ICAS-88-2.6.1

3D FLOW COMPUTATIONS IN A CENTRIFUGAL COMPRESSOR WITH SPLITTER BLADE INCLUDING VISCOUS EFFECT SIMULATION

Valérie Millour*
Office National d'Etudes et de Recherches Aérospatiales
BP 72 - 92322 Châtillon Cedex, France

Abstract

Optimization of centrifugal compressors is based on the knowledge of the inner aerodynamic characteristics of the impeller and more precisely the rotor "alone". This paper describes the results of a 3D non-viscous steady flow computation in a centrifugal rotor with splitter vanes. The splitter may be in various positions and have any shape. The software used has been developed at ONERA and tested in several turbomachines cases. It consists of a general computer code solving the full Euler equations directly in the absolute physical space in cylindrical coordinates.

Introduction

This work aims at a better understanding of the inner phenomena of a transonic or supersonic flow through a centrifugal rotating wheel with high pressure ratio and large specific mass flow. The flow in a centrifugal rotor is quite complex because of the centrifugal and Coriolis forces action in the radial outlet on a flow which is already degraded by 3D deflection imposed by the curvature.

The well-known 2.5D approach combines a so-called blade-to-blade calculation on streamsheets which are supposed to be axisymmetric and a meridional calculation which gives those streamsheets. It appears that this approach which deals with 2D flows, partially represents 3D effects. A full 3D computation seems then absolutely necessary because it takes into account the warping of the streamsheets through the rotor. Moreover, the higher the load and the longer the blade-to-blade channel, the greater the phenomenon is.

Computational method

The time marching code for a 3D full Euler equations integration in turbomachinery cascades uses an explicit finite-difference predictor-corrector scheme. The equations are discretized using McCormack scheme directly in the physical space on an arbitrary curvilinear mesh[1]. An artificial dissipation stabilizes the calculation. Boundary conditions are treated using the characteristics theory, by means of compatibility relations. At the upstream boundary stagnation conditions of pressure P_t and temperature T_t, and the absolute flow angles are imposed. At the downstream boundary, a static pressure condition is fixed.

In turbomachine calculations, absence of viscous effects can lead to great differences with the real flow. We use a simplified representation of the viscosity to improve the validity of the method. Instead of very sophisticated calculations using Navier-Stokes solvers, viscous effects are simulated by introducing in the Euler equations a surface force on the whole wet walls. Therefore, losses are introduced through a parietal slowing down by skin friction and diffusion[2] and they can induce a separation. The numerical implementation of this model is quite easy. However as the mesh transverse size has an influence on the calculation, thus some coefficients have to be introduced and optimized ; they can be defined as characteristics thicknesses of viscous layer.

Application on centrifugal compressors

It is very important to have a great number of grid points to compute such a configuration and obtain results with high precision. Indeed, gradients are much stronger in a centrifugal than in an axial impeller and a fine mesh of roughly 45000 points ($150\times26\times12$) is necessary for a good discretization ; that implies a longer time for one iteration; besides, time step is shorter and the number of iteration greater. At least 30000 iterations are required to achieve convergence (which corresponds to more than one rotation of the rotor, necessary for the steady phenomena to be established). To run such a computation, time and memory requirements are rather considerable. On CRAY XMP, we need approximately 7 to 8 hours CPU time, and the maximum file length (MFL) is about 1200000 words. Moreover, a large amount of data is obtained. Therefore a special effort has been made to ensure the presentation of these results is easy and clear, and to get a quick interpretation of the operating field compressor.

The blade-to-blade and meridian meshes are given respectively on Fig 1 and Fig 2. The upstream mesh is classical and similar to an axial one. On the contrary, at the outlet, where we apply a simplified radial equilibrium and simulate a downstream infinite, we find the following problem: it is necessary to go far from the rotor in order to avoid the influence of the arbitrary conditions imposed at the downstream boundary (one supposes the uniformity of the static pressure P_s in the tangential direction, and moreover on the complete section as the flow is radial at the outlet). Besides the higher the radius, the more important the blade-to-blade distance is. In opposite to axial impellers for which the flow is not very deflected, gradients are quite important in centrifugal impellers because of the radius increase. More especially, if we consider a constant channel evolution, the flow rate velocity decreases dramatically and the absolute angle tends to 90°; that can lead to numerical difficulties. To offset those difficulties, a decrease in the channel width has ben induced.

