AERODYNAMIC DESIGN AND OPTIMIZATION OF BLADE CONFIGURATION IN AN INLET STAGE OF AN AIRCRAFT ENGINE COMPRESSOR

Olga Marakueva
Department of Plasma- and Gas Dynamics and Heat Engineering
Baltic State Technical University

Keywords: *axial compressor, blade geometry, CFD-simulation, aerodynamic optimization*

Abstract

Design of turbomachines includes aerodynamic computation and optimization of blades. The present paper describes the optimization process and the results for an inlet compressor stage of a turbojet engine. Very often optimization is aimed at a simultaneous improvement of the efficiency and minimization of the total pressure loss in every compressor row. Objective of this work is to construct a rotor blade geometrical configuration on the basis of the mentioned performance criteria

1 Introduction

At the present time the use of CFD simulation flow in turbomachines is widely applied in the research and developments, and also in designing for defining the characteristics and optimization.

In common the flow of working gas in turbomachines is three dimensional, however 3D simulation requires a great deal of CPU time, so practically in the process of flows calculation in axial machines the quasi 3D approach is often applied which is directed to perform 2D calculations in some sections according to the height of blades, herewith the calculation time is being decreased by a decade in comparison with full 3D simulation process. The big number of papers is devoted to applying of quasi 3D approach. (see, e.g. [1–4]).

The other important and often used simplification in the process of multistage turbomachines simulation is quasi-steady state method, when the averaged in circumferential direction output parameters of one row are set as input data for another row [5]. Quasi-steady state approach gives the possibility to consider the flow in turbomachines as well to approximation of Reynolds equations as using full Navier–Stokes equations in the relative frame connected with blades. Notwithstanding that above mentioned averaging introduces noticeable error the given approach has come in wide use in the process of multistage turbomachines integral characteristics evaluation. [6–9].

Parameters optimization approaches of turbomachines can be subdivided into 2 major classes: gradient and stochastic. Gradient methods have obvious advantages, for example the realization simplicity, but also the number of disadvantages. The major disadvantage is the necessity to calculate the derivates of objective function on each step. The second disadvantage is the absence of precision to absolute optimum.

Among the stochastic methods the major is the method based on the so called genetic algorithms [10, 11]. This method is significantly more universal; however in comparison with gradient methods it requires the big quantity of calculations, what makes the limits for its practical use.

From the aerodynamics point of view the turbomachines optimization is associated with providing a non-separated flow around blades and vanes that must decrease the losses caused by eddies dissipation (profile losses). In general, the total losses include the tip clearance, endwalls, and the secondary flows losses, as well as the shock losses. Not all above mentioned effects can be simulated in 2D calculations; however the profile and shock losses are simulated with a high precision.

The optimization of low pressure turbines blades with the use of quasi 3D method was examined in [2]. The criteria of optimization was represented by Mach number, the limits were represented by some geometrical parameters, maximal Mach number and minimal Mach number. However the quasi-3D calculations with the results of full 3D simulation process weren't represented.

Dutta *et..al*. [1] performed multi-objective optimization of single-stage compressor, using the quasi-3D method and stochastic method of optimization, as the result the Pareto front was found.

The given work was devoted to optimization of first stage of air craft engine compressor on the basis of genetic algorithm upon use of quasi-3D flow simulation and quasi-steady state approach. The main objective is the examination of blade geometrical parameters impact on the flow structure.

2 Compressor 3D calculation

In this paper the blade geometry of turboprop aircraft engine TV3-117 axial compressor first stage was chosen as the object for optimization. Geometrical characteristics and blades stagger angles of inlet guide vanes (IGV) and other vanes were fixed. The optimization process was implemented according to compressor working parameters, which corresponds "operating point" of aircraft cruise mode.

On the first stage the compressor characteristics calculation with blades original set geometry, obtaining of operating point parameters and comparison of obtained results with the test data are performed. In 3D calculations were examined the spokes, IGV and 12 stages "rotor-stator" (Fig.1).

In the whole computational area the structured hexagonal mesh with total number of cells app. 15.5 mln was built with the help of software module AutoGrid5(see Fig 2.). Mesh in blade channel is shown on the Fig.3.

