ICAS-84-1.1.2

ADVANCED NUMERICAL METHODS FOR ANALYSIS AND DESIGN IN AIRCRAFT AERODYNAMICS

Bernhard Wagner* and Wolfgang Schmidt**
Dornier GmbH, D-7990 Friedrichshafen, F.R. Germany

Abstract

A review is presented of the developments in recent years in computational methods for aerodynamic design and analysis. The discussion is mainly influenced by the industrial requirements and developments at Dornier. The need and use of computational aerodynamics in the design of aircraft and missile configurations is explored through several examples. These include synthesis-programs for predesign and evaluation work of aircraft and missile weapon systems, airfoil and high lift analysis and design methodologies, threedimensional transport- and fighter aircraft wing-body analysis methods for the complete speed range from subsonic to supersonic speed even including leading edge vortex flows, engine-inlet flows and interference problems. Besides the importance of advanced numerical schemes and fast large computers the cost-limiting factor of complex geometry handling and data pre- and postprocessing is discussed.

The use of these numerical methods has proven to substantially increase airplane performance capabilities while reducing risk, flow time, and testing requirements and thus total costs. At the same time such methods are in use to analyse and improve current and future wind tunnel limitations like wall effects, flow angularity, and Reynolds number.

1. Introduction

Aircraft development costs have escalated exceedingly within the last years. Greater emphasis must be placed on exploring analytically and experimentally new configuration concepts aimed at substantially expanding airplane and missile performance capabilities. The past and partially even present state of the art in aerodynamic analysis and design requires extensive configuration iterations through repeated wind tunnel testing that is costly, time consuming, and relies heavily on inhouse experiences and expertise. Significant advances have been achieved in the last ten years in aerodynamic computational methods which allow the numerical simulation of complex flows around two- and three-dimensional configurations and components. They provide valuable guides to those seeking understanding of specific problems and those pursuing innovative design concepts.

There are three major motivations for rigorously developing computational aerodynamics. One is to provide important new technological capabilities that cannot be provided by experimental facilities. Because of basic limitations, wind tunnels suffer e.g. from wall interference, flow angularity, Reynolds number limitations and dynamic-aerodynamic iteration problems for instance for aircraft-store compatibility problems like safe release. Numerical flow simulations, on the other hand, have none of these fundamental limitations and/or error sources, but have their own: no method can produce results beyond the validity of the physical model on which the mathematical modelling is based, and last but not least the complexity in geometry of complete configurations can easily exceed present day mesh generation strategies. These latter limitations seem to have a larger potential to be overcome in the future based on the progress in computer technology as well as method efficiency and software strategy.

A second compelling motivation concerns total configuration analysis cycle time and cost. It is evident that the time to design, build, and test a model in a wind tunnel is limiting the configurational space in industrial analysis. Numerical simulation requires no model construction time and even larger configurational changes can be verified at the computer model in a very short time. Therefore computational methods are extremely well suited to configurational studies and performance enhancements in advanced design.

The third major motivation for developing and using computational methods in industry relates to economics. Since computer speed and algorithm improvements have reduced net cost to conduct a given numerical simulation drastically, all aerospace industries can effort their own computers and skilled personal to analyse designs with computational methods, in the whole speed regime. Based on the high investment and operating costs only very few companies can effort their own wind tunnels. Competitive design, however, needs fast and continuous access to simulation facilities.

At Dornier a selection of methods for numerical flow simulation has been developed and is being applied in past and present designs which provide tools for analysis and design of transport as well as fighter type aircraft configurations and missiles in the whole speed regime. To some extent methods have been successfully used also for road vehicle aerodynamics, turbomachinery, and water based vehicles. A great amount of effort and emphasis has been placed on the validation of these methods and to establish limits of their applicability. Results to date have been encouraging and the use of such methods does provide a substantial reduction in development cycle time as well as cost required to achieve a good design.

In this paper we will attempt a brief review of computational methods and our view of their implication in designs as well as future developments.

---

* Head of Theoretical Aerodynamics Group
** Head of Aerodynamics Department

2. The Rule of Aerodynamics in Modern Aircraft and Missile Design

In recent years, air vehicle design has been complicated substantially by the trade-offs that must be made to accomodate conflicting requirements. For commercial aircraft these involve performance, cost, noise, and exhaust pollution, all of which are driven by economic and societal pressures. The main aerodynamic factors for these requirements are large C_L/C_D at high speed (new classes of airfoil sections) to improve the cruise efficiency and range, increased C_L^3/C_D^2 to improve climb rate, increased $C_{L_{max}}$ (new flap designs) to reduce minimum speed and thus landing field length, and new propulsion concepts and integration, e.g. new propellers, fan engines, prop-fans. The Dornier Do228-Series utility aircraft and the Airbus A-310 are flying examples of efficiency enhancements based on new aerodynamic wing designs each in its own flight regime.

In the case of military aircraft, different mission profiles require trade-offs at multiple design points. For example, an aircraft may be required both to cruise and manoeuver efficiently at transonic speeds and to accelerate rapidly to supersonic speeds and perform effectively in that regime. In addition, different missions for the same aircraft may require a large variety of different external stores. The optimum aerodynamic configurations that correspond to each of these design points are significantly different. More recently, reduced signal signature requirements impose additional configurational requirements.

Reliable trade-offs and configurational decisions can only be made if the differences in performance and thus aerodynamics can be predicted properly. Present efforts on a new generation of tactical fighters led at Dornier to a blend of lifting surfaces and thrust for stability and control. This demands excellent aerodynamic design capabilities.

The process by which design requirements are converted into a production aircraft is illustrated schematically in figure 1. The system requirements, as determined from customer requirements, market investigation, and mission analysis, feed into the conceptual design phase.

From simple analyses, parametric variation studies, high level of in house design expertise, and sizing considerations a conceptual baseline emerges. Assessment matrices help to decide on the most promising concept and compromise. During the preliminary design phase, the concept is refined by means of more detailed analyses using computational methods or exploratory tests or both. A complete data set must be established to allow for digital flight simulation. Design baselines are allocated to each of the technical specialities and high risk areas have been identified and at least conceptually solved. In the final design phase, an efficient blend of detailed computational analysis and testing leads to a prototype phase to verify the production baseline.

All three phases need their own expertise and tools. However, the best expertise in pre-design stems from successful production planes. In aerodynamics, this demands a strong link between project engineers, wind tunnel specialists and computational fluid mechanicists to provide the most appropriate and efficient tools.

3. Computational Tools in Aerodynamic Design

As stated in the previous chapter, only an efficient blend of computational and experimental tools will lead to a successful and cost effective design. During all design stages this blend must supply the project manager with the necessary information in time. We do not agree with a philosophy allowing computational studies as an off-line job not meeting necessarily the project schedules as mentioned in reference 1, since the best ideas will not be incorporated in a design if they are late or require duplicate work in other disciplines. The following chapters will only discuss computational tools since this is expected to be the subject of the present paper. However, efficient and improved experimental testing techniques are also mandatory for future aircraft designs.

