Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
(ICIDES-III), Editor: G.S. Dulikravich, Washington D.C., October 23-25, 1991.

N92-13938

"RECENT PROGRESS IN INVERSE METHODS IN FRANCE"

PIERRE-FRANÇOIS BRY (SNECMA)

Société Nationale d'Etude et de Construction
de Moteurs d'Aviation
(S.N.E.C.M.A.)
Cincinnati, Ohio, U.S.A.

OLIVIER-PIERRE JACQUOTTE
MARIE-CLAIRE LE PAPE

Office National d'Etudes et de Recherches Aérospatiales
(O.N.E.R.A.)
Aerodynamics Department
92322 CHATILLON FRANCE

ABSTRACT

Given the current level of jet engine performance, improvement of the various turbomachinery components requires the use of advanced methods in aerodynamics, heat transfer, aeromechanics as well as in other fields.
In particular, successful blading design can only be achieved via numerical design methods which make it possible to reach optimized solutions in a much shorter time than ever before. The present paper focuses on two design methods which are currently being used throughout the French turbomachinery industry to obtain optimized blading geometries. Examples are presented for compressor and turbine applications. The status of these methods as far as improvement and extension to new fields of applications is also reported on.

1. INTRODUCTION

The ever-increasing performance requirements for jet engines, together with the fast pace of development programs, have led designers to rely more than ever on computations when defining their products. The field of Fluid Mechanics has naturally been at the forefront of this evolution both in external and internal aerodynamics. A great deal of effort has been devoted to the development of powerful numerical tools which allow both the design and analysis of geometries with the obvious goal of obtaining optimized shapes that can enhance performance.
The present paper focuses on two design methods which are currently used throughout the French industry for turbomachinery applications. After a brief review of the general inverse problem in the turbomachinery field, examples of what can be achieved are presented both for compressor and turbine blading. In addition, the versatility of one of the methods is demonstrated by using the example of a jet engine inlet design.

2. A PROPER FORMULATION FOR THE INVERSE PROBLEM

The idea of inverse design methods is obviously not new. Once the blade or wing designers had the knowledge and the understanding of the flow around an airfoil, isolated or not, it was natural to try to define profiles not from the purely geometrical standpoint but rather by using this very knowledge of the fundamental profile aerodynamics. It was recognized early on that there was a direct relationship between the surface velocity distributions and overall performance. Hence the idea of defining the profile starting from the velocity distribution itself.

The inverse problem for isolated profiles in incompressible flows was first formulated by LIGHTHILL [1]. It consists in determining an airfoil that produces a given speed distribution prescribed on the unknown airfoil profile. It was shown that closed profiles could exist only if the prescribed velocity W^{o} satisfies three integral constraints. In this early work, these were chosen as the upstream velocity W^{∞} and two parameters related to the closure of the profile.
More recently, Volpe and Melnik [2] proposed several possible choices for the design of isolated profiles. In particular, they showed that it was possible to obtain closed profiles via introduction of two modification functions for the target velocity.

For turbomachinery applications, the problem is slightly different in the sense that 1) the flow is quasi-three dimensional (in first approximation) and 2) the profiles are entrapped between adjoining blade rows which partly determine the upstream and downstream boundary conditions. In particular, the upstream (or downstream) velocity as well as the upstream/downstream flow angles cannot be set as free parameters.

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
ICIDES-III, Editor G.S. Dulikravich, Washington D.C., October 23-25, 1991

The prescribed velocity distribution is defined on the pressure and suction sides of the blade and is given by two separate functions of the arc lengths S (pressure) and S (suction). The relative lengths of the two sides or, equivalently, the position of the stagnation point, can therefore be considered as the first necessary parameter.

The second parameter is a direct consequence of the fact that, in general, a flow evolves (in basic approximation) on a quasi-three-dimensional surface throughout a blade row. Whereas it is possible to set free two parameters defining the trailing edge closure (Δx and Δy) for isolated 2-D profiles, it is obvious that a trailing edge cannot be reasonably defined if both suction and pressure side trailing edge points are not located at the same radius (the same m-coordinate in the standard ($m-\theta$) quasi-three-dimensional blade-to-blade representation). Therefore, the remaining parameter pertaining to profile closure is the circumferential gap in the θ-direction at the trailing edge.

Finally, like in all turbomachinery problems, the solidity is a governing parameter, directly related to the circulation around the profile and the flow turning. It comes as no surprise that it is the third parameter to be computed by the algorithm since inlet/outlet angles as well as velocity distributions are given data for the design method.

Based on these general considerations, many methods were proposed in the past to deal with the problem of profile or blade design. It is not the purpose of this paper to review all these methods and we will instead refer the readers to overall summaries such as proposed by Sloof [3] or Meauzé [4].

A commonly used method for two-dimensional applications was proposed years ago by Stanitz [5] to determine analytically a profile from a given velocity distribution. It is still being used successfully for specific two-dimensional applications at a reasonably low Mach number. More recent developments by Cedar and Stow [6] in England and Jacquotte [7] in France allow the definition of high Mach number profiles within the quasi-three-dimensional and potential approximations. Finally Meauzé [8] in France and Leonard et al. [9] in Belgium have proposed solutions for the non-viscous quasi-three-dimensional problem solving the Euler equations that allow for the occurrence of strong shocks within the flow field.

In the following paragraphs, we will discuss the recent developments in France that concern both the potential design method by Jacquotte [7] and the Euler method by Meauzé [8].

3. FINITE ELEMENT INVERSE METHOD FOR POTENTIAL FLOW [7]

A thorough description of the method is given in [7] and [9]. We will therefore only give here a general outline while concentrating on concepts and applications.

The method was developed by Jacquotte in 1989. It makes use of the concept introduced in the previous paragraph pertaining to the constraints that must be taken into account in order to arrive at a solution.

3.1 Basic assumptions

Three very basic approximations are retained:

a) the flow is inviscid;
b) it is considered quasi-three-dimensional and computed on stream surfaces in the computational space (m, θ), where m is the arc length of the meridian line defining the stream surface, and θ is the polar angle around the axis Oz. It is therefore assumed that the characteristics of the through-flow are known and given by a function r (z) defining a stream surface and the stream tube thickness b(z);
c) third, the calculation is carried out within the potential flow approximation. Even though the entropy production through shocks cannot be taken into account, such a model is still valid for compressible transonic flows where strong shocks do not occur i.e., for relative Mach numbers that do not exceed 1.3 or 1.4. The advantage of using such a potential flow approach is to be found in the small CPU times necessary to obtain solutions. This turns out as a very strong point for a design method which can therefore be used on an interactive basis.

3.2 Computational domain and boundary conditions

In order to take advantage of the periodicity of the problem, the computation is of course restricted to a blade-to-blade channel. The profile is prolonged by a pseudo-wake, without lift and with a constant angular thickness equal to the trailing edge gap.
A C-topology is used to describe the computational domain since it is well adapted to profiles with relatively thick leading edges.

