APPLICATIONS OF ADVANCED CFD CODES FOR AIRCRAFT DESIGN AND DEVELOPMENT AT NAL

S. K. Chakrabartty*, J. S. Mathur* and K. Dhanalakshmi*

Abstract

Development of Computational Fluid Dynamics (CFD) has now reached a state of maturity, which enables it to simulate the flow past practical aircraft configurations. It can thus play a very useful role in aircraft design and development. CFD codes can be used to determine the aerodynamic loads on aircraft and thus provide a necessary input for the structural design. They can also be used to study and optimize the flow patterns and thus help design aerodynamically efficient shape. Advanced level multi-block Euler and Navier-Stokes codes with multi-block grid generation package developed at National Aerospace Laboratories (NAL) are introduced here and their applications in analysis and design & development of practical aerospace configurations are described in detail.

Introduction

In the design and development of aerospace vehicles, it is essential to have an accurate estimate of aerodynamic data and an in-depth knowledge of the flow-structure close to the body surface. Complex flow-fields with tip-vortices, roll-up-vortices, boundary-layer separation, shock waves etc. are difficult to be simulated using approximate models of the governing equations. CFD as a whole is a set of computer programs consisting of geometrical data processing, grid generation, solution of the governing equations, and the post processing of the solution obtained to get the relevant aerodynamic data / flow visualization. The state-of-the-art CFD codes which are being used for practical purposes solve Euler equations for inviscid flows and Reynolds Averaged Navier-Stokes (RANS) equations with proper turbulence models for viscous flows. The aerodynamic loads at different flight conditions are needed for the structural design of the vehicle, while the total aerodynamic forces and moments determine overall performance. Earlier, this data could be obtained only by experimental methods, but now, developments in high-speed digital computers, numerical algorithms and computer graphics have made it possible to simulate the flow past realistic aerospace configurations within a reasonable amount of time. Thus, it is obvious that CFD can play a major role in the design process.

A three-dimensional RANS code, JUMBO3D [1] and the corresponding Euler code, JUEL3D [2], have been developed at NAL. These codes use a vertex-based finite volume space discretisation algorithm with five-stage Runge-Kutta time integration. They have been developed, tested and validated systematically starting from two-dimensional flow past airfoils [3, 4, 5], to simple three-dimensional flows like subsonic and transonic flow past wings, space vehicles, etc. [1, 2, 6]. Complexities with respect to three-dimensional geometry and also with respect to flow structure were gradually introduced and carefully studied. These studies included the study of grid consistency and the effect of other parameters involved in the flow algorithm. Recently a multi-block grid generation code JUMGRID [7, 8], developed at NAL is being routinely used along with the imported GRIDGEN [9] code. Programs for geometrical data processing before generating grids have to be developed on a case-to-case basis depending on the nature of the data available. For example, a complete set of (x, y, z) data in a fixed co-ordinate system is generally not available necessitating the data to be generated from a drawing sheet. The present paper summarizes some of the recent applications of these codes to the design and development activities of different aircraft configurations at National Aerospace Laboratories.

* Scientist, Computational and Theoretical Fluid Dynamics Division, National Aerospace Laboratories, Post Box No. 1779, Bangalore-560 017, India
Manuscript received on 07 Feb 2002; paper reviewed, revised, re-revised and accepted on 25 Nov 2002

Paper presented at the 4th Annual CFD Symposium of the Aeronautical Society of India held on 10-11 August 2001 at Bangalore, India
Description of Multi-Block Flow Solver and Grid Generation Codes

A number of algorithms have been developed for the computation of the flow over complex geometries by the numerical solution of Euler and RANS equations. The finite volume methods, where the solution procedure is decoupled from the generation of the grid, have proved successful for the computation of flows past complex geometries. These methods can be either of the cell-centered type or of the cell-vertex type, depending on whether the flow variables are stored at the centers or vertices of the cells. A novel vertex-based finite volume space discretisation scheme developed in [5] for the viscous terms, facilitates computation of full RANS equations, as it needs about the same numerical effort as that needed for any thin-layer type of approximation. The NAL codes use a vertex based finite volume scheme and are multi-block structured. The computations can be carried out block-wise after dividing the computational domain into smaller blocks to reduce the memory requirement for a single processor computer and also to facilitate parallel computing. The codes use an explicit five-stage Runge-Kutta time stepping scheme to advance the solution in time. Enthalpy damping, implicit residual smoothing, local time stepping and grid sequencing can be used to accelerate the convergence to steady state. The multi-block structure of the codes enables them to handle any arbitrary three-dimensional geometry.

