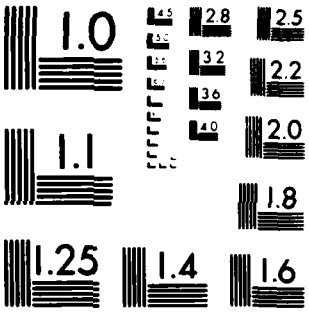
AD-A144 323 APPLICATION OF COMPUTATIONAL FLUID DYNAMICS IN AIRCRAFT 1/1
DESIGN(U) FOREIGN TECHNOLOGY DIV WRIGHT-PATTERSON AFB
OH 19 JUL 84 FTD-ID(RS)T-0741-84

UNCLASSIFIED F/G 20/4 NL

END
DATE
FILMED
9 84
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963 A

FTD-ID(RS)T-0741-84

AD-A144 323

FOREIGN TECHNOLOGY DIVISION

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS
IN AIRCRAFT DESIGN

DTIC FILE COPY

DTIC
ELECTE
AUG 15 1984
S D

Approved for public release;
distribution unlimited.

84 08 09 119

Accession For
NTIS GRA&I ☒
DTIC TAB ☐
Unannounced ☐
Justification
By
Distribution/
Availability Codes
Dist | Avail and/or Special
A/1

DTIC COPY INSPECTED 2

FTD-ID(RS)T-0741-84

EDITED TRANSLATION

FTD-ID(RS)T-0741-84 19 July 1984

MICROFICHE NR: FTD-84-C-000716

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS
IN AIRCRAFT DESIGN

English pages: 11

Source: Guoji Hanghong, Nr. 3(253), March 1984, pp. 6-8

Country of origin: China
Translated by: SCITRAN
F33657-81-D-0263
Requester: FTD/TQTA
Approved for public release; distribution unlimited.

THIS TRANSLATION IS A RENDITION OF THE ORIGINAL FOREIGN TEXT WITHOUT ANY ANALYTICAL OR EDITORIAL COMMENT. STATEMENTS OR THEORIES ADVOCATED OR IMPLIED ARE THOSE OF THE SOURCE AND DO NOT NECESSARILY REFLECT THE POSITION OR OPINION OF THE FOREIGN TECHNOLOGY DIVISION.

PREPARED BY:

TRANSLATION DIVISION
FOREIGN TECHNOLOGY DIVISION
WP.AFB, OHIO.

GRAPHICS DISCLAIMER

All figures, graphics, tables, equations, etc. merged into this translation were extracted from the best quality copy available.

Application of Computational Fluid Dynamics in Aircraft Design /6

Author Unknown

In the past, the only tool in the aerodynamic design of an aircraft was the wind tunnel. Aerodynamic engineers have been using wind tunnels for 50 years and accumulated a lot of experience. It is very direct to use a wind tunnel to measure the resultant force and moment of force on the model. However, many problems, such as Reynolds number, M number, wall interference, and support interference, exist. Furthermore, there are difficulties in understanding the details in the air flow. The most significant drawback of the wind tunnel is that it can only be used as a verification but not in designing. The appearance will have to be repeatedly modified and it takes very long to verify a design in a wind tunnel. The cost of experimentation in the wind tunnel is also rising continuously. These factors limit the aircraft designers to obtain one feasible design within a given period with limited funding.

Computational Fluid Dynamics (CFD) was rapidly developed since the late sixties and significant progress was made in the seventies. Presently, considerably reliable results have been obtained in the subsonic, transonic, and supersonic bound flow. Especially in the design of transport aircrafts, the main objective is to improve the aerodynamic efficiency, i.e., to raise the lift to drag ratio L/D. The design of the aircraft provides that the air flow is bound when the aircraft is cruising. Therefore, computational fluid dynamics gradually becomes an important design tool and the function of the wind tunnel is downgraded for verification.

Many detailed data in the flow field can be obtained by using computational fluid dynamics, including pressure distribution, position and intensity of shock wave, streamline of the air flow, and the boundary layer which can be used as a guide to improve the design. Some computer programs even have the capability to design backwards, i.e., to find the required aerodynamic appearance for certain given flow field

characteristics. The time and cost of computation are rapidly decreasing to allow the designers to have a powerful means to deliver designs with the optimal aerodynamic characteristics.