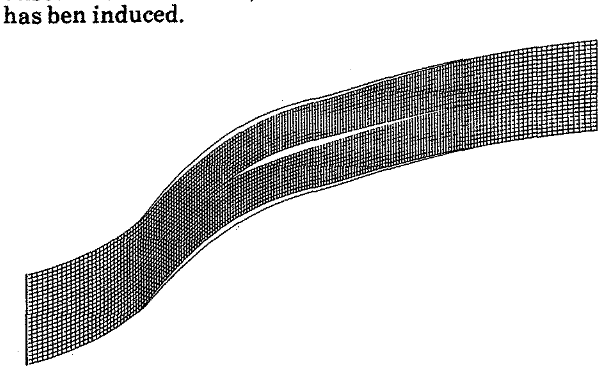

Fig. 1. Blade-to-blade mesh.

* Research Engineer, Energetics Department

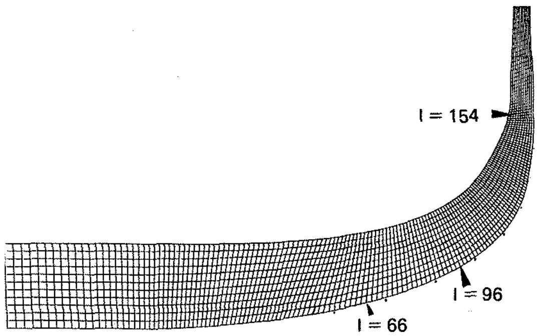

Fig. 2. Meridian mesh.

An additional problem appears with the value of that static pressure necessarily assigned on the downstream boundary: on the one hand, experience shows that the iso-velocity characteristic of a centrifugal rotor corresponds to a slight variation of P_S. On the other hand, strong gradients, shock wave and local supersonic flows make it difficult to choose the pressure corresponding to a realistic configuration. For an over estimated pressure, the flow is completely unshocked, further it is impossible. Too low a pressure level corresponds to a configuration with strong shock waves. In both cases, numerical difficulties can occur.

Besides, at a given pressure, at least 20000 iterations are required to obtain stabilized results and submit them to analysis. To avoid such delays, achievement of the convergence can be accelerated by first using a coarse mesh. With a coarse mesh of about 20000 points, we reach convergence after 10000 iterations for less precision but much less memory and time. Then, the initialization of the new calculation corresponding to the fine mesh is defined by using a special interpolation code on the previous points. Finally, high level precision points are computed for less than 10000 iterations.

The introduction of splitter blade brings no basic modification in the calculation scheme. Let us notice that centrifugal compressors usually require a small number of blades in the inducer in order to avoid high loss levels with large flow incidence angles. On the other hand,the use of splitter blades in the radial part enables the reduction of the blade-to-blade loading while providing a better energy transfer to the flow. Therefore, the blades can be thinner than in a configuration without splitter vanes.

Results without losses

Mach contour lines are shown on Fig 3a and Fig 3b for rotor hub and tip sections at a fixed downstream pressure.

Analysis of the tip blade-to-blade channel (Fig. 3a) shows a supersonic incident flow till behind the blade entrance. Then a supersonic expansion and before the splitter vane a normal shock wave occur. Beyond the flow is subsonic in both subchannels. Notice the concentric organization of the Mach contours because of the Coriolis force action and the flow leading. At the outlet, the sudden change of this configuration is associated with the flow leading rupture.