Fig. 1. Flow path

The calculations for each row were carried out using equation solver Fine/Turbo with quasi-steady approach and the use of "mixing plane" treatment at an interface between adjacent rows. The mixing-plane treatment provides the conservation in mass, momentum, and energy across an interface. The Reynolds-averaged Navier–Stokes equations were spatially discretized in a cell-centred finite volume method framework with inviscid fluxes calculated using the central difference scheme. Time integration is achieved by using the four-stage Runge-Kutta method. The equations were solved with the k-ε turbulence model. As the boundary condition on the blades surface the wall function is applied for the tangent velocity component.

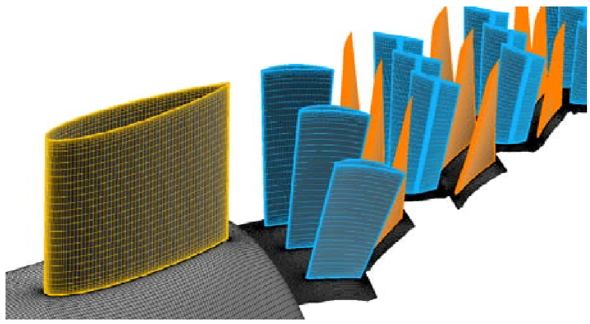


Fig. 2. 3D solid mesh

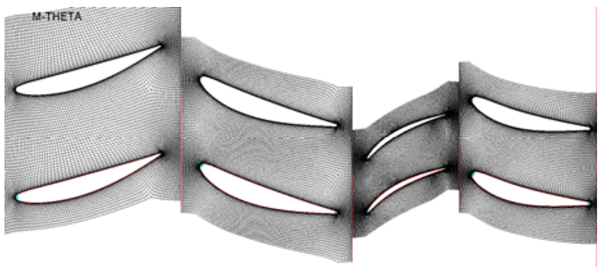


Fig. 3. Blade-to-blade mesh

Operating gas is air. In the capacity of boundary conditions the absolute total pressure and temperature were set as P*=101325 Pa and T*=288, 15 K in the inlet; and also speed axial direction. Rotational speed $n = 18525$ rpm. Upon the calculation of diagram vertical reach of characteristic curve π(G) (π – pressure ratio, G – flow rate) on the compressor outlet boundary was set static pressure, upon the calculation of horizontal reach –flow rate G (see diagram Fig.4).

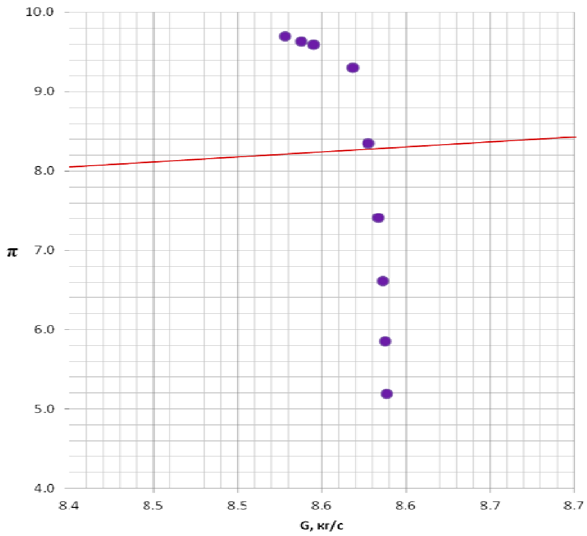


Fig. 4. Compressor characteristic obtained in calculations.

The max difference in flow rate G obtained in CFD simulation process and in the experiment upon the fixed pressure ratio π was about 4%, this can be explained according to the following reasons:

- Incorrect size and clearance configuration between blades and shroud, these parameters change in the process of exploitation because of blades material deformation;
- Clearance between regulated vanes were not taken into account in the process of connection;
- Non stationary effects were not taken into account in the process of rows interaction.

Operating point parameters were defined as crossing of the working regimes line (see the curve Fig.4), which characterize both the work of compressor and engine turbine with obtained projected characteristic curve.

3. 2D calculation

To minimize the error due to 3D effects, 2D turbomachinery calculations are usually made on meridional streamlines. In present paper in 2D calculations 3 cylindrical cuts with different spanwise location (h=25%, 50% and 75%) were considered. The geometry model included row of spokes, IGV, first and second stages (see Fig.5 and 6).

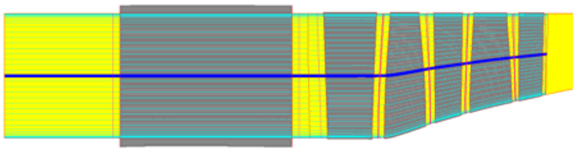


Fig. 5.Compressor flow path with streamlines.

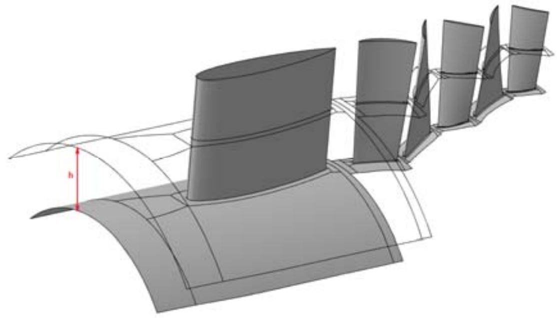


Fig. 6. 3D geometry model.

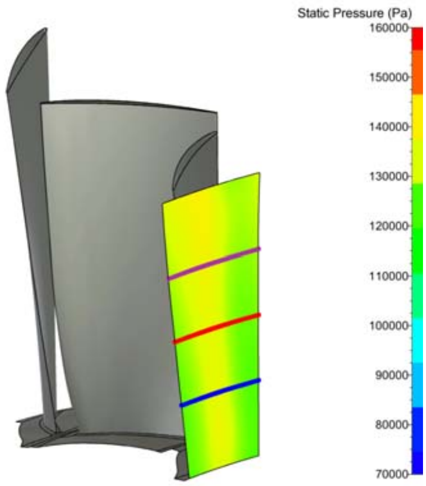


Fig. 7. Interface downstream the 2nd stage.

Static pressure was set as outlet boundary condition. It was obtained in 3D calculation and then averaged in circumferential direction for each considered cut (Fig.7 and 8).

Other boundary conditions, parameters and assumptions of task were the same as in 3D calculation.

On Fig. 9 and 10 you can see the distribution of static pressure in 2D and 3D calculations for section with height $h = 75\%$.

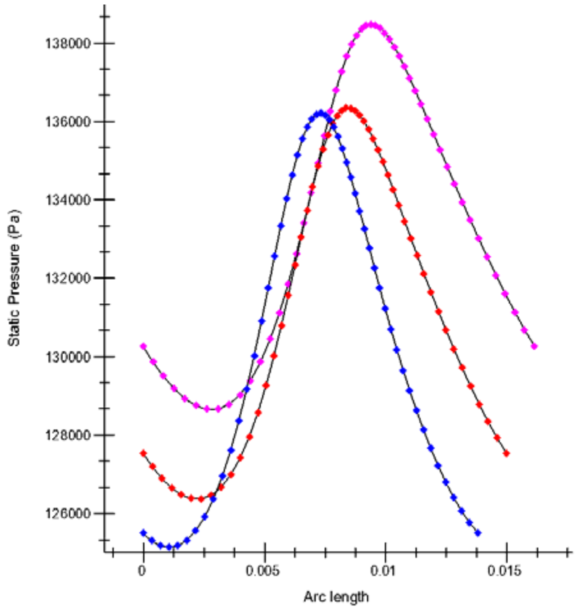


Fig. 8. Graphics of static pressure on interface downstream the 2nd stage for each considered section.

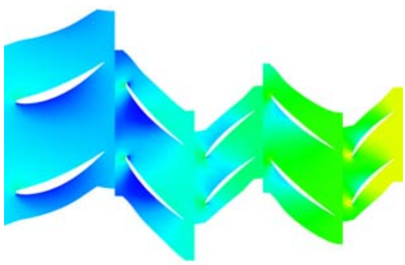
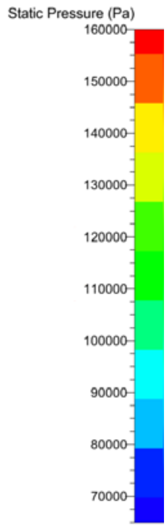


Fig. 9. Static pressure in 3D calculation



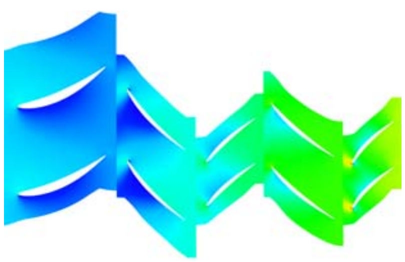


Fig. 10. Static pressure in 2D calculation.