Three-dimensional extensions of algebraic turbulence models [10] based on Baldwin and Lomax [11] are used to determine the turbulent eddy viscosity. For viscous flow, the no-slip condition is used on solid wall boundaries. The adiabatic wall condition is applied by setting the normal derivative of the temperature equal to zero. The complete details of the no-slip, cut and far-field boundary conditions, as implemented in the JUMBO3D code, are available in [1]. For inviscid flow, the flow tangency condition is applied at the solid wall, and details of the implementation of this boundary condition in the JUEL3D code are available in [2].

The JUEL3D and JUMBO3D codes are independent of the grid topology used, and the only necessary input is the grid data. The multi-block structure of the codes makes them very flexible, and allows the same code to solve a variety of flow problems with modifications only in the turbulence-modeling part. The computational domain can be subdivided into smaller sub-domains/blocks and the computation can be carried out block-wise to reduce the memory requirement and to facilitate parallel computation. The blocks are connected by overlapping one layer of grid cells, and the boundary data are transferred by using, an a-priori specification of the proper connectivity, in the input data. One layer of image cells is necessary to implement the boundary conditions at wall, cut, symmetry, far-field and degenerate faces.

Where a grid face degenerates to a line, an imaginary cell-face is considered and extrapolated values of the variables are assigned to this cell-face. The cut condition arises at inter-block boundaries, and continuity is enforced here by connecting cells of one face segment of the computational block to the corresponding face segment of the connecting block, with proper orientation of the cell numbering. The symmetric boundary conditions are applied at the symmetry plane by adding dummy grid cells, and reflecting the values of the flow variables from the interior grid points to the dummy grid points. The treatment of the far-field boundary is based on Riemann invariants for one-dimensional flow normal to the boundary. The Riemann invariants of incoming and outgoing characteristics are calculated by using free-stream values, and values extrapolated from the interior of the field, respectively. Adding and subtracting the Riemann invariants gives the boundary values for the normal velocity component and speed of sound. At an inflow boundary, the tangential velocity components and the entropy are prescribed from their free stream values, while at an outflow boundary these values are extrapolated from the interior.

The three-dimensional multi-block grid generation code, JUMGRID [7,8], has the capability to generate a structured grid in a particular block (whereas the blocks may be unstructured) and can be used for any arbitrary, geometrically complex body. Here, initial grids are generated algebraically and a three-dimensional elliptic solver is used for grid smoothing. The grid is directly compatible to the flow analysis codes JUEL3D and JUMBO3D.

Any structured grid generation technique essentially maps the given physical domain to a cube in computational space. This is not always possible or desirable for a closed complex body in a three-dimensional simply connected physical domain. The multi-block approach divides the domain into a number of blocks where a simple H-H type grid can be generated algebraically. The overall topology of the grid is determined by the distribution and relative positions of the blocks. After distributing grid points on all the faces of each block, the interior grid is generated within the block taking care to maintain conti-
nuity of grid lines across two adjacent blocks. The multi-block technique can be used to generate grids for extremely complex shapes. The procedure followed is briefly described below.

The geometry data for a complex configuration is generally available for each component separately. Sometimes this data needs to be rewritten or modified in order to redefine the complete geometry in a global system. The next step is to decide the number of blocks to be used and their relative position and orientation to cover the entire computational domain. This is decided by the geometry of the configuration and the desired topology of the grid (e.g. the block structure needed for a C-H grid is different from that needed for a C-O grid). This is a very important step where the skill and experience of the user plays a significant role. The number of points in each of the three coordinate directions is chosen carefully for each block depending on its position and the expected flow features in that region. Defining its six faces then forms each block. Then comes the redistribution of grid points on each face of every block such that the points are suitably clustered in regions where a dense grid is necessary. The interior grid points are then generated using simple interpolation techniques. The algebraic grid thus generated may have slope discontinuities in the grid lines, crossing of grid lines etc. These problems are removed by grid smoothing using elliptic equations with control functions [12]. The control functions as suggested by Yu [13] are used here to maintain the clustering of grid lines. Proper cut boundary conditions ensure slope continuity of grid lines across the block boundaries. Dirichlet boundary conditions are applied at the solid wall and far-field boundaries.

