3D INVERSE PROBLEM SOLUTION USED TO REDESIGN SIX-STAGE HIGHLY LOADED HIGH PRESSURE COMPRESSOR WITH THE VIEW OF DESIGNED PARAMETERS ACHIEVEMENT

Mileshin V. - Orekhov I. - Kozhemyako P. - Shchipin S.

Central Institute of aviation motors, Moscow, Russia, mileshin@ciam.ru

ABSTRACT
Flow structure and integral performances for a six-stage high pressure compressor (HPC) are found based on 3D viscous through-flow computation within RANS aided by the modified S.K. Godunov's implicit scheme, “3D-IMP-MULTI” software developed at CIAM.

To increase the surge margin a decision was made to modify (R1) operates on the left branch of the characteristic line in the mode with a detached shock.

To increase the surge margin a decision was made to modify R1, R3, R4 and R6 of HPC on the basis of 3D inverse problem solution by “3D-INVERSE.EXBL” software developed at CIAM. Results of HPC computations with the modified R1, R3, R4 and R6 showed an increase in the surge margin by 14.0%.

Comparison of numerical and experimental data for individual stages and for the whole modified HPC are to be presented for verification of the numerical method of HPC performance computation and redesign.

KEY WORDS
HIGH PRESSURE COMPRESSOR, 3D INVERSE PROBLEM, EXPERIMENTAL PERFORMANCES

NOMENCLATURE
ΔR_{clearance} relative tip clearance
P_{out} static pressure P/\rho^*a^*^2 in the outlet
P static pressure
a^* critical sonic speed of incoming flow
\rho^* critic density of incoming flow, \rho^*=f(M)
\rho_0 stagnation density of incoming flow
\gamma specific heat ratio
X longitudinal coordinate which is coincident with the compressor axis
LE; TE leading edge; trailing edge
Cu Curant number
Z_{bl} number of blades
G flow rate, kg/s
G_{ref} reference flow rate
G_{cor}^* corrected flow rate
PR, \pi^* total pressure ratio
\eta_{ad}^* adiabatic efficiency

\rho^* = \rho_0 \left( \frac{2}{\gamma + 1} \right)^{\frac{1}{\gamma-1}}
$G_{\text{des}}$, $\text{PR}_{\text{des}}$, $\text{Eff}_{\text{des}}$ parameters in design point

PS, SS profile pressure side; profile suction side

OL, SL operating line, surge line

MP mixing plane

$M$ Mach number

$\Delta SM$ surge margin $\Delta SM = \left(\frac{\pi_{\text{SL}}}{\pi_{\text{OL}}} \cdot \frac{G_{\text{OL}}}{G_{\text{SL}}} - 1\right) \cdot 100\%$

$\kappa$ turbulent kinematic energy, $(u'^2 + v'^2 + w'^2)/2$

$\omega$ specific dissipate rate

$u', v', w'$ fluctuating velocity components

$\mu$ molecular viscosity

$\mu_t$ turbulent viscosity

$Tu$ inlet free-stream turbulence level, $100 \frac{\sqrt{2/3} \kappa}{U_{\infty}}$

$U_{\infty}$ inlet velocity

ini; mod initial; modified

INTRODUCTION

Multistage axial compressor often put a strain on CFD modeling, especially when dealing with off-design operating conditions. However, the need for increasing efficiency and operating range of existing machines pushed the diffusion of CFD tools able to handle multistage computational domains within industrial R&D departments (Mansour et al. (2008); Denton, J.D., (1983); Gisbert F. and Corral R., (2016)). Multistage 3D steady-state RANS simulations started to appear and be used in the 1990’s (Adamczyk J.J., (1999)), but some decisive aspects of this kind of analysis are still open. Probably the most crucial one is the way to model the interface between stator and rotor rows, in which a circumferential averaging of some sort must be introduced to remove the effects of unsteadiness that are inherent in turbomachinery applications.

The averaging process is performed at the interface between adjacent rows in relative motion, called “mixing plane”. The example of multi-row steady-state simulation with the use of mixing planes was published by Cozzi L., (2017).