3.1 Conceptual Design Phase

During the conceptual design phase information is required within a very short time about a high amount of different configurations. This requires inhouse data base systems to make the expertise based on previous designs readily available. In addition, sizing programs allow parametric optimization and trade-offs between thrust, weight, size, performance and other mission criteria. Based on a minimum set of information about mission requirements and aircraft or missile size, geometry configuration studies, trade-offs and optimizations can be performed. The longitudinal and lateral aerodynamic characteristics within these programs are determined by semi-empirically based computer programs. Reference 2 describes in detail the Dornier-method for aerodynamic data of general aviation, transport, and fighter aircraft from take-off through manoeuver and landing and through the subsonic, transonic, and supersonic speed regimes. As input only the configuration geometry and the flight condition are needed. However, user expertise can be added by scaling. Computer response time for a complete polar is of the order of seconds on an IBM 3083 computer. Low cost, input simplicity, and reliability in use prove such methods very useful. However, aerodynamicists have to survey use and improvements carefully since the integrated program system is based on semi-empirical methods for the components which in general implies limitations in configurational space.

Although missile configurations look much simpler in shape, their aerodynamics are highly nonlinear and very hard to predict. The prediction method as described in reference 3 proved to be very general in application and accurate enough to serve as aerodynamic module within a missile sizing program. Since computer time is extremely short, it is used within optimization cycles on line.

These methods have to be updated against the most appropriate and advanced data to provide reliable and actual information to the designer. in parallel research efforts numerical methods and/or systematic wind tunnel tests can provide better physical insight and improved simple modelling.

3.2 Preliminary Design Phase

During preliminary design extensive computational and experimental analysis has to be performed to explore and optimize the baseline-design. This phase will lead to the overall definition of the aircraft under consideration and aerodynamics have to be evaluated fair enough to establish a complete dataset for digital flight simulation in the whole flight envelope and to allow for comparison against the norm. To meet those requirements efficiently in cost and time, computational methods are playing an important role during this phase at Dornier. Windtunnel tests are performed complimentary to evaluate conceptionally new ideas and to validate computational designs. The computational methods being used in this phase have in common CPU times of the order of minutes or hours on standard computers like IBM 4341. For high or medium aspect ratio configurations two-dimensional analysis methods are still playing an important role. Interactive wing section design and analysis is a key technology in high performance wing aerodynamics. Very fast general viscous airfoil solvers have been developed at Dornier to design optimal airfoil sections for given requirements. In reference 4 a detailed description of the baseline airfoil method is given. Figure 2 portrays some typical results for the Dornier A1 (CAST 7) section.

The agreement with experimental data is very good and all trends are predicted properly. During a recent trainer development at Dornier the airfoil design has been based on this approach. Without any single two-dimensional wind tunnel testing hour the airfoil has been chosen on the basis of about 100 section simulations on the cost and time basis of one single experimental airfoil analysis. Furthermore, the method has been linked to a general optimization procedure resulting in an efficient tool for optimizing the viscous flow problem under very arbitrary constraints (reference 5). The method, however, is limited to attached flow or mild separation. More advanced methods based on the solution of the Euler or Navier Stokes equations are presently only used in the final design phase and will be discussed later.

The performance of subsonic and/or transonic mechanical high lift devices is of high importance for the overall economy and operational efficiency of present day aircraft. The preliminary design of such systems is heavily based on in house expertise. Valuable assistance is being obtained from high lift section methods as described in reference 6 and 7. Due to the complexity of the flowfield with a variety of separated regions all such component methods suffer from the inherent modelling. Figure 3 presents some typical results for both methods on the TST experimental aircraft section Dornier A-4.

Potential flow methods in combination with three-dimensional boundary layer approaches are the standard tools in wing-body and wing-body-tail design in order to limit the wind tunnel testing in the preliminary design to off-design problems mainly. Standard and higher order panel methods are routine type tools in subsonic linear angle of attack range designs. To reduce the man power cost and time requirements, the interactive graphic system CADAM® of Lockheed Corporation is being used at Dornier in combination with inhouse developed macros to perform panel generation. Not only has this yielded productivity factors of 5 - 10, but also gives a visual check on the integrity of the model before processing. Once created, the baseline model is stored for instant retrieval and can be easily modified during the preliminary design. Figure 4 shows such a panel generation, the corresponding isobar-plot of the body-pressure, and the boundary layer development (reference 8).

Interaction viscous flow is being computed by solving iteratively the panel method and the three-dimensional laminar and turbulent boundary layer methods of reference 9 and 10. Since both methods are based on the integral formulation in general curvilinear coordinate systems, e.g. the panel arrangement, their use together with panel methods is without significant additional cost. Both Dornier methods have been tested extensively against finite difference results and experiments and have proved highly reliable and accurate for attached flows. Figure 5 portrays typical results for an automobile application (C 111 research car model of Daimler Benz AG, reference 11).

The main interest in applying boundary layer analysis during preliminary design is to identify critical areas for separation and to validate design concepts which are based on laminar or turbulent boundary layer development.

Complementary to panel methods vortex lattice methods are very easy to handle and fast tools for design studies, not only for simple wing shapes but also for winglets, high lift devices, shrouded propellers, jet effects and wing-wing interference problems. They provide not only good lift and moment curves which are of importance for structural and aeroelastic design, but also quite good induced drag results. The Dornier method as described in reference 12 and 13 allows for nonlinear vortex lift based on a modified Polhamus analogy. Induced drag optimization for twist and camber can be performed by Lagrange multipliers of different shape modes. Figure 6 presents a typical comparison for a wing body combination with leading edge separation (reference 13).

Subsonic high lift wing or wing body analysis still relies heavily on wind tunnel testing. However, component-type computational methods as the one described in reference 14 are very helpful tools for high lift design of general aviation-transport-, and moderate aspect ratio fighter configurations. It is a fast tool for the estimation of maximum lift as well as stall characteristics in preliminary design to complement wind tunnel results. The use of such a method demands an experienced aerodynamicist since quite complex configurations with part span flaps or boundary layer fences can be involved.

The transonic speed range has become the most important one for efficiency improvements of transport aircraft as well as manoeuver capabilities of modern fighters. Efficient computational techniques are the most powerful tools for optimized designs since they help understanding the flow nonlinearities apparent at transonic speeds. Since valuable answers are highly depending on the configuration complexity which can be treated, mesh strategies and generation techniques for complex geometries are as important as efficient numerical algorithms. Reference 15 is giving an overview over the different techniques used at Dornier. All have in common to provide meshes for finite volume methods which guarantee high flexibility.

This mesh generation is using the same CADAM system as for panel methods, the man power for mesh generation is highly reduced and configurational changes can be adapted easily. Since transonic speed implies strong interference between wing and body or other components, the wing-body configuration possibly with the air intake is the standard configuration being evaluated computationally even during preliminary design. Standard flow solvers are presently the full potential solver of reference 16 and the Euler solver of reference 17. A collection of results is presented in reference 18. Optionally, viscous effects can be added by the previously mentioned boundary layer package.

Standard pre-design tools for supersonic speeds are again panel methods and to some extend Mach box methods for wings. Both are mainly dedicated to the analysis and design of the lifting surfaces for optimum cruise or manoeuver. Because of their linearity simple optimization strategies can be used (e.g. reference 19).

