Aerodynamic Designing of Supersonic Centrifugal Compressor Stages

Y. Galerkin, A. Rekstin, K. Soldatova

Abstract**—**Universal modeling method well proven for industrial compressors was applied for design of the high flow rate supersonic stage. Results were checked by ANSYS CFX and NUMECA Fine Turbo calculations. The impeller appeared to be very effective at transonic flow velocities. Stator elements efficiency is acceptable at design Mach numbers too. Their loss coefficient versus inlet flow angle performances correlates well with Universal modeling prediction. The impeller demonstrates ability of satisfactory operation at design flow rate. Supersonic flow behavior in the impeller inducer at the shroud blade to blade surface Φ_{des} deserves additional study.

Keywords**—**Centrifugal compressor stage, supersonic impeller, inlet flow angle, loss coefficient, return channel, shock wave, vane diffuser.

NOMENCLATURE

D	Diameter
k	Isoentropic coefficient
$M_u = \dfrac{u_2}{\sqrt{kRT_0^*}}$	Mach number
$\overline{m}$	Mass flow rate
R	Gas constant
T	Temperature
u	Blade speed
π^*	Total pressure ratio
ρ	Gas density
$\Phi = \dfrac{\overline{m}}{0,785 D_2^2 u_2 \rho_0^*}$	Flow rate
$\eta^* = \dfrac{\ln \pi^*}{\frac{k}{k-1} \ln\left(\frac{T_2^*}{T_0^*}\right)}$	Total efficiency
ψ_T	Work coefficient

Subscripts

0, 1, 2, 3, 4, 5, 6, 0'	indexes of control sections
max	maximum

Yuri Borisovich Galerkin is with S.-Peterbugd Poltechnical Yniversity, Russian Federaton (phone: +7-921-942-73-40; fax: 8-812-552-86-43; e-mail: yuiri_galerkin@ mal.ru).
Aleksey Feliksovich Rekstin is with S.-Peterbugd Poltechnical Yniversity, Russian Federaton (phone: +7-921-923-51-49; fax: 8-812-552-86-43; e-mail: rekst2K7@ mal.ru).
Kristina Valerievna Soldatova is with S.-Peterbugd Poltechnical Yniversity, Russian Federaton (phone: +7-905-220-50-70 fax: 8-812-552-86-43; e-mail: buck02@list.ru).

I. Design Method General Features

Purely analytical approach to gas dynamic design of centrifugal compressors is impossible. The effective engineering methods are based on generalized results of physical experiments (and numerical ones partially). The Universal modeling method [1] is the set of rules for main dimensions of a flow path definition and the set of computer programs to realize it, optimize design by candidates' comparison and calculate performances. The head loss model is based on physical and math models derived from vast experiments. The validity of the method was proven by successful design of several dozen types of industrial compressors with total installed power close to 5 000 000 KWt. The Authors present below an attempt to extrapolate this method for design of a supersonic stage for GT. Specific problems of supersonic flow were taken into account.

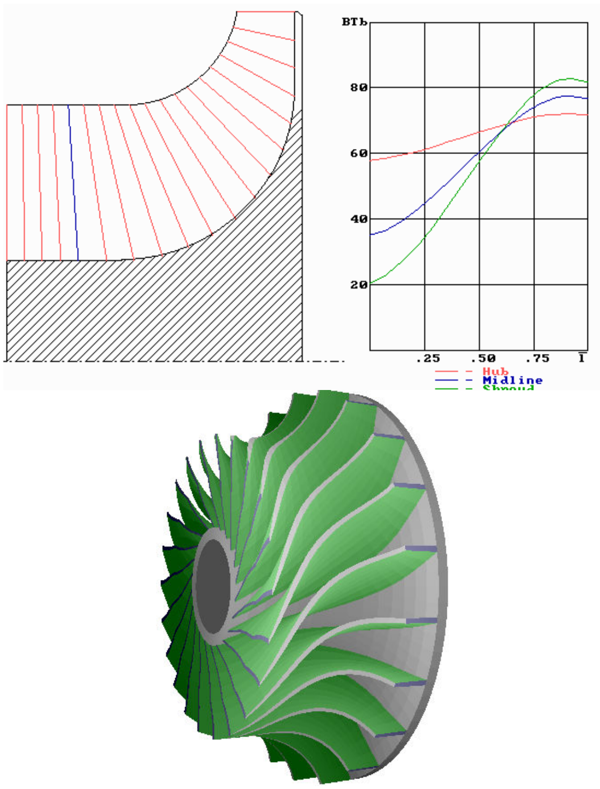


Fig. 1 The scheme of the impeller flow path in meridian plane, blade angles along three blade to blade surfaces, blade cascade view

II. Q3D and CFD Analysis of Design

Main dimensions of the impeller with flow rate Φ_{des} =0,12 and ψ_{Tdes} = 0,80 were optimized by means of 5th and 6th generations [2], [3] computer programs. The blade cascade has been designed for work at moderate conditional Mach number by calculations of a quasi three-dimensional non viscous flow. The scheme of a flow path in meridian plane, blade angles along three blade to blade surfaces and blade cascade view are shown on Fig. 1.

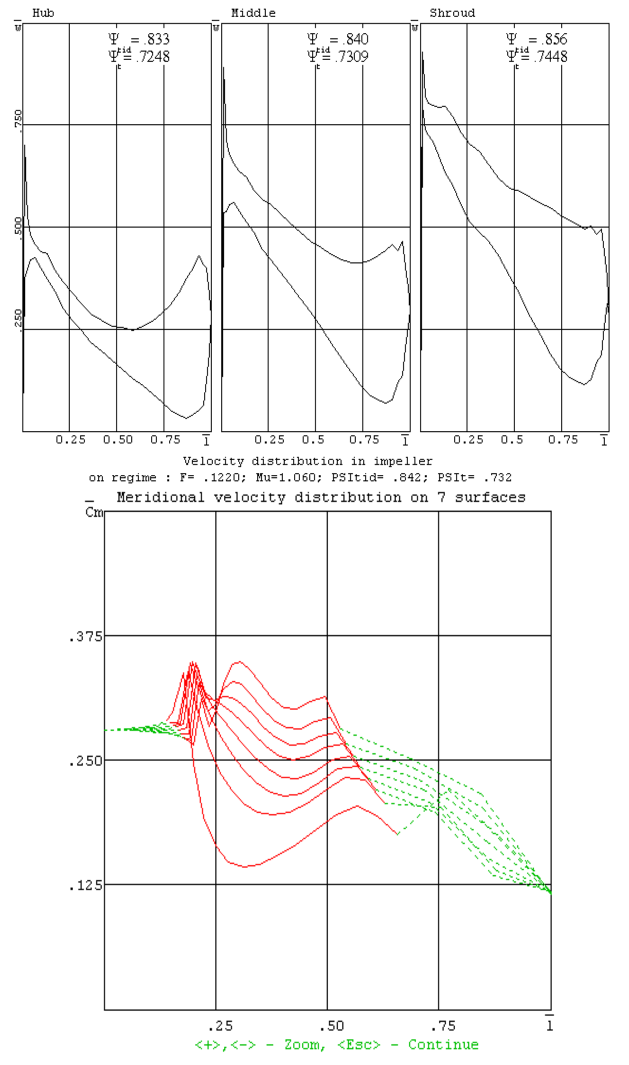


Fig. 2 Velocity diagrams on three axisymmetric flow surfaces on blade height (above) and meridian velocities on seven axisymmetric flow surfaces (below). Calculation of a non-viscous Q3D flow

The impeller design was checked by CFD calculations. The ANSYS CFX program was applied for calculations at M_u =0,5 $M_{u\,des}$.

The non viscous velocity diagrams on impeller blades along three blade to blade surfaces are shown on Fig. 2 (above). Meridian velocities along eight blade to blade surfaces are shown at Fig. 2 below. Velocity diagrams show conformity to proven design principles [1]:

- minimal velocity peak at a leading edge demonstrates closeness to non incidence inlet condition. This condition takes place at the shroud blade top blade surface where flow kinetic energy is maximal;
- blade load (velocity difference at suction and pressure sides) is minimal near leading edges, therefore the maximum local flow velocity at the suction side is close to the velocity at the cascade inlet;
- mean blade load and load distribution along blade periphery are under control. It is impossible practically to control blade load at the mean and hub surfaces though.

The pressure diagram on the shroud blade to blade surface is shown at Fig. 3.

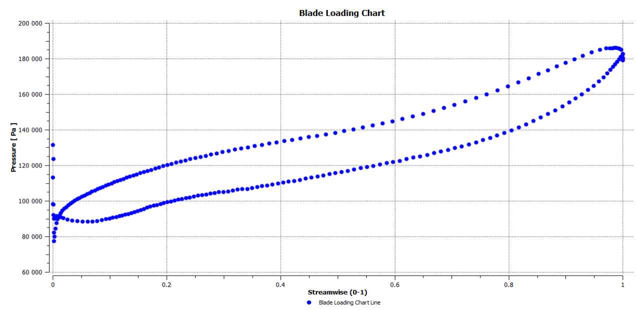


Fig. 3 The pressure diagram on the periphery blade to blade surface (ANSYS CFX)

The pressure load character shows that there is a small negative incidence angle. It is unlike the velocity diagram at Fig. 2. Boundary layers thickness blocks the inlet area. It could be the reason of non-viscid and viscid calculations mismatch. The difference is not sufficient practically though.

III. Impeller Flow Structure and Performance

Mach number fields in impeller on shroud and hub blade to blade surfaces are shown on Fig. 4.

Quick flow deceleration resembles an oblique shock at the entrance of the blade cascade at the periphery blade to blade surface.

Wake zone is present usually at the cascade exit on the suction side of blades. This zone is not visible on Fig. 4. The considerable thickening of boundary layers takes place at the end of blades' pressure side instead. There is known that activation of the turbulence pulsations normal component on pressure side takes place due to blade load. Elevated friction forces can be responsible for this low energy zones formation. High velocity jets in the vaneless diffuser correspond to low energy zones near trailing edges of the impeller blades. Viscosity leads to quick mixing of the flow in the vaneless space after the impeller.

The impeller is quite effective in accordance with calculation results. The maximum total efficiency of axial inlet + impeller + vaneless diffuser ($D_3 / D_2 = 1{,}35$) is 0,974 – no disk friction and gap impeller-body losses are taken into account. The efficiency performance is presented at Fig. 5.

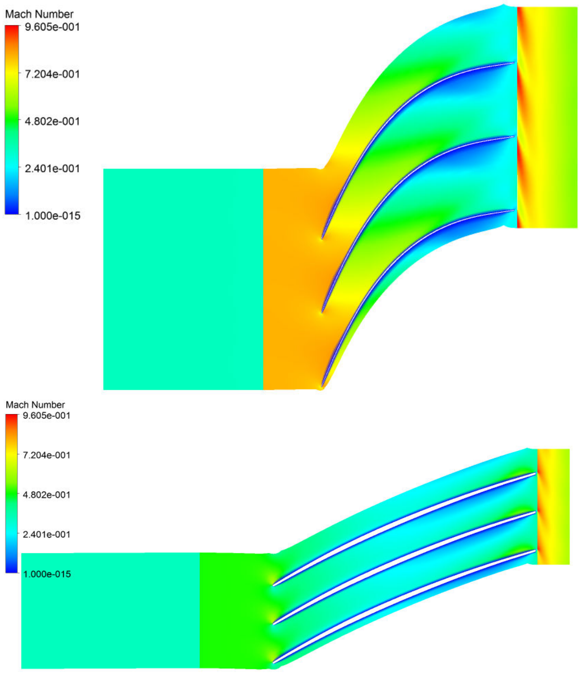


Fig. 4 Mach number field on shroud and hub blade to blade surfaces M_u =0,5 $M_{u\,des}$

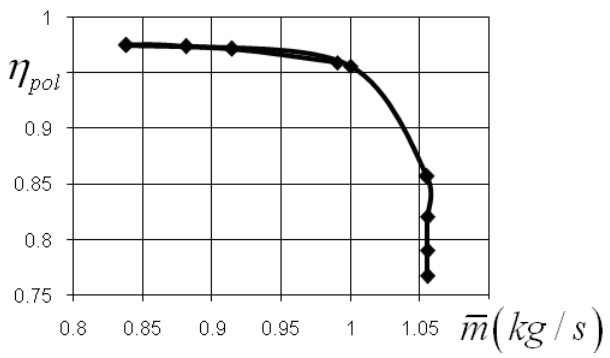


Fig. 5 Total efficiency of axial inlet + impeller + vaneless diffuser versus relative mass flow rate

For calculations at design Mach number $M_{u\,des}$ = 1,85 the program NUMECA Fine/Turbo has been used [4]. The information for flow rate $\overline{m} = 0{,}99\overline{m}_{max}$ is presented below. Flow character has undergone considerable changes in comparison with lower Mach number. Mach number fields on three blade to blade surfaces - shroud, mean, hub – are presented at Fig. 6.

The flow accelerates visibly along suction side of blades. The intensive shock occurs at the shroud blade to blade surface with subsonic flow after it. The shock is followed by flow separation that blocks the channel practically. More favorable are flow conditions at the mean surface. The flow becomes subsonic after oblique and direct shocks. The separation zone near the exit of the suction side is the usual phenomenon for this kind of impellers. Lower Math numbers at the hub surface guarantee good flow conditions.

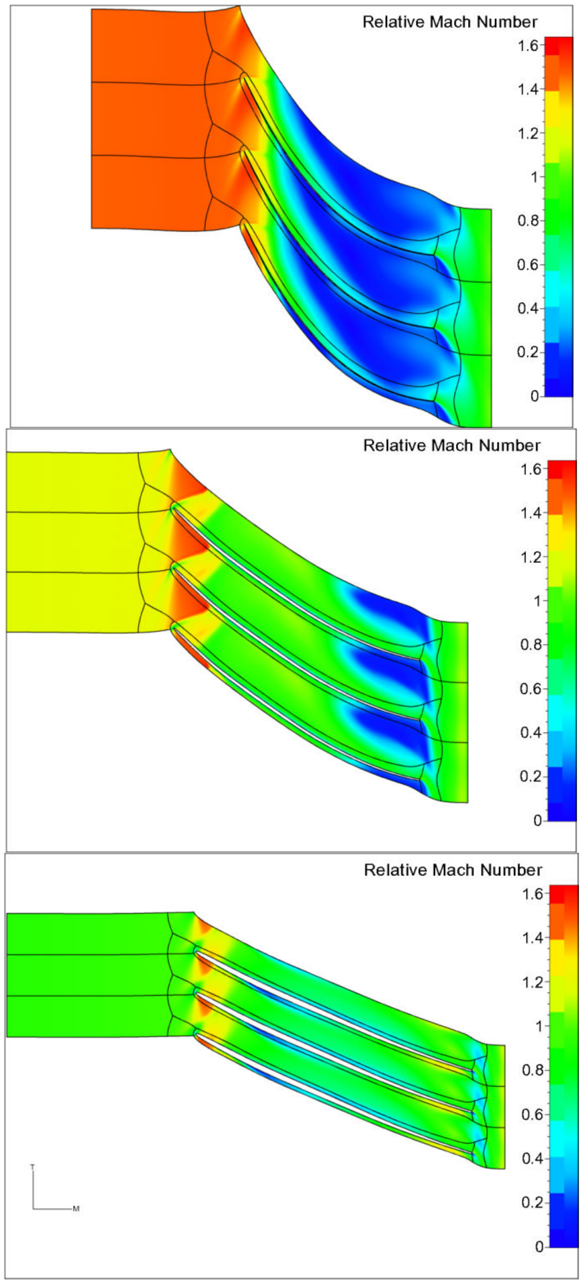


Fig. 6 Mach number fields on three blade to blade surfaces: shroud, mean, hub. $M_{u\,des}$ = 1,85

IV. Stator Part Study

The maximum total efficiency of axial inlet + impeller + vaneless diffuser ($D_3/D_2 = 1{,}35$) is 0,85 only (no disk friction and gap impeller-body losses are taken into account). It is 12% less than at $M_u = 0{,}5M_{u\,des}$. The main increase of losses takes part in the vaneless diffuser. Fig. 7 demonstrates meridian projection of flow trajectories in the impeller and diffuser.

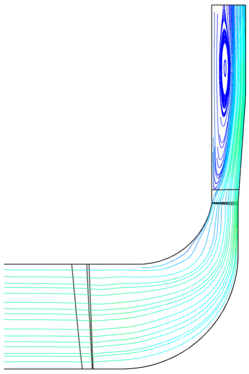


Fig. 7 Meridian projection of flow trajectories, M_u = 1,85

The meridian flow is rather uniform in the impeller. The separation zone on the outer wall of the diffuser blocks the area of the diffuser on 2/3. But the vaneless diffuser is not the element of the stage under analysis. It was applied for the impeller calculation analysis only. