

Re-engineering the design process through computation

Antony Jameson
Princeton Univ., NJ

AIAA, Aerospace Sciences Meeting & Exhibit, 35th, Reno, NV, Jan. 6-9, 1997

This paper reviews the use of computational flow simulation for aircraft design. While computational simulation has already yielded great benefits, its impact continues to be restricted by computational and human costs. The requirements of the design process are examined in detail. In current practice there are bottlenecks in geometry modelling, data handling, and mesh generation. To exploit computational simulation more effectively there is also a need to combine it with automatic optimization procedures. A case study is presented of a recent industrial experience with optimization. Finally it is suggested that developments in information technology now offer the opportunity to produce an integrated numerical design environment and to re-engineer the design process. (Author)

Re-Engineering the Design Process through Computation

Antony Jameson[†]

Princeton University

Department of Mechanical and Aerospace Engineering
Princeton, New Jersey 08544 USA

ABSTRACT

This paper reviews the use of computational flow simulation for aircraft design. While computational simulation has already yielded great benefits, its impact continues to be restricted by computational and human costs. The requirements of the design process are examined in detail. In current practice there are bottlenecks in geometry modelling, data handling and mesh generation. To exploit computational simulation more effectively there is also a need to combine it with automatic optimization procedures. A case study is presented of a recent industrial experience with optimization. Finally it is suggested that developments in information technology now offer the opportunity to produce an integrated numerical design environment, and to re-engineer the design process.

1 INTRODUCTION

During the past 25 years the entire process of engineering design has been revolutionized as computational simulation has come to play an increasingly dominant role. Nowadays engineers spend most of their time at workstations.

Most notably, computer aided design (CAD) methods have essentially replaced the drawing board as the basic tool for definition and control of the configuration. Software systems such as CATIA and Unigraphics provide a solid modelling capability that enables designers to prepare complex layouts without the need to build mockups. The visualization provided by three dimensional graphics enables the designer to verify that there will be no interference between different parts in the layout, and greatly facilitates decisions on the routing of all the electrical wiring and hydraulic piping.

Similarly, structural analysis is now entirely carried out by computational methods typically based on the finite element method. Commercially available software systems such as NASTRAN or ELFINI have been progressively developed and augmented by new features, and can treat the full range of requirements for aeronautical structures, including analysis of stressed skin structures into the nonlinear range.

They are also very carefully validated before each new release against a comprehensive suite of test cases, and engineers can place complete confidence in the results. Accordingly the structural design is routinely committed on the basis of computational analysis, while structural testing is limited to the role of verification that the design truly meets its specified requirements of ultimate strength and fatigue life.

Computational simulation of fluid flow has not yet reached the same level of maturity. While commercial software for the simulation of fluid flow is offered by numerous vendors, aircraft companies continue to make substantial investments on the in-house development of their own methods, such as Boeing's TRANAIR program, or Lockheed's TEAM program. At the same time there are major ongoing efforts to develop the science of computational fluid dynamics (CFD) in government research agencies such as NASA, Japan's ARL, or in Europe, France's ONERA, Germany's DLR, Holland's NLR and Sweden's FFA, all of which are a source of industrially used computer programs. This reflects the fact that fluid flow is generally more complex and harder to predict than the behavior of structures. The complexity and range of phenomena of fluid flow is well illustrated in Van Dyke's *Album of Fluid Motion* [13].

The concept of a numerical wind tunnel, which might eventually allow computers "to supplant wind tunnels in the aerodynamic design and testing process", was already a topic of discussion in the decade 1970-1980. In their celebrated paper of 1975 [9], Chapman, Mark and Pirtle listed three main objectives of computational aerodynamics:

1. To provide flow simulations that are either impractical or impossible to obtain in wind tunnels or other ground based experimental test facilities.
2. To lower the time and cost required to obtain aerodynamic flow simulations necessary for the design of new aerospace vehicles.
3. Eventually, to provide more accurate simulations of flight aerodynamics than wind tunnels can.

Copyright ©1997 by the Author.

Published by the AIAA Inc. with permission

[†]James S. McDonnell Distinguished University Professor of Aerospace Engineering, AIAA Fellow

Chapman, Mark, and Pirtle also noted that the inherent limitations of computational and wind tunnel simulations are complementary. Wind tunnels are limited by the size of the models that can be placed in them, and by the density, temperature and velocity of the flow that they can sustain, with the consequence that flight-Reynolds numbers cannot be realized with complete models. Their accuracy is also limited by wall and support interference, and by aeroelastic distortion. Computers are not limited in any of these ways, but they are limited in speed and memory, which in turn limit the attainable complexity and resolution of the simulations.

Computational Fluid Dynamics (CFD) has now matured to the point at which it is widely accepted as a key tool for aerodynamic design. Algorithms have been the subject of intensive development for the past two decades. The principles underlying the design and implementation of robust schemes which can accurately resolve shock waves and contact discontinuities in compressible flows are now quite well established. It is also quite well understood how to design high order schemes for viscous flow, including compact schemes and spectral methods. Adaptive refinement of the mesh interval (h) and the order of approximations (p) has been successfully exploited both separately and in combination in the h - p method [34].

Despite these advances, CFD is still not being exploited as effectively as one would like in the design process. This is partially due to the long set-up times and high costs, both human and computational, of complex flow simulations. A continuing obstacle to the treatment of configurations with complex geometry has been the problem of mesh generation. Several general techniques have been developed, including algebraic transformations and methods based on the solution of elliptic and hyperbolic equations. In the last few years methods using unstructured meshes have also begun to gain more general acceptance.

The fidelity of mathematical modelling of high Reynolds number flows continues to be limited by computational costs. Thus, accurate and cost-effective simulation of viscous flow at high Reynolds numbers associated with full scale flight remains a challenge. Several routes are available toward the reduction of computational costs, including the reduction of mesh requirements by the use of higher order schemes, improved convergence to steady state by sophisticated acceleration methods, and the exploitation of massively parallel computers. In the present state of the art however, it is still cheaper to obtain massive quantities of data such as the loads data over the flight envelope by wind tunnel testing, since the incremental cost of obtaining additional data is very small once a wind tunnel model has been built. With computational simulation, the cumulative cost of generating data for the full flight envelope becomes very large since a separate run is required for each data point. Computational simulation has the key advantage, on the other hand, that it allows the rapid

exploration of numerous alternative designs. Thus CFD and wind tunnel testing can be effectively used in complementary roles, with CFD the prime tool for the initial design studies, and wind tunnel testing the prime tool for final verification of the design concept and acquisition of the full aerodynamic data required for completion of the detailed design.

This paper examines ways to exploit computational simulation more effectively in the overall design process, with the primary focus on aerodynamic design, while recognizing that this should be part of an integrated multi-disciplinary process. The design process itself is surveyed in the next section. The following two sections examine the industrial requirements for effective and trustworthy CFD software, and the way in which optimization techniques can be integrated with CFD. Section 5 discusses recent industrial experience in the application of CFD and optimization techniques to a major project for a commercial aircraft. Finally Section 6 discusses ways in which the design process might be reengineered to exploit computational simulation more effectively.

2 THE DESIGN PROCESS

The design process can generally be divided into three phases: conceptual design, preliminary design, and final detailed design, as illustrated in Figure 1. The conceptual design stage defines the mission in the light of anticipated market requirements, and determines a general configuration capable of performing this mission, together with first estimates of sizing, weight and performance. In the preliminary design stage the aerodynamic shape and structural skeleton progress to the point where detailed performance estimates can be made and guaranteed to potential customers. The design is sufficiently refined to provide the basis for making formal offers to customers and signing contracts. At this stage the development costs are still fairly moderate, in the range of 50 - 100 million dollars. In the final design stage the structure must be defined in complete detail, together with complete systems, including the flight deck, control systems (involving major software development for fly-by-wire systems), electrical and hydraulic systems, landing gear, weapon systems for military aircraft, and cabin layout and systems for commercial aircraft. Major costs are incurred at this stage, during which is also necessary to prepare a detailed manufacturing plan, together with appropriate facilities and tooling. The development costs to reach the point of initial production are in the range of 3 - 10 billion dollars. Thus the final design would normally be carried out only if sufficient orders have already been received to indicate a reasonably high probability of recovering the return on the investment.

Figure 2 provides a closer look at the conceptual design stage. In the case of commercial aircraft the mission is defined on the basis of airline requirements. Desired payload-range characteristics follow from route analysis between representative city pairs such as Los Angeles - Tokyo, including

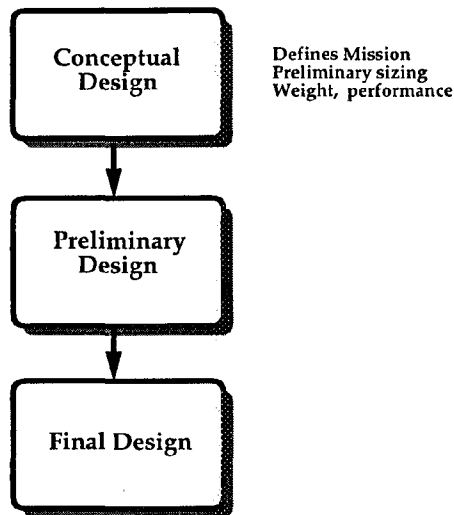


Figure 1: The Overall Design Process

data on expected traffic volume, desired frequency, and prevailing weather patterns. At the same time it is necessary to consider issues of airport compatibility, including constraints on gate size and noise regulations. A preliminary synthesis using simplified aerodynamic and structural models and statistical data bases provides an initial configuration and sizing, together with performance estimates, taking into account requirements for stability and control. Software for aircraft synthesis such as NASA Ames' ACSYNT program is available to assist this process. For commercial aircraft it is necessary to estimate both the operating cost and the cost of ownership, while for military aircraft the life time cycle cost may be a determining factor. In either case it is generally assumed that the selling price is likely to be proportional to the gross weight of the aircraft.

The result of the initial synthesis may confirm the feasibility of the proposed mission. On the other hand it may suggest that it is too ambitious, requiring an excessively large and expensive aircraft, or alternatively that a more testing mission could be accomplished with an aircraft of acceptable size. Thus the process will generally be iterated until it arrives at a mission and corresponding design that can be expected to attain the desired market capture and return on investment. Concurrently discussions will proceed both with potential customers to verify market interest, and with major vendors such as the engine manufacturers to assure the availability of appropriate power plants and systems. These discussions may well lead to further iteration of the mission and design concept in an ongoing process. Vendors may also be approached to share in the development costs as risk sharing partners, or to undertake substantial development costs of their own to provide components which meet the design

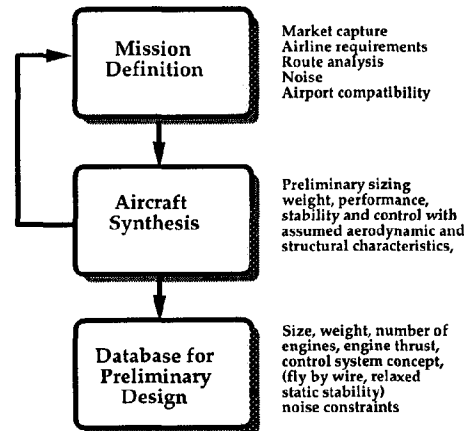


Figure 2: The Conceptual Design Process

requirements.

In the development of commercial aircraft, aerodynamic design is in the lead during the preliminary design stage. The definition of the external aerodynamic shape may actually be finalized in the preliminary design. The aerodynamic lines of the Boeing 777 were frozen, for example, when initial orders were accepted before the initiation of detailed design of the structure. Figure 3 illustrates the way in which the aerodynamic design process is embedded in the overall preliminary design. The starting point is an initial CAD definition resulting from the conceptual design. The inner loop of aerodynamic analysis is contained in an outer multi-disciplinary loop, which is in turn contained in a major design cycle involving wind tunnel testing. In recent Boeing practice three major design cycles, each requiring about 4 - 6 months, are used to finalize the wing design. Improvement in CFD which would allow the elimination of a major cycle would significantly shorten the overall design process and reduce costs. In the development of the MDXX Douglas planned to rely on high level CFD together with the experimental database which had been developed for the MD12; and expected to eliminate the need for a sequence of major design cycles.

The inner aerodynamic design loop is used to evaluate numerous variations in the wing definition. In each iteration it is necessary to generate a mesh for the new configuration prior to performing the CFD analysis. Computer graphics software is then used to visualize the results, and the performance is evaluated. The first studies may be confined to partial configurations such as wing-body or wing-body-nacelle combinations. At this stage the focus is on the design of the clean wing. Key points of the flight envelope include the nominal cruise point, cruise at high lift and low lift to allow for the weight variation between the initial and final cruise as the fuel is burnt off, and a long range cruise point

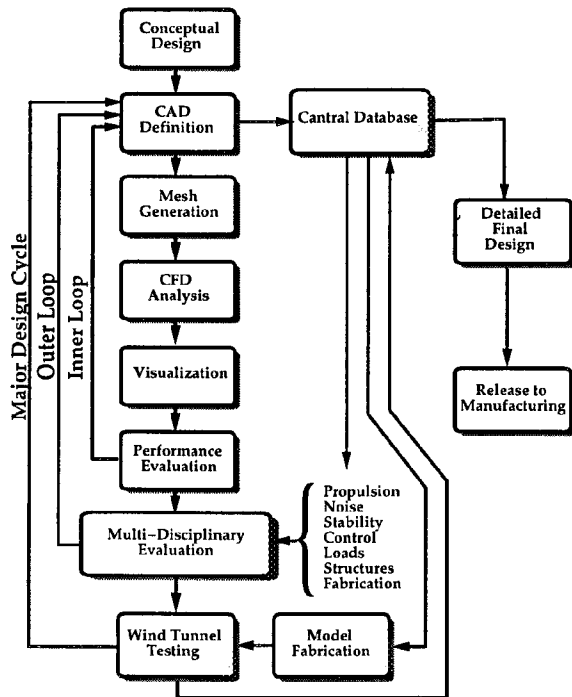


Figure 3: The Aerodynamic Design Process

at lower Mach number, where it is important to make sure there is no significant drag creep. Other defining points are the climb condition, which requires a good lift to drag ratio at low Mach number and high lift coefficient with a clean wing, and the buffet condition. This is typically taken as the high lift cruise point increased to a load of 1.3 g to allow for maneuvering and gust loads. Both wing section modifications such as the thickness to chord ratio, and planform variations such as the sweepback angle or aspect ratio may be considered. While the detailed design of the high lift system and control surfaces may be deferred to a later stage, the planform must provide the necessary space for both high lift systems and control surfaces outside the main structural box, and it must also accommodate the landing gear. This generally requires an extension of the in board trailing edge to form a "yehudi".

The aerodynamic analysis interacts with the other disciplines in the next outer loop. These disciplines have their own inner loops, not shown in Figure D3. For an efficient design process the fully updated aero-design data base must be accessible to other disciplines without loss of information. For example, the thrust requirements in the power plant design will depend on the drag estimates for take-off, climb and cruise. In order to meet airport noise constraints a rapid climb may be required while the thrust may also be limited. Initial estimates of the lift and moments allow preliminary sizing of

the horizontal and vertical tail. This interacts with the design of the control system, where the use of a fly-by-wire system may allow relaxed static stability and tail surfaces of reduced size.

First estimates of the aerodynamic loads allow the design of an initial structural skeleton, which in turn provides an estimate of the structure weight. One of the main trade-off is between aerodynamic performance and wing structure weight. The requirement for fuel volume may also be an important consideration. An increase in the thickness to chord ratio both increases fuel volume, and allows the same bending moment to be carried with reduced skin thickness, with an accompanying reduction in weight. On the other hand it will lead to a decrease in the drag rise Mach number. The induced drag, which typically contributes around 40 percent of the cruising drag, varies inversely as the square of the span. Thus a 5 percent increase in the wing span could produce a total drag reduction of the order of 4 percent, but would lead to an increase in wing weight because of the increase in the root bending moment. The wing span may in fact be limited by airport gate constraints.

The taper ratio and span load distribution also affect the trade-off between aerodynamic performance and wing weight. While an elliptic span load distribution minimizes the induced drag for a given span, a more triangular load distribution reduces the root bending moment. A large root chord may be dictated by the need to accommodate the landing gear and flaps, but it also has the advantage of increasing the root thickness for a fixed thickness to chord ratio, yielding a weight reduction. For example, the root chord of the MDXX was increased at a late stage in the design to accommodate larger flaps, and this contributed a significant weight reduction. In order to maintain a moderately efficient span load distribution with a highly tapered planform the outboard wing must operate with higher local section lift coefficient than the inboard wing. This can have an adverse effect on the behavior near buffet, as the outboard wing will incur a shock stall before the inboard wing, leading to a reduction of lift behind the center of gravity, and consequently a high speed pitch-up. This is unacceptable for certification if it is too severe.

An increase in the wing sweepback angle may be used to increase the drag rise Mach number. Alternatively it allows an increase in the thickness to chord ratio for the same drag rise Mach number, with a resulting weight reduction. This is partially offset by the increase in the length of the wing. Moreover, an increase in the sweep back angle will aggravate the problem of high speed pitch-up. Most modern highly loaded wings have sweep back angles no greater than 35 degrees at the $\frac{1}{4}$ chord line.

Manufacturing constraints must also be considered in the final definition of the aerodynamic shape. For example, the section changes in the span wise direction must be limited.

This avoids the need for shot peening which might otherwise be required to force curvature in both the span wise and chord wise directions.

From the complexity of these trade-offs it can be seen that a crucial requirement for aerodynamic analysis is to make trustworthy predictions with fast enough turn around not to delay the outer multidisciplinary cycle. In order to allow the completion of the major design cycle in 4 - 6 months, the cycle time for the multidisciplinary loop should not be greater than about 2 weeks. Considering the need to examine the performance of design variation at all the key points of the flight envelope, this implies the need to turn around the aerodynamic analyses in a few hours. The computational costs are also important because the cumulative costs of large numbers of calculations can become a limiting factor.

It is also evident that the number of possible design variations is too large to permit their exhaustive evaluation, and thus it is very unlikely that a truly optimum solution can be found without the assistance of automatic optimization procedures. Ultimately there is a need for multi-disciplinary optimization (MDO), but this can only be effective if it is based on sufficiently high fidelity modelling of the separate disciplines. As a step in this direction there could be significant pay-offs from the application of optimization techniques within the disciplines, where the interactions with other disciplines is taken into account through the introduction of constraints. For example the wing drag can be minimized at a given Mach number and lift coefficient with a fixed planform, and constraints on minimum thickness to meet requirements for fuel volume and structure weight.

3 INDUSTRIAL CFD

In order to carry out the inner loop of the aerodynamic design process the main requirements for effective CFD software are:

1. Sufficient and known level of accuracy
2. Acceptable computational and manpower costs
3. Fast turn around time

Performance estimation in the cruise condition is crucial to the design of transport aircraft and the error should be in the range of $\pm\frac{1}{2}$ percent. The drag coefficient of a long range transport aircraft such as the Boeing 747 is in the range of .0275 (275 counts), depending on the lift coefficient, which is in the range of .5. The drag coefficient of proposed supersonic transport designs is in the range of .0120 to .0150 at much lower lift coefficients in the range of .1 - .12. Thus one should aim to predict drag with an accuracy of the order of $\pm .0001$ (± 1 count). Manufacturers have to guarantee performance, and errors can be very expensive through the costs of redesign, penalty payments and lost orders.

A first consideration is the choice of appropriate mathematical models of fluid flow which are adequate for trustworthy flow predictions. Many critical phenomena of fluid flow, such as shock waves and turbulence, are essentially non-linear. They also exhibit extreme disparities of scales. While the actual thickness of a shock wave is of the order of a mean free path of the gas particles, on a macroscopic scale its thickness is essentially zero. In turbulent flow energy is transferred from large scale motions to progressively smaller eddies until the scale becomes so small that the motion is dissipated by viscosity. The ratio of the length scale of the global flow to that of the smallest persisting eddies is of the order $Re^{\frac{1}{3}}$, where Re is the Reynolds number, typically in the range of 30 million for an aircraft. In order to resolve such scales in all three space directions a computational grid with the order of $Re^{\frac{2}{3}}$ cells would be required. This is beyond the range of any current or foreseeable computer. Consequently mathematical models with varying degrees of simplification have to be introduced in order to make computational simulation of flow flow feasible and produce viable and cost-effective methods.

Figure 4 (supplied by Pradeep Raj) indicates a hierarchy of models at different levels of simplification which have proved useful in practice. Efficient flight is generally achieved by the use of smooth and streamlined shapes which avoid flow separation and minimize viscous effects, with the consequence that useful predictions can be made using inviscid models. Inviscid calculations with boundary layer corrections can provide quite accurate predictions of lift and drag when the flow remains attached, but iteration between the inviscid outer solution and the inner boundary layer solution becomes increasingly difficult with the onset of separation. Procedures for solving the full viscous equations are likely to be needed for the simulation of arbitrary complex separated flows, which may occur at high angles of attack or with bluff bodies. In order to treat flows at high Reynolds numbers, one is generally forced to estimate turbulent effects by Reynolds averaging of the fluctuating components. This requires the introduction of a turbulence model. As the available computing power increases one may also aspire to large eddy simulation (LES) in which the larger scale eddies are directly calculated, while the influence of turbulence at scales smaller than the mesh interval is represented by a subgrid scale model.

Computational costs vary drastically with the choice of mathematical model. Panel methods can be effectively used to solve the linear potential flow equation with higher-end personal computers (with an Intel 80486 microprocessor, for example). Studies of the dependency of the result on mesh refinement, performed by this author and others, have demonstrated that inviscid transonic potential flow or Euler solutions for an airfoil can be accurately calculated on a mesh with 160 cells around the section, and 32 cells normal

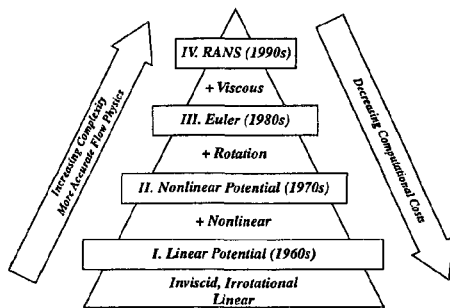


Figure 4: Hierarchy of Fluid Flow Models

to the section. Using multigrid techniques 10 to 25 cycles are enough to obtain a converged result. Consequently airfoil calculations can be performed in seconds on a Cray YMP, and can also be performed on 486-class personal computers. Correspondingly accurate three-dimensional inviscid calculations can be performed for a wing on a mesh, say with $192 \times 32 \times 48 = 294,912$ cells, in about 5 minutes on a single processor Cray YMP, or less than a minute with eight processors, or in 1 or 2 hours on a workstation such as a Hewlett Packard 735 or an IBM 560 model.

Viscous simulations at high Reynolds numbers require vastly greater resources. Careful two-dimensional studies of mesh requirements have been carried out at Princeton by Martinelli [30]. He found that on the order of 32 mesh intervals were needed to resolve a turbulent boundary layer, in addition to 32 intervals between the boundary layer and the far field, leading to a total of 64 intervals. In order to prevent degradations in accuracy and convergence due to excessively large aspect ratios (in excess of 1,000) in the surface mesh cells, the chordwise resolution must also be increased to 512 intervals. Reasonably accurate solutions can be obtained in a 512×64 mesh in 100 multigrid cycles. Translated to three dimensions, this would imply the need for meshes with 5–10 million cells (for example, $512 \times 64 \times 512 = 16,777,216$ cells as shown in Figure 5). When simulations are performed on less fine meshes with, say, 500,000 to 1 million cells, it is very hard to avoid mesh dependency in the solutions as well as sensitivity to the turbulence model.