4. Parameterization

Shape parameterization plays a very important role in turbomachinery blade aerodynamic design optimization.

The commercial software AutoBlade is used to parametrically describe the blade geometry in this study case. The blade shape is defined by polynomial interpolation of section parameters in the radial direction from several blade sections. Camber line of blade profiles, pressure and suction sides were considered as Bezier curves.

5. Optimization

The optimization procedure was carried out using software IOSO NM for each considered section. The design target is to increase the blade isentropic efficiency while applying constraints to the mass flow rate G and pressure ratio π.

Inlet β_1 and outlet β_2 metal angles of profile in each cut were varied in range ±10% (Fig. 10).

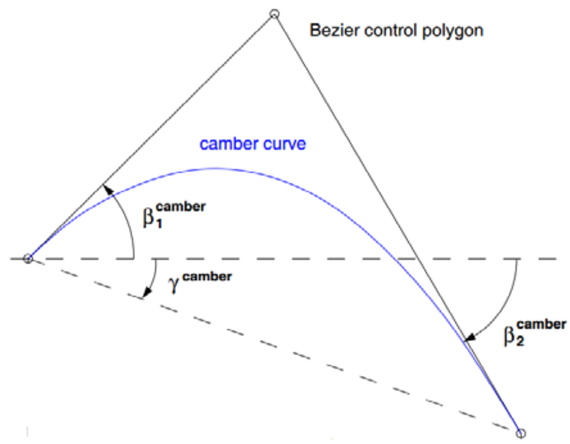


Fig. 10. Camber line and angles of profile.

6. Results

For each section were performed about 100 calculations. After that the point with maximal efficiency was chosen. Results for sections (h=25% and h=75%) are in Tables 1-2.

Table 1. Parameters for section with h=75%

	initial	optimized
π	1.6242	1.6229
η, %	89.89	90.07
$G*10^4$, kg/s	94.65	94.69
β_1	-15.846	-14.507
β_2	13.074	12.161

It's very important to analyze how geometry changed during a process of optimiza-

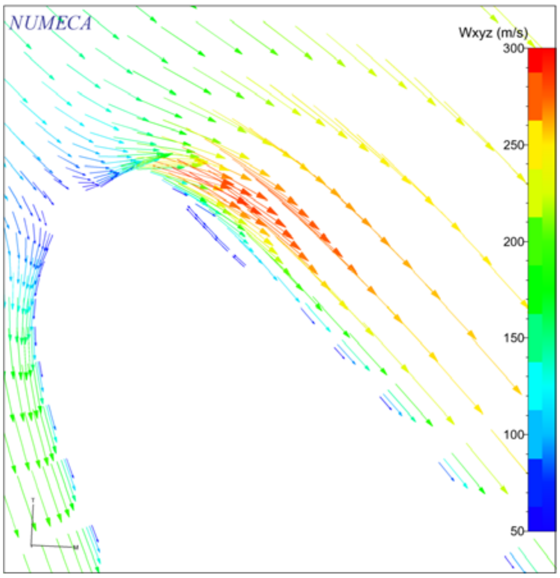


Fig. 11. Vectors of relative velocity on leading edge of initial geometry for the section with *h*=75%.