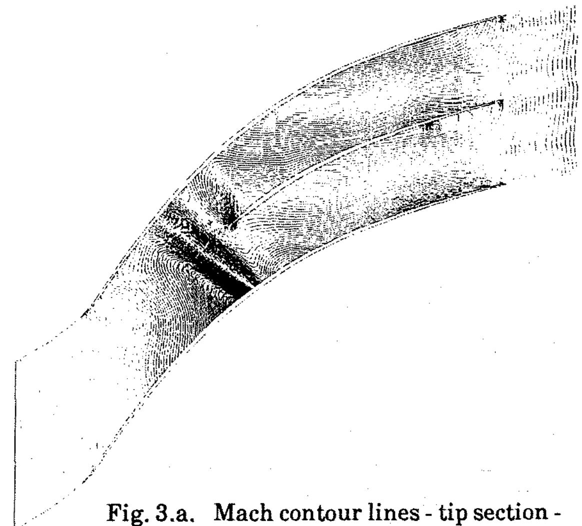

Fig. 3.a. Mach contour lines - tip section - without loss.

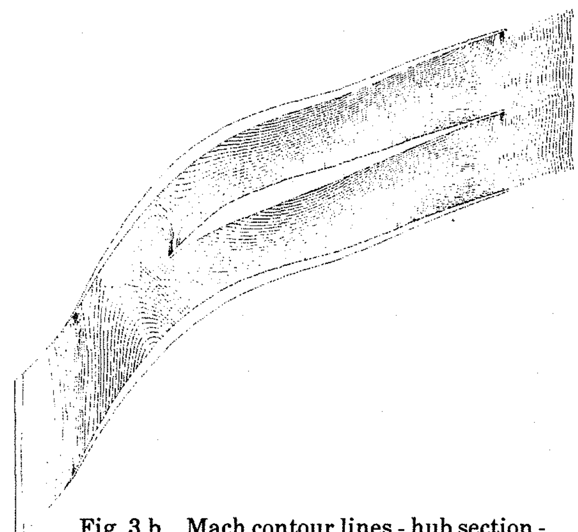

Fig. 3.b. Mach contour lines - hub section - without loss.

The hub section on Fib 3b is completely subsonic. The Mach contour organization is similar to the tip section. A strong slow down area is located at the main blade suction side towards the outlet.

Fig 4.a,b,c,d give the Mach number distribution on the blade suction and pressure sides and confirms the previous analysis. One observes that the introduction of the splitter blade allows to keep moderate values of the lift on the blades.

Fig 4a shows the position of the normal shock wave.

The mass flow balance from hub to tip (Fig 5) is described for 3 locations indicated on Fig 1. The first section is located at the splitter blade entrance (I=66); the second is an oblique, which is more axial than radial and the last one is at the rotor outlet. At the splitter blade entrance, the mass flow is more important on the tip, due to the action of the centrifugal and Coriolis forces. This phenomenon increases till the mixed section (I=96). At the outlet (I=154), the flow sets against the hub section. The streamlines pattern on Fig. 6.a,b confirms those statements. Moreover, the warping effect obviously appears on the blades. On the suction sides the flow sets against hub. This phenomenon reverses on the pressure sides. Notice the appearance of vortices on the splitter blade pressure side.

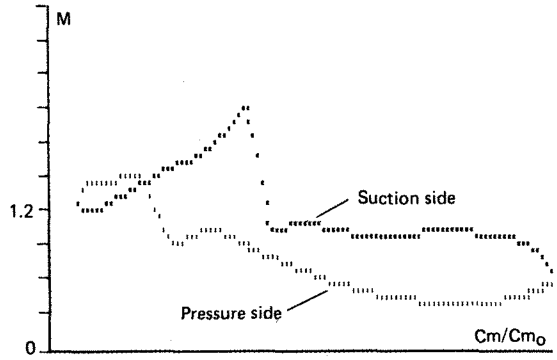

Fig. 4.a. Relative Mach on the main blade - tip section.

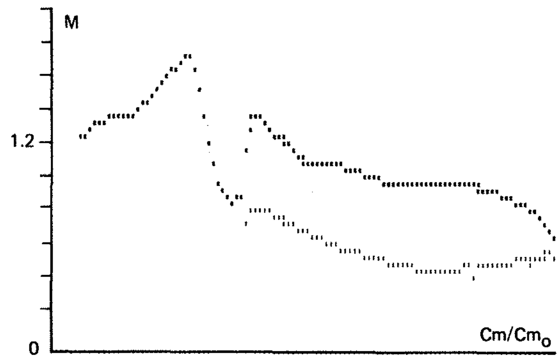

Fig. 4.b. Relative Mach on the splitter vane - hub section.