More important, however, is a reliable prediction of wave drag at supersonic speed. In the past our standard preliminary design tool has been the integrated analysis and design system of reference 20. In combination with empirical expertise this proved to be a valuable tool. For conceptionally new configuration, such a method, however, can fail to predict the Mach number trend of drag properly due to its linear flow assumptions. Figure 7 indicates the usefulness for weapon integration and optimization studies as reported in reference 21.

The results clearly indicate the importance of optimizing the configuration already in preliminary design under presence of the most likely available stores. To improve the accuracy of wave drag prediction further, more recently space marching Euler methods as in reference 22 and 23 tend to displace linear methods in preliminary design.

3.3 Final Design Phase

During final design all details have to be designed and optimized. Performance and handling guaranties will base on the results of this phase. Strong links between aerodynamics, flight mechanics, structures, aeroelastics and weights provide constraints for sub-optimization and part design. Computational methods have to provide contributions to the detailed designs, wind tunnel tests should be limited to verification. Efficient use of appropriate computational methods can highly reduce turnaround time and tests even in the later prototyp flight phase. The computational methods involved in final design are partially those of preliminary design and updates of the semiempirical conceptual design tools but mainly the most advanced and accurate ones available. Typical final design tools at Dornier are the full potential solver of reference 16 and the more recent Euler- and Navier Stokes solvers as described in references 17,18,23, and 24. Recently, the threedimensional Euler solver became extremely versatile with respect to complex geometries by adaption of the block structured mesh concept (references 15, 18).

Since present day wind tunnels exhibit limitations in Reynolds number computational methods are partially used also to extrapolate wind tunnel data to free flight. At manoeuver boundaries for airfoils such analysis is made either by solving the time dependent Euler equations plus the boundary layer equations using an inverse method or by solving the time dependent Reynolds averaged Navier-Stokes equations.

For both approaches methods have been developed which are in practical use if shock induced or larger trailing edge separation is expected. The corresponding finite volume methods are described in detail in references 25, 26, and 24. Figure 8 shows the good agreement with experimental data without any Mach- or angle of attack correction rather than the one given by the experimentalist as tunnel correction.

Although this method can predict separated flow of moderate extent properly, it is limited to steady flow in the mean. For strong shocks, also a higher order interaction model might be missing. We, however, feel Navier-Stokes solutions to be more appropriate to such problems. Figure 9 portrays typical high Reynolds number results which also indicate the importance of proper mesh spacing (e.g. by solution adaptive grids) in order to resolve the physically important flow characteristics (reference 24, 27).

To provide detailed design information on axisymmetric afterbodies with jets, Navier-Stokes solvers have been used successfully to simulate the interacting flow. Details can be found in reference 28 and a typical result is presented in figure 10.

Final design requires detailed design of components in the presence of the complete aircraft. Such details can be wing-body fairings, airframe-inlet integration, propeller-wing interaction and wake, supersonic drag optimization, detailed leading edge design to establish certain vortex flow characteristics or safe store release and icing-problems. Main tools for these problems are three-dimensional finite volume full potential solvers (reference 16) and three-dimensional Euler solvers. The Euler solvers can be either time dependent as described in references 17 and 29, or space marching for supersonic flow (see e.g. references 22 and 23)...

Complex wing body interaction can cause strong shocks on the lower surface of a high mounted wing at low lift conditions. Figure 11 portrays a design modification at the wing-body intersection which completely changed the wing lower surface pressure distribution without any wing section change (reference 30). Based on classical empirical designs this would have been an enormous job.

Airframe-engine inlet integration is highly configuration dependent and still more on art. Three-dimensional Euler solvers, however, provide valuable insight and help if fixed geometry pitot-type engine inlets have to be optimized from the outer inlet lip region up to the inner tube and compressor entrance plane. On figure 12 typical results for a supersonic underbody-mounted pitot-type air intake are shown. More details can be found in reference 31.

For large transport aircraft engine cowl of high bypass ratio engines can cause large integration problems. As shown in reference 32 our three-dimensional Euler solver is a very valuable tool for detailed analysis.

Engine inlets can cause severe problems during icing conditions. Therefore ice-prediction and de-icing design is an important task during final design. The computation of water droplet paths in the whole speed and engine condition range is basis for icing-prediction in the inlet lip and tube area. Figure 13 presents some typical results from reference 33.

An other interesting task is the analysis of missile plume effects during release on the air intake and the prediction of possible flow distortion in the inlet tube up to the compressor plane. In reference 34 this problem has been solved successfully. Figure 14 shows some typical results for an application within the Alpha-Jet program.

An important detailed design problem for small and large transport aircraft with propeller driven propulsion units is the change in wing and tail loading due to the overspeed and swirl of the rotating propeller and different thrust conditions. Recently, this problem has been analysed properly as described in reference 35 by solving the Euler equations.

For new generations of fighter aircraft excellent transonic manoeuver capabilities and sustained supersonic cruise and manoeuver are mandatory. This requires powerfil tools for optimization of wave drag at supersonic speeds and leading edge shape and device for transonic and supersonic manoeuver. The recently developed Euler methods provide excellent tools to optimize final details. Reference 22 presents a very attractive example for drag prediction using an Euler space marching scheme. Geometry complexity is unlimited only depending on mesh generation.

Figure 15 portrays an example for leading edge design with leading edge vortex flow depending on the nose sharpness. As described in reference 36 this approach is valid over the whole speed range and again unlimited in wing or body complexity. First results even indicate the features of vortex burst effects to be well predicted which can severely limit the angle of attack and sideslip range with respect to stability and control.

All these methods can be easily combined with structural methods to predict aeroelastic effects or to perform aeroelastic tayloring of highly manoeuverable wings.

Finally, safe store release will be mentioned briefly. Such computational simulations in general start after the final design when a fighter aircraft is decided to carry certain external stores. Such external stores exhibit quite different characteristics and an aircraft qualification program with all possible stores is extremely expensive. Therefore an efficient blend of numerical simulation, wind tunnel testing, and flight testing is mandatory. Store release pattern depend on carrier interference, release disturbances and store and release unit characteristics as well as carrier manoeuver. In reference 37 the standard Dornier procedures are described for release of stores ranging from tanks to missiles to towed targets. The carrier interference field during release studies is predicted by the most appropriate method ranging from vortex lattice to panel and Euler method. Figure 16 presents a typical release result.

4. Conclusions

The last ten years have been very exciting for researchers in computational aerodynamics. Advances in solution algorithms, complex mesh generation strategies, computer power and pre- and postprocessing packages have led not only to research applications but to an involvement of CFD in preliminary and final design. The generation of aircraft appearing now or in the near future has been highly influenced by designs based on the use of computational methods. Such simulations will gain even more importance for the aircraft designers in the next decade. Ongoing work will allow much more detailed simulation of even more complex configurations and aerodynamic phenomena. Improvements in data handling man power as well as pre- and postprocessing will enable aircraft designers to use sophisticated CFD as routine tools during all design phases. Computer aided design systems will connect all the different disciplines like aerodynamics, structures, propulsion, and design.