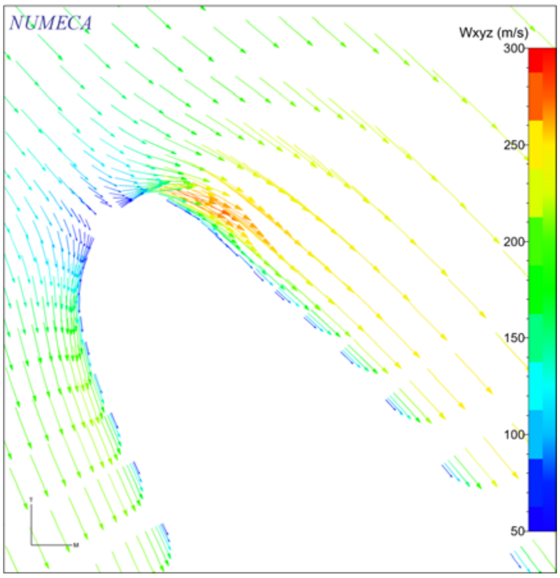


Fig. 12. Vectors of relative velocity on leading edge of optimized geometry for the section with *h*=75%.

tion impacts on downstream flow. That's why 2 stages were considered for evaluation of efficiency and pressure ratio. But flow through the vane of 2nd stage in the section with *h*=75% has a big area of low Much number and separated flow on the leading edge(Fig. 14), it says about necessity of optimization geometry of blade and vane in 2nd stage. And it also influences on evaluation of performance characteristics, that's why the difference between initial efficiency and optimized is small. It managed to remove the vortex on the leading edge of blade in the section with *h*=75% (Fig. 11 and 12). And also maximal Relative Mach Number on suction side of optimized blade is lower than on initial one (Fig. 13 and 14).

Table 2. Parameters for section with *h*=25%

	initial	optimized
π	1.4649	1.4653
η, %	82.96	82.98
$G*10^4$, kg/s	78.85	78.89
β_1	-21.63	-20.05
β_2	12.78	11.06

7 Concluding remarks

The present research has shown that the efficiency of a compressor depends on the performance of every row. The effect of the 1st stage aerodynamic optimization by varying metal angles β_1 and β_2 of profiles was reduced by flow separation in the 2nd stage. Thus, optimization of the 2^{nd} and the following stages is needed to improve the efficiency of the whole compressor.

8 Acknowledgements

This research was supported by the company "NUMECA RU". The author wish to thank Prof. Yu. Tsirkunov for discussions of setting up the problem and computational results.

References

[1] Dutta, A. K., Flassig, P. M., and Bestle, D.. A non-dimensional quasi-3d blade design approach with respect to aerodynamic criteria. *ASME Conference Proceedings*, 2008(43161), pp. 2445–2455, 2008

[2] Goel, S. Turbine airfoil optimization using quasi-3d analysis codes. *International Journal of Aerospace Engineering*, 2009.

[3] Ellbrant, L., Eriksson, L.-E., and Martensson, H. Design of compressor blades considering efficiency and stability using cfd based optimization. *Proceedings of ASME Turbo Expo*, Copenhagen, Denmark, Vol.8, GT2012-69272, pp. 371-382, 2012.

[4] Gezork, T., Volke. Applicability of quasi-3D blade design methods to profile shape optimization of turb-

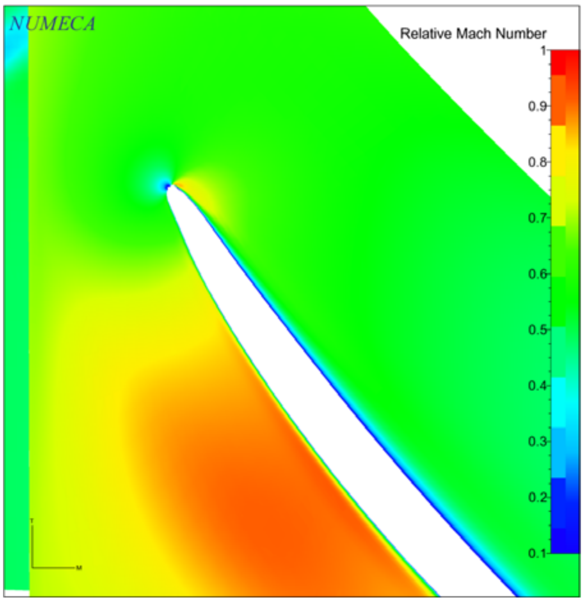


Fig. 13. Field of relative Mach number around the initial geometry blade of the 1st stage for the section with *h*=75%.