A typical algorithm requires of the order of 5,000 floating point operations per mesh point in one multigrid iteration. With 10 million mesh points, the operation count is of the order of 0.5×10^{11} per cycle. Given a computer capable of sustaining 10^{11} operations per second (100 gigaflops), 200 cycles could then be performed in 100 seconds. Simulations of unsteady viscous flows (flutter, buffet) would be likely to require 1,000–10,000 time steps. A further progression to large eddy simulation of complex configurations would require even greater resources. The following estimate is due to W.H. Jou [24]. Suppose that a conservative estimate

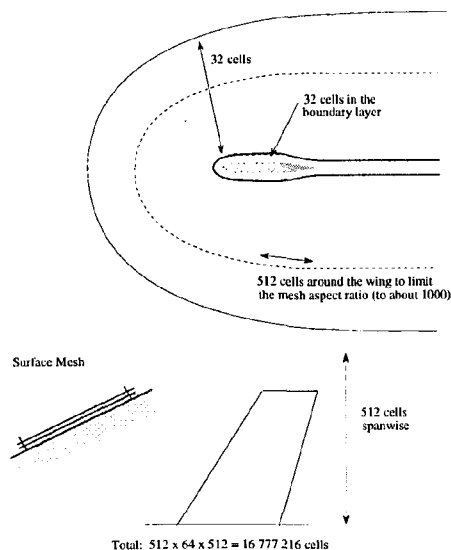


Figure 5: Mesh Requirements for a Viscous Simulation

of the size of eddies in a boundary layer that ought to be resolved is $1/5$ of the boundary layer thickness. Assuming that 10 points are needed to resolve a single eddy, the mesh interval should then be $1/50$ of the boundary layer thickness. Moreover, since the eddies are three-dimensional, the same mesh interval should be used in all three directions. Now, if the boundary layer thickness is of the order of 0.01 of the chord length, 5,000 intervals will be needed in the chordwise direction, and for a wing with an aspect ratio of 10, 50,000 intervals will be needed in the spanwise direction. Thus, of the order of $50 \times 5,000 \times 50,000$ or 12.5 billion mesh points would be needed in the boundary layer. If the time dependent behavior of the eddies is to be fully resolved using time steps on the order of the time for a wave to pass through a mesh interval, and one allows for a total time equal to the time required for waves to travel three times the length of the chord, of the order of 15,000 time steps would be needed. Performance beyond the teraflop (10^{12} operations per second) will be needed to attempt calculations of this nature, which also have an information content far beyond what is needed for engineering analysis and design. The designer does not need to know the details of the eddies in the boundary layer. The primary purpose of such calculations is to improve the calculation of averaged quantities such as skin friction, and the prediction of global behavior such as the onset of separation. The current use of Navier-Stokes and large eddy simulations is to try to gain an improved insight into the physics of turbulent flow, which may in turn lead to the development of more comprehensive and reliable turbulence models.

TURBULENCE MODELLING

It is doubtful whether a universally valid turbulence model, capable of describing all complex flows, could be devised [14]. Algebraic models [8, 2] have proved fairly satisfactory for the calculation of attached and slightly separated wing flows. These models rely on the boundary layer concept, usually incorporating separate formulas for the inner and outer layers, and they require an estimate of a length scale which depends on the thickness of the boundary layer. The estimation of this quantity by a search for a maximum of the vorticity times a distance to the wall, as in the Baldwin-Lomax model, can lead to ambiguities in internal flows, and also in complex vortical flows over slender bodies and highly swept or delta wings [11, 31]. The Johnson-King model [22], which allows for non-equilibrium effects through the introduction of an ordinary differential equation for the maximum shear stress, has improved the prediction of flows with shock induced separation [35, 25].

Closure models depending on the solution of transport equations are widely accepted for industrial applications. These models eliminate the need to estimate a length scale by detecting the edge of the boundary layer. Eddy viscosity models typically use two equations for the turbulent kinetic energy k and the dissipation rate ϵ , or a pair of equivalent quantities [23, 43, 40, 1, 33, 10, 28]. Models of this type generally tend to present difficulties in the region very close to the wall. They also tend to be badly conditioned for numerical solution. The $k - l$ model [38] is designed to alleviate this problem by taking advantage of the linear behavior of the length scale l near the wall. In an alternative approach to the design of models which are more amenable to numerical solution, new models requiring the solution of one transport equation have recently been introduced [3, 39]. The performance of the algebraic models remains competitive for wing flows, but the one- and two-equation models show promise for broader classes of flows. In order to achieve greater universality, research is also being pursued on more complex Reynolds stress transport models, which require the solution of a larger number of transport equations.

The selection of sufficiently accurate mathematical models and a judgment of their cost effectiveness ultimately rests with industry. As the design progresses through the three phases of conceptual design, preliminary design, and detailed design. The appropriate CFD models will vary in complexity. In the conceptual and preliminary design phases, the emphasis will be on relatively simple models which can give results with very rapid turn-around and low computer costs, in order to evaluate alternative configurations and perform quick parametric studies. The detailed design stage requires the most complete simulation that can be achieved with acceptable cost.

ALGORITHMS AND MESH GENERATION

The computational simulation of fluid flow presents a number of severe challenges for algorithm design. At the level of inviscid modelling, the inherent nonlinearity of the fluid flow equations leads to the formation of singularities such as shock waves and contact discontinuities. Moreover, the geometric configurations of interest are extremely complex, and generally contain sharp edges which lead to the shedding of vortex sheets. Extreme gradients near stagnation points or wing tips may also lead to numerical errors that can have global influence. Numerically generated entropy may be convected from the leading edge for example, causing the formation of a numerically induced boundary layer which can lead to separation. The need to treat exterior domains of infinite extent is also a source of difficulty. Boundary conditions imposed at artificial outer boundaries may cause reflected waves which significantly interfere with the flow. When viscous effects are also included in the simulation, the extreme difference of the scales in the viscous boundary layer and the outer flow, which is essentially inviscid, is another source of difficulty, forcing the use of meshes with extreme variations in mesh interval. For these reasons CFD, has been a driving force for the development of numerical algorithms.

An essential requirement for industrial CFD is the capability to treat extremely complex geometric configurations. A key choice that must be made is the nature of the mesh used to divide the flow field into discrete subdomains. The discretization procedure must allow for the treatment of complex configurations. The principal alternatives are Cartesian meshes, body-fitted curvilinear meshes, and unstructured tetrahedral meshes. Each of these approaches has advantages which have led to their use. The Cartesian mesh minimizes the complexity of the algorithm at interior points and facilitates the use of high order discretization procedures, at the expense of greater complexity, and possibly a loss of accuracy, in the treatment of boundary conditions at curved surfaces. This difficulty may be alleviated by using mesh refinement procedures near the surface. With their aid, schemes which use Cartesian meshes have recently been developed to treat very complex configurations [32, 36, 7, 26].

Body-fitted meshes have been widely used and are particularly well suited to the treatment of viscous flow because they readily allow the mesh to be compressed near the body surface. With this approach, the problem of mesh generation itself has proved to be a major pacing item. In order to treat very complex configurations it generally proves expedient to use a multiblock [42, 37] procedure, with separately generated meshes in each block, which may then be patched at block faces, or allowed to overlap, as in the Chimera scheme [5, 6]. While a number of interactive software systems for grid generation have been developed, such as EAGLE, GRIDGEN, GRAPE, and ICEM, the generation of a satisfactory grid for a very complex configuration may

require months of effort.

The alternative is to use an unstructured mesh in which the domain is subdivided into tetrahedra. This in turn requires the development of solution algorithms capable of yielding the required accuracy on unstructured meshes. This approach has been gaining acceptance, as it is becoming apparent that it can lead to a speed-up and reduction in the cost of mesh generation that more than offsets the increased complexity and cost of the flow simulations. Two competing procedures for generating triangulations which have both proved successful are Delaunay triangulation [12, 4], based on concepts introduced at the beginning of the century by Voronoi [41], and the moving front method [29].

For a detailed review of CFD algorithms in current use the reader is referred to reference [17]. Another key issue is the validation of CFD software for industrial use. For a better understanding of this issue it is important to distinguish the different sources of error. These include modelling errors because the mathematical model does not adequately represent the true physics of the flow, numerical errors and programming errors. Numerical errors include discretization errors, and errors in the numerical solution of the discrete model, if for example, an iterative procedure is not fully converged. The asymptotic behavior of discretization errors may be estimated by numerical analysis, and their magnitude in practice can be estimated by mesh refinement studies. It is hard to guarantee the elimination of programming errors, but their likelihood can be reduced by the use of modular programming. Then it should be possible to obtain the same result when alternative implementations are substituted for each module. Mesh refinement studies may also help the detection of programming errors by exposing discrepancies from the predicted asymptotic behavior as the mesh spacing is reduced, or discrepancies from known results for special cases, such as the fact that the drag should be zero in two dimensional subsonic inviscid flow. It is only after the correctness of the program and the accuracy of the numerical solution procedure have been independently verified that it is possible to assess the modelling errors which may arise, for example, from the use of an inappropriate turbulence model. For a more detailed discussion of validation procedures the reader is referred to reference [19].

4 AERODYNAMIC SHAPE OPTIMIZATION

Traditionally the process of selecting design variations has been carried out by trial and error, relying on the intuition and experience of the designer. It is not at all likely that repeated trials in an interactive design and analysis procedure can lead to a truly optimum design. In order to take full advantage of the possibility of examining a large design space the numerical simulations need to be combined with automatic search and optimization procedures. This can lead to automatic design methods which will fully realize the

potential improvements in aerodynamic efficiency.

The simplest approach to optimization is to define the geometry through a set of design parameters, which may, for example, be the weights α_i applied to a set of shape functions $b_i(x)$ so that the shape is represented as

$$f(x) = \sum \alpha_i b_i(x).$$

Then a cost function I is selected which might, for example, be the drag coefficient or the lift to drag ratio, and I is regarded as a function of the parameters α_i . The sensitivities $\frac{\partial I}{\partial \alpha_i}$ may now be estimated by making a small variation $\delta \alpha_i$ in each design parameter in turn and recalculating the flow to obtain the change in I . Then

$$\frac{\partial I}{\partial \alpha_i} \approx \frac{I(\alpha_i + \delta \alpha_i) - I(\alpha_i)}{\delta \alpha_i}.$$

The gradient vector $\frac{\partial I}{\partial \alpha}$ may now be used to determine a direction of improvement. The simplest procedure is to make a step in the negative gradient direction by setting

$$\alpha^{n+1} = \alpha^n - \lambda \frac{\partial I}{\partial \alpha},$$

so that to first order

$$I + \delta I = I + \frac{\partial I}{\partial \alpha} \delta \alpha = I - \lambda \frac{\partial I}{\partial \alpha} \frac{\partial I}{\partial \alpha}.$$

More sophisticated search procedures may be used such as quasi-Newton methods, which attempt to estimate the second derivative $\frac{\partial^2 I}{\partial \alpha_i \partial \alpha_j}$ of the cost function from changes in the gradient $\frac{\partial I}{\partial \alpha}$ in successive optimization steps. These methods also generally introduce line searches to find the minimum in the search direction which is defined at each step. The main disadvantage of this approach is the need for a number of flow calculations proportional to the number of design variables to estimate the gradient. The computational costs can thus become prohibitive as the number of design variables is increased.