One important characteristic in designing fighters is to have multiple design points. At certain critical design points, unbound vortex, shock wave, and separation are significant in the air flow. For example, in the development of the F-16, 12,000 hours of wind tunnel were performed in a ten year period, of which only 15% was used to study bound flow. Therefore, computational fluid dynamics is only a supplement to wind tunnel experiments in designing fighters. Despite this fact, computational fluid dynamics has a significant effect on reducing the cost of developing high performance aircraft. This method still has a high potential for further development. It will be more important in the future.

The Navier-Stokes equation (N-S equation) for aerodynamic motions was first published in 1823. It can be used to solve many details of the flow, including the transition from laminar to turbulent flow. However, it is a set of non-linear second order partial differential equations. When the Cartesian coordinate system is used, it includes 60 partial derivatives. Its full solution may require a network with 1,000,000,000,000 points, which needs a computer with 3,000,000 times the capacity of the largest computer today. Therefore, some approximations are used. There are three classes, i.e., the surface element method, finite difference method and Navier-Stokes method.

1. Surface Element Method

The basic approximation of the aerodynamic equation is to treat it linearly without viscosity. Thus, the subsonic and supersonic pressure and vortex drag can be calculated. Such an approximation only preserves three partial derivatives and has been used to solve some simple problems since the thirties before computers existed.

Because of the small attack angles and yawing when an aircraft is cruising, the viscosity is not significant.

Therefore, the inviscid method is primarily used to solve the cruising state. However, the boundary layer theory has frequently been used to make viscosity corrections in recent years to improve the approximation. Furthermore, the high lift state also agrees with the incompressible flow limitation. Therefore, the boundary layer correction method has also been widely used in the computation of high lift states.

The surface element method discertizes the surface of the object. A singular point model is used to solve the equation for the incompressible flow. The linearity of this equation allows the determination of the flow by considering the aerodynamic surface alone. This is a mature method which is widely used in designing, including the allocation method, vortex lattice method, and higher order surface element method. They are basically used to solve the LaPlace or Prandtl equation and the Kutta condition is used at the trailing edge. This method can be used to calculate multiple segment aerofoil and simulate flow characteristics such as the interference of the wind tunnel wall, propulsion system, rolled up trailing edge vortex, leading edge vortex separation, and supersonic flow.

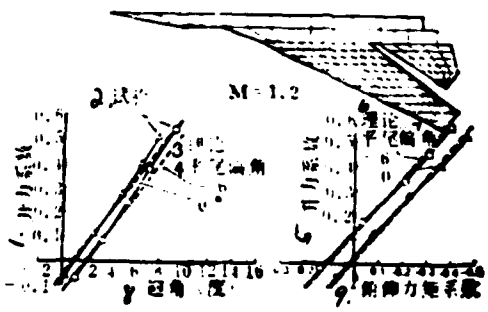


Figure 1. Simulating Fuselage with Plate Using Linearized Surface Element Method

1. lift coefficient
2. experimental
3. theoretical
4. even tail deflection angle
5. lift coefficient
6. theoretical
7. even tail deflection angle
8. angle of attack (degree)
9. pitching moment index

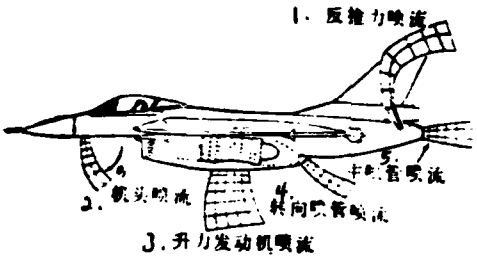


Figure 2. Simulation of Jet in Higher Order Surface Element Method /7

1. reverse thrust jet
2. nose jet
3. lift engine jet
4. turning nozzle jet
5. main nozzle jet

Surface element methods based on the Woodward, Carmichael, and Hess Linearization theories are frequently used in aircraft design. These methods have a lot of limitations. However, engineers can overcome these drawbacks with experience and judgement. Figure 1 is employed to calculate the force and moment at a supersonic speed by simulating the fuselage as a flat plate. In the range of small attack angles, the agreement

of lift is good. However, the major shortcoming of this method is that the drag is not accurately estimated, which will require empirical correction.

The reliability of the first generation surface element method is dependent upon the technique and experience of the users. Therefore, the users are required to be trained thoroughly. This limitation is relaxed in the second generation of methods. The most prominent ones are the Hess and PANAIR programs. A higher order surface element method, especially PANAIR, is a technique to solve the flow problem beyond the potential state. It can usually be realized by defining an equivalent appearance of an object which can simulate the flow potential effect. The calculated results are also modified empirically. Figure 2 is a method to calculate the jets of a vertical and short distance take off aircraft by a flow potential method. An incoming flow method is under development to combine the potential program with the boundary layer estimation program in order to include the viscosity effect in the computation.