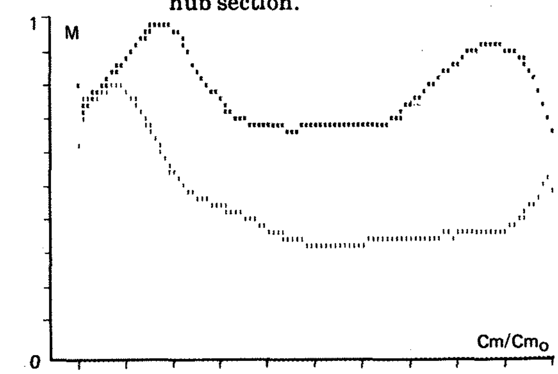

Fig. 4.c. Relative Mach on the main blade - hub section.

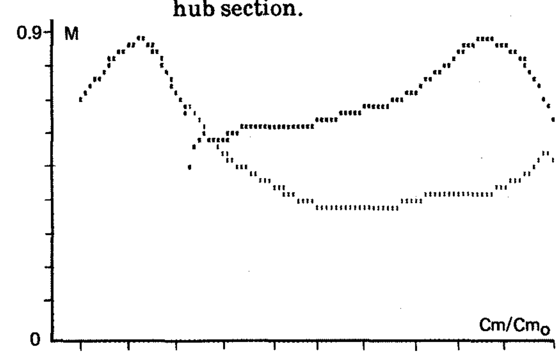

Fig. 4.d. Relative Mach on the splitter vane - Tip section.

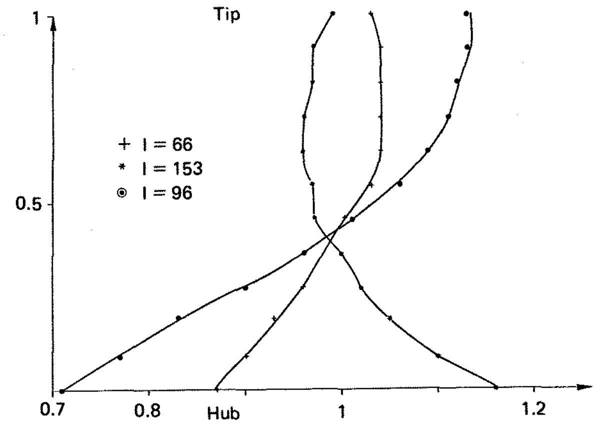

Fig. 5. Mass flow distribution from hub to tip.

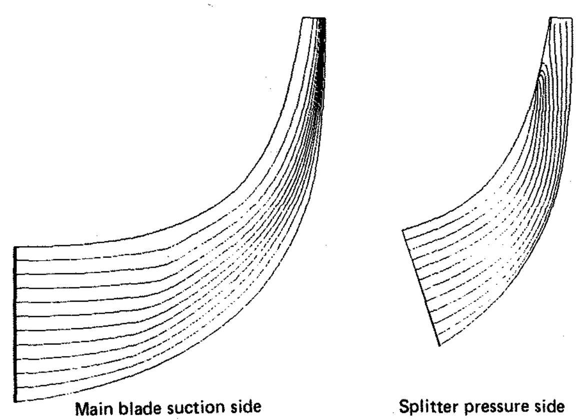

Fig. 6.a. Streamsheets pattern.

Fig. 6.b. Streamsheets pattern.

Fig 7 and Fig 8 show the 3D characteristic of the flow at the outlet of the rotor for 6 sections from hub to tip, respectively in absolute and relative coordinates.

- The variations of the ratio T_{t_2}/T_{t_0} of the absolute total temperature to the inlet total temperature T_{t_0}, are more an more important from tip to hub sections. As for P_{t_2} the absolute stagnation pressure, normalized by the reference P_{t_0}, the deviations are stronger on the boundary sections (Fig 7).