Upstream and downstream conditions are obtained from any standard through-flow computation; the upstream flow is prescribed via inlet angle and inlet relative Mach number while the downstream flow is defined only via the exit angle.
The other exit quantities are naturally obtained through the continuity equation. These boundary conditions are taken far enough upstream and downstream so that the flow can be considered as uniform.
The method can operate both in direct and inverse mode depending on the kind of boundary conditions which are applied on the profile: whereas a Neumann condition corresponding to a zero normal velocity is usually applied in direct calculations, a Dirichlet condition is imposed in the inverse method. This condition corresponds to the fact that the tangential velocity (to the profile) must be equal to a given W^o.

3.3 Profile modification

The goal here is to find the shape which satisfies both constraints:
a) zero normal velocity
b) tangential velocity equal to a given W^o.

The solution of the inverse problem leads to a flow that follows the prescribed tangential velocity W^o on the profile but does not necessarily satisfy the zero normal velocity condition. The non-zero normal velocity obtained from the algorithm is used to modify the profile via a transpiration model: the displacement of the blade surface is accounted for by injection of fluid through the original blade surface such that the new surface becomes a stream surface [6]. The displacement normal to the profile is then obtained simply by expressing the mass conservation between two elements of length ds on the profile (see figure 1).

3.4 Inverse design algorithm

The inverse method consists therefore of a sequence of the following three-step iterations:
a) computation of the potential on the profile by integration of the prescribed velocity;
b) computation of the potential in the domain by solution of the continuity equation with a Dirichlet boundary condition on the profile;
c) computation of the normal displacement of the blade surface as described above and modification of the profile.

While the first and third steps are simple one-dimensional integrations, the second step corresponds to the resolution of a two-dimensional, second order, non linear partial differential equation. The numerical method used to solve this equation is a finite-element method developed by Bredif [10] which will not be described here. Transonic flows can be handled by using a density upwinding also presented in [10].

3.5 Numerical results for turbomachinery applications

Starting from an initial profile, three modifications are generally needed in order to obtain good agreement between the prescribed velocity distribution and the one corresponding to the computed profile. The inverse computation is automatically followed by a direct calculation only to verify the convergence of the procedure. With a 10 x 117 point C-grid (used in most applications) the total computing time is about 15s on an IBM 3090 computer.

Three examples are presented: one for a highly loaded compressor rotor hub section, one for the root section of a strongly quasi-three-dimensional turbine nozzle and the last one corresponding to a case where the robustness of the method is demonstrated.

Compressor rotor hub section

Figure 2 shows the compressor flow path. The stream tube thickness is obtained from a through-flow calculation which also provides all the input parameters:

- inlet Mach number = 0.95
- inlet flow angle = 61.7^{o}
- outlet flow angle = -2.5^{o}

The initial geometry came from a previous calculation and the initial velocity distribution (see figure 3) was obtained by running the inverse code in its analysis mode.

The objective for the calculation was to reduce the peak Mach number on the suction side while retaining the same solidity and maximum thickness. Figure 3 shows the prescribed velocity distribution vs the original one as well as the new profile that was obtained after three successive modifications. The pitch angle and the thickness distribution have changed in a substantial manner.

Hub section of a turbine nozzle

The case considered here corresponds to a strongly quasi-three-dimensional section of a turbine nozzle with a large stream-wise variation of the stream tube thickness (outlet to inlet ratio of 1.3). Designing such blading with a two-dimensional inverse method results invariably in the occurrence of non-uniformities in the velocity distributions.

For the present computation, the inlet and exit flow angles are 31.4^{o} and -61^{o} respectively and the inlet Mach number is 0.424.

The velocity distribution on the initial blade and the target velocity distribution are shown in figure 4 together with the blade profiles. For this case, five blade modifications were necessary to reach convergence. The resulting profile remains very smooth.

Example with a poor initialization

The case in Figure 5 involves large changes in the profile from the initialization and demonstrates the robustness of the method. Starting with a geometry having a relative maximum thickness of 3% and a pitch angle of 25°, the code is capable of converging to a new profile with a thickness of 7% and a pitch angle of 5°. After one iteration, a very large displacement is observed but nonetheless the calculation remains stable.

3.6 Extension of the method to the nacelle design

For the design of transonic blades presented up to now, the complete 3D blade is obtained by stacking a series of 2D profiles; this procedure leads to a reasonable blade if the input pressure (or velocity) distributions vary smoothly, and, most importantly, if the flow is essentially two dimensional, in the sense that there is a preferential direction where little happens in comparison to the other two directions. A complete 3D calculation using a more accurate model (Euler or Navier-Stokes) is the definite proof that the blade obtained by the inverse method possesses the desired features.
The flow around a commercial aircraft inlet (nacelle) demonstrates the "essentially 2D" quality mentioned above and therefore the stacking procedure can be used about its axis for the design of this type of geometry. The method has been extended with the following characteristics:

- basic assumptions:

a & c): same as in 3.1
b) the flow is considered to be axisymmetrical and the potential equation is written and discretized in the (z, r) plane.

- computational domain and boundary conditions: a C-topology is used to describe the computational domain extending around the inlet from the compressor plane to the downstream plane behind the nacelle. The four boundaries and the conditions applied thereon are the following:

o the inlet profile and its continuation until the downstream plane; boundary condition: either no mass flow for the direct calculation, or Dirichlet condition on the profile in the inverse mode;
o the compressor plane, with a prescribed velocity distribution (varying Neumann condition);

- o a three-segment boundary, including the axis (no-mass-flow condition), the upstream plane (prescribed velocity) and a far field boundary (no-mass-flow condition);
- o the downstream plane, with a prescribed velocity computed from the mass balance equation between this boundary, the upstream plane and the compressor plane.
- the profile modification is carried out in the same way as before, using the transpiration model mentioned in 3.3 [6].
- the inverse design algorithm also remains the same as in 3.4

We will now present a result proving once again the robustness of the method with respect to arbitrary initializations. A velocity repartition (so-called "ideal velocity" on figures) is computed by direct calculation around a given profile ("ideal profile"); this profile is modified into the "initial profile" by thickening. The velocity distribution around this profile is represented in figure 6. It clearly shows an aspect different from that of the ideal velocity. The inverse method has been used in order to recover the ideal profile from this initialization. The convergence of the inverse algorithm is monitored by the decrease of the mass flow across the profile for each inverse calculation. The normal velocity distributions for the first three iterations are shown in Figure 7. After these iterations, the normal velocity is zero on most of the profile, except in the neighborhood of the leading edge. These initial iterations determine therefore the overall shape of the profile. The final iterations (there are four of them here) tend to precisely shape the leading edge of the profile.
The final geometry of the nacelle is compared to the initial one in Figure 8.
This example has been carried out around the H208 nacelle, (an Aerospatiale nacelle which was tested in a windtunnel at ONERA) in a subsonic case (Minf = 0.30). It required 7 profile modifications performed in one minute on an Alliant FX2800. Transonic cases have also been tested and have led to similar conclusions with a slight increase in CPU time.