NAL has also acquired the GRIDGEN [9] software. This is a highly interactive and user-friendly package for three-dimensional grid generation, developed by M/s Pointwise Inc., USA. The main features of this package are

1. The package has a user-friendly graphical interface, which enables the grid to be visible at every stage. This is of immense help in generating complex multi-block grids with a large number of blocks.
2. Intersections of different components e.g. fuselage and wing are easily determined.
3. The user can control the number and distribution of grid points on all the block edges of the grid.

4. The initial grid is generated with transfinite interpolation. Grid smoothing by elliptic equations, with control functions is possible.
5. A block can be checked for the presence of negative volumes and skewed cells. The grid axes can be independently set for each block and a warning is given if a left-handed system is prescribed. The blocks can be assembled in the desired order for the flow solver.

Both the indigenous JUMGRID code and the commercial GRIDGEN code have similar capabilities in the generation of multi-block grids for complex geometries. The graphical interface makes the GRIDGEN code user friendly, particularly for grid smoothing, since the grid is continuously visible, in contrast to JUMGRID where some other post-processing software is needed for visualization. The JUMGRID code generates a grid based on the input data, which is read from data files. Hence, modifications to the grid, due to minor changes in geometry, can be made very easily using the JUMGRID code. Both of these codes are capable of producing grids of similar grid quality. Even if a grid is generated using GRIDGEN, the JUMGRID code has to be used to create image cells at the boundaries for each block and make the grid compatible for the solver codes.

Applications

The flow solver codes JUEL3D and JUMBO3D have been extensively used to analyze the flow past a variety of aerospace configurations, some of which are described in [6]. Some recent applications of these codes used for design and development of aircraft geometries are described below.

Design and analysis of SARAS

The design and development of a 9-14 seat Light Transport Aircraft named SARAS is in progress at the CSIR Center for Aircraft Design and Development (C-CADD) at NAL. It is a low-winged aircraft with a large fairing. A thirty-block grid was generated around the wing-fuselage configuration using the JUMGRID code. The Euler code, JUEL3D, was used to compute inviscid flow past the SARAS wing-fuselage configuration [14]. Later, the solver code and the grid data of the basic SARAS model were installed in C-MMACS computer for the routine use by the design group of C-CADD. Further generation of the grid for modified geometries and validation of the results using the same solver code continued in
the CTFD Division. From time to time, the updated grid data for the modified geometry was transferred to C-CADD for their production runs.

Figure 1a shows the grid on the surface and Fig.1b shows a perspective view of the grid in the symmetry plane as well as the elliptic outer boundary. Experiments were performed in NAL for this configuration and the comparison of lift and moment coefficients with the Euler results are shown in Fig.1c and Fig.1d respectively. The lift coefficient, $CL$, agrees very well with the experimental data. The moment coefficient, $CM$, versus angle of incidence, $\alpha$, curves also maintain good agreement in their slopes which is desirable. Comparisons of the surface pressure coefficient, $C_p$, with experiment were done at C-CADD for various free stream conditions and a typical case is shown in Fig.1e. As is seen in this figure, leading edge suction is over predicted but overall comparison up to the trailing edge is satisfactory as expected from inviscid flow computation.

The shape of the fuselage and fairing near the wing-body junction has to be designed very carefully to minimize the mutual interference effects and have a smooth flow pattern. The design procedure consists of an iterative process, with each iteration involving a number of steps; first modifying the contours of the fuselage cross-sections, then generating the wing-fuselage intersection curve, creating the grid for this configuration, computing the flow and finally visualizing the surface flow pattern. The fuselage modifications are also subject to the design constraints. In fact, these are restricted to the fairing that fits under the fuselage. Fig.2a and Fig.2b show the streamlines on the original and on the modified configurations respectively. The streamline patterns show that the modified configuration has a smoother flow pattern and does not have the clustering of streamlines on the rear fuselage as seen in the original configuration. Subsequently, the design group changed the shape and length of the fuselage fairing. After detailed analysis of the flow and the streamline pattern, modifications were carried out systematically and a final configuration was obtained. Fig.2c shows the streamlines of the modified configuration showing a smooth flow pattern.
Aerodynamic Load Distribution on a Fighter Aircraft