5. References

1 Ballhaus, W.F., Jr. (1982)
Computational Aerodynamics and Design.
Lecture Note in Physics, Vol. 170, Eighth International Conference on Numerical Methods in Fluid Dynamics, pp 1 - 20 (Springer-Verlag Berlin-Heidelberg)

2 Fritz, W. (1979)
Aerodynamikbaustein für ein Programmsystem zur parametrischen Entwurfs- und Leistungsrechnung von Flugzeugen.
Dornier FB 79/46 B

3 Mathauer, H. (1979)
Erweiterung des Rechenprogramms LARV zur Bestimmung der aerodynamischen Beiwerte von Flugkörpern.
Dornier FB 79/44 B

4 Longo, J., Schmidt, W. and Jameson, A. (1982)
Numerical Simulation of Transonic Airfoil Flow.
ICAS-82-6.1.2, Seattle

5 Misegades, K.P. (1984)
Airfoil Optimization
AIAA-Paper 84-0053

6 Arlinger, B. and Schmidt, W. (1978)
Design and Analysis of Slat-Systems at Tranconic Flow.
11th ICAS Congress, Lisboa

7 Leicher, S. (1980)
Viscous Flow Simulation of High Lift Devices at Sub- and Transonic Speed.
AGARD CP 291, Paper 6

8 Seibert, W., Stock, H.-W. and Schmidt, W. (1982)
Aerodynamische Flugzeugheckgestaltung.
Dornier FB 82 BF/7B

9 Stock, H.-W. (1977)
Integralverfahren zur Berechnung dreidimensionaler, laminarer und turbulenter Grenzschichten.
Dornier FB 77/51 B

10 Stock, H.-W. (1980)
Computation of the Boundary Layer and Separation Lines on Inclined Ellipsoids and of Separated Flows on Infinite Swept Wings.
AIAA-Paper 80-1442

11 Seibert, W. (1982)
Aerodynamische Nachrechnung der Daimler Benz Fahrzeugkonfiguration C 111.
Dornier FB 82 BF/16 B

12 Lucchi, C.W. and Schmidt, W. (1976)
Vortex Lattice Approach for Computing Overall Forces on V/STOL Configurations.
AGARD CP 204, pp 19-1 - 19-8

13 Wedekind, G. (1980)
Verbesserung der Berechnung der Widerstandspolaren von gewölbten Konfigurationen mit größerer Vorderkantenpfeilung unter Verwendung der Saugkraftanalogie.
Dornier Note BF10-1660/80

14 Proksch, H.-J. (1976)
Berechnung von Flügelauftrieb, -momenten und -widerstand ohne und mit Klappen in reibungsbehafteten Unterschallströmung einschließlich abgelöster Strömung.
Dornier FB 76/41 B

15 Leicher, S., Fritz, W., Grashof, J., and Longo, J. (1982)
Mesh Generation Strategies for CFD on Complex Configurations. Lecture Notes in Physics, Vol. 170, Eighth International Conference on Numerical Methods in Fluid Dynamics, pp 329 - 334 (Springer-Verlag Berlin-Heidelberg)

16 Leicher, S. and Jameson, A. (1980)
Finite Volume Transsonik-Verfahren zur Lösung der vollen Potentialgleichung.
Dornier Note BF30-1716/80

17 Schmidt, W. and Jameson, A. (1983)
Euler Solvers as an Analysis Tool for Aircraft Aerodynamics.
In: W.G. Habashi (ed.), "Advances in Computational Transonics", Volume 4 in "Recent Advances in Numerical Methods in Fluids", Pineridse Press, Swansea, U.K.

18 S. Leicher (1984)
Analysis of Transonic and Supersonic Flows around Wing-Body Combinations,
ICAS-Paper 84-1.2.2

19 Wedekind, G. (1978)
Methoden der Tragflügeloptimierung für ein taktisches Kampfflugzeug.
Dornier FB 78/50 B

20 Middleton, W.D. and Lundry, J.L. (1976)
A Computational System for Aerodynamic Design and Analysis of Supersonic Aircraft.
NASA-CR-2715

21 Misegades, K. (1980)
Optimale Außenlastintegration an Kampfflugzeugen im Überschall.
Dornier FB 80/38 B

22 Arlinger, B. (1982)
Computation of Supersonic Flow Around Three-Dimensional Wings.
ICAS-82-6.1.3, Seattle

23 Wagner, B. and Schmidt, W. (1981)
Berechnungsverfahren für stationäre Überschallströmungen.
Dornier FB 81 BF/16 B

24 W. Haase, B. Wagner, and A. Jameson (1983)
Development of a Navier-Stokes Method Based on a Finite Volume Technique for the Unsteady Euler Equations.
Proceedings of the 5th GAMM Conf. on Numerical Methods in Fluid Mechanics, Rome, Italy.

25 Schmidt, W., Jameson, A. and Whitfield, D. (1981)
Finite Volume Solution for the Euler Equations for Transonic Flow over Airfoil and Wings Including Viscous Effects.
AIAA-Paper 81-1265

26 Whitfield, D., Thomas, J.L., Jameson, A. and Schmidt, W. (1983)
Computation of Transonic Viscous-Inviscid Interacting Flow. Second Symposium on Numerical and Physical Aspects of Aerodynamic Flows, Long Beach, CA, USA

27 W. Haase, K. Misegades, and M. Naar (1983)
Adaptive Grids in Numerical Fluid Dynamics. Amended Version of a paper given at the 3rd Intern. Conf. on Numerical Methods in Laminar and Turbulent Flow, Seattle, Wash., submitted for publication to International Journal of Numerical Methods in Fluids

28 Wagner, B. (1984)
Calculation of Turbulent Flow about Missile Afterbodies Containing an Exhaust Jet.
AIAA-Paper 84-1659

29 Grashof, J. and Schmidt, W. (1982)
Computational Methods for Two-Dimensional and Three-Dimensional Inlets in Subsonic and Supersonic Flow.
13th ICAS Congress, Seattle

30 Leicher, S. (1982)
Computational Analysis of Body Modifications on Wing pressures.
Dornier Note BF30-2144/82

31 Grashof, J. (1983)
Pitot-Einlauf, Phase II. Triebwerkseinläufe für Überschallflugzeuge - Theoretischer Teil.
Dornier FB BF 11/83 B

32 Chen, H.C., Rubbert, P., Yu, N.J. and Jameson, A. (1983)
Flow simulation for General Nacelle Configurations Using Euler Equations.
AIAA Paper 83-0539

33 Stock, H.-W. (1983)
Water Droplet Trajectory Computation Around an Air Intake.
To be published in the Zeitschrift für Flugwissenschaften und Weltraumforschung

34 Grashof, J. (1983)
Numerical Investigation of Three-Dimensional Transonic Flow Through Air Intakes Disturbed by a Missile Plume.
AIAA-Paper 83-1854

35 Whitfield, D. and Jameson, A. (1983)
Three-Dimensional Euler Equation Simulation of Propeller-wing Interaction in Transonic Flow.
AIAA-Paper 83-0236

36 Hitzel, St. and Schmidt, W. (1983)
Slender Wings with Leading-Edge Vortex Separation - A Challenge for Panel-Methods and Euler-Codes.
AIAA-Paper 83-0562

37 Kapp, R. (1983)
Abgangsverhalten und aerodynamische Interferenz Trägerflugzeug/Flugkörper.
Dornier FB BF 3/83 B.

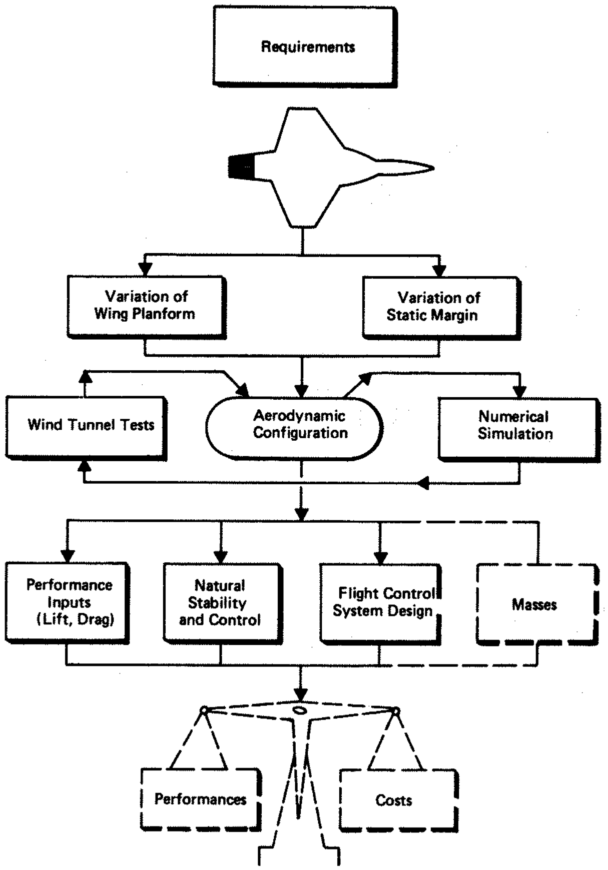


Fig. 1: Steps in aerospace design

CRAY I

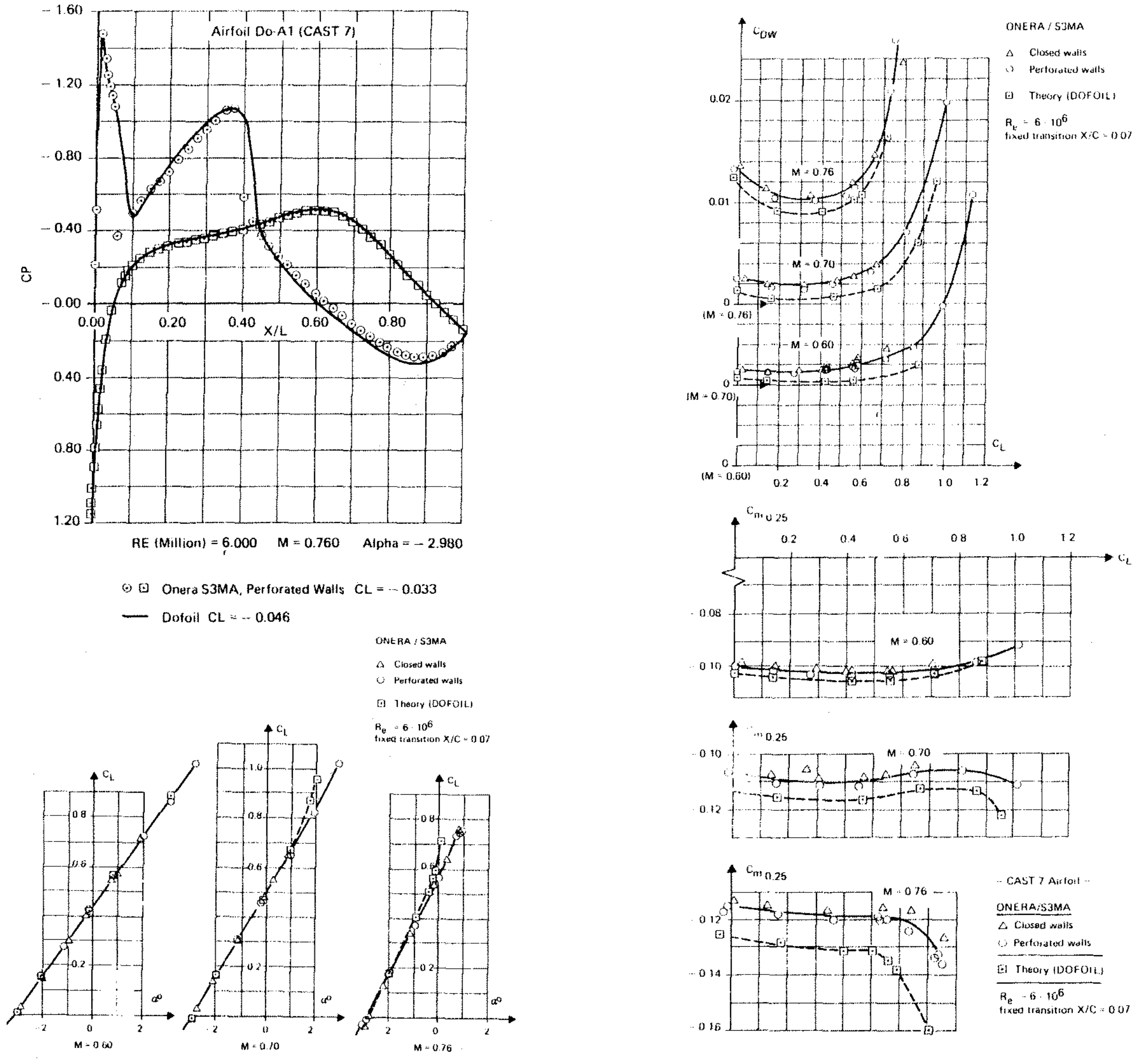


Fig. 2: Comparison of DOFOIL results against wind tunnel data

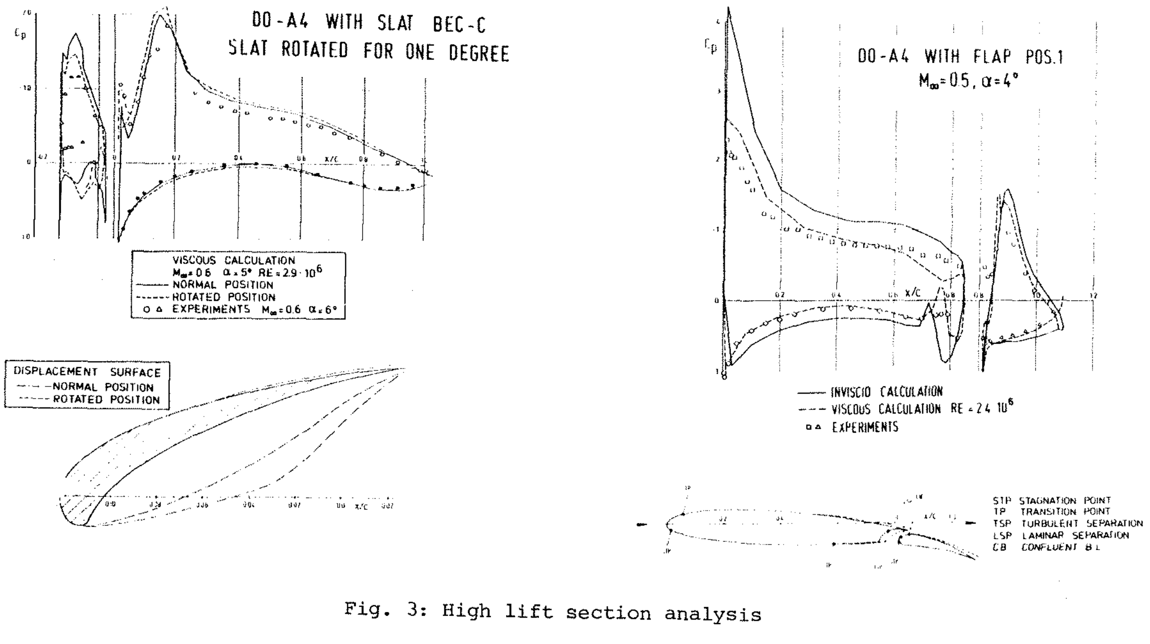


Fig. 3: High lift section analysis

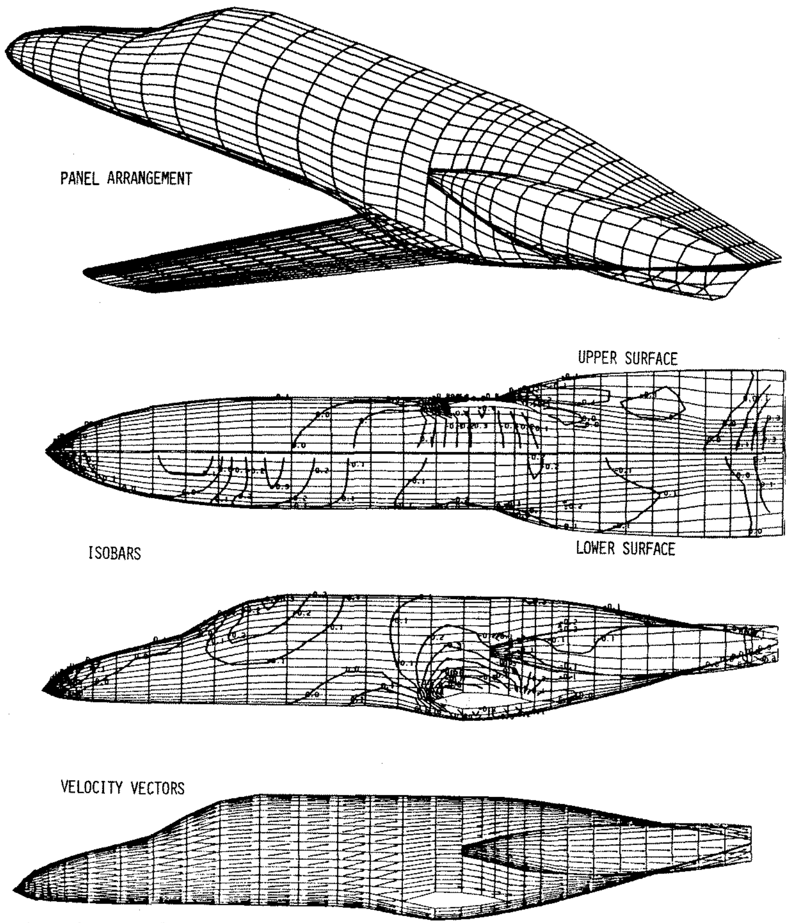


