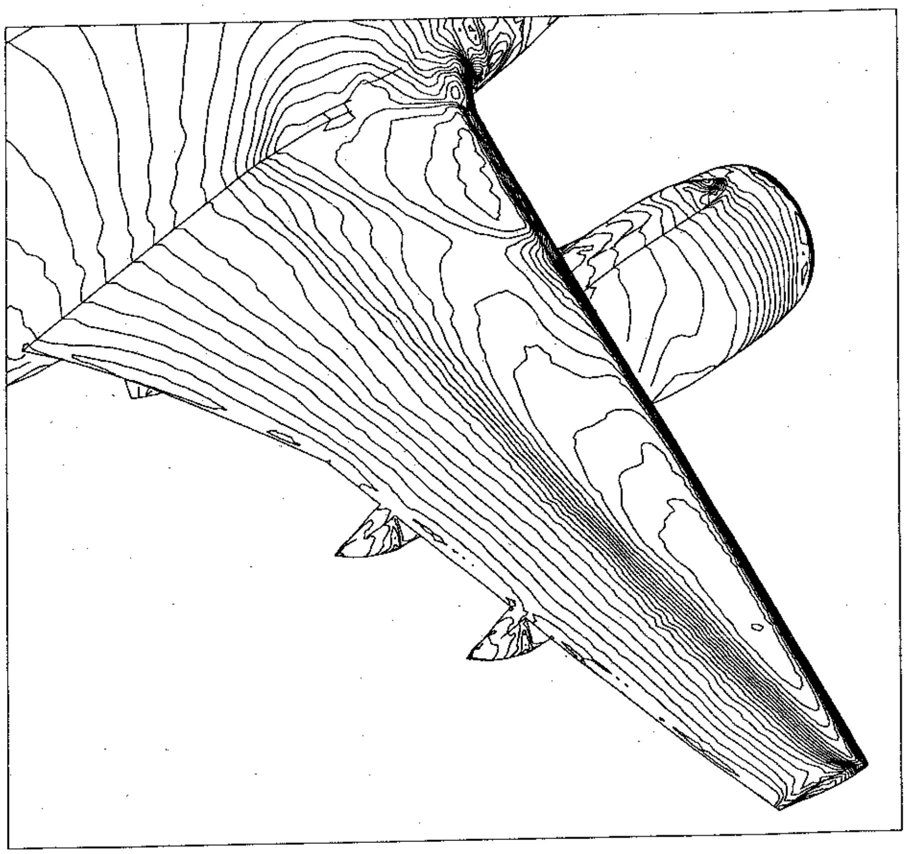


Figure 1 Surface grid (top) and pressure contours on the wing (bottom).