An alternative approach is to cast the design problem as a search for the shape that will generate the desired pressure distribution. This approach recognizes that the designer usually has an idea of the kind of pressure distribution that will lead to the desired performance. Thus, it is useful to consider the inverse problem of calculating the shape that will lead to a given pressure distribution. The method has the advantage that only one flow solution is required to obtain the desired design. Unfortunately, a physically realizable shape may not necessarily exist, unless the pressure distribution satisfies certain constraints. The difficulty that the target pressure may be unattainable may be circumvented by treating the inverse problem as a special case of the optimization problem, with a cost function which measures the error in

the solution of the inverse problem. For example, if p_d is the desired surface pressure, one may take the cost function to be an integral over the body surface of the square of the pressure error,

$$I = \frac{1}{2} \int_{\mathcal{B}} (p - p_d)^2 d\mathcal{B},$$

or possibly a more general Sobolev norm of the pressure error. This has the advantage of converting a possibly ill posed problem into a well posed one. It has the disadvantage that it incurs the computational costs associated with optimization procedures.

APPLICATION OF CONTROL THEORY

In order to reduce the computational costs, it turns out that there are advantages in formulating both the inverse problem and more general aerodynamic problems within the framework of the mathematical theory for the control of systems governed by partial differential equations [27]. A wing, for example, is a device to produce lift by controlling the flow, and its design can be regarded as a problem in the optimal control of the flow equations by variation of the shape of the boundary. If the boundary shape is regarded as arbitrary within some requirements of smoothness, then the full generality of shapes cannot be defined with a finite number of parameters, and one must use the concept of the Frechet derivative of the cost with respect to a function. Clearly, such a derivative cannot be determined directly by finite differences of the design parameters because there are now an infinite number of these. Using techniques of control theory, however, the gradient can be determined indirectly by solving an adjoint equation which has coefficients defined by the solution of the flow equations. The cost of solving the adjoint equation is comparable to that of solving the flow equations. Thus the gradient can be determined with roughly the computational costs of two flow solutions, independently of the number of design variables, which may be infinite if the boundary is regarded as a free surface.

For flow about an airfoil or wing, the aerodynamic properties which define the cost function are functions of the flow-field variables (w) and the physical location of the boundary, which may be represented by the function \mathcal{F} , say. Then

$$I = I(w, \mathcal{F}),$$

and a change in \mathcal{F} results in a change

$$\delta I = \frac{\partial I^T}{\partial w} \delta w + \frac{\partial I^T}{\partial \mathcal{F}} \delta \mathcal{F}, \quad (1)$$

in the cost function. Using control theory, the governing equations of the flowfield are introduced as a constraint in such a way that the final expression for the gradient does not require reevaluation of the flowfield. In order to achieve this

δw must be eliminated from (1). Suppose that the governing equation R which expresses the dependence of w and \mathcal{F} within the flowfield domain D can be written as

$$R(w, \mathcal{F}) = 0. \quad (2)$$

Then δw is determined from the equation

$$\delta R = \left[\frac{\partial R}{\partial w} \right] \delta w + \left[\frac{\partial R}{\partial \mathcal{F}} \right] \delta \mathcal{F} = 0. \quad (3)$$

Next, introducing a Lagrange Multiplier ψ , we have

$$\begin{aligned} \delta I &= \frac{\partial I^T}{\partial w} \delta w + \frac{\partial I^T}{\partial \mathcal{F}} \delta \mathcal{F} - \psi^T \left(\left[\frac{\partial R}{\partial w} \right] \delta w + \left[\frac{\partial R}{\partial \mathcal{F}} \right] \delta \mathcal{F} \right) \\ &= \left\{ \frac{\partial I^T}{\partial w} - \psi^T \left[\frac{\partial R}{\partial w} \right] \right\} \delta w + \left\{ \frac{\partial I^T}{\partial \mathcal{F}} - \psi^T \left[\frac{\partial R}{\partial \mathcal{F}} \right] \right\} \delta \mathcal{F}. \end{aligned}$$

Choosing ψ to satisfy the adjoint equation

$$\left[\frac{\partial R}{\partial w} \right]^T \psi = \frac{\partial I^T}{\partial w} \quad (4)$$

the first term is eliminated, and we find that

$$\delta I = \mathcal{G} \delta \mathcal{F}, \quad (5)$$

where

$$\mathcal{G} = \frac{\partial I^T}{\partial \mathcal{F}} - \psi^T \left[\frac{\partial R}{\partial \mathcal{F}} \right].$$

The advantage is that (5) is independent of δw , with the result that the gradient of I with respect to an arbitrary number of design variables can be determined without the need for additional flow-field evaluations. In the case that (2) is a partial differential equation, the adjoint equation (4) is also a partial differential equation and appropriate boundary conditions must be determined.

After making a step in the negative gradient direction, the gradient can be recalculated and the process repeated to follow a path of steepest descent until a minimum is reached. In order to avoid violating constraints, such as a minimum acceptable wing thickness, the gradient may be projected into the allowable subspace within which the constraints are satisfied. In this way one can devise procedures which must necessarily converge at least to a local minimum, and which can be accelerated by the use of more sophisticated descent methods such as conjugate gradient or quasi-Newton algorithms. There is the possibility of more than one local minimum, but in any case the method will lead to an improvement over the original design.

The adjoint method can be applied to a variety of measures of performance. It should be remembered, however, that gradient search methods depend on the assumption that the cost-function depends continuously on the design parameters. This can be violated, if, for example, on attempts to

calculate the sensitivity of the pressure at a fixed location, because there is the possibility that a shape modification could result in a shock moving over that location. The movement of the shock, however, is continuous as the shape changes, with the consequence that integrated quantities such as the drag coefficient also depend continuously on the shape. The adjoint equation allows the sensitivity of the drag coefficient without the explicit evaluation of pressure sensitivities.

In reference [16] the author derived the adjoint equations for transonic flows modelled by both the potential flow equation and the Euler equations. The theory was developed in terms of partial differential equations, leading to an adjoint partial differential equation. In order to obtain numerical solutions both the flow and the adjoint equations must be discretized. The control theory might be applied directly to the discrete flow equations which result from the numerical approximation of the flow equations by finite element, finite volume or finite difference procedures. This leads directly to a set of discrete adjoint equations with a matrix which is the transpose of the Jacobian matrix of the full set of discrete nonlinear flow equations. On a three-dimensional mesh with indices i, j, k the individual adjoint equations may be derived by collecting together all the terms multiplied by the variation $\delta w_{i,j,k}$ of the discrete flow variable $w_{i,j,k}$. The resulting discrete adjoint equations represent a possible discretization of the adjoint partial differential equation. If these equations are solved exactly they can provide the exact gradient of the cost function which results from the discretization of the flow equations, which is itself inexact. This may facilitate the asymptotic convergence of the search procedure. On the other hand any consistent discretization of the adjoint partial differential equation will yield the exact gradient in the limit as the mesh is refined.

There are a number of benefits to be gained from developing the theory for the partial differential equations of the flow. First, the true optimum shape belongs to an infinitely dimensional space of design parameters, and the theory provides an indication in principle of how such a solution could be approached if sufficient computational resources are available. Second, it provides insight into the nature of the adjoint equations, and the connection between the formulation of the cost function and the boundary conditions needed to assure a well posed problem. Third, in certain circumstances the discrete solution may lose the property of continuous dependence of the design parameters. It may, for example, contain non-differentiable flux limiters. Also, if adaptive mesh refinement is used, there will be a discontinuous change in the solution whenever a mesh point is added or deleted. Finally, the differential equation theory provides a guideline for the design of iterative solution methods for the adjoint equation, both in the case when the adjoint equation is separately discretized and in the case when the discrete adjoint equations are derived directly from the discrete flow equations. The

theory for standard multigrid methods, for example, depends on the property that the discrete equations on a sequence of meshes all represent the same differential equation. It turns out that the same multigrid solution method can readily be used for both the flow and the adjoint equation.

The adjoint method has recently been extended to treat the compressible Navier Stokes equations [20]. As an illustration of the power of the method Figures 8 and 9 illustrate the redesign of a wing representative of wide-body transport aircraft in current use. The redesign was performed by modifying the wing sections with a fixed planform, subject to the constraint that the thickness could not be reduced. Due to the high computational costs of viscous design, a two stage strategy was adopted. In the first stage, a design calculation was performed with the Euler equations on a mesh with $192 \times 32 \times 48$ cells to minimize the drag at a fixed lift coefficient. In the second stage the pressure distribution of the Euler solution was used as the target pressure for inverse design with the Navier Stokes equations, using a mesh with $192 \times 64 \times 48$ cells, including 32 intervals normal to the wing concentrated inside the boundary layer region. Comparatively small modifications were required in the second stage, so that could be accomplished with a small number of design cycles.

The design point was taken as a lift coefficient of .55 at a Mach number of .83. Figure 8 illustrates the Euler redesign, displaying both the geometry and the upper surface pressure distribution, with negative C_p upwards. The initial wing shows a moderately strong shock wave across most of the top surface, as can be seen in Figure 8a. 60 design cycles were needed to produce the shock free wing shown in Figure 8b, with an indicated drag reduction of 15 counts from .0196 to .0181. Figure 9 shows the viscous redesign at a Reynolds number of 12 million. In Figure 9a it can be seen that the Euler design produces a weak shock due to the displacement effects of the boundary layer. 10 design cycles were needed to recover the shock free wing shown in Figure 9b. It is interesting that the wing section modifications between the initial wing of Figure 8a and the final wing of Figure 9b are remarkably small.

5 INDUSTRIAL EXPERIENCE: A CASE STUDY

During the summer a group consisting of the author, J. Alonso, J. Reuther and L. Martinelli participated in design studies for the McDonnell Douglas MDXX. We interfaced with the project principally through J. Vassberg. The MDXX was a promising successor to the MD11. Despite significant airline interest, it was cancelled by the McDonnell Douglas Board in late October.

We were brought into the project to augment the Douglas design effort by applying advanced CFD and aerodynamic optimization techniques. These methods were used to evaluate attainable values of Mach number and L/D in cruise while satisfying other design constraints, including:

- Drag creep.
- Buffet (≥ 1.3 g to buffet from the maximum cruise CL).
- Maximum cruise CL.
- High speed pitch-up
- Suitability for high lift.
- Low speed characteristics
- Fuel volume.
- Wing weight.

In particular, the goals of the study were:

1. To prove the validity and feasibility of adjoint based design methods in the context of a real design environment.
2. To improve the existing DAC configuration which is recognized to be highly refined, by small modifications to extract maximum performance.
3. To independently design a family of optimized wings as an alternative to the DAC configuration.

These goals could provide DAC with options for alternative designs which might yield improvements in L/D , cruise Mach number, and thickness (for fuel volume and structural weight). They could also establish a bound on attainable limits which could be used as a yardstick to measure the DAC configuration and to determine whether or not there was room for significant improvement. From our side we also recognized that direct exposure to a project environment could give us the insight and awareness of practical requirements that could enable us to develop better software for future use.

The two design improvement criteria used in this study were:

1. Improvements in $M_\infty L/D$.
2. Reduction in weight: 3,300 lbs was estimated to be equivalent to 1 percentage point in $M_\infty L/D$.

The existing Douglas wing design was used as the baseline against which any improvements were to be measured.

Some of the key questions to be addressed were as follows:

- Could L/D be increased by either reducing the shock drag, varying the spanload, or improving the wing-body-engine integration? Possible improvements to L/D would yield a significant advantage from the point of view of fuel efficiency and aircraft range.

- Could the cruise Mach number be increased? This could produce both a reduction of airline operating costs and an increase in passenger comfort derived from shorter flight times.
- Could the wing thickness be increased without penalizing the current design? Several options are available in order to take advantage of a wing with increased thickness, among them one has: decreased structure weight for the same wing loading, increased fuel volume, optimized span loading for the same structural weight, or the possibility of installing larger winglets.
- Could the loading be moved forward to reduce trim drag and reduce hinge moments on control surfaces?
- Could the design be made less sensitive to small changes in CL, Mach number and Reynolds number?
- Would a shock-free design necessarily have undesirable off-design characteristics?
- Could the benefits of the divergent trailing edge technology developed by Douglas [15] be combined with optimization?
- What compromises are needed to assure satisfactory maneuver and buffet margins, as well as good high lift characteristics?
- Would there be any benefits in planform variations (sweep, taper)?
- Could capacity for stretch be built into the system? Would span extensions be necessary for this purpose?

In order to support the project we used a variety of computer programs for both flow analysis and aerodynamic design. Some of these tools were very recent, and were the subject of ongoing development during the study, as we tried to respond to the project requirements within the very short time available. Due to the cancellation of the MDXX the study was not brought to full fruition. Prior to the cancellation, there had been plans to carry out wind tunnel tests to evaluate an alternate wing designed by optimization in comparison with the Douglas baseline design. Nevertheless a number of valuable lessons were learned from the experience.

In the initial phase of the study we focused on the development of the optimization tools for isolated wings. Aside from difficulties with data handling, file conversation and observation of the same conventions as Douglas, for example, in the definition of reference quantities such as wing area, it proved necessary to modify the codes in various ways. In particular the visualization was greatly improved by incorporating an interface to Vassberg's COMPLOT program. The codes had to be modified to allow for thick trailing edges. It also

proved worthwhile to introduce terms measuring the pressure gradient into the cost function to prevent the appearance of the pressure gradients in the optimized designs from becoming unacceptably large in the rear upper surface. Access to offsite supercomputers was limited, and subject to serious delays due to the queues from many users. It was demonstrated, however, that optimizations could be completed overnight on work stations.

During the initial study three major issues soon became apparent:

1. The body effect was too large to be ignored and must be included for the optimizations to be useful.
2. Supercritical wings of the type contemplated for the MDXX are sensitive to viscous effects, which should also be included in the optimization.
3. Single point designs could be too sensitive to small variations in the flight condition.

Therefore in the second phase of the study we concentrated in the optimization of wing-body combinations, proceeded to 3 point optimizations, and carried out optimizations with the Reynolds averaged Navier Stokes equations. Only a preliminary version of a viscous design code was available, and it had to be pressed into action. In order to enable quick turn around the strategy was adopted of first carrying out a 3 point wing body optimization with SYN 88, which models the flow with the Euler equations. This could be accomplished in about $2\frac{1}{2}$ days on a C-H mesh with $256 \times 32 \times 48$ cells using a workstation. The preliminary Euler design was then fed to SYN 107 for Navier Stokes redesign, using the pressure distribution of the Euler result at the principal design point as a target, with the constraint that the thickness could be increased but not reduced. The inverse mode was preferred because of doubts about the accuracy of viscous drag prediction. The Navier Stokes calculations are much more expensive, requiring a mesh with at least twice as many points, and 5 - 10 times as many iterations at each design cycles. Usually a fairly close approximation to the Euler target pressure could be obtained in 10 - 20 design cycles. This could be accomplished in about 3 days on a work station.

The Douglas design team were using the OVERFLOW program, originally developed by P. Buning, for Navier Stokes analysis. This could treat complete configurations if enough time was taken to generate the required overlapping meshes over all the components. The use of overlapping meshes also facilitates the concentration of mesh points to resolve the viscous regions, and results obtained with OVERFLOW had been validated against wind tunnel data obtained from tests of earlier Douglas designs. However, each OVERFLOW run required about 25 hours of CPU time on a Cray C90, and had to be broken up into 6 hour shifts on separate nights. A proper evaluation of the designs emerging from

the optimization would require analyses at numerous points through the flight envelope, including a series of points to establish the drag rise characteristics. It was clear that this was impossible in the time available. In fact it was impossible even with the Douglas baseline design to achieve turn around times compatible with the attempt to complete the multidisciplinary design loop in 2 weeks.

In order to alleviate the situation we accelerated the development of a parallel implementation of our own multi-block analysis program FLO107MB, which solves the full Reynolds averaged Navier Stokes (RANS) equations. With this we were able to complete a RANS analysis in a mesh with 1.5 - 2 million mesh cells in about $1\frac{1}{2}$ hours using 32 processors of an IBM SP2. This enabled us to evaluate the performance of 4 different candidate designs over the flight envelope with the aid of 60 Navier Stokes calculations during the weekend September 6-8. The results allowed us to eliminate one design. We also learned that the wings were carrying too much outboard load near the buffet point, and would be susceptible to shock stall near the tip. This led us to increase the wing twist to reduce the angle of attack of the tip.

It also became apparent that there were discrepancies between the results of the RANS design calculations, which had been performed on coarser meshes with about 600000 mesh cells, and the analysis on finer meshes. Therefore we also implemented a parallel version of the single block design code SYN107, which enabled us to carry out RANS designs on meshes with 1.8 million mesh points. 20 design cycles were usually found to be sufficient, and these could be completed in a run of about $7\frac{1}{2}$ hours using 48 processors of the IBM SP2.

As an illustration of the results that could be obtained, Figures 10 - 15 show an alternate wing-body design with increased sweep back of about 38 degrees at the $1/4$ chord. Starting from the result of an Euler design, the RANS optimization produced an essentially shock free wing at a cruise design point of Mach .86, with a lift coefficient of .6 for the wing body combination. Figure 10 shows the design point, while the evolution of the design is shown in Figure 11, using John Vassberg's COMPPLOT. In this case the pressure contours are for the final design. This wing is quite thick, actually thicker than the Douglas baseline design across the span, with a thickness to chord ratio of more than 14 percent at the root and 9 percent at the tip. The design offers excellent performance at the nominal cruise point. Figures 12 and 13 show the results of a Mach number sweep to determine the drag rise. It can be seen that a double shock pattern forms below the design point, while there is actually a slight increase in the drag coefficient of about $1\frac{1}{2}$ counts at Mach .85. The drag is still low, however, and the double shocks remain quite weak. Figure 15 shows a comparison of the design point with alternate cruise points at lower and

higher lift. Finally, Figure 14 indicates that the pressure distribution at the buffet point is acceptable. Provided that the high speed pitch-up associated with the high sweepback angle is controllable, this is a promising candidate design. It is a subject of ongoing research whether the sensitivity near the design point could be reduced by forcing the presence of a shock at the design point.

One difficulty of the study was that there were discrepancies between the predictions of OVERFLOW and FLO107MB. These can be attributed to a combination of mesh effects, turbulence modelling, and differences in the discretization scheme. FLO107MB was normally run with the CUSP scheme [18] which we considered to be more accurate. We were able to verify this by mesh refinement studies in which the CUSP solution on a mesh with 1 million mesh points was found to approach closely the solution with the standard scalar dissipation [21] on 2 million mesh points. A weakness of the present implementation of FLO107MB is its use of the Baldwin Lomax turbulence model. This model is generally considered to be reasonably accurate for attached flows in the neighborhood of the cruise point, but unsuitable for the prediction of separated flows. In future work it is planned to provide options for a variety of turbulence models.

The prediction of fuselage drag was another source of difficulty. The pressure drag on the fuselage can be quite large, of the order of 40 counts, because the fuselage contributes about 15 percent of the lift, and the downwash distribution of a swept wing causes a transfer of the induced drag to the inboard part of the wing, while the tip region experiences a thrust. With the C-H mesh used in the wing-body design codes the fuselage pressure drag was drastically over predicted. In the drag optimization studies only the wing drag was included. This leads to the possibility that the optimization might transfer drag from the wing to the fuselage. The multiblock analysis calculations indicated that drag savings on the wing were partially offset by an increase in fuselage drag, but further studies are needed to clarify this issue.

The final phase of the study, which was truncated by the cancellation of the MDXX, addressed the performance of the wing-body combination with engines and winglets included. Using GRIDGEN several weeks were needed to generate a mesh with 234 blocks and more than 5 million mesh cells. RANS calculations could then be performed in 5 or 6 hours with 48 processors of an IBM SP2. An example of such a calculation is presented in Figure 16 and 17, in which the shading indicates the surface pressure, with darker shading corresponding to higher pressure. The overall turn around for mesh generation and flow analysis is still too slow. A multiblock optimization code in which the flow is modelled by the Euler equations is already operational. A multi block viscous design code is clearly needed and we plan to undertake its development. In the long run unstructured meshes may be needed to treat complete configurations with rapid

turn around.

Two major lessons of the studies were:

1. Useful simulations in the design of a wing for a commercial transport must treat at least wing-fuselage combinations and include viscous effects: more complete simulations ought to treat the engines, and also winglets if they are featured in the design.
2. To be fully accepted by the design team both CFD and optimization methods need to be validated before their use in the project.

6 THE OPPORTUNITY TO REENGINEER THE DESIGN PROCESS

In the long run, computational simulation should become the principal tool for aerodynamic design because of the flexibility it provides for the rapid and comparatively inexpensive evaluation of alternative designs, and because it can be integrated with a multi-disciplinary design process. To be effective in this role high fidelity aerodynamic simulation needs to be used at an early stage in the process, when it can be used to make crucial trade-off decisions before the principal features of the design have been frozen.

Presently available computer programs for design integration incorporate only crude and simplified aerodynamic models such as vortex lattice methods. Long set up and turn-around times continue to restrict the use of high fidelity simulation methods. The opportunity now exists to take advantage of developments in information technology to completely re-organize the design process. The basic flow simulation software is only one of the needed ingredients. The flow solver must be embedded in a user-friendly system for geometry modelling, output analysis and data management that will provide a complete numerical design environment. The objective should be to provide fast, cost-effective computational analysis and optimization capabilities at an early stage in the design cycle. Some of the principal goals are as follows:

1. Analysis turn-around of less than an hour for a full aircraft configuration
2. Geometry manipulation via a computer aided design (CAD) system with access to a central data base
3. Automated optimization of the design
4. Multidisciplinary analysis

An important element in an integrated system is the need for good geometry modelling. When computational simulations were first attempted, geometric definitions of aircraft were still generally provided by drawings. This made it very difficult to obtain an adequate digital description of

the configuration. Modern developments in computational geometry, such as Bezier patches and non-uniform rational B-splines (NURBS) have made it possible to provide a complete and precise digital definition of the geometry. In principle this should allow manufacturing engineers, structural designers, and aerodynamicists to access the same unique definition of the design in a central database. The definitions provided by current computer-aided design (CAD) systems are intended to meet appropriate manufacturing tolerances. Unfortunately they sometimes prove to be inadequate for aerodynamic analysis because they do not always guarantee a sufficient degree of smoothness across boundaries of geometric patches. Moreover, accurate viscous simulations require mesh points to be placed extremely close to the surface to resolve the inner part of the boundary layer. The first mesh point should be at a distance of the order of $y^+ = 1$, where y^+ is a dimensionless coordinate based on the viscous length scale, and this distance may be smaller than the usual manufacturing tolerances.

Given an adequate geometric model, it becomes crucially important to compress the time spent in mesh generation. There has been rapid progress in methods based on overlaying separately generated meshes for different components, including the development of software which automatically calculates the coefficients needed to transfer data between the meshes. It seems difficult, however, to automate the choice of the component meshes, though expert systems might prove useful for this purpose. At the same time, the use of unstructured meshes is becoming an increasingly attractive option through the emergence of improved triangulation techniques, which are both fast and can also assure the satisfaction of various criteria of mesh quality. These methods should make it possible to completely automate the mesh generation process, removing one of the principal bottlenecks of current flow simulation systems. Ultimately one can anticipate the use of hybrid meshes which allow the use of beneficial combination of hexahedral, prismatic and tetrahedral cells in the same simulation.

Turn-around times even of viscous simulations for complex configurations can now be reduced to at most a few hours through the emergence of stable and reliable parallel computing systems, together with the software needed to support compilation, message passing and memory management. In addition to the availability of powerful central servers with hundreds of processors, it is now possible to link large numbers of workstations through fast networks to operate in groups as parallel computers. This allows the more effective utilization of workstations which might otherwise be idle at night, and can significantly increase the available computational resources at a moderate cost. The costs of software conversion for parallel use can be very large, however. Therefore it is crucially important to establish uniform standards for parallel software, and the new message passing

interface (MPI) protocol is rapidly gaining acceptance.