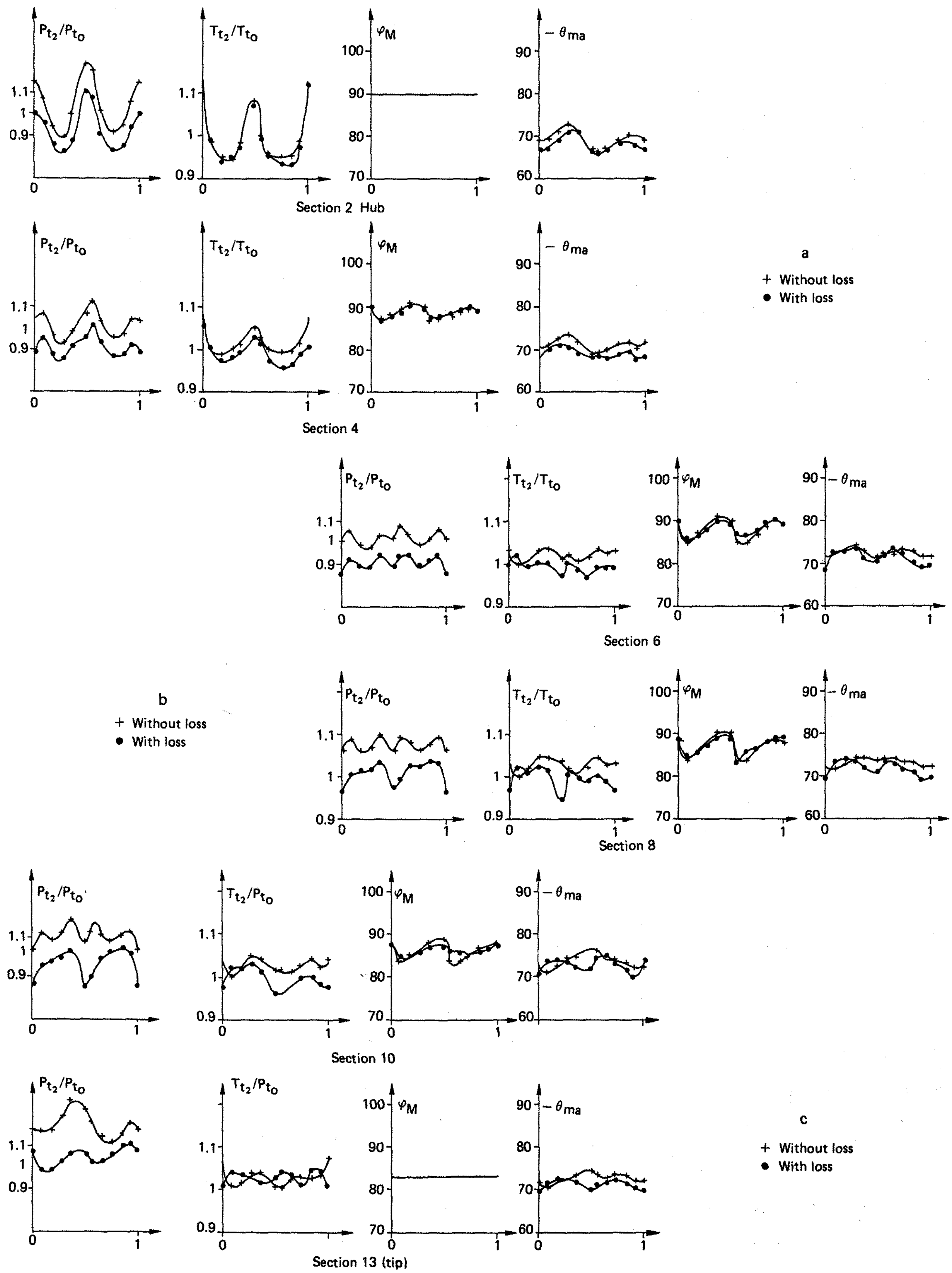

Fig. 7.a.b.c. Absolute flow at the rotor outlet.