In order to estimate the fatigue life of an aircraft structure, the aerodynamic loads experienced by the aircraft in flight must be accurately simulated in the laboratory. This requires the calculation of the flow past the aircraft in order to determine the pressure distribution on its surface. In general, the flow on the wing is influenced to a large extent by the presence of the fuselage; hence the computation of the flow past at least the wing-fuselage configuration becomes essential. Since the computation of flow past complete aircraft requires a considerably larger effort in grid generation, a simpler geometry, consisting of the wing attached to the fuselage with canopy, has been used for the computations. Two grids with different grid topologies (one with 30 blocks and the other with 22 blocks) were generated for the actual wing-fuselage configuration using GRIDGEN. It was found that the second grid, which had a smaller number of grid points on the wing tip than the root, was better suited for the low aspect-ratio fighter type wing. The inlet nose cone was considered in the above model, but the computations have
not considered the flow through this inlet. To study this further, a third grid was generated in which the nose cone was merged with the fuselage. This grid topology and size were identical with the 22-block-grid used for the earlier model. No significant difference in computed results was observed. While comparing the results with experimental data, it was found that for surface pressure and force measurement in NAL tunnel, a model with axisymmetric fuselage with contoured nose and actual wing was used. To further validate the code and be more confident about the computed results, a fourth grid was generated for this geometry and the results agreed better with experimental data near the leading edge of the wing for supersonic flow. Fig. 3a, 3b and 3c show a view of the 22 block grids of the three models considered respectively. From all the results obtained so far for different models and for different grids at various free stream conditions [15], it is observed that for the low angle of incidence cases there is reasonably good agreement between the computed values and experimental data, except near the wing tip. It is apparent that there is a considerable disagreement between the experimental and computational results for the flows at high angle of incidence. The flow past a delta wing at high angle of incidence is dominated by leading edge vortices, and this is shown in Fig. 3d and 3e at $\alpha = 15$ deg and free stream Mach numbers, $M_{inf} = 0.6$ and 0.9 respectively. Fig. 4a and 4b show the surface pressure distribution at $M_{inf} = 0.6$, $\alpha = 5$ deg and $M_{inf} = 0.9$, $\alpha = 15$ deg respectively. Fig. 4c

Fig. 3a A view of the grid for a fighter aircraft - actual wing body configuration (total grid size ~ 0.63 million grid points)

Fig. 3b A view of the grid for a fighter aircraft - merged nose - cone configuration

Fig. 3d Streamlines from leading edge of a fighter aircraft, $M_{inf} = 0.6$, $\alpha = 15$ deg

Fig. 3e Streamlines from leading edge of a fighter aircraft, $M_{inf} = 0.9$, $\alpha = 15$ deg
Fig. 4a Surface pressure distribution on a fighter aircraft, $M_\infty = 0.6$, $\alpha = 5$ deg

Fig. 4b Surface pressure distribution on a fighter aircraft, $M_\infty = 0.9$, $\alpha = 15$ deg

Fig. 4c Comparison of computed $C_p$ distribution with experiment at four different span stations for $M_\infty = 0.9$, $\alpha = 5$ deg
(i) 19.21%, (ii) 38.22%, (iii) 57.23%, (iv) 76.24%

Fig. 5a Dowstream cross-sections of the three bodies

Fig. 5b Surface grids on the radome and its extended bodies, with and without canopy

compares the computed $C_p$ distribution with experiment for the case $M_\infty = 0.9$, $\alpha = 5$ deg. Further details of the results and comparisons are available in [15].
Analysis of the Nose Radome of a Fighter Aircraft

In many cases, analysis of flow past individual components of the full aircraft is necessary to find their requirement of structural strength to sustain the aerodynamic load. The nose radome of a fighter aircraft is one such important component whose shape and the material used should be chosen such that it can bear the aerodynamic load at transonic and supersonic speed range. NAL has initiated an inter-divisional program for an improved design and development of the nose radome of a fighter aircraft using composite material. For the analysis of a cone-cylinder type body of revolution of finite length, the computational domain should be extended beyond the trailing edge unless the flow field is fully supersonic, since in that case, small disturbances propagate along the characteristics and the downstream conditions do not affect the upstream region. The present problem is of a different nature because the nose radome is not an isolated body but a body of revolution of finite length attached to the front part of the fuselage of an aircraft. It is expensive and time consuming to compute the flow past the full aircraft in order to study the solution for only the nose radome region. So it was decided to study the flow past a nose radome considering some effect of the fore-body of the aircraft, which may contain the canopy also. Hence, the body is extended in a simple conical way and an approximate canopy is added at the top of the downstream end. Fig. 5a and 5b show the downstream cross-sections of the three bodies and surface grids on the radome and its extended bodies, with and without canopy. JUMGRID has been used to generate a C-O-type grid with (31 x 53 x 105) points in the normal, wrap-around and flow directions respectively for the radome and 151 points in the flow direction for the extended bodies. Solutions have been obtained for all the models using JUEL3D code and a detailed analysis is reported in [16]. For a typical case, Fig. 6a shows the comparison of the pressure distributions on the surface of the nose radome and its extensions for the flow with a free-stream Mach number, $M_{\infty} = 1.1$ and an angle of incidence, $\alpha = 22$ deg. Fig. 6b and 6c show the Mach contours on the extended bodies. It has been observed that though there is a significant effect on the nose radome due to the downstream extension of the body, i.e. the extension of the computational domain, no significant effects are observed due to the presence of the canopy whose distance is sufficient enough not to induce any disturbances at the nose region. Grid consistency was demonstrated using a finer grid, created by increasing the grid points in all directions resulting in an increase of the
Fig. 6b Mach contours on extended nose radome without canopy, and symmetry plane $M_\infty = 1.1$, $\alpha = 22\,\text{deg}$

Fig. 6c Mach contours on extended nose radome with canopy, and symmetry plane $M_\infty = 1.1$, $\alpha = 22\,\text{deg}$

Fig. 7a Planform of the cropped delta wing with aileron

Fig. 7b Schematic of block arrangement of first seven blocks (total grid size – 0.37 million grid points for inviscid flow – 1.2 million grid points for viscous flow)

Fig. 7c $C_p$ contours on upper surface of cropped delta wing with 6 deg deflected aileron Euler computation. $M_\infty = 0.9$, $\alpha = 1\,\text{deg}$

Fig. 7d $C_p$ contours on upper surface of cropped delta wing with 6 deg deflected aileron Navier-Stokes computation. $M_\infty = 0.9$, $\alpha = 1\,\text{deg}$, $Re = 17\times10^6$
total number of grid points by about three times. In [16], a fully supersonic case, \( M_\infty = 3.0 \) was also considered to demonstrate that the flow past the nose radome does not really get influenced whether the body has been extended further or not.

**Computation of Flow Past a Delta Wing With Deflected Aileron**

The determination of the aerodynamic characteristics of a wing with control surface deflections is one of the most important and difficult tasks in aircraft design and development. Present day aircraft make extensive use of high lift devices, which greatly improve their performance during the take-off and landing phases. Hence, there is a great interest in developing the computational tools required to accurately simulate the effects of control surface deflections.

The geometry considered here is a thin, sharp-edged, highly swept and cropped delta wing whose planform shape is shown in Fig.7a. Some experimental data for this configuration is available in [17]. The JUMGRID code has been used to generate a grid containing twenty-eight blocks, around the delta wing. Fig.7b shows the schematic arrangement of the first seven blocks on a plane of constant span. The wing cross-section is shown both without and with (dashed line) the deflected aileron. The wing is divided into four zones in the span-wise direction, the first extending from the symmetry plane up to the outboard end of the aileron at 56.6\% semi-span, the second lying on the aileron, the third extending from the outboard end of the aileron (82.9\% semi-span) up to the wing tip, and the fourth extending beyond the wing tip. Detailed analysis of the results of Euler and Navier-Stokes codes are reported in [18, 19]. Fig.7c and 7d show the computed contours of the coefficient of pressure on the upper surface of the wing with an aileron deflection of 6\(^\circ\) obtained by Euler and Navier-Stokes computations respectively. In this problem a thin, cropped delta wing with a sharp leading edge and deflected aileron has been considered as a solid model (without gaps), in order to study its suitability in representing the actual wing with deflected control surfaces. It has been observed that the inviscid flow computation using Euler equations simulates the flow very well as observed in practice/experiment, Fig.7e. The solid model gives a separated flow on the aileron, which is physically correct for this model and is predicted by viscous flow computation, shown in Fig.7f (refer page No. 85). A detailed analysis of the Euler and Navier-Stokes computations on this model is available in [19].

**Analysis of Wing Flap Position for HANSA**

A two-seater trainer aircraft named HANSA has been designed and built at NAL. For any aircraft, the effectiveness of the control surfaces is of critical importance. In order to determine an effective flap configuration, a study of the flow past the two-dimensional airfoil with flap has been initiated. Some results have been obtained for the viscous flow past the airfoil-flap configuration with two different flap configurations. A multi-block grid has been generated using the GRIDGEN package. The two-dimensional version of the Navier-Stokes code, JUMBO2D [5], has been used for the computations.