Three-dimensional surface element methods can be used to simulate the flow surrounding a complicated geometry. Figure 3 is an example consisting of nine aerodynamic parts including the wing, fuselage, shield, short cabin, hooks, leading edge flap, trailing edge flap and upper and lower wings at the tip. The calculated pressure distribution of the wing along the chord direction shown in the figure is in good agreement with the experimental result.

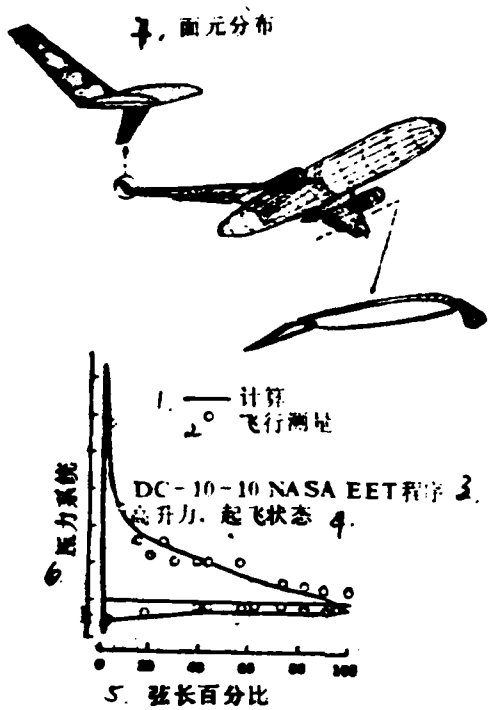


Figure 3. Using Three Dimensional Method to Calculate Complex Geometry

1. calculated
2. flight measurement
3. DC-10-10 NASA EEF Program
4. high lift, taking off
5. chord length percentage
6. pressure system
7. surface element distribution

2. Finite Difference Method

The second class of methods are used to solve the transonic flow. In this case, the equations are non-linear. The mixed elliptical-hyperbolic property requires the handling of the transonic flow phenonenon from the point of view of "volume", instead of relying on "surface" as in the linearization theory. This method employs the difference method to simplify the

inviscid equations at various points of discretization, including the small perturbation equation, full velocity potential equation or Euler equation. There are two and three dimensional difference methods which can include the effect of viscosity by using the boundary layer theory.

As the speed of the computer increases and a discrete network can automatically be generated in a three dimensional space, this type of method is developed. The automatic formation of a network is not a trivial matter. There are a number of ways at the present moment. One of them is to divide the entire external appearance. Others treat each component separately, which requires the conversion from network to network.

In the past decade, significant progress has been made in the computation of transonic flow. The capability of aerodynamic design has been improved. Each computation requires a network of approximately 10,000 points. Large computers can handle this type of calculation. Therefore, the finite difference method is widely used in transonic design, primarily to calculate the aerodynamic characteristics of an aircraft in a cruising stiuation or to improve the external appearance for cruising. Earlier, this method was used to analyze the model, and calculate the pressure distribution and cruising drag of a given wing design. The method is reliable. Later, this method is used to solve problems in the forward and reverse directions.

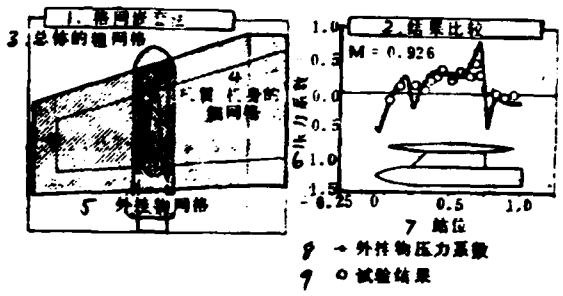


Figure 4. Using Embedded Network to Calculate Wing-Fuselage Plus External Stores

1. embedded network method
2. comparison of results
3. coarse network of the whole unit
4. fine network wing and fuselage
5. network of external stores
6. pressure index
7. position
8. pressure index of external stores
9. experimental results

Finite difference methods to solve small perturbation and full speed potential equations have been used for some time. They are very effective for the wing-fuselage combinations which are simple, smooth, and without separation. After estimating the viscosity effect based on the displacement thickness of the boundary layer, it is also possible to improve the results of high lift situations. However, the deviation is relatively large at greater attack angles or higher flap deflections.