To conclude this section, it may be stated that the method presented here is a powerful tool for the design of turbomachinery blading. It is currently being applied in the French industry for the definition of high performance turbomachinery.

Parallel research has been going on with the goal of opening a new field of application in the domain of engine inlet design for which the methcd has proven suitable. Improvements are still being worked on especially in the field of mesh definition for turbine applications. The method has naturally some limitations. One of these is the built-in potential approximation which in fact leads us to the next section devoted to the transonic inverse and semi-inverse method developed initially by Meauzé at ONERA.

4. TRANSONIC INVERSE AND SEMI-INVERSE METHOD [8]

Whereas the method described above solved the potential equation, the one under consideration here deals with the Euler equations which allow for the occurrence of shock waves within the flow field .
This method was first developed by Meauzé in the early eighties as a follow-up of the transonic blade-to-blade direct calculation developed at ONERA by Viviand and Veuillot [11].
These authors made a valuable contribution to the resolution of the Euler equations by using time-marching methods where time is only a computational parameter and the final asymptotic flow field is obtained as the steady solution of the equations.

4.1 Overall description and concepts

The basic features of these methods can be summarized as follows:

- the quasi-three-dimensional Euler equations are discretized in the physical plane;
- a McCormack type predictor-corrector numerical formulation is used;
- when strong pressure or velocity gradients occur, an artificial viscosity is used to smooth out numerical instabilities;
- boundary conditions (wall boundary conditions or inlet/outlet boundary conditions) are treated via compatibility relations which are derived from the theory of characteristics.

Using this framework, Meauzé developed an inverse method in which the standard zero normal velocity boundary condition on the profile can be completely or only partly replaced by a static pressure (or velocity) condition. Whatever the case, the boundary condition problem is always dealt with via the compatibility relations. When operating in inverse mode, the profile and consequently the grid system must be updated. This can be accomplished either through reconstruction of the blade surface by using the flow angle computed at each wall grid point or, more rigorously, via a transpiration model like in the previous method.

4.2 Inverse and semi-inverse methods

What makes the method especially attractive for the designers is the fact that not only does it allow the defining of blading in the transonic regime, but it can also operate in the semi-inverse mode. This makes it possible to apply a given boundary condition on one part of the profile - say a pressure distribution - while retaining for instance the initial geometry on another portion of the blade. Localized corrections of the geometry can therefore be implemented in order to improve the overall aerodynamics of the blade.
Of course, for such applications, special care must be taken at the junction between the direct and inverse calculations. This is especially true when the flow is locally subsonic; then a smooth transition from the prescribed to the computed pressure distribution is required.
On the other hand, for locally supersonic flows, jumps in static pressures are allowed which would correspond to crossing shock waves or expansions.
One interesting version of the code allows prescribing of the pressure distribution on only one blade surface - generally the suction surface - while the other surface is determined from purely geometrical considerations, such as a thickness distribution.
One may note that, in this case, the cascade solidity may be chosen in advance since the profile is automatically closed. However, one drawback is the lack of control over the velocity distribution on the surface for which the pressure distribution was not prescribed. Moreover, two solutions to the problem can exist. Numerical experiments have demonstrated that only solutions corresponding to small flow deflections are stable. Therefore, this method is really only suitable for compressor applications.

4.3 Numerical results

Three examples will be presented: the first corresponds to the definition of a high supersonic blading on the second stage of a rocket turbopump; the second one is devoted to the design of a high pressure ratio turbine cascade; finally, the third application deals with the definition of a supersonic compressor profile.
These three cases have been selected to give examples of the various modes of operation of the method and will demonstrate its versatility.

High supersonic turbopump rotor

The case considered here corresponds to the redesign of the mean section of the high supersonic rotor of a rocket turbopump. For this configuration, the direct calculation on the original blade showed that the upstream flow was started, i.e., the inlet flow angle is fixed by the unique incidence phenomenon. An inlet relative Mach number of 1.22 was obtained for an inlet angle of 48.5°. The results of this direct blade-to-blade calculation are shown in Figure (9a). Strong shock waves are observed throughout the blade channel with a strong normal shock on the suction side.

An attempt was made to improve the situation with the inverse method operating in its semi-inverse mode. The pressure distributions were prescribed on the pressure and suction surfaces but only over part of the blade. In fact, for this case of supersonic inlet flow, the goal was to leave the inlet conditions undisturbed in order to guarantee adequate matching between the blade rows. The blade entrance region consists of a straight part on the suction side. The slope of this straight portion is chosen such as to obtain the specified unique incidence computed with the original blade. The pressure distribution is then prescribed downstream of this entrance region. Figure (10a) shows the selected distributions; on the pressure side, the flow becomes subsonic and the pressure gradient is chosen so as not to cause boundary layer separation. A smooth pressure distribution is prescribed on the suction side where the impingement of the shock has been deleted.

The resulting pressure field is presented in Figure (10b). An oblique shock is observed at the leading edge on the pressure side. On the suction side, a sharp change in the slope of the surface is observed which compensates for the impinging shock. The calculated relative inlet Mach number is 1.21 and the computed inlet angle 48.3°. These are in good agreement with the results of the direct blade-to-blade calculation on the original blade.

The result of the direct blade-to-blade calculation on the redesigned rotor profile is shown on Figure 11. Good agreement is likewise observed between the inverse and direct calculations.

High pressure ratio turbine cascade

Here again, the code is used in its standard semi-inverse mode for which the pressure distribution was prescribed on both blade surfaces but only downstream of certain points on the surfaces.

Upstream of these points, the initial geometry of the blade is retained and the method operates as a direct blade-to-blade computation. Figure (12a) shows, as broken lines, the initial pressure distributions with suction side non-uniformities. Also presented are the prescribed pressure distributions shown in solid lines. The blade shapes corresponding to these pressure distributions are shown in Figure (12b). Again, the solid line corresponds to the modified blade. Note that the solidity has changed, with a slight increase of the pitch.