Steady-state multistage analysis is nowadays a standard for industrial design and design-validation purposes, but understanding the limitations of this kind of modeling is important for contemporary designers as pointed out by Denton, J.D., (2010). Some experiences in accounting for unsteady effects in turbomachinery can be found in Rubechini et al. (2015) and Holmes et al. (2011). As the computational cost of unsteady simulations on high stage count turbomachines is not yet suitable for industrial needs, assessing the level of accuracy of steady-state approximation still has a great interest.

Simulations of an 8-stage shrouded, high speed axial compressor using steady state RANS are presented by Wang et al. (2017). The objective of the paper is to examine the effect of turbulence models and end wall features on the prediction of the compressor performance. The paper also shows that an accurate representation of the end wall geometry and an effective turbulence model, together with a good quality and sufficiently refined grid result in credible predictions of the performance with steady state mixing planes both at design speed and at part speed. Shroud cavities are found to be essential to capture the flow features towards the hub at both design and part speeds. It has also been shown that the capability of existing meshing techniques and turbulence models is such that empirical models can be confidently replaced by CFD calculations for these components. The direct effect of the shroud cavities at the design speed is confined close to the hub but this effect is more pronounced at part speed. The simulations without the shroud cavities at part speed under-predict the flow separation at S1 hub and this leads to a failure in predicting the hub corner
stall at R2. Because of this, the shape of the radial profiles of total pressure at S2 leading edge is inaccurately predicted. 

Presented by Dent et al. (2017) is a comparison between steady and unsteady CFD results by performing a CFD investigation on a 3-stage industrial transonic compressor (Li et al. (1999)). The overall efficiency is broken down into individual blade row efficiencies so that it can be determined where the efficiency predictions for steady and unsteady simulations differ. Results are presented from isolated simulations of the front 1.5 stages of the compressor, with an analysis of each blade row individually for the design speed. Then results for off-design speeds are presented to show how the difference between steady and unsteady simulations changes as the speed varies. Finally, results are presented from multistage simulations to assess the impact of the downstream blade rows on the front stage efficiency predictions.

The idea of this work stems from the analysis of the experimental data accumulated in tests of six-stage HPC. The HPC tests at $\Delta R_{\text{clearance}}=100\%$ (the nominal tip clearance) revealed a contact between R1 blade and the casing. It is worth noting that the design value of the tip clearance is approximately $\Delta R_{\text{clearance}}=35\%$ of nominal tip clearance. HPC for initial tests was assembled with an increased tip clearance up to $\Delta R_{\text{clearance}}=100\%$ to prevent possible contacts between rotating blades and the casing, nevertheless they were found by tests. An obvious decision was taken to avoid these contacts - an increase of the R1 tip clearance by 50%, i.e. $\Delta R_{\text{clearance}}=150\%$. But again, contacts between R1 blades and the casing as well as a sudden increase in pressure pulsations at the blade tips and an increase in R1 blade vibration amplitude resulted in blade damage were registered in tests with the increased tip clearance by 50% (relatively to the nominal value).

To analyze physical reasons of this phenomena causing failure of several HPC R1 blades, a HPC mathematical model was developed on the basis of 3D viscous through-flow computation. Based on this mathematical model within RANS aided by S.K. Godunov's modified implicit scheme, flow structure and integral performances of the six-stage HPC were found at four tip clearance values. As a result of these calculations (Mileshin et al. (2007)) it was found that the tip clearance has a strong effect not only on HPC surge margin but also flow structure in HPC. First of all, this effect becomes apparent in supersonic and transonic HPC rotors.

Mileshin et al. (2007) discovered that HPC R1 at $75\%\leq \Delta R_{\text{clearance}} \leq 100\%$ operates in modes with detached shock, and a decrease in the tip clearance down to $35\%\leq \Delta R_{\text{clearance}} \leq 50\%$ causes a shift of the detached shock inside the blade passage towards the trailing edge. Existence of detached shock at the HPC inlet leads to HPC stall (decrease in surge margin) and consequent contacts between blades and the casing as the casing over R1 has decreasing tip diameter. Subsequent tests of the six-stage HPC (Fig. 1) with $\Delta R_{\text{clearance}}=50\%$ of the nominal value gave the evidence that there are no contacts between R1 and the casing in operating modes near to the HPC operating line.

![Figure 1: HPC under study](image-url)
To increase $\Delta SM$ surge margin at $\Delta R_{\text{clearance}}=50\%$, a decision was made to modify R1 on the basis of 3D inverse problem solution for Navier-Stokes equations. For this purpose 3D-\textsc{inverse}.\textsc{exbl} software developed at CIAM was used (Mileshin et al. (2004); Mileshin, Fateev et al. (2007)). Examples of the specified algorithm application for 3D inverse problem solution in an effort to modify fan blade rows are presented in (Mileshin et al. (2007)). With the use of 3D-\textsc{inverse}.\textsc{exbl} software HPC R1 was redesigned for the purpose of shifting the detached shock inside R1 blade passage. Flow in R1 near the stall line at $\Delta R_{\text{clearance}}=100\%$ was taken as the initial flow field. Results of HPC computations with the modified R1 showed an increase in the surge margin by $\Delta SM \geq 9\%$ as compared with the initial version at $\Delta R_{\text{clearance}}=50\%$.

Several HPC blade rows (R2, R3, R4 and R6) were redesigned for an additional increase of the surge margin up to $\Delta SM \geq 24\%$ at $n = 100\%$ rpm on the basis of 3D inverse problem.

1. **3D viscous flow computation in the context Navier-Stokes equations in blade rows of axial compressors**

A marked success was achieved recently in the development of numerical methods of 3D viscous compressible flow computations and, first of all, owing to application of implicit differential schemes reflecting physical specific features of flow and allowing for a large integration step with respect to time that considerably cuts down computation hours. In this work 3D viscous flow computation was based on "3D-\textsc{imp}-\textsc{multi}" software package developed by CIAM's compressor department. This software package uses the solution technique on the basis of a modified S.K. Godunov's scheme (Godunov et al. (1979)) having 3$^{rd}$ order of approximation with respect to 3D variables, being conservative and monotonous and enabling through-flow computations of subsonic and supersonic flows.

The 3D viscous flow computation is based on the solution of non-stationary Navier-Stokes complete equations with use of a time-marching technique. These equations are written in a dimensionless form, their type is well-known and presented, for example, by Mileshin et al. (2004). An implicit factorized operator is used for solution of Navier-Stokes equations. For simultaneous linear equations this technique makes possible to use any large step with respect to time, i.e. formally the deferential scheme is unconditionally stable. In practice, because of nonlinearity of the initial equations and due to approximation errors this technique is conditionally stable with Curant’s number within $10\cdot10^3$. The computation starts with small Curant’s numbers - $Cu \sim 5\cdot10$ with a subsequent increase in the process of solution convergence.

A modified Godunov’s scheme (Godunov et al. (1979)) with 3$^{rd}$ order of approximation with respect to 3D variables is used for calculations of fluxes through cell planes therewith the application of TVD limiting factor (Anderson et al. (1985); Brailko et al. (2010); Mileshin et al. (2010)).

The "3D-\textsc{imp}-\textsc{multi}" software package allows flow calculations not only for the individual blade row but for a stage consisting of non-rotating and rotating rows as well as for a multistage compressor. For this purpose flow along a surface which is perpendicular to the meridian plane and located in the middle between the rotor and the stator is averaged in the circumferential direction in such a way that mass flow, 3 components of impulse and energy are kept constant for any streamline. In this case there is an increase in entropy that simulates losses caused by flow non-uniformity rectification at the rotor outlet due to mixing. This type of boundary conditions is known as «mixing plane». It is used in this software package as a non-reflecting version. The computation starts with flow field hydraulic approximation and then proceeds with iterations with respect to time coordinate until the point when the initial error decreases by a factor of $10^2 \div 10^3$. Resulted flow in the relative coordinate system in any blade row is steady because unsteady interaction between the rotor and the stator is excluded due to flow averaged in the transition area from the rotor to the stator. Nevertheless, the combined computation of two rows takes into account their interaction and gives correct averaged velocity profile across the duct at the stator inlet.
The "3D-IMP-MULTI" software enables 3D viscous flow computations with account of tip clearances as well as radial clearances at the stator hub in case of cantilever-type mounting of stator vanes. The two-parameter differential model of turbulence $k-\omega$ (Wilcox D.C., 1986)) was used in the computations.

2. BOUNDARY CONDITIONS
Flow angles in the circumferential direction and in the meridian plane in the direction from the hub to the tip as well as total pressure and temperature distributions along the radius are specified at the inlet; static pressure at the hub is specified at the outlet; and the static pressure distribution from the hub to the tip is derived from the radial balance equation. A boundary condition for a wall is sticking condition and for a periodic boundary - periodicity condition. Non-reflecting boundary conditions can be used at the inlet and the outlet in case of troubles with problem convergence and implemented by specifying corresponding Riemann’s invariants. To calculate kinetic energy of pulsations, $\kappa$, and specific dissipation rate, $\omega$, the turbulent-to-laminar viscosity ratio, $\mu_t/\mu=50$, and the intensity of turbulent pulsations at the inlet, $Tu=5\%$, are specified at the computational domain inlet.

3. RESULTS OF 3D VISCOUS FLOW COMPUTATIONS FOR THE INITIAL VERSION OF SIX-STAGE HPC
Computations were based on the upgraded “3D-IMP-MULTI” code. The computation goal is assessment of the whole HPC and its separate rows local and integral performances for the rotational speed 100% ($n_{cor}=100\%$, $u_{1R}=370$ m/s) and for partial speeds $n_{cor}=25\%÷100\%$.

Computations of viscous flow in the compressor (13 blade rows including IGV) were made for the corrected parameters at the inlet ($T^*=288^°K$, $P^*=101325$ Pa).

In the computations it was taken into consideration that nominal (100 %) clearances at rotor tip and stator hub and stator tip are specified according to the values shown in the table below:

<table>
<thead>
<tr>
<th>Rotors cold clearances</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
<th>R4</th>
<th>R5</th>
<th>R6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z bl</td>
<td>40</td>
<td>52</td>
<td>64</td>
<td>72</td>
<td>78</td>
<td>82</td>
</tr>
<tr>
<td>Tip Clearance mm</td>
<td>1.0</td>
<td>0.95</td>
<td>0.95</td>
<td>0.92</td>
<td>1.04</td>
<td>1.05</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Stators cold clearances</th>
<th>IGV</th>
<th>S1</th>
<th>S2</th>
<th>S3</th>
<th>S4</th>
<th>S5</th>
<th>S6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z bl</td>
<td>46</td>
<td>66</td>
<td>82</td>
<td>100</td>
<td>106</td>
<td>116</td>
<td>82</td>
</tr>
<tr>
<td>Hub Clearance mm</td>
<td>0.8</td>
<td>0.75</td>
<td>1.22</td>
<td>1.20</td>
<td>1.00</td>
<td>1.00</td>
<td>-</td>
</tr>
<tr>
<td>Tip Clearance mm</td>
<td>1.05</td>
<td>1.10</td>
<td>1.07</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

All clearances are shown in the “cold” state. IGV, S1 and S2 guide vanes are tip-shrouded and variable and, therefore, they have tip clearances both at the hub and at the tip. S3, S4 and S5 guide vanes are unshrouded with tip clearances at the hub. The S6 outlet guide vanes are tip-shrouded and non-variable.

In compliance with the compressor design in the computations it was assumed that compressor hub rotation was from the rotor leading to the trailing edges for all stages, excluding rows with stators with two supports.

The 3D viscous flow computation was made with the help of a computational grid composed of 3594240 cells. The tip clearance occupies 8 cells. The computational grid was drawn with highlighting the leading and trailing edges of blade rows that was achieved by concentration of the grid nodes near the edges.

In the center of the cell closest to the wall, $y+\approx 10–15$, and to determine the coefficient of friction on the wall, the wall functions (Menter F.R., 2004) were used. Taking into account the wall functions the grid used in the calculations is minimally acceptable. Double the mesh size in each
direction, i.e. a total of 8 times, leads to an increase in adiabatic efficiency of 1.0 ± 1.2%. However the flow rate and the degree of increase in the total pressure change slightly, the flow pattern remains the same.

At the operating point of the characteristic line for the initial version of HPC at 100 % speed the following parameters (see Fig. 20) are found: adiabatic efficiency - $\eta_{\text{ad}}^*/\eta_{\text{des}}^*$=0.94 at $\pi^*/\pi_{\text{des}}^*$=0.87 total pressure ratio, i.e. the HPC under study demonstrates low parameters as to pressure ratio and efficiency. The compressor surge margin is equal only to $\Delta SM=1.716\%$ that is not high, $(G_{\text{des}}-G_{\text{INI}})/G_{\text{des}}=0.1-1.100\%\approx 10\%$. The operating point is the cross-point of $\pi^*=\pi^*(G_{\text{cor}})$ characteristic line and the operating line.

4. AERODYNAMICS OF THE INITIAL COMPRESSOR VERSION AT 100 % SPEED AND NOMINAL TIP CLEARANCE

The detailed study of compressor aerodynamics at 100% r.p.m. was performed at the operating point. Aerodynamics of the HPC initial version is presented by Mach number contours (see Fig. 2a, Fig. 3a) at the operating point at 100% rpm and tip clearance $\Delta R_{\text{clearance}}=50\%$. $\Delta R_{\text{clearance}}(50\%)=0.5 \cdot \Delta R_{\text{clearance}}(100\%)$, that is approximately equal to tip clearances in the “hot” state in the design condition.

Mach number distributions on the rotor suction side and stator pressure side as well as Mach number distributions on rotor pressure side and stator suction side are shown in Fig. 2 for the meridian plane. Mach number distributions in Fig. 2a show that there is noticeable flow acceleration on R1 suction side with subsequent deceleration in the detached shock. The supersonic area in R1 tip section is terminated by a shock with pre-shock M=1.2. An improvement of R1 aerodynamics, namely, R1 redesigning aiming at shifting the detached shock inside the blade passage is the main source for an improvement of compressor performances.

As evident from Fig. 2a and Fig. 3a, the operating point for this HPC version is on the left branch of HPC characteristic line. In this area of characteristics there is a risk of unstable HPC operation that can result in strong vibrations of the detached shock and stall of HPC. The flow in the vicinity of leading edges of R1, R2, R3 and R4, presented in Fig. 2a, has increased incidence angles (concentration of vertical contours).

R2 is transonic, not choked at the operating point and provides high efficiency - $\eta_{\text{ad}}^*$=0.88.
Efficiency of R3 is high enough because of low losses in Stator3. Efficiency of last stages at the operating point of the characteristic line smoothly decreases from the inlet to the outlet and efficiency of the 6-th stage is equal to $\eta_{ad}^*=0.8$. Mainly, this is caused by a decrease in efficiency of rotors because of strengthening the adverse effect of the tip clearance in last stages. Losses in guide vanes are kept constant and even slightly decrease in last stages.

Initial compressor aerodynamics in the design point at 100% r.p.m. is presented by Mach number contours (see Fig. 3 a).

Fig. 3a shows Mach number distributions in the tip section for the first 3 stages of the six-stage HPC. Mach number distributions show the detached shock in R1. The supersonic area in R1 tip section is terminated by shock with pre-shock $M=1.2$.

![Mach number distributions](image)

**Figure 3:** Mach number distributions on rotational surface near blade tip. Initial (a) and modified (b) compressor. Stages 1–3. Step 0.1. Near operating line.

5. **HPC R1, R2, R3, R4 MODIFICATION BASED ON THE SOLUTION OF 3D INVERSE PROBLEM AIMING AT AN INCREASE IN HPC SURGE MARGINS**

This work highlights an attempt to increase surge margins of the six-stage HPC without application of any casing treatment. For an increase in the surge margins we made a decision to redesign R1 on the basis of the 3D inverse problem solution for Navier-Stokes equations with the purpose of shifting the detached shock inside the blade passage.

3D inverse problem is solved by 3D- INVERSE.EXBL code developed by CIAM (Mileshin, Fateev et al. (2007); Mileshin et al. (2004)). In this software package the inverse problem is based on specification of the preferable static pressure distribution on the blade surfaces, preset blade thickness and differences of static pressure on the blade suction and pressure sides (so-called “loading”) in corresponding blade points. Input and output data of gasdynamic parameters are received from the solution of the direct problem of flow in a multistage compressor and are kept constant in the process of the inverse problem solution CIAM (Mileshin et al. (2012)).

The inverse problem solution is found with the help of a mobile computational grid (with account of changes in flow, impulse and energy caused by movement of the cell sides). So each step with respect to time starts with definition of changes in blade surfaces with subsequent drawing a new computational grid. Analysis of the initial HPC R1 version at the point of the operating line at $n = 100\%$ r.p.m. shows that max aerodynamic loading is located near to the R1 leading edge along the whole blade height $0 \leq h \leq 100\%$ as shown in Fig. 4, 6. The decisive factors for this loading distribution are the increased flow incidence angles near to the leading edge and R1 operation on the left branch of HPC characteristic line in modes with the detached shock (Fig. 2 a, 3 a).

Based on 3D inverse problem solution, R1 was redesigned to avoid high incidence angles at the R1 leading edge and shift the detached shock inside the blade passage. For this purpose static pressure distributions on the blade suction side in all R1 blade sections from the hub to the tip were modified. Figures 4-7 show an example of this modification for two R1 blade sections - 4% and 83% of the blade height. Only static pressure distribution along the suction side is modified in Fig. 4,6,8,10,12,14,16,18 (red color). As to static pressure distribution along the pressure side, it remains unmodified and corresponds to the initial pressure distribution (blue). As evident from these figures,
max pressure near to the blade leading edge for the modified static pressure distribution is cut off. Then, the line of the modified static pressure distribution is drawn in such a way that max loading enters into the blade passage but total loading in each blade section is unchanged to avoid HPC stages mismatching.

Figure 4: Aerodynamic loading of R1 blade. Section 4% of blade height

Figure 5: Mach number distributions for blade-to-blade channel of modified R1. Section 4%

Thereafter, the static pressure difference on blade suction and pressure sides is calculated for each cell of the finite-difference grid at X=const that is kept constant during the solution of 3D inverse problem and is used in this problem as a boundary condition on the blade surface. Some results of this problem solution are shown in Fig. 5,7 presenting profiles of initial and modified blades for two sections along the height \( h = 4\% \) and 83%.

It is worth noting that 3D flow field found by solution of the direct problem is taken as an initial flow field for the inverse problem solution and loading distribution along R1 blade height was corresponding to the operating point of the characteristic line at \( \Delta R_{\text{clearance}} = 50\% \).

\( \Delta R_{\text{clearance}}(50\%) = 0.5 \Delta R_{\text{clearance}}(100\%) \), that is approximately equal to tip clearances in the “hot” state in the design condition.

An example of the inverse problem solution for modification of R2, R3 and R4 by 3D-INVERSE.EXBL software is shown in Fig. 5,7,9,11,13,15,17,19. Analysis of the initial HPC version R2, R3 and R4 at the point of the operating line at \( n = 100\% \) r.p.m. shows that max
Aerodynamic loading is located near to the R2, R3 and R4 leading edge along the whole blade height as shown in Fig. 8,10,12,14,16 and 18. The decisive factors for this loading distribution are the increased flow incidence angles near to the leading edge and R2, R3 and R4. On the base of the inverse problem solution by 3D-INVERSE.EXBL R2, R3 and R4 were modified in order to eliminate the increased flow incidence angles near to the leading edge and R2, R3 and R4. Figures 4,6,8,10,12,14,16 and 18 present static pressure distributions along the profile suction and pressure side of initial and modified blades. Figures 5,7,9,11,13,15,17 and 19 present comparison of initial and modified blades profiles for R1, R2, R3 and R4. Also Mach number distributions for modified blade-to-blade channel are showed in Fig. 5,7,9,11,13,15,17.
Figure 12: Aerodynamic loading of R3 blade. Section 16% of blade height

Figure 13: Mach number distributions for blade-to-blade channel of modified R3. Section 16% of blade height

Figure 14: Aerodynamic loading of R3 blade. Section 83% of blade height

Figure 15: Mach number distributions for blade-to-blade channel of modified R3. Section 83% of blade height

Figure 16: Aerodynamic loading of R4 blade. Section 4% of blade height

Figure 17: Mach number distributions for blade-to-blade channel of modified R4. Section 2% of blade height
Based on the 3D inverse problem solution, the modified R1, R2, R3 and R4 blades were mounted in the six-stage HPC. HPC characteristic line at $\Delta R_{\text{clearance}}=50\%$ was computed by «3D-IMP-MULTI» software as shown in Fig. 20. From the comparison of the received characteristics in Fig. 20 it is evident that HPC with the modified R1, R2, R3 and R4 has a long vertical section because the terminating shock at the operating point of the characteristic line is very close to the blade trailing edge. This is clear from Fig. 2b and 3b showing streamlining in tip sections of first 3 HPC stages including IGV with terminating shock inside R1 blade-to-blade channel (without detached shock at the HPC inlet). There is an increase in HPC surge margins up to $\Delta SM=18.6\%$ determined by stall phenomena in R4 and R6. Modification of these rotors as well as R3 based on 3D inverse problem solution allows an increase in the surge margin up to $\Delta SM\geq24\%$. Modification of R2, R3 and R4 blades based on the 3D inverse problem solution resulted in increased surge margin up to $\Delta SM\geq24\%$ and adiabatic efficiency $\eta^*_{\text{ad}}\geq85\%$.

6. COMPARISON OF CALCULATED AND EXPERIMENTAL DATA OF THE REDESIGNED HPC

Fig. 20 shows calculated and experimental characteristics for HPC within $\bar{n}=(25\div105)\%$ corrected rotational speeds. The differences between calculated and experimental values are the following: for max. airflow - $\Delta G\sim1\%$, for total pressure ratio on the surge line - $\Delta \pi^*\sim(2\div3)\%$. The difference in max. adiabatic efficiency is $\Delta \eta^*_{\text{ad}}\sim(1\div1.5)\%$ within $n=(80\div105)\%$, $\Delta \eta^*_{\text{ad}}\sim(1.5\div3.0)\%$ within $n=(55\div75)\%$, and $\Delta \eta^*_{\text{ad}}\sim(3.0\div4.5)\%$ within $n=(25\div45)\%$.

Total pressure and temperature pickups in tests are installed at the leading edge of the guide vanes (S1–S6), so it is possible to compare the calculated and experimental characteristics of separate groups, each consisting of two rows, namely, (IGV+R1), (S1+R2), (S2+R3), (S3+R4), (S4+R5) and (S5+R6). The results of this comparison for total pressure ratio as a function of airflow at the compressor inlet are shown in Fig. 21-24 for $\bar{n}=(65\%, 75\%)$ and in Fig.4 and Fig. 5 for $\bar{n}=(90\%, 100\%)$, respectively. It is clear from these Figures that calculated and experimental characteristics of the first and second groups - (IGV+R1) and (S1+R2) - are in good agreement within the total range of rotational speeds. The calculated and experimental characteristics for the third group of rows - (S2+R3) differ at rotational speeds lower than 80%. The calculated and experimental characteristics of the last (sixth) group of rows - (S5+R6) differ at rotational speeds lower than 90%. The best agreement within the total range of rotational speeds is demonstrated by the group of rows in the middle of the compressor, namely (S3+R4) and (S4+R5).
Figure 20: Total calculated (light color) and experimental (solid color) characteristics for modified HPC within the range of rotational speeds from 25% to 105%.

INI – initial version of HPC. $\bar{P}_R = \frac{P_R}{P_{R_{des}}}$, $\bar{E}_{ff} = \frac{E_{ff}}{E_{ff_{des}}}$, $\bar{G} = \frac{G}{G_{des}}$

The low value of calculated adiabatic efficiency, $\eta^*_{ad}$, as compared with the experimental value at low rotational speeds is caused, most likely, by higher calculated total pressure losses in the group of rows (IGV+R1), and, as a consequence, in the group of rows (S1+R2). The cause of these losses is the off-design operation mode of the first rotor with high incidence angles and high turning angle of IGV at low speeds. An increase in the incidence angle at the R1 leading edge leads to a powerful flow separation with high losses. Due to these losses, air in the groups of rows (IGV+R1) and (S1+R2) is under-compressed. In this case, the R3 rotor operation is with increased axial
component of the velocity at the inlet and low incidence angles; the group of rows (S2+R3) has a lower calculated pressure ratio as compared with the experimental value. However, this situation is partially eliminated at the R3 rotor outlet and the groups of rows (S3+R4) and (S4+R5) run in conditions similar to the experimental (see Fig. 21-24). The last group (S5+R6) runs with an increased load to compensate for total pressure losses in the first three groups of rows (Fig. 21-24). Despite this difference in operation of different groups of rows at low rotational speeds, total compressor characteristics, both calculated and experimental, in the G–π* coordinates are in good agreement for airflow, total pressure ratio, and a surge line position (Fig. 20). In the G–η* ad coordinates, the calculated value of η* ad is in satisfactory agreement with the experimental value at rotational speeds higher than 75%, and it is lower than the experimental value at rotational speeds lower than 75% for the reasons described above.

Figure 21: Characteristics for groups of rows at n=65% - calculations versus tests

Figure 22: Characteristics for groups of rows at n=75% - calculations versus tests

Figure 23: Characteristics for groups of rows at n=90% - calculations versus tests
CONCLUSIONS

1. Based on 3D viscous through-flow computation in the context of Navier-Stokes equations, it was found out that in the design rotational speed $n_{\text{corr}} = 100\%$ ($u_R = 370$ m/sec) and in the design point the initial HPC operates in the modes with the detached shock caused decrease of its parameters: pressure ratio $\Delta P \approx 13.5\%$, corrected mass flow $\Delta \dot{m} \approx 10\%$, efficiency $\Delta \eta \approx 6\%$, and $\Delta \Sigma M \approx 22\%$.

2. Based on 3D inverse problem solution, HPC R1 was redesigned in order to shift the detached shock inside R1 blade passage. HPC with the modified R1 demonstrated a noticeable increase in the surge margin up to $\Delta \Sigma M \geq 18\%$.

3. Subsequent modification of R2, R3 and R4 on the basis of 3D inverse problem solution made possible an increase in the surge margin up to $\Delta \Sigma M \geq 24\%$ and provided other required design parameters of HPC.

4. Comparison of calculated and experimental data for HPC within $n=(25\div105)\%$ corrected rotational speeds showed differences: for max. airflow $\Delta \dot{m} \approx 1\%$, for total pressure ratio on the surge line $\Delta P \approx (2\div3)\%$. The difference in max. adiabatic efficiency is $\Delta \eta_{\text{ad}} \approx (1\div1.5)\%$ within $n=(80\div105)\%$, $\Delta \eta_{\text{ad}} \approx (1.5\div3.0)\%$ within $n=(55\div75)\%$.

5. Comparison of calculated and experimental characteristics of separate groups, each consisting of two rows, (IGV+R1), (S1+R2), (S2+R3), (S3+R4), (S4+R5) and (S5+R6) for total pressure ratio as a function of airflow at the compressor inlet and adiabatic efficiency showed that calculated and experimental characteristics of the first and second groups - (IGV+R1) and (S1+R2) - are in good agreement within the total range of rotational speeds. The calculated and experimental characteristics for the third group of rows - (S2+R3) differ at rotational speeds lower than 80%. The calculated and experimental characteristics of the last (sixth) group of rows - (S5+R6) differ at rotational speeds lower than 90%. The best agreement within the total range of rotational speeds is demonstrated by the group of rows in the middle of the compressor, namely (S3+R4) and (S4+R5).

REFERENCES