The computed streamlines for the flow past the airfoil at free-stream Mach number \( M_\infty = 0.3 \) and angles of incidence, \( \alpha = 0 \) deg and 10 deg are shown in Fig.8a and 8b respectively. In these figures streamlines are shown for
flap deflection angle $\delta = 0$ deg and $\delta = 20$ deg for both an existing and a proposed configuration. The streamlines show that there are large regions of separated flow on the deflected flap in both the configurations considered, which reduce the effectiveness of the flap. Efforts are underway to determine a better and optimum configuration for the flap.

Wing Tip Models of HANSA

The shape of the tip of an aircraft wing has a significant effect on the flow in that region, particularly on the formation of the wing-tip trailing vortices. Hence, a study was made of the effect of two different types of wing tips for the HANSA wing; one was a tip closed by a wedge and the other a tip with a vertical plane (sawn-off tip). Two different multi-block grids were generated for these two cases using GRIDGEN. The multi-block approach allowed the creation of a block such that one face mapped onto the sawn tip.

Figure 9 shows the surface grid and pressure distribution on the wing with the two different wing tips. Also shown are two typical streamlines near the tip, one on the upper surface of the wing and the other on the lower surface. The figure shows that the streamlines deviate further from each other in the case of the wedge tip, leading to an increase in vorticity and a larger induced drag factor. Hence, the wing with the sawn tip has aerodynamic characteristics similar to a wing with higher aspect ratio giving higher lift coefficient $C_L$ and lift-induced-drag coefficient $C_{D_i}$, but lower induced drag factor $K$.

Effect of Reynolds Number on a Probe

Aircraft usually have a probe that measures the static and dynamic pressure and thus helps to determine the air speed. This probe needs to be properly designed and calibrated and this requires the determination of the pressure distribution on the surface of the probe at different flight conditions. The pressure distribution at selected positions on the probe can be obtained experimentally, but
Fig. 9  Study of the effect of wing-tip model on Hansa wing, $M_{\infty} = 0.2, \alpha = 6\,\text{deg}$
(a) surface grid  (b) pressure contours with streamlines close to tip (total grid size ~ 0.25 million grid points)

Fig. 7f  Mach contours at 68\% semi-span, $M_{\infty} = 0.9, \alpha = 1\,\text{deg}$, $Re_{\infty} = 17 \times 10^6$
Fig. 10 Reynolds number effect on Cp distribution on a probe
(a) perspective view of the grid (104 x 30 x 51)
(b) surface Cp distribution at \( M_\infty = 0.2, \alpha = 0 \text{ deg} \)
(c) surface Cp distribution at \( M_\infty = 0.3, \alpha = 0 \text{ deg} \)
(d) surface Cp distribution at \( M_\infty = 0.4, \alpha = 0 \text{ deg} \)
it is often difficult to simulate the flight Reynolds number in the wind tunnel. So, the question arises about the effect of the Reynolds number on the measured pressure. The pressure at a particular point on the probe had been measured experimentally at NAL tunnel.

A single block grid was generated for the probe using the JUMGRID code and JUMBO3D code was used to compute the flow. Fig.10 shows a perspective view of this grid along with the computed Cp distribution for three different free-stream Mach Numbers M∞ = 0.2, 0.3 and 0.4 and angle of incidence, α = 0 deg. Each case was computed for two different free-stream Reynolds Numbers and compared with corresponding experimental result. The results show that the Reynolds number has a negligible effect in all the cases considered. Although there is only one experimental data point, it agrees quite well with the computations.

Conclusions

The results presented in this paper have shown the capability of CFD codes to simulate the flow past a variety of realistic aerospace configurations. CFD can play a major role not only in the design and development of aircraft but also in the study of complex fluid flow structure. The multi-block approach enables the analysis of fairly complex configurations. In the near future, with further improvement in computer technology and numerical algorithms, CFD will be able to simulate even more complex flows with less turnaround time, and thus become a robust and reliable design tool by making the design process faster, more accurate and less expensive.

Acknowledgement

The authors take this opportunity to thank Dr. Rajeswari S. Ramamurthy of C-CADD for providing the data for Fig.1e. Many fruitful discussions the authors had with their colleagues in Computational and Theoretical Fluid Dynamics (CTFD) Division and the continuous encouragement they received from Dr. S. S. Desai, Head, CTFD Division are gratefully acknowledged.

References


12. Thompson, J. F., "A General 3-Dimensional Elliptic Grid Generation System on a Composite Block