Supersonic compressor cascade

This case is a typical example of the method described above where the blade is defined using a mixed type of aerodynamic and geometrical data. Here, the method is applied to the design of a supersonic compressor profile with an inlet Mach number of 1.2. Figure(13a) presents the initial pressure distribution where a shock at a peak Mach number of 1.6 occurs on the suction side near the trailing edge causing an increase in the loss and probable separation. The new blade is now obtained by tailoring the suction side pressure distribution so as not to exceed a peak Mach number of 1.42. The initial blade thickness distribution is retained.
Figure (13b) shows the new profile compared to the initial one. As can be seen, the difference between the two geometries is very small (which, by the way, ought to make us wonder what really happens in the machine when all manufacturing deviations have been taken into account).
The newly computed pressure distribution on the pressure side is also presented in Figure (13a). It exhibits a rather irregular shape especially in the trailing edge region. This is due to the evolutions of the pressure side curvature in this rear part of the blade which necessarily "follow" those of the suction side since the thickness distribution is prescribed.
This is one of the drawbacks of the method although a local correction of the blade on the pressure side can usually improve the situation without deteriorating the suction side pressure distribution.

4.4 Current developments

As stated earlier, this inverse Euler code must really be considered as a by-product of the direct blade-to-blade calculation. As a consequence, a major overhaul of the code is under way which reflects the improvement brought to the direct flow computation.

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
ICIDES-III, Editor G.S. Dulikravich, Washington D.C., October 23-25, 1991

Most of these improvements have been obtained on the mesh itself where the standard H-grid has been replaced by an H-C or H-C-H one with a multi-domain approach (compatibility relations are used at the boundaries between the domains). The improvement is especially to be found in turbine applications where round leading edges can be properly modelled (see Figure 14).
In a parallel effort, the algorithm has been modified in such a way that the inverse mode and the profile modification procedures are now only applied after convergence has been achieved on a given intermediate geometry. Although this brings about some penalization of the computing time, this approach gives better quality solutions.

CONCLUSION

Two quasi-three-dimensional inverse methods have been described above. Taken as a whole, they allow the defining of turbomachinery blade profiles throughout the entire Mach number range of interest for jet engine (even rocket engine) rotating components. Examples have been presented for compressor and turbine profile designs.
Both methods are currently being used throughout the French industry. A parallel research effort is still under way to improve them and extend their fields of application. The next step will certainly include coupling with a boundary layer calculation in order to better predict viscous effects.
It is obvious, however, that even such improved methods will have their limitations. The next significant step in turbomachinery design will have to be found in optimization techniques similar to the ones developed for external aerodynamics. Although some progress has been observed in this domain in the recent past, it is still widely believed that a breakthrough in the field of fundamental mathematical analysis will be required in order to formulate this complex multi-parameter problem.

ACKNOWLEDGMENT

The authors wish to thank D. Nicoud and B. Petot from SNECMA and G. Meauzé from ONERA for their contributions to this paper.

References

[1] Lighthill, M.J., "A New Method of Two-Dimensional Aerodynamical Design," R. & M. No.2112, Aeronautical Research Council, London, 1945.

[2] Volpe, G. and Melnik, R.E., "The Role of Constraints in the Inverse Design Problem for Transonic Airfoils," AIAA Paper No.81-1233, AIAA 14th Fluid and Plasma Conf., 1981.

[3] Sloof, J.W., "A Survey of Computational Methods for Subsonic and Transonic Aerodynamic Design," Proceedings of the International Conference on Inverse Concepts in Engineering, Austin, TX, U.S.A., 1984.

[4] Meauzé G., "Overview of Blading Design Methods," Agard Lecture Series No. 167, Blading Design for Axial Turbomachines, June 1989.

[5] Stanitz, J.D., "Design of Two-Dimensional Channels with Prescribed Velocity Distributions along the Channel Walls," NACA, Rep 1115, 1952.

[6] Cedar, R.D., and Stow, P., "A Compatible Mixed Design and Analysis Finite Element Method for the Design of Turbomachinery Blades," International Journal for Numerical Methods in Fluids, Vol. 5, 1985, pp. 331-345.

[7] Jacquotte, O.P., "An Inverse Finite Element Method for the Design of Turbomachinery Blades," 7th Int'l Conference on Finite Element Methods in Flow Problems, Huntsville, U.S.A, April 1989.

[8] Meauzé, G., "An Inverse Time-Marching Method for the Definition of Cascade Geometry," Journal of Engineering for Power, Vol. 104, July 1982.

[9] Nicoud, C., Le Bloa, C., and Jacquotte, O.P., "A Finite Element Inverse Method for the Design of Turbomachinery Blades," ASME Paper 91-GT-80, Orlando, June 1991.

[10] Bredif, M., "A Fast Finite Element Method for Transonic Potential Flow Calculations," AIAA Paper 83-0507, AIAA 21st Aerospace Sciences Meeting, Renc, U.S.A 1983.

[11] Viviand H., and Veuillot, J-P., "Methodes Pseudo-Instationnaires pour le Calcul d'Ecoulements Transsoniques," Publications ONERA No. 1978-4. English Translation ESA TT 549.

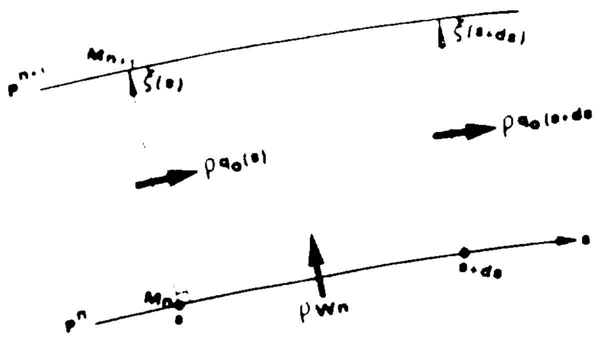

Fig.1 : Profile modification

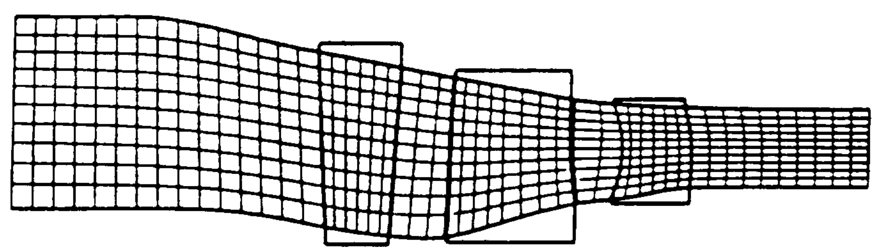

Fig.2 : Through-flow calculation

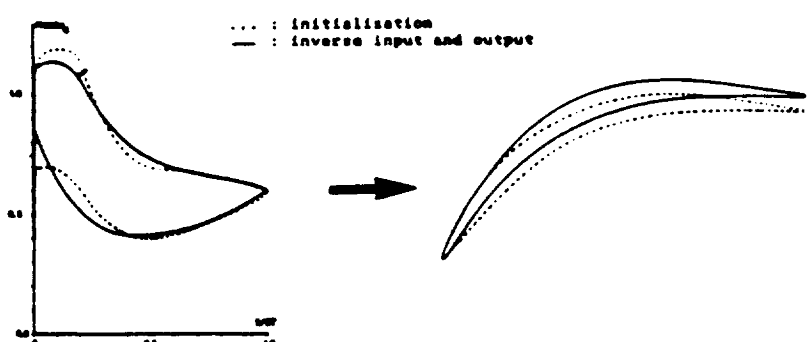

Fig.3 : Required Mach number distribution
and final profile after 3 modifications

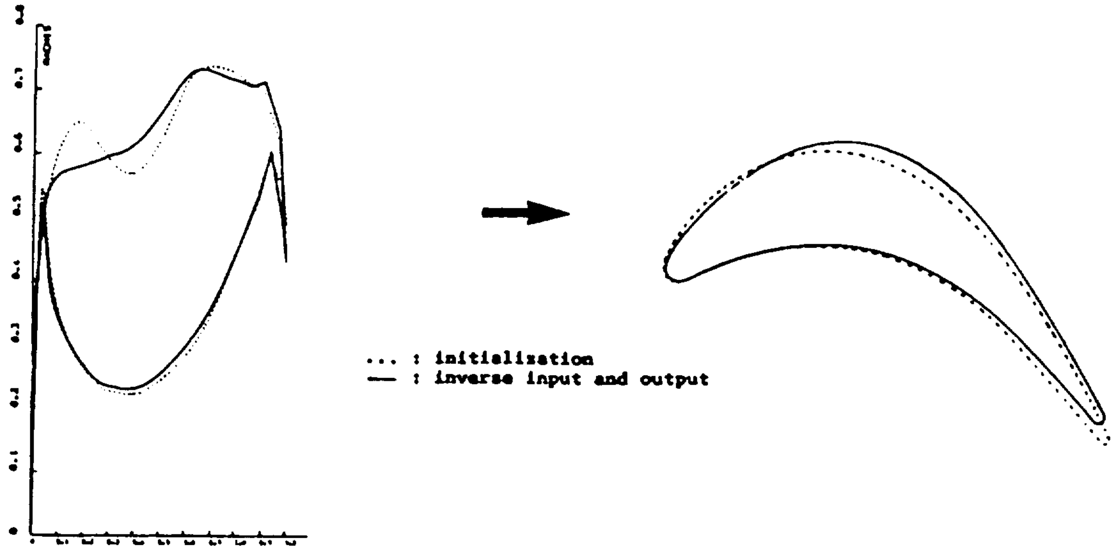

Fig.4: Required Mach number distribution and final profile after five modifications

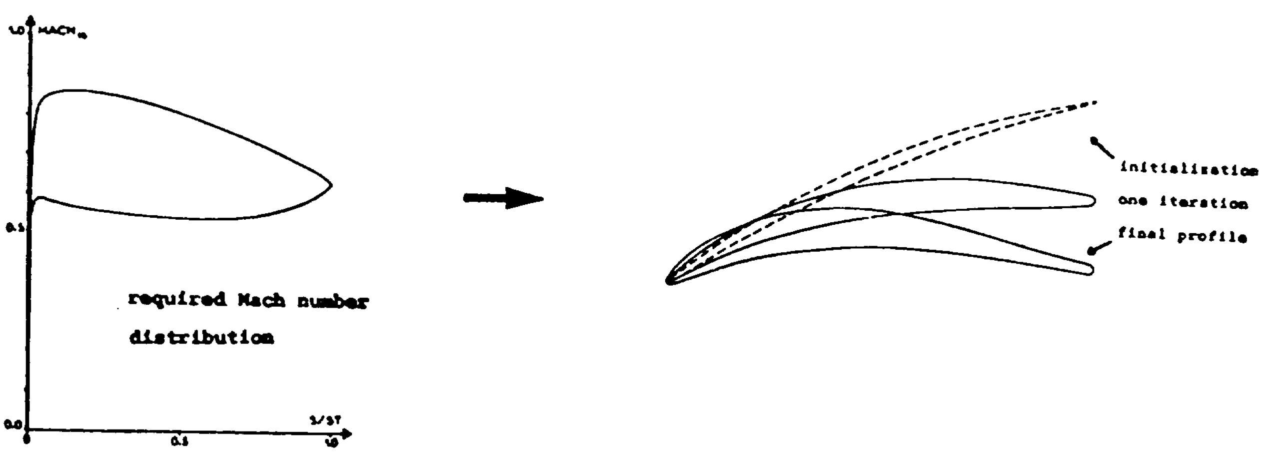

Fig.5: Subsonic example with a poor initialization

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences (ICIDES-III), Editor G.S. Dulikravich, Washington D.C., October 23-25, 1991.