Fig. 4: Panel arrangement, surface pressures and surface velocity vectors for Dornier ANT research aircraft ($M_\infty = 0.6$; $\alpha = 4^\circ$)

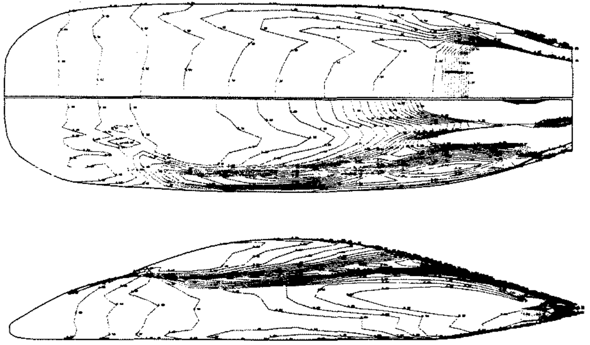


Fig. 5: Three-dimensional boundary layer analysis on a car (iso displacement thickness lines)

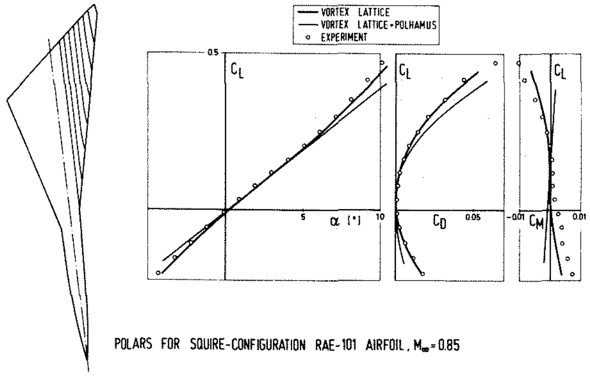
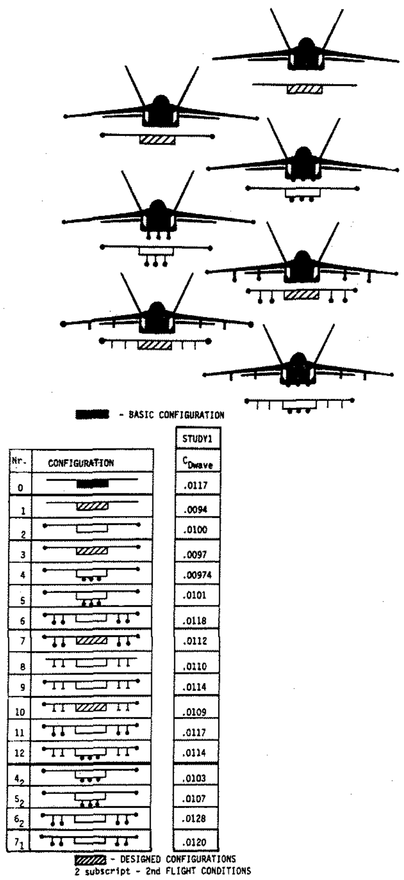


Fig. 6 Vortex lattice analysis for slender wing-body flow



Nr.	CONFIGURATION	STUDY1 C_{Dwave}
0		.0117
1		.0094
2		.0100
3		.0097
4		.00974
5		.0101
6		.0118
7		.0112
8		.0110
9		.0114
10		.0109
11		.0117
12		.0114
4_2		.0103
5_2		.0107
6_2		.0128
7_1		.0120