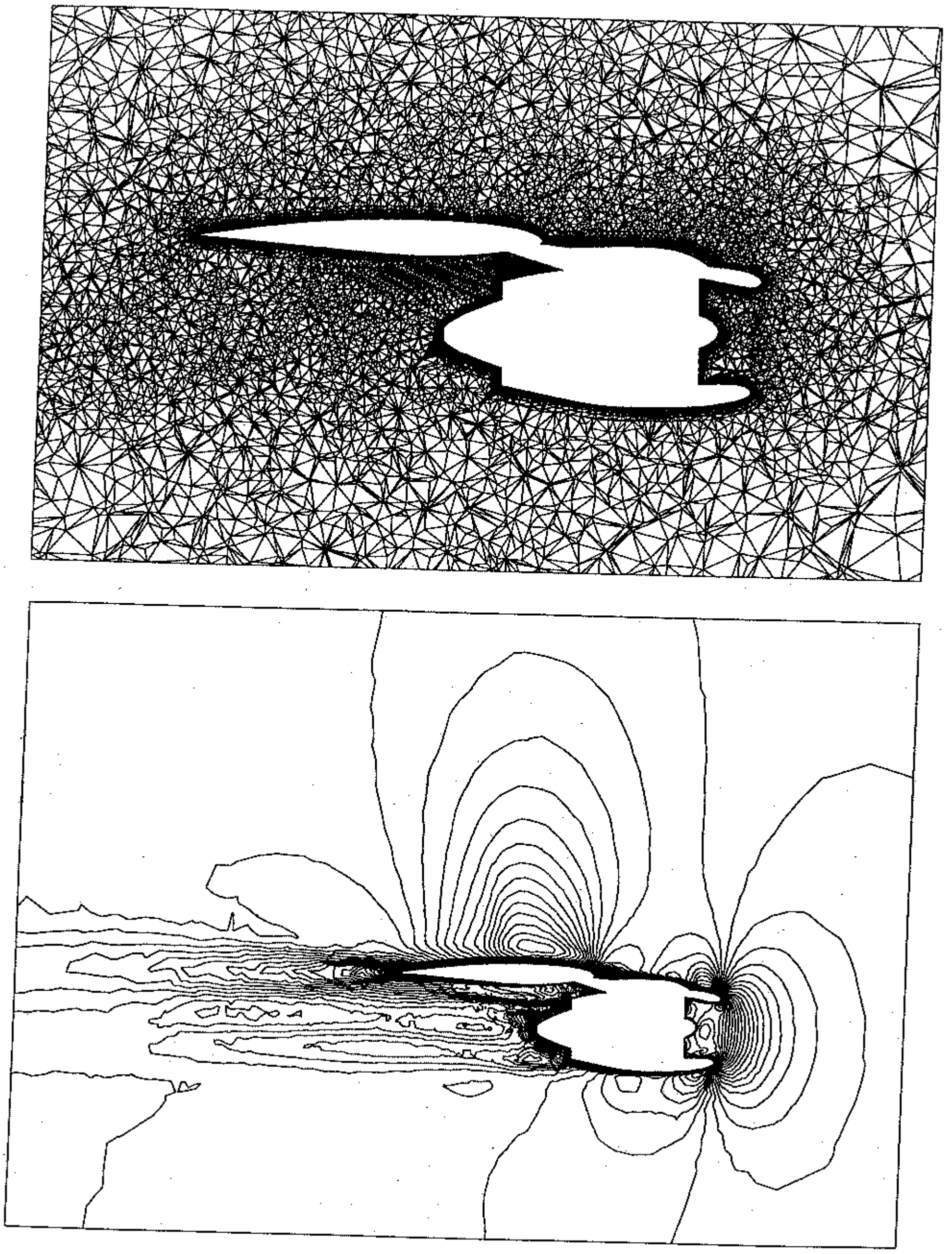


Figure 2 A plane cut through engine (top: mesh, bottom: mach contours).