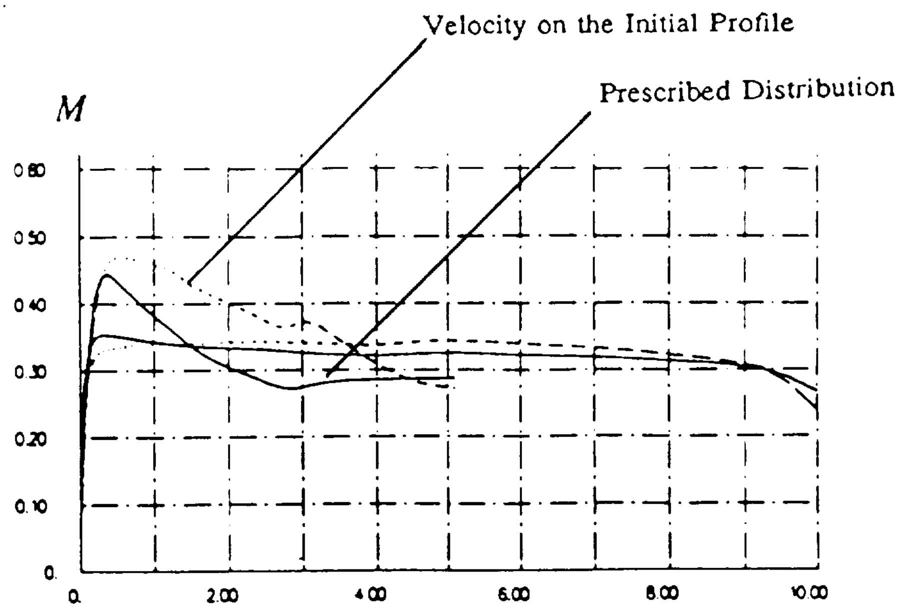

Fig.6 : Prescribed Velocity and Velocity around the Initial Profile

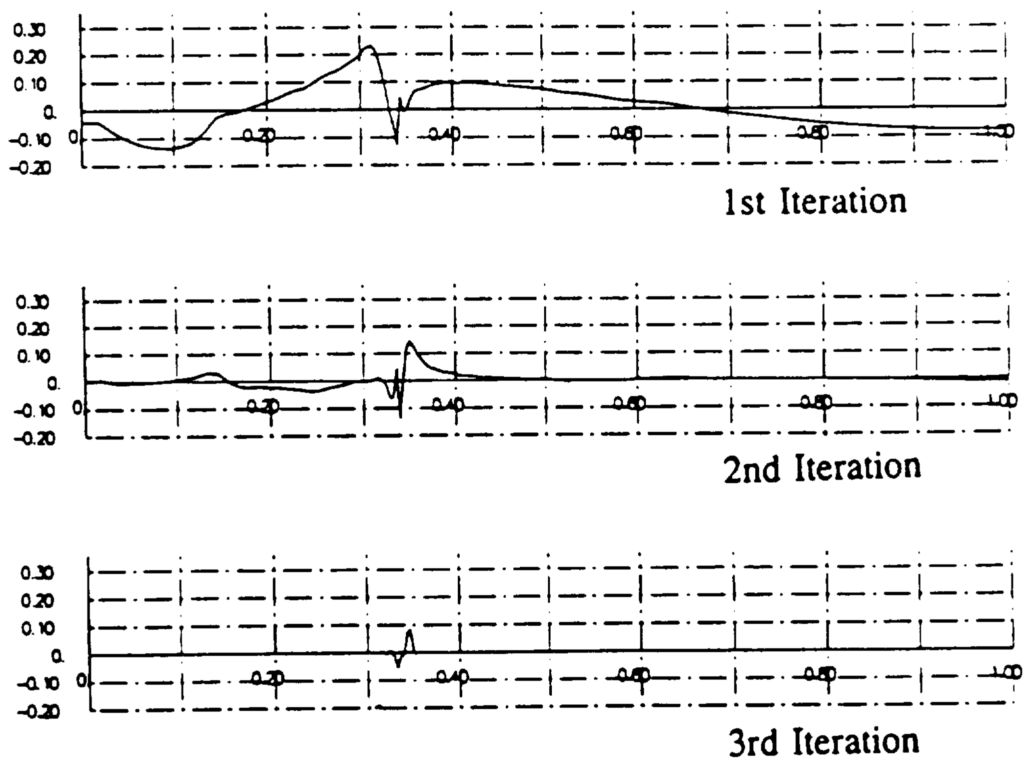

Fig. 7 : Velocity Normal to the Profile

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences (ICIDES-III), Editor G.S. Dulikravich, Washington D.C., October 23-25, 1991.

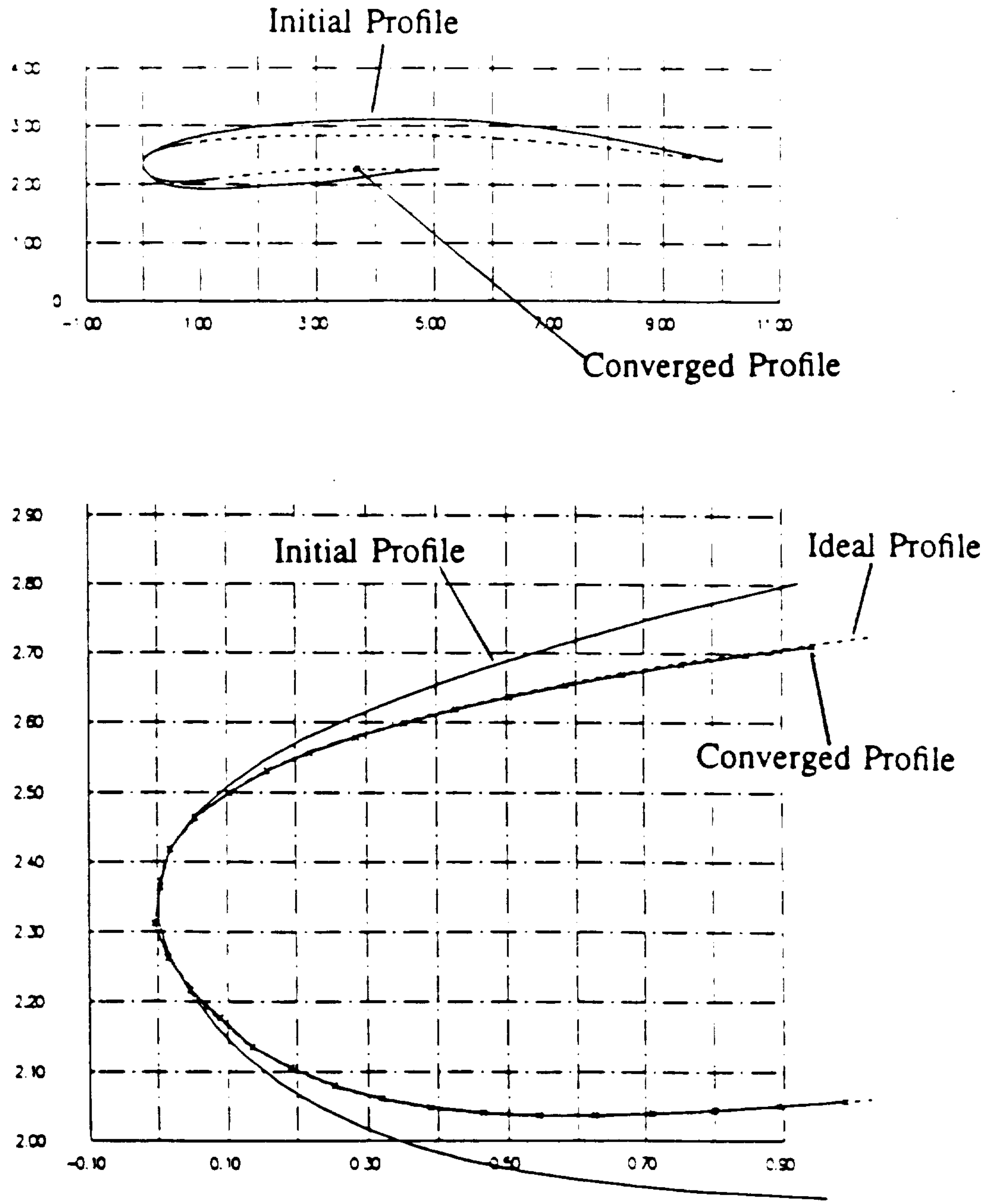

Fig.8 : Modification of the Profile

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
(ICIDES-III), Editor: G.S. Dulikravich, Washington D.C., October 23-25, 1991

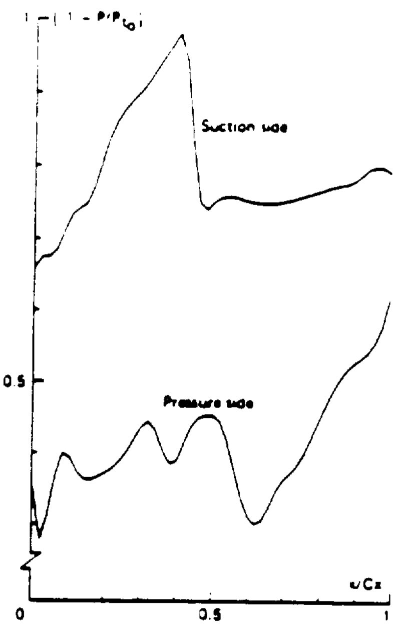

Fig.9a – *Mid span section of the second rotor : pressure distribution.*

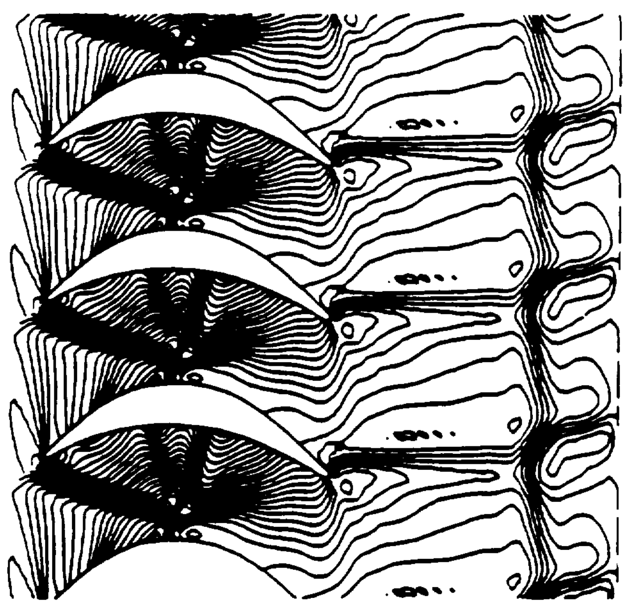

Fig.9b – *Mid span section of the second rotor : isobaric lines.*

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
(ICIDES-III), Editor: G.S. Dulikravich, Washington D.C., October 23-25, 1991.

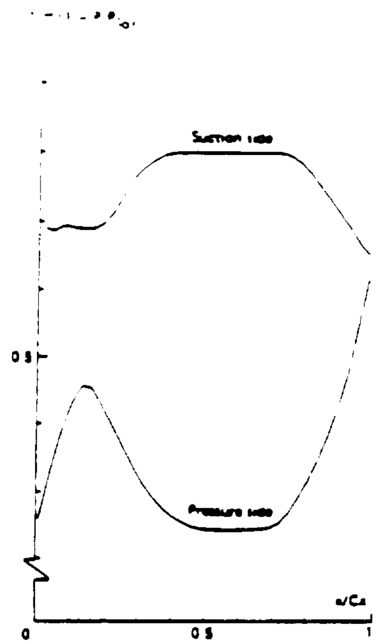

Fig.10a – *Second rotor design : pressure distribution (mid span section).*

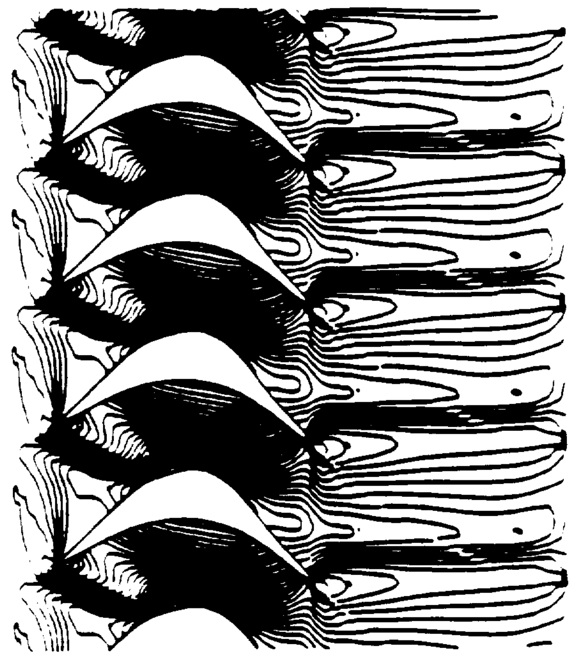

Fig.10b – *Second rotor design : isobaric lines. (mid span section).*

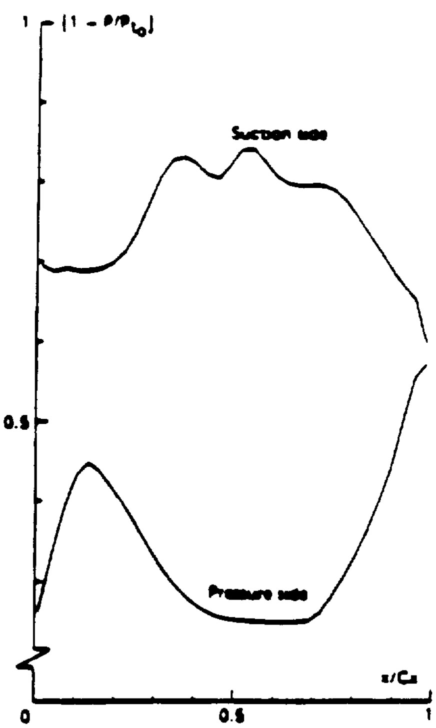

Fig.11 – *Second rotor design : pressure distribution (direct mode calculation).*