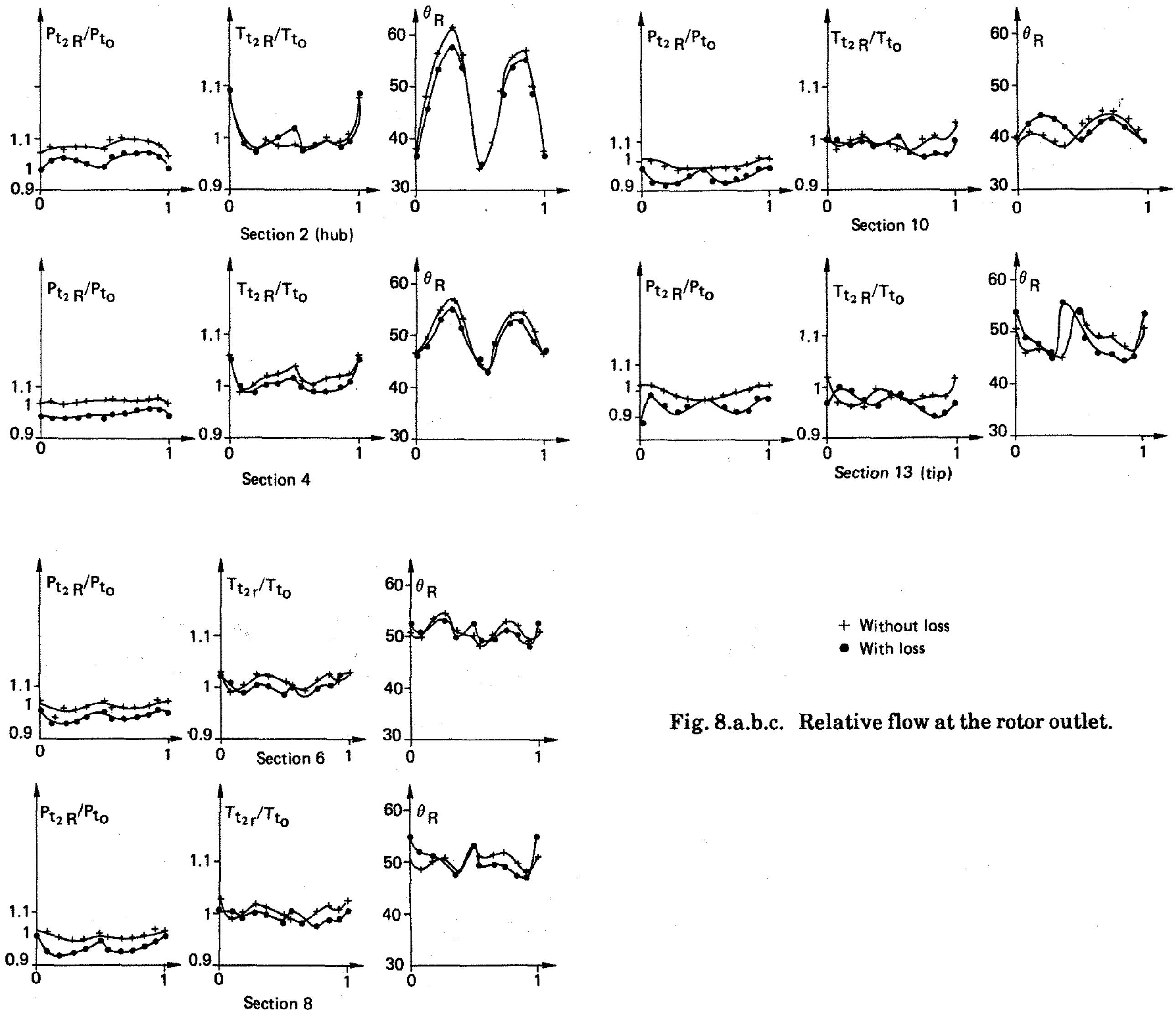

Fig. 8.a.b.c. Relative flow at the rotor outlet.

- The meridional angle ϕ_M is the best representation of the 3D warping effects, influent at the mid section (Fig 7).