Advanced geometry modelling, automatic mesh generation and parallel computing provide the basic building blocks for an integrated system. Figures 6 and 7 illustrate the way in which a numerical wind tunnel might evolve from current techniques to a fully integrated numerical design environment. Figure 6 is representative of current practice, both at the NASA Ames National Aerodynamic Simulation (NAS) system, and at the Japanese National Aeronautical Laboratory's Numerical Wind Tunnel (NWT). The massive data handling activities in the left hand box are human intensive. In the advanced system shown in Figure 7 many of these tasks have been automated and transferred to the right hand box of numerically intensive activities. An advanced system

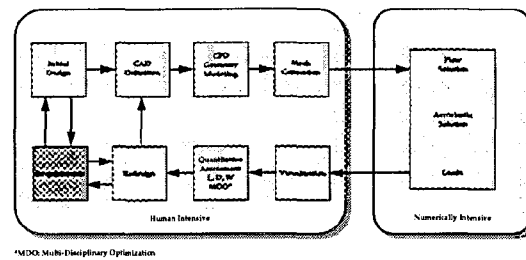


Figure 6: Concept for a numerical wind tunnel.

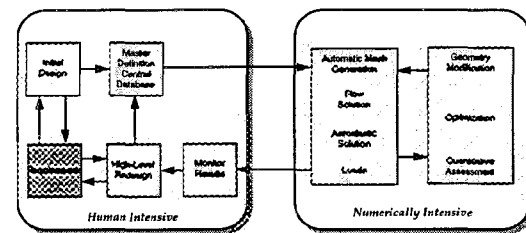


Figure 7: Advanced numerical wind tunnel.

should also eventually provide for optimization and multi-disciplinary analysis. Adjoint methods based on control theory can provide the capability of aerodynamic shape optimization with quite moderate computational costs. Effective design optimization must, however, account for trade-offs in the complete system. The design of a wing, for example must include trade-offs between drag, structure weight, fuel volume, take-off and landing field lengths, the need to retract the undercarriage, and gate width restrictions. Multi-disciplinary analysis requires as a first step the linkage of the relevant disciplines through a uniform integrated database. This will provide the basis for a tighter coupling where it is needed. For example, integrated aerodynamic and structural analysis would allow engineers to take early account of

aeroelastic effects, such as structural deflection under aerodynamic load which vary as the weight changes with fuel burn-off.

Proceeding beyond multi-disciplinary analysis, a future goal is the development of effective tools for multi-disciplinary optimization (MDO). This subject is becoming the focus of extensive research. The results of numerical optimizations which use low fidelity models of the different disciplines should be treated with caution, as they may be quite misleading. In order to establish confidence in the conclusions it is therefore important to determine the sensitivity of the results to modelling inaccuracies.

7 CONCLUSIONS

The basic techniques of computational flow simulation are now quite well established. Simulation techniques can both compress the design process, and enhance it by allowing engineers to explore a larger range of options. This potential can only be fully realized by improved integration of the whole process from geometry definition to analysis of the final output. Developments in a broad spectrum of information technology including computer aided design, geometry modelling database control, and parallel computing can all contribute to the re-engineering of the design process.

Companies can become more competitive through reduced cycle times and improved products. In the long run the integration of computational simulation with information technology may also have a significant social impact on the engineering profession. Historically many engineers were employed on large projects to perform routine calculations. Engineers of the future, relieved of the burden of computing, will be able to spend more of their time in creative thinking to find innovative solutions to problems, and to produce new design concepts.

As the power of microprocessors continues to increase, it will become possible to perform most engineering simulations on inexpensive workstations, and comparatively small groups will be able to afford competitive computational resources. This may facilitate the emergence of small independent groups to take over specialized design tasks. It is already a common practice, for example, for independent studios to design automobile bodies for the major manufacturers. Thus in the future an increasing number of engineers may fill a role more like that played today by architects and consulting engineers in the construction industry.

ACKNOWLEDGEMENTS

The research described in this paper has benefited greatly from the sponsorship of the AFOSR under Grant Number AFOSR F49620-95-1-0259, monitored by Major Scott Schreck. In the development of parallel implementations we have been very fortunate to have the support of IBM

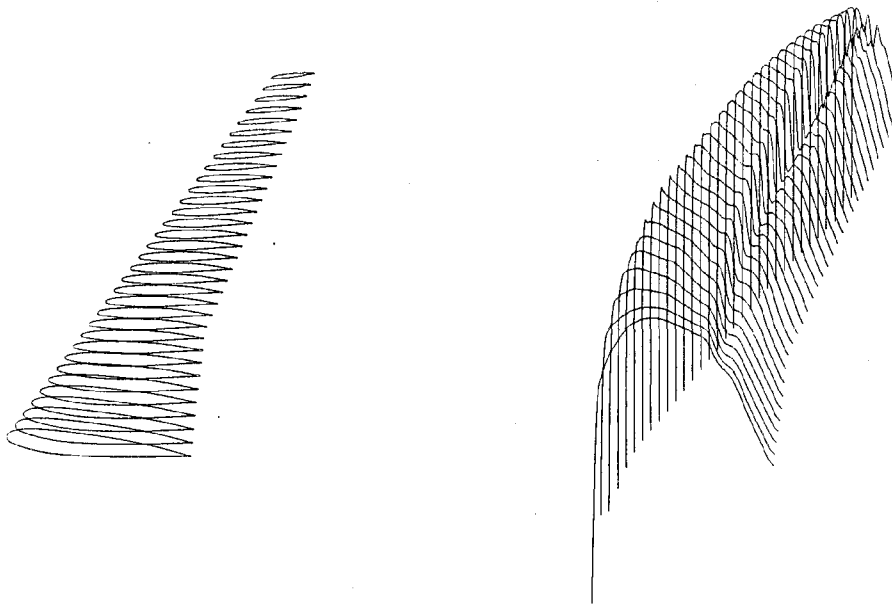
and NASA through the Cooperative Research Agreement. We are also very grateful for the support of the McDonnell-Douglas Corporation and the opportunity to experience at first hand the realities of an industrial project.

REFERENCES

- [1] R. Abid, C.G. Speziale, and S. Thangam. Application of a new $k-\tau$ model to near wall turbulent flows. AIAA Paper 91-0614, AIAA 29th Aerospace Sciences Meeting, Reno, NV, January 1991.
- [2] B. Baldwin and H. Lomax. Thin layer approximation and algebraic model for separated turbulent flow. AIAA Paper 78-257, 1978.
- [3] B.S. Baldwin and T.J. Barth. A one-equation turbulence transport model for high Reynolds number wall-bounded flows. AIAA Paper 91-0610, AIAA 29th Aerospace Sciences Meeting, Reno, NV, January 1991.
- [4] T. J. Barth. Aspects of unstructured grids and finite volume solvers for the Euler and Navier Stokes equations. In *von Karman Institute for Fluid Dynamics Lecture Series Notes 1994-05*, Brussels, 1994.
- [5] J.A. Benek, P.G. Buning, and J.L. Steger. A 3-D Chimera grid embedding technique. In *Proceedings AIAA 7th Computational Fluid Dynamics Conference*, pages 507-512, Cincinnati, OH, 1985. AIAA Paper 85-1523.
- [6] J.A. Benek, T.L. Donegan, and N.E. Suhs. Extended Chimera grid embedding scheme with applications to viscous flows. AIAA Paper 87-1126, AIAA 8th Computational Fluid Dynamics Conference, Honolulu, HI, 1987.
- [7] M. Berger and R.J. LeVeque. An adaptive Cartesian mesh algorithm for the Euler equations in arbitrary geometries. AIAA Paper 89-1930, 1989.
- [8] T. Cebeci and A.M.O. Smith. *Analysis of Turbulent Boundary Layers*. Academic Press, 1974.
- [9] D.R. Chapman, H. Mark, and M.W. Pirtle. Computers vs. wind tunnels in aerodynamic flow simulations. *Astronautics and Aeronautics*, 13(4):22-30, 35, 1975.
- [10] T.J. Coakley. Numerical simulation of viscous transonic airfoil flows. AIAA Paper 87-0416, AIAA 25th Aerospace Sciences Meeting, Reno, January 1987.
- [11] D. Deganiand and L. Schiff. Computation of turbulent supersonic flows around pointed bodies having cross-flow separation. *J. Comp. Phys.*, 66:173-196, 1986.

- [12] B. Delaunay. Sur la sphere vide. *Bull. Acad. Science USSR VII: Class Scil, Mat. Nat.*, pages 793–800, 1934.
- [13] M. Van Dyke. *An Album of Fluid Motion*. The Parabolic Press, Stanford, 1982.
- [14] M.H. Ha. The impact of turbulence modelling on the numerical prediction of flows. In M. Napolitano and F. Solbetta, editors, *Proc. of the 13th International Conference on Numerical Methods in Fluid Dynamics*, pages 27–46, Rome, Italy, July 1992. Springer Verlag, 1993.
- [15] P.A. Henne and R. D. Gregg. A new airfoil design concept. AIAA Paper 89-2201, AIAA 7th Applied Aerodynamics Conference, Seattle, Wa, July 1989.
- [16] A. Jameson. Aerodynamic design via control theory. *J. Sci. Comp.*, 3:233–260, 1988.
- [17] A. Jameson. The present status, challenges, and future developments in Computational Fluid Dynamics. Technical report, 77th AGARD Fluid Dynamics Panel Symposium, Seville, October 1995.
- [18] A. Jameson. Analysis and design of numerical schemes for gas dynamics 2, artificial diffusion and discrete shock structure. *Int. J. of Comp. Fluid Dyn.*, To Appear.
- [19] A. Jameson and L. Martinelli. Mesh refinement and modelling errors in flow simulation. *AIAA paper 96-2050*, AIAA 27th Fluid Dynamics Conference, New Orleans, June 1996.
- [20] A. Jameson, N. Pierce, and L. Martinelli. Optimum aerodynamic design using the Navier-Stokes equations. *AIAA paper 97-0101*, January 1997.
- [21] A. Jameson, W. Schmidt, and E. Turkel. Numerical solutions of the Euler equations by finite volume methods with Runge-Kutta time stepping schemes. *AIAA paper 81-1259*, January 1981.
- [22] D. Johnson and L. King. A mathematically simple turbulence closure model for attached and separated turbulent boundary layers. *AIAA Journal*, 23:1684–1692, 1985.
- [23] W.P. Jones and B.E. Launder. The calculation of low-Reynolds-number phenomena with a two-equation model of turbulence. *Int. J. of Heat Tran.*, 16:1119–1130, 1973.
- [24] W.H. Jou. Boeing Memorandum AERO-B113B-L92-018, September 1992. To Joseph Shang.
- [25] T.J. Kao, T.Y. Su, and N.J. Yu. Navier-Stokes calculations for transport wing-body configurations with nacelles and struts. AIAA Paper 93-2945, AIAA 24th Fluid Dynamics Conference, Orlando, July 1993.
- [26] A.M. Landsberg, J.P. Boris, W. Sandberg, and T.R. Young. Naval ship superstructure design: Complex three-dimensional flows using an efficient, parallel method. *High Performance Computing 1993: Grand Challenges in Computer Simulation*, 1993.
- [27] J.L. Lions. *Optimal Control of Systems Governed by Partial Differential Equations*. Springer-Verlag, New York, 1971. Translated by S.K. Mitter.
- [28] F. Liu and X. Zheng. A strongly coupled time-marching method for solving the Navier-Stokes and $\kappa - \omega$ turbulence model equations with multigrid. *J. Comp. Phys.*, 128:289–300, 1996.
- [29] R. Lohner and P. Parikh. Generation of three-dimensional unstructured grids by the advancing front method. AIAA Paper 88-0515, Reno, NV, January 1988.
- [30] L. Martinelli and A. Jameson. Validation of a multigrid method for the Reynolds averaged equations. *AIAA paper 88-0414*, 1988.
- [31] L. Martinelli, A. Jameson, and E. Malfa. Numerical simulation of three-dimensional vortex flows over delta wing configurations. In M. Napolitano and F. Solbetta, editors, *Proc. 13th International Conference on Numerical Methods in Fluid Dynamics*, pages 534–538, Rome, Italy, July 1992. Springer Verlag, 1993.
- [32] J.E. Melton, S.A. Pandya, and J.L. Steger. 3D Euler flow solutions using unstructured Cartesian and prismatic grids. AIAA Paper 93-0331, Reno, NV, January 1993.
- [33] F. Menter. Zonal two-equation $k-\omega$ turbulence models for aerodynamic flows. AIAA Paper 93-2906, AIAA 24th Fluid Dynamics Meeting, Orlando, July 1993.
- [34] J.T. Oden, L. Demkowicz, T. Liszka, and W. Rachowicz. $h-p$ adaptive finite element methods for compressible and incompressible flows. In S. L. Venneri A. K. Noor, editor, *Proceedings of the Symposium on Computational Technology on Flight Vehicles*, pages 523–534, Washington, D.C., November 1990. Pergamon.
- [35] C.L. Rumsey and V.N. Vatsa. A comparison of the predictive capabilities of several turbulence models using upwind and centered - difference computer codes. AIAA Paper 93-0192, AIAA 31st Aerospace Sciences Meeting, Reno, January 1993.