ORIGINAL PAGE IS
OF POOR QUALITY

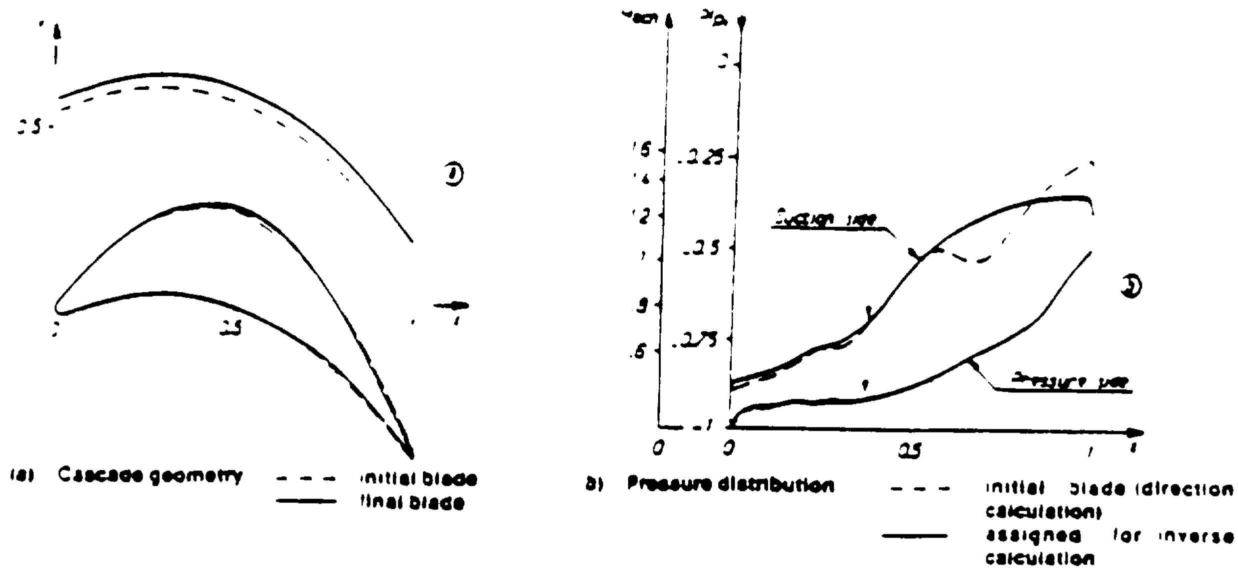

Fig.12 - Definition of a turbine blade and the cascade pitch, for pressure distributions assigned on the pressure and suction sides

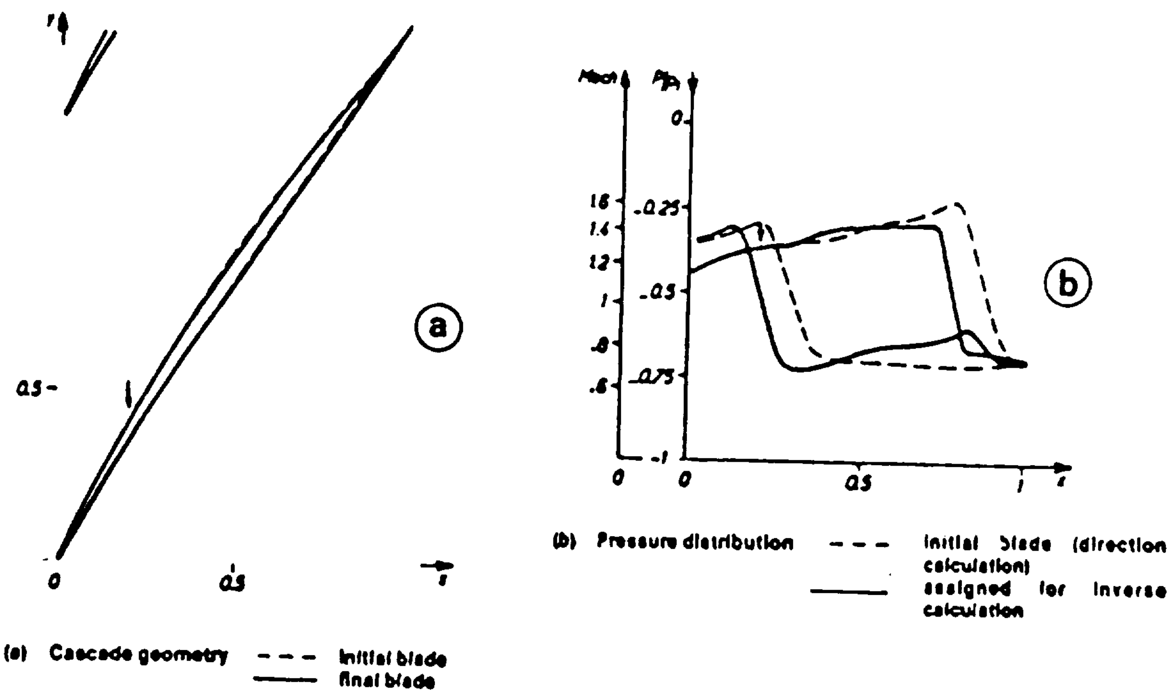

Fig.13 - Definition of a section of supersonic compressor

Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences
(ICIDES-III), Editor G.S. Dulikravich, Washington D.C., October 23-25, 1991

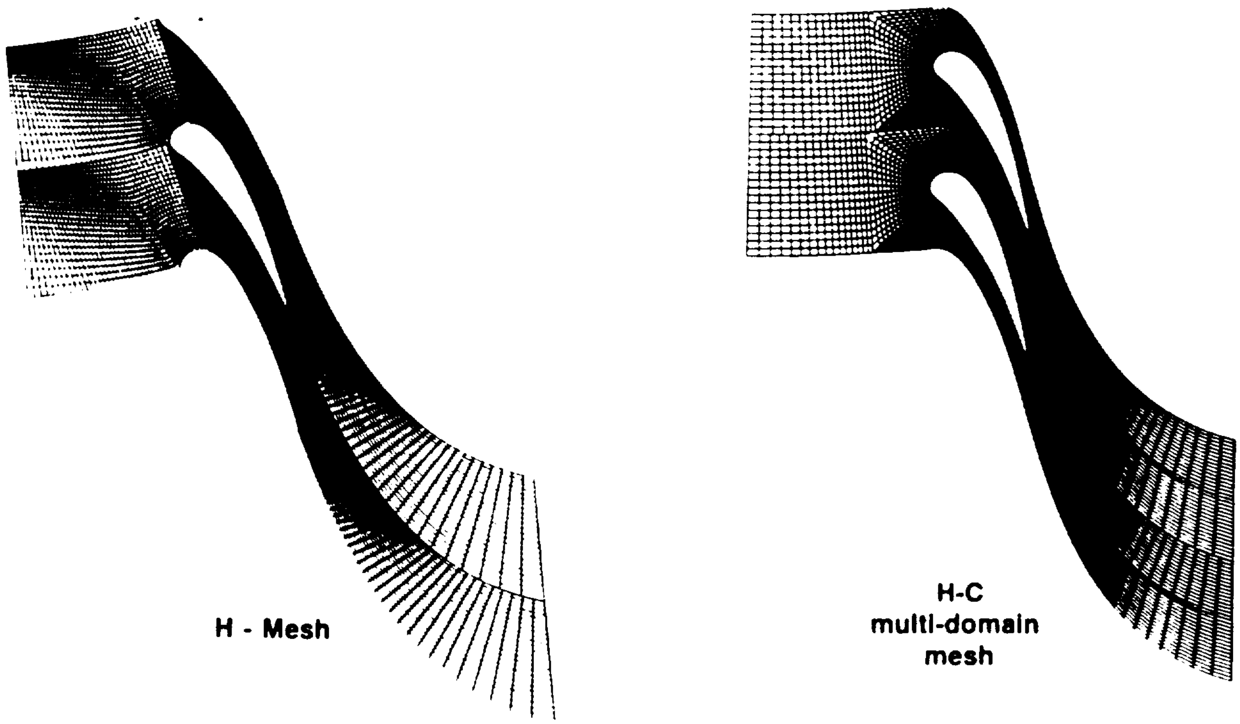

Fig.14 : Comparison of mesh types
for turbine applications