- In relative cordinates, the slight oscillations of the total pressure $P_{t_{2R}}$ normalized by the reference P_{t_0} come from entropy production phenomena. Relative flow angle (θ_R) evolution explains the great variations obtained for the ratio $P_{t_{2R}}/P_{t_0}$ in absolute coordinates: at the radial outlet, a small variation of θ_R *corresponds* in the velocity triangle to a great variation of the absolute flow. Though the great wake obtained in absolute describes the action of the Coriolis force.

- The relative total temperature $T_{t_{2R}}$, normalized by T_{t_0} shows a bad conservation, on the main blade at the hub section. We are conscious of that lack but we have no obvious explanation. However, this particularity appears negligeable compared to the extremely satisfactory global results.

Results with losses

An approximate simulation of the losses due to the viscous effects in a turbomachinery through flow is allowed by taking into account a friction force limited to the wall surfaces. This force is represented by a friction coefficient which is adapted for a given stage.

Initially this treatment was developed for axial impellers: in the case of axial rotors, losses are quite important. On the contrary losses in centrifugal are weak. Indeed all what occurs in the quasi-radial part and the centrifugal effect hold a benefic action on the losses. Therefore, the losses level which we introduce is quite weak and the impeller efficiency excellent (more than 0.9). That explains the fact that we have such slight discrepancies between the calculations with and without losses (Fig 7,8,9).

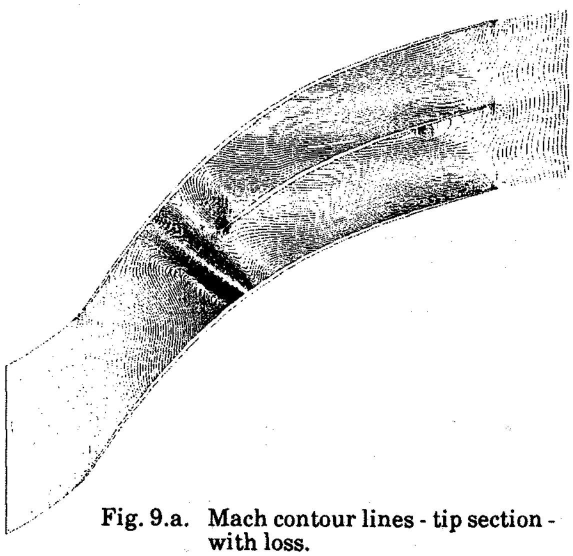

Fig. 9.a. Mach contour lines - tip section - with loss.

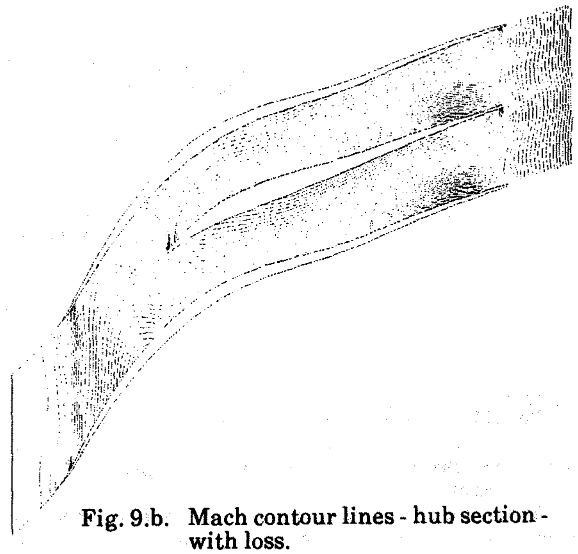

Fig. 9.b. Mach contour lines - hub section - with loss.

Conclusions

This code can be used by manufacturers to predict and estimate the performance of new machines.

Although the computations have been made for perfect fluid flows, the 3D code allows various investigations. Moreover, as the code makes it possible to model certain viscous effects, the computations and experiments should be improved.

References

1. Veuillot, J.P. and Meauzé, G., "3D method for internal transonic flows computation with a multi-domain approach". AGARD/PEP L.S. No. 140, Paris, 13-14, June 1985.

2. Meauzé, G. and Fourmaux, A., "Numerical simulation of flows in axial and radial turbomachines using Euler solvers ". (VKI 15-18 June, 1987).