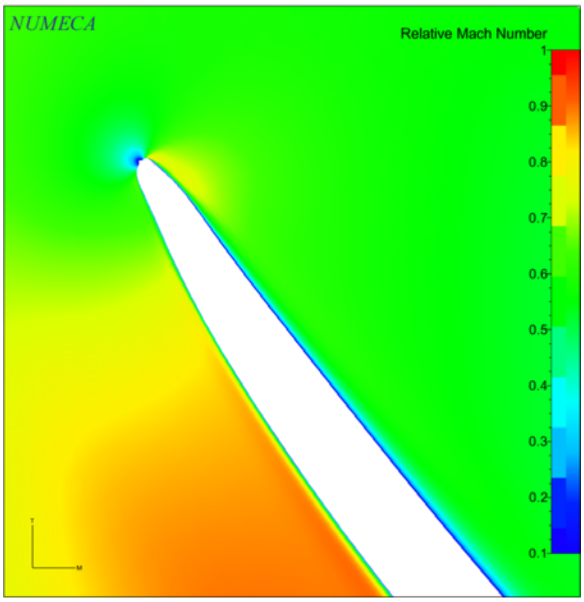


Fig. 14. Field of relative Mach number around the optimized geometry blade of the 1st stage for section with *h*=75%.

ine blade. *Proceedings of ASME Turbo Expo*, San Antonio, Texas, USA, GT2013-95467, 2013.

[5] Denton, J. D. The calculation of Three-Dimensional Viscous Flow Through Multistage Turbomachines. *ASME J. Turbomach.*, pp.18-26, 1992.

[6] Herve, S., Reutter, O., Wieler, M., Nicke., E. Numerical investigation of the origin og losses in the rotor hub region of a multistage axial compressor. *Proceedings of ASME Turbo*, Proceedings of ASME Turbo, GT2013-95013, 2013.

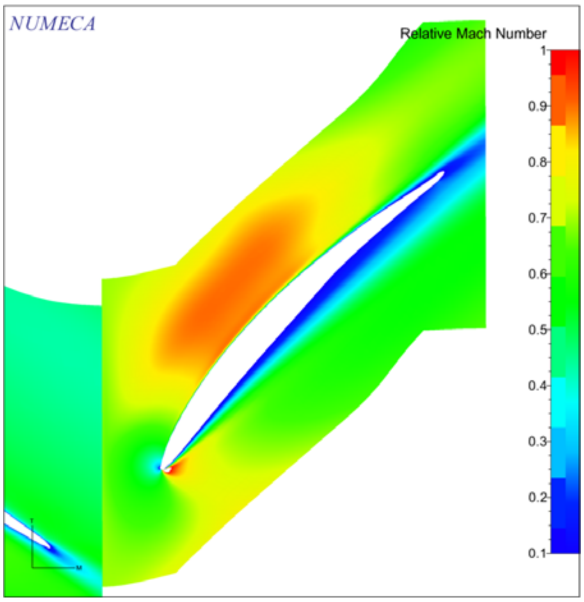


Fig. 15. Field of relative Mach number around the initial geometry vane of the 2st stage for the section with *h*=75%.

[7] Wang, D. X., He, L. Adjoint aerodynamic design optimization for blades in multistage turbomachines - part I: methodology and verification. *Journal of Turbomachinery*, Vol. 132/ 021011-1, 2010.

[8] Wang, D. X., He, L, Li., Y. S. Adjoint aerodynamic design optimization for blades in multistage turbomachines - part II: validation and application. *Journal of Turbomachinery*, Vol. 132/ 021012-1, 2010

[9] Bonaiuti, D., Zangeneh, M. On the coupling of inverse design and optimization techniques for the multiobjective, multipoint design of turbomachinery blades. *Journal of Turbomachinery*, Vol. 131/ 021014-1, 2009.

[10] Lotfi, O., Teixeira, J. A., Ivey, P. C., Kinghorn, I., and Sheard, A. G. Shape optimization of axial fan turbines using genetic algorithm and a 3D Navier – Stockes Solver. *ASME*, GT2006-90659, 2006.

[11] Yi, W., Huang, H., and Han, W. Design optimization of transonic compressor rotor using CFD and genetic Algorithm. *ASME*, GT2006-90155, 2006.

Contact Author Email Address

Mail to: olga_marakueva@mail.ru

Copyright Statement