After using the embedded network in the transonic small /8 perturbation program, it can be extended to calculate the leading wing, tail wing, short cabin and external stores. Figure 4 is the calculated result of the wing, fuselage, and external stores. The pressure data agrees with the wind tunnel test result.

This method can be combined with an optimization method to form a program to optimize the design.

There are methods using the Euler equations, which are capable of solving air flow with annular quantities. They can be used to calculate the vortices generated at the leading edge and the tip of a lifting object. Figure 5 is a typical example of using the Jameson method to solve the Euler equations of the external appearance of a wing. The Euler equation permits flow separation and vortex formation. This example seems to exhibit the characteristics of a separated vortex to obtain the maximum lift. However, some artificial viscosity was added in the calculation. The effect of this added viscosity still requires full evaluation.

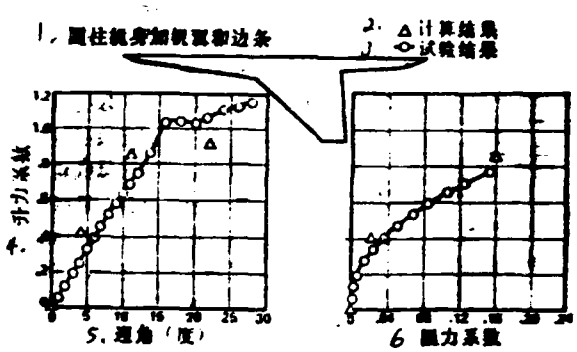


Figure 5. Euler Solution to Fuselage Plus Wing and Strake

1. cylindrical fuselage plus wing and strake
2. calculated results
3. experimental results
4. lift index
5. attack angle (degree)
6. drag index

Some preliminary results are very encouraging. However, people are still evaluating the modelling of air flow and the effect of mathematical damping (which has an artificial viscosity effect). In summary, this type of method is not as mature as the surface element method. It is still in an evaluation stage.

3. Navier-Stokes Method

The next important topic in computational fluid dynamics is solving the Navier-Stokes equations. The highest hope in the design of fighters involving complex flow calculations is solving a complete set of N-S equations. The turbulent flow model is a major difficulty which can be simplified by solving the turbulent dynamics under each flow situation. However, this is in an early exploratory stage.

A lot of work has been done to solve the average Reynolds N-S equations with respect to simple geometries. However, more powerful supercomputers and effective numerical methods will be needed when these methods are officially used in designing aircrafts. Automatic formation of networks is required. Before larger computers are available, it may be necessary to study methods which calculate by regions.

Presently, this method can only be applied to a few limited flow fields. Figure 6 is an axially symmetric jet calculated by a method developed by Cline. Many compressibility characteristics were simulated. A CD-Cyber 174 computer was used for 12 hours without reaching convergence.

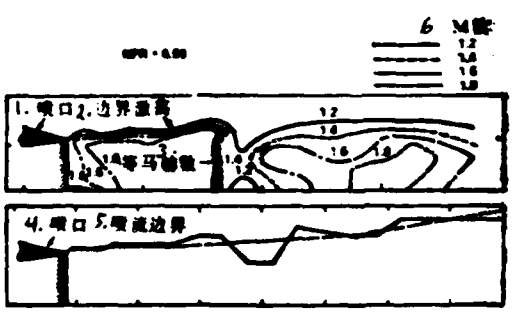


Figure 6. Jets Solved Using N-S Method

1. nozzle
2. boundary agitation
3. equi-Mach number
4. nozzle
5. jet boundary
6. M number

If Reynolds average N-S equations are used to calculate the three dimensional flow field of the entire aircraft, including transonic situation and viscosity effect which is most important in maneuvering a high performance aircraft, a computer approximately 3 times larger than the capability of the existing Class 6 computer will be required. It needs a processing speed of approximately 1 billion floating points per second and a capacity of 25,000 words of 64 characters to execute this job normally. The Reynolds average approximation simplifies the calculation because the instantaneous scale in an actual flight is replaced by an average turbulent scale. The range of turbulent scale in reality can be as high as 50:1. However, the mean aircraft drag is a function of the mean turbulence. This simulation must be materialized in stages. The United States planned to use this high speed Cray 2 computer by 1986. The capability will be even better by 1988. New computers have not yet been developed to date.

Because the Reynolds average approximation can simulate unsteady flows such as separation flow, fluttering and total drag, the optimum aerodynamic design can be obtained in combination with an optimizaton program. Therefore, this method has many advantages. But, it will require 10 million grid points. Presently, it is only used to calculated wing models.

(Cheng Bushi)