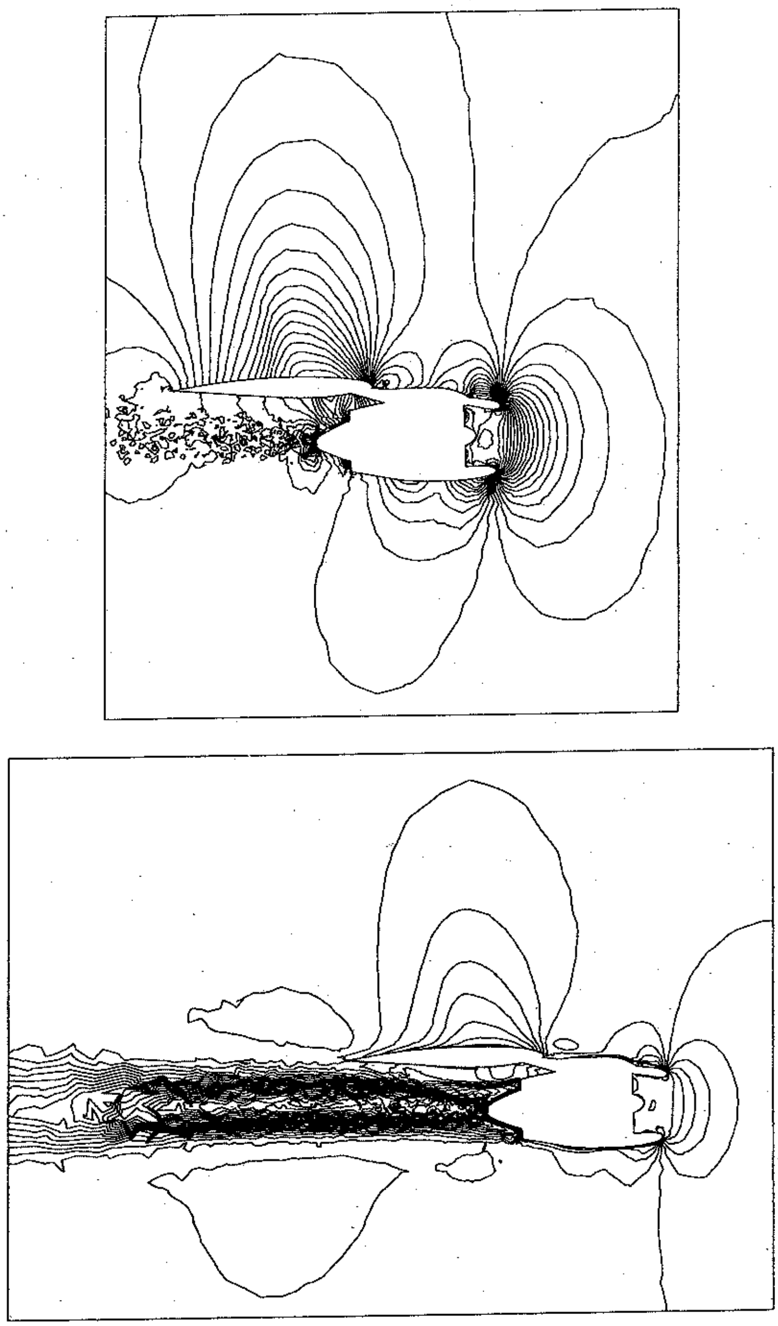
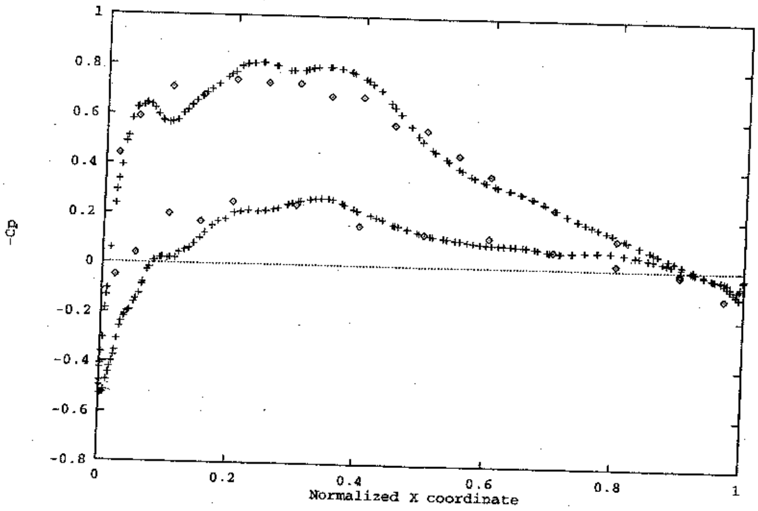
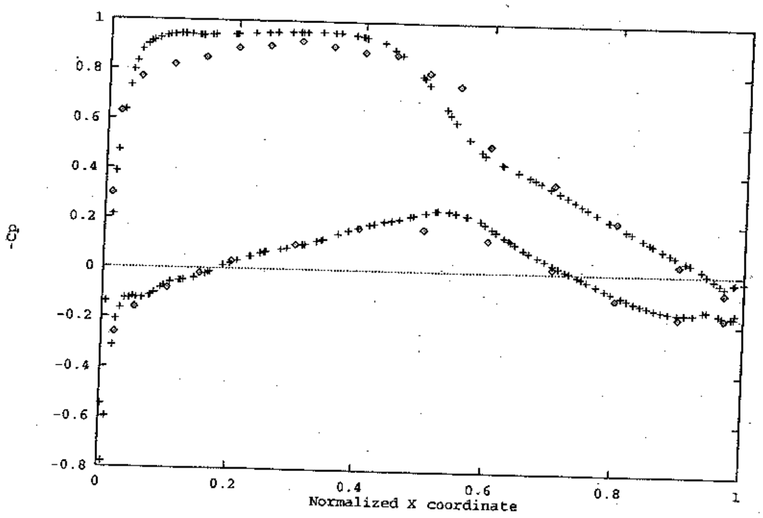


Figure 3 A plane cut through engine
(top: pressure contours, bottom: density contours).



Section $Y = 91$



Section $Y = 380$

Figure 4 Comparison of computed pressure coefficient distributions (plus) with experimental data (diamonds).