Fig. 7: Store integration study by drag optimization

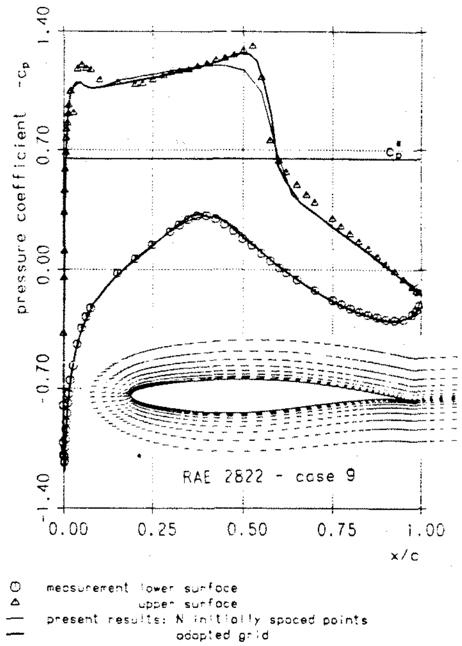


Fig. 9: Navier-Stokes solution for transonic flow using solution adaptive grid

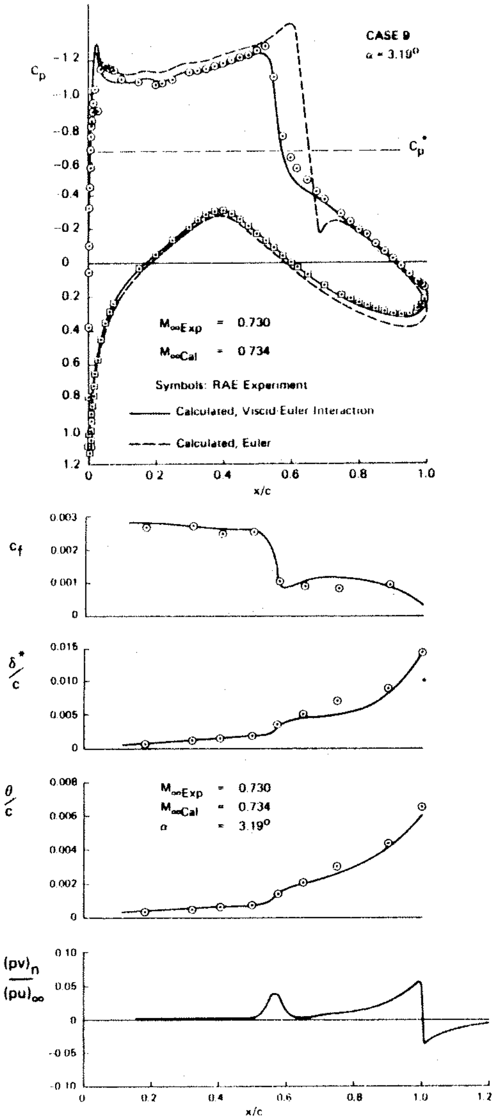


Fig. 8: Viscous-inviscid interacting flow solution using Euler and inverse boundary layer methods

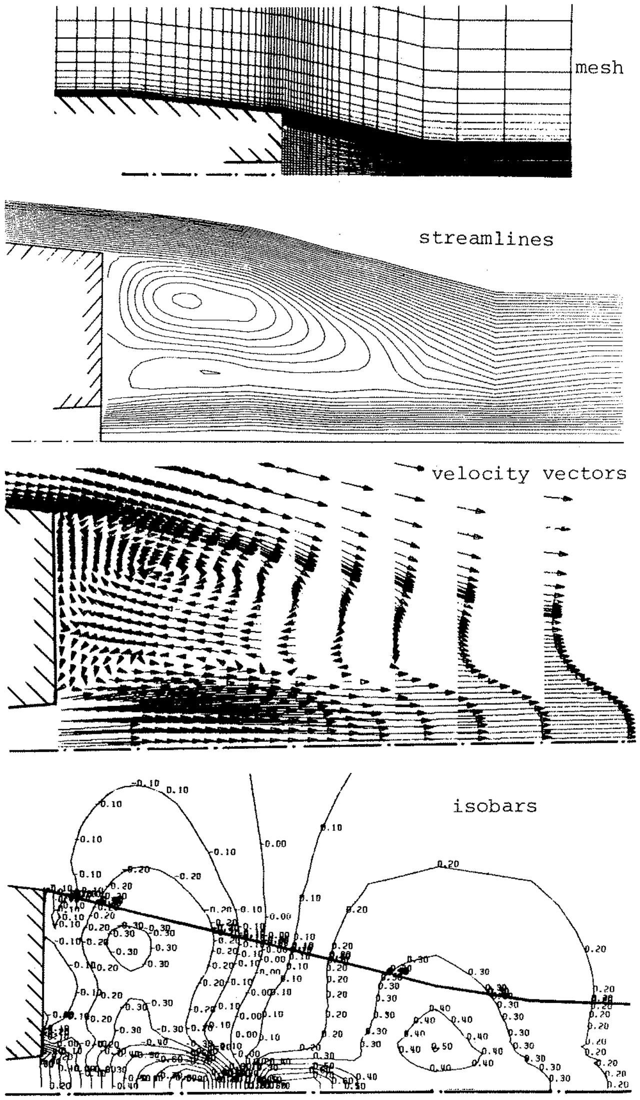


Fig. 10: Navier-Stokes solutions for transonic missile afterbody flow ($M_\infty = 0.85$; $\beta_e = 6^\circ$; $M_j = 2.9$; $\beta_j = 3^\circ$; $p_j/p_\infty = 1.17$)