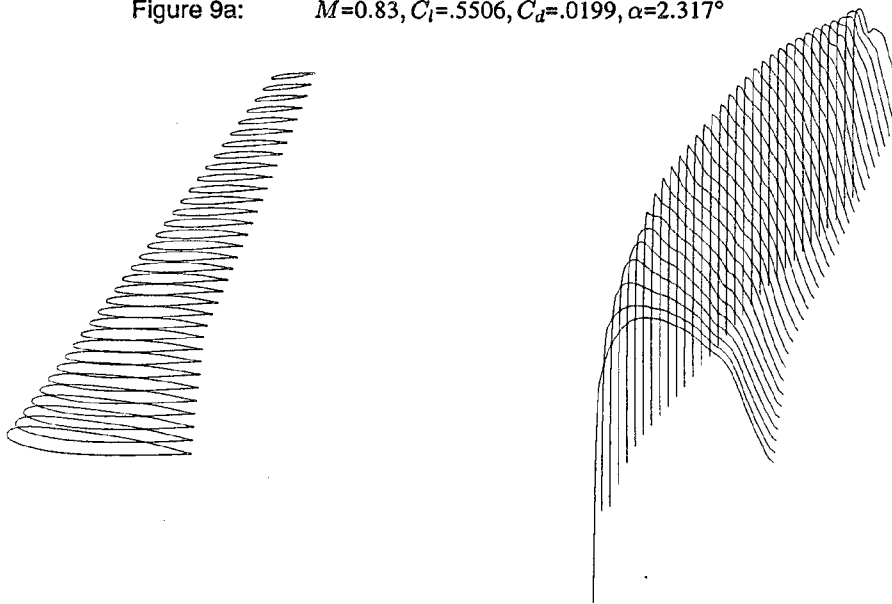
- [36] S.S. Samant, J.E. Bussoletti, F.T. Johnson, R.H. Burkhart, B.L. Everson, R.G. Melvin, D.P. Young, L.L. Erickson, and M.D. Madson. TRANAIR: A computer code for transonic analyses of arbitrary configurations. AIAA Paper 87-0034, 1987.
- [37] K. Sawada and S. Takanashi. A numerical investigation on wing/nacelle interferences of USB configuration. In *Proceedings AIAA 25th Aerospace Sciences Meeting*, Reno, NV, 1987. AIAA paper 87-0455.
- [38] B. R. Smith. A near wall model for the $k - l$ two equation turbulence model. *AIAA paper 94-2386*, 25th AIAA Fluid Dynamics Conference, Colorado Springs, CO, June 1994.
- [39] P. Spalart and S. Allmaras. A one-equation turbulent model for aerodynamic flows. AIAA Paper 92-0439, AIAA 30th Aerospace Sciences Meeting, Reno, NV, January 1992.
- [40] C.G. Speziale, E.C. Anderson, and R. Abid. A critical evaluation of two-equation models for near wall turbulence. AIAA Paper 90-1481, June 1990. ICASE Report 90-46.
- [41] G. Voronoi. Nouvelles applications des parametres continus a la theorie des formes quadratiques. Deuxieme memoire: Recherches sur les paralleloedres primitifs. *J. Reine Angew. Math.*, 134:198-287, 1908.
- [42] N.P. Weatherill and C.A. Forsey. Grid generation and flow calculations for aircraft geometries. *J. Aircraft*, 22:855-860, 1985.
- [43] D.C. Wilcox. A half a century historical review of the $k - \omega$ model. AIAA Paper 91-0615, AIAA 29th Aerospace Sciences Meeting, Reno, NV, January 1991.



Initial Wing.

C_p on Upper Surface.

Figure 9a: $M=0.83, C_l=.5506, C_d=.0199, \alpha=2.317^\circ$

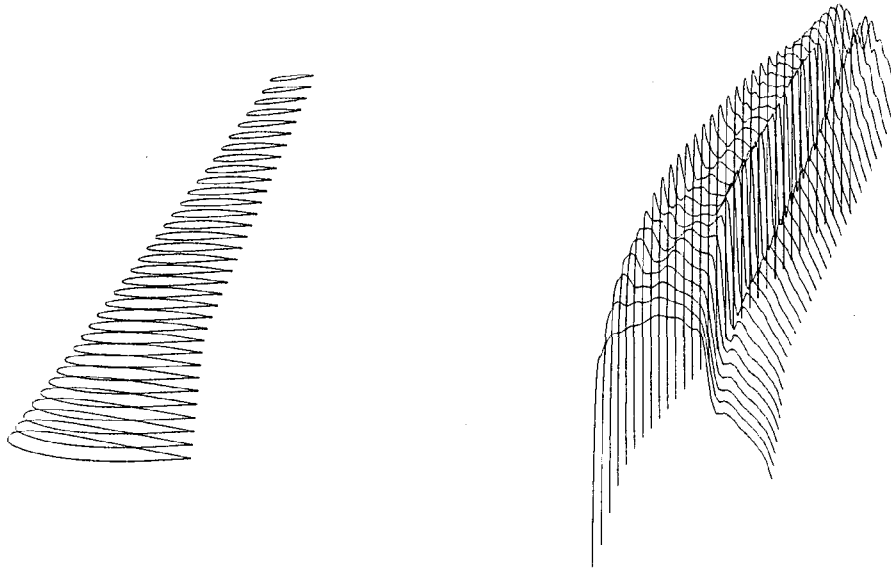


Redesigned wing.

C_p on Upper Surface.

Figure 9b: $M=0.83, C_l=.5508, C_d=.0194, \alpha=2.355^\circ$

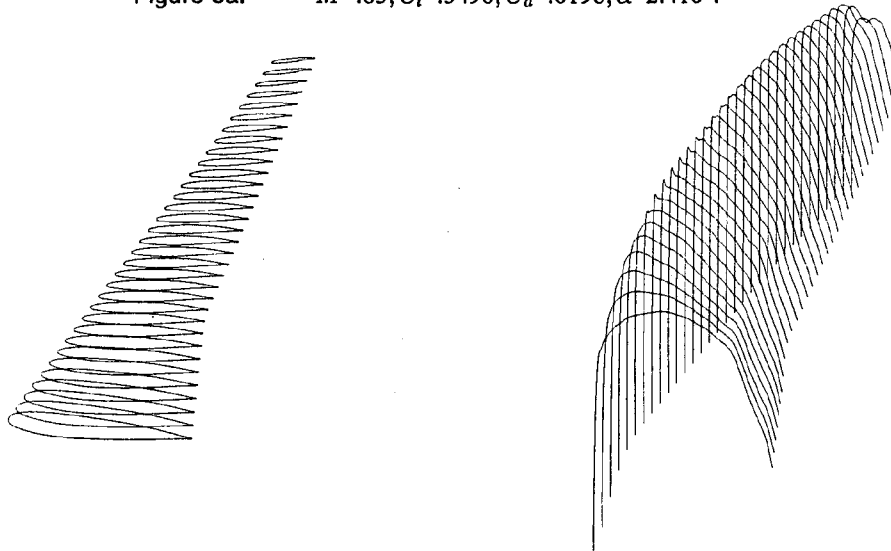
Figure 9: Redesign of the wing of a wide transport aircraft. Stage 2: Viscous re-design. 10 design cycles in inverse mode.



Initial Wing.

C_p on Upper Surface.

Figure 8a: $M=.83, C_l=.5498, C_d=.0196, \alpha=2.410^\circ$.



Redesigned wing.

C_p on Upper Surface.

Figure 8b: $M=.83, C_l=.5500, C_d=.0181, \alpha=1.959^\circ$.

Figure 8: Redesign of the wing of a wide transport aircraft. Stage 1 Inviscid design : 60 design cycles in drag reduction mode with forced lift.

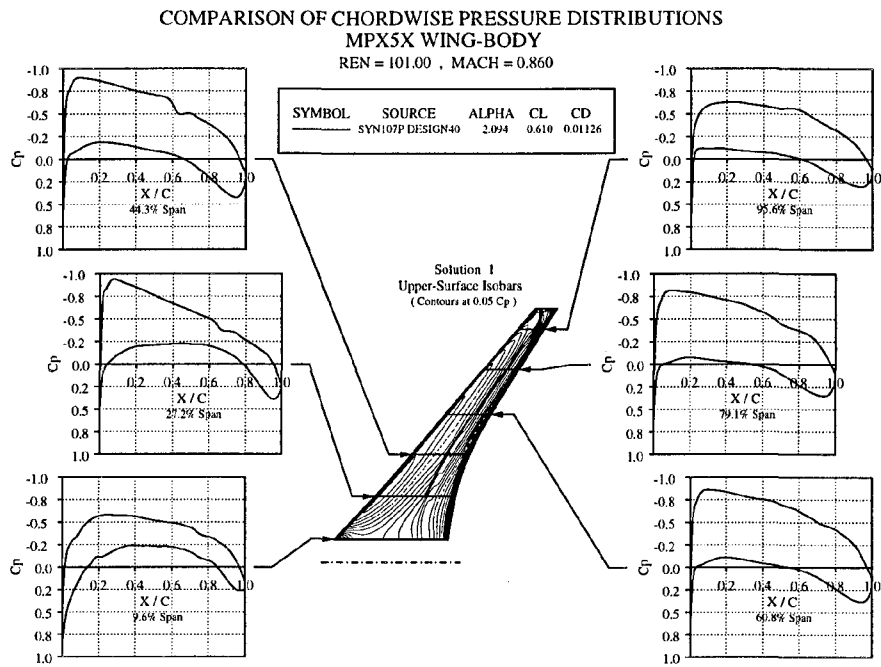


Figure 10: Pressure distribution of the MPX5X at its design point.

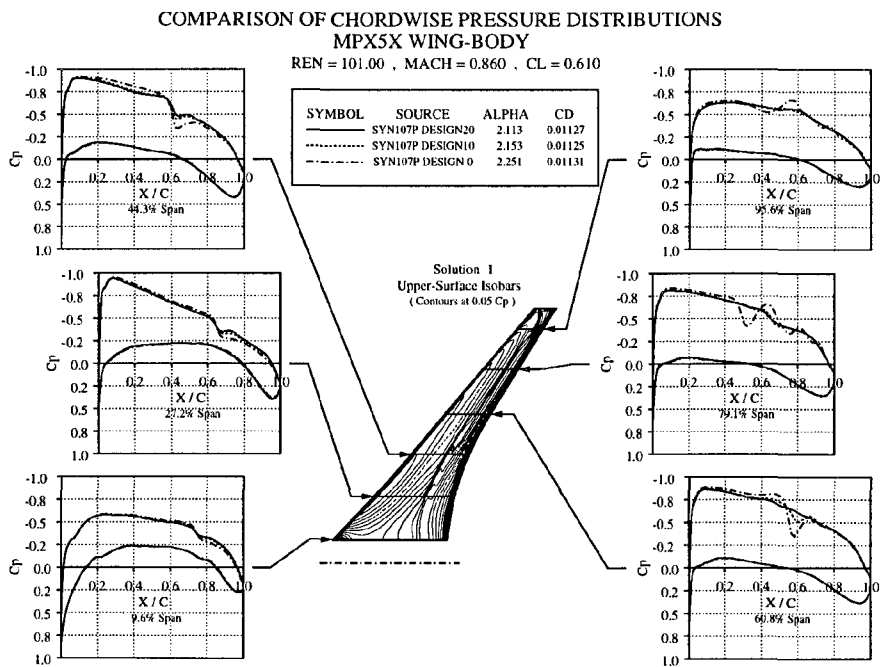


Figure 11: Optimization Sequence in the design of the MPX5X.

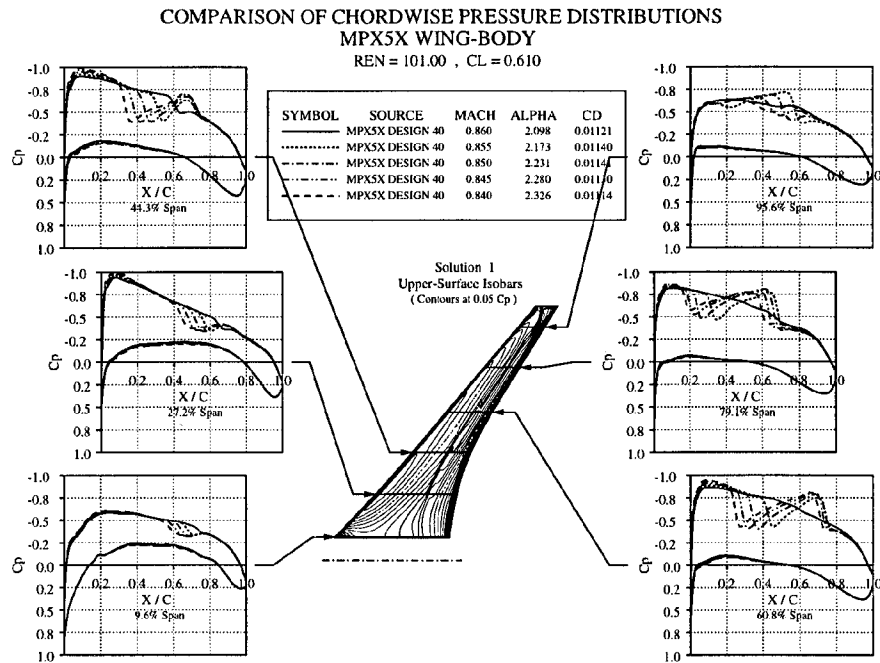


Figure 12: Off design performance of the MPX5X below the design point.

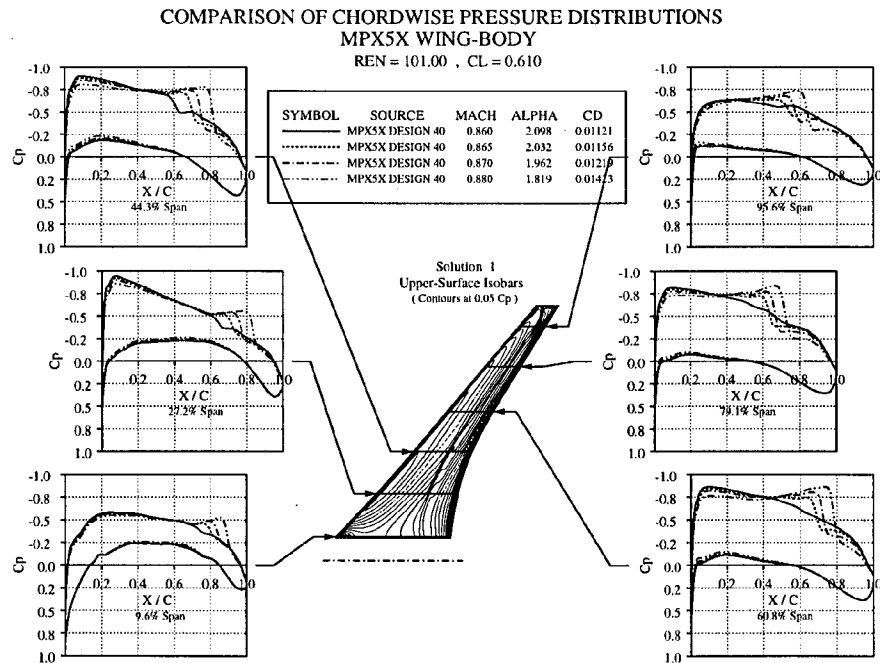


Figure 13: Off design performance of the MPX5X above the design point.

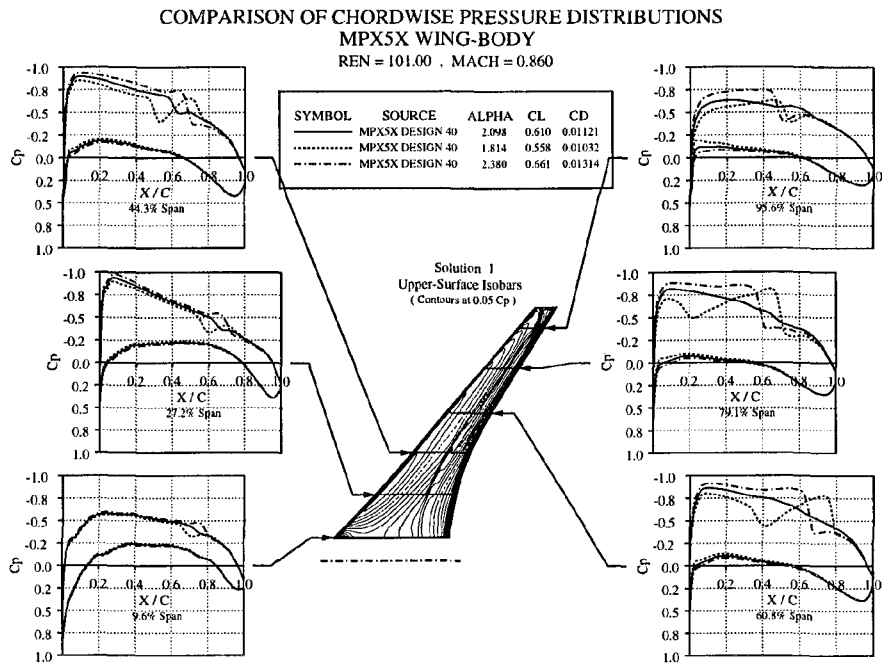


Figure 14: Comparison of the MPX5X at its design point and at lower and higher lift.

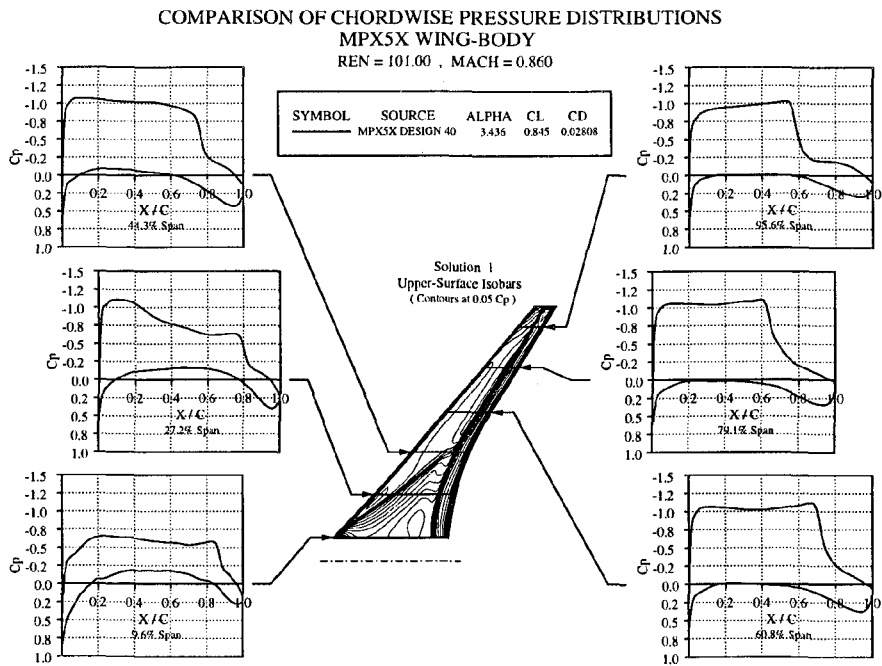


Figure 15: Off design performance of the MPX5X at the buffet point.



Figure 16: MPX3R wing-body-nacelle- winglet combination at Mach .85, CL = .6 . View from below.

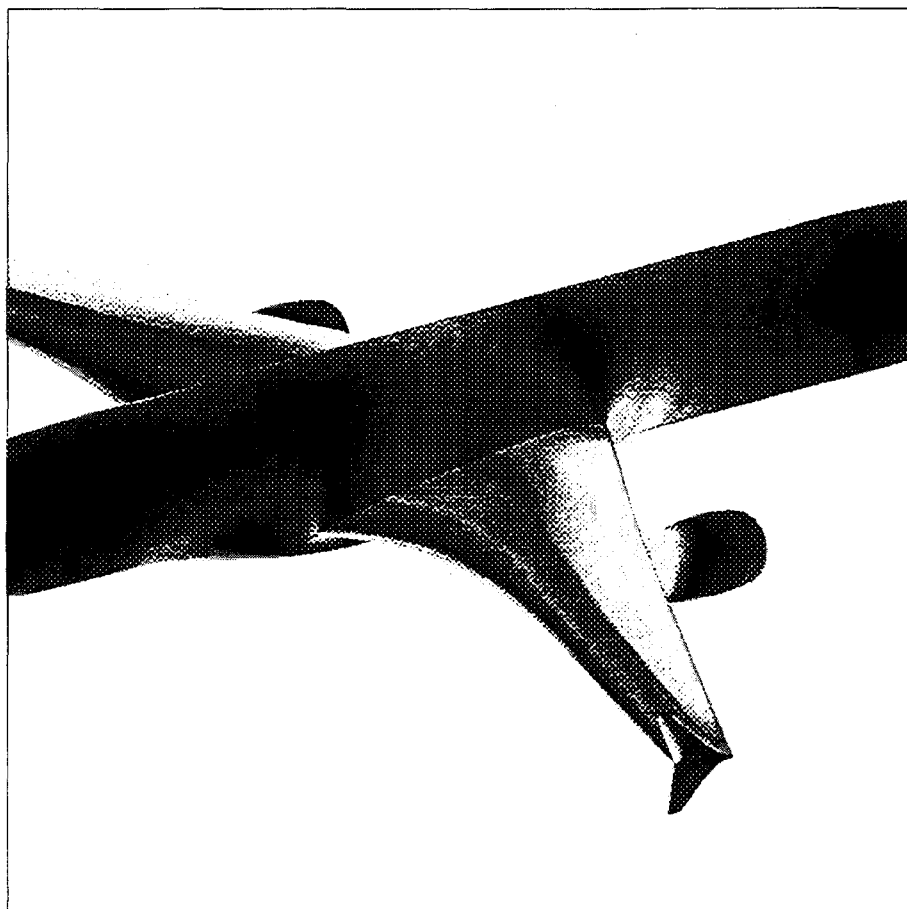


Figure 17: MPX3R wing-body-nacelle-winglet combination at Mach .85, $CL = .6$. View from above.