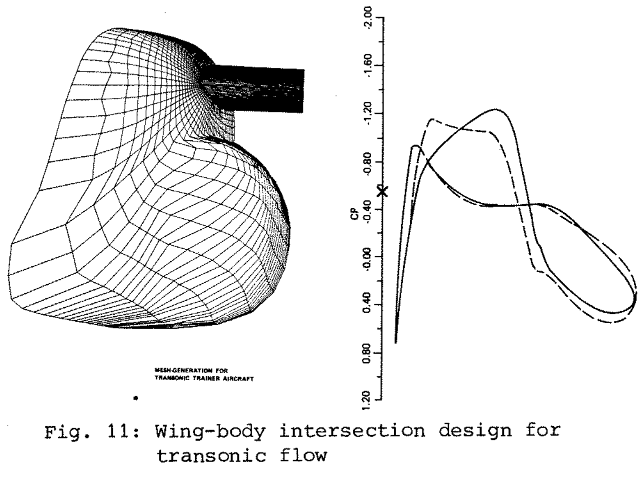


Fig. 11: Wing-body intersection design for transonic flow

—— original design

---- modified design, based on computational results

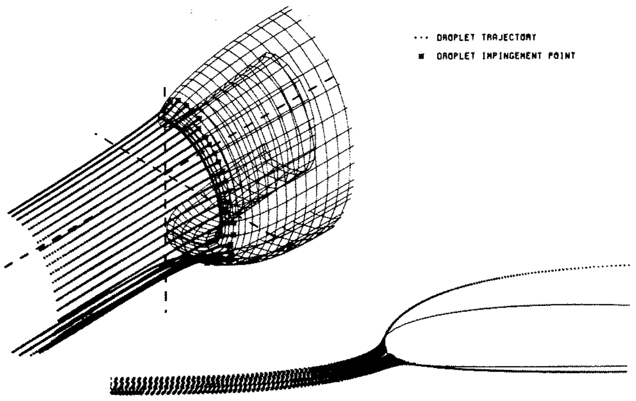


Fig. 13: Water droplet path calculation for inlet-deicing design

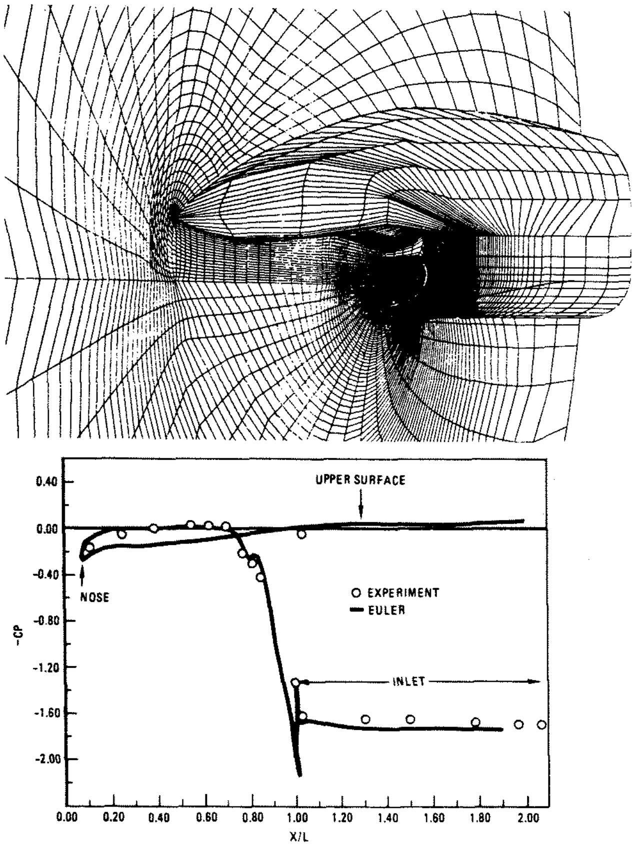


Fig. 12: Analysis of inlet-forebody combination in supersonic flow

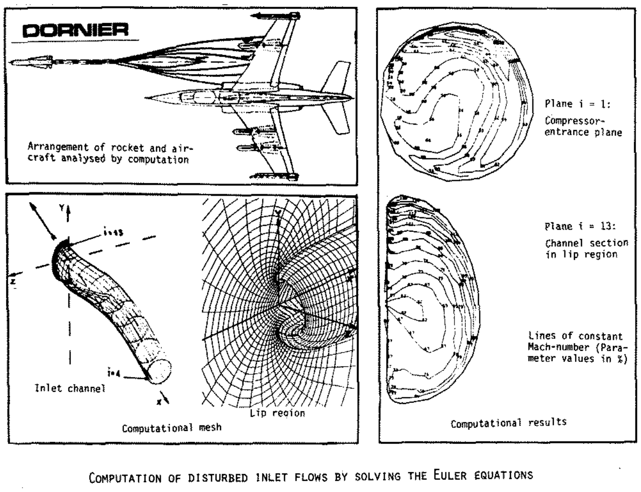


Fig. 14: Inlet distortion due to missile launch

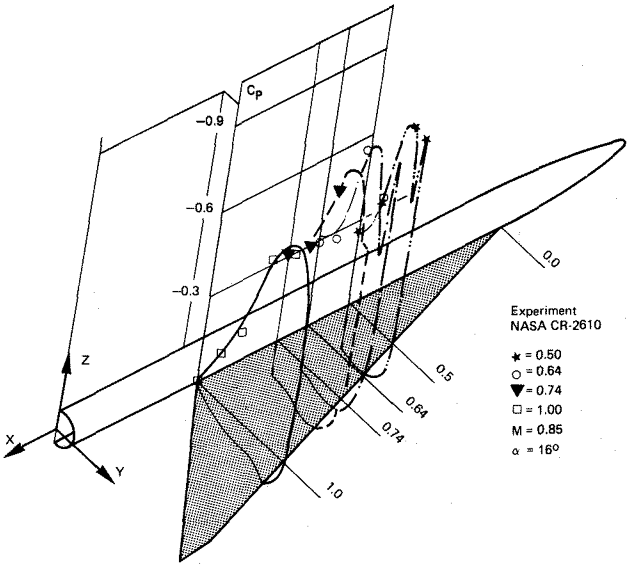


Fig. 15: Leading edge vortex analysis in transonic flow (arrow wing, round leading edge)

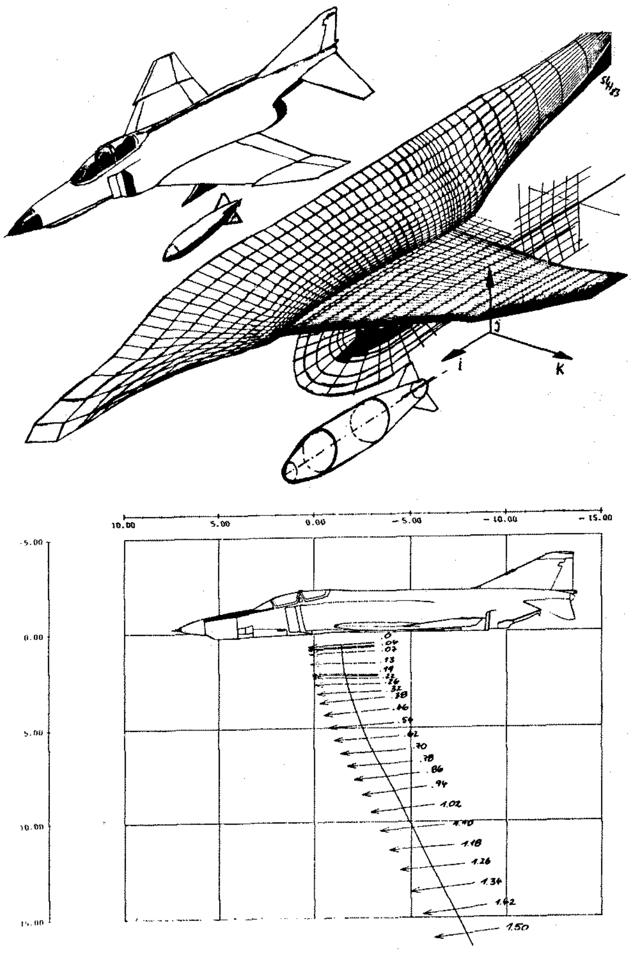


Fig. 16: Safe release simulation using flow field prediction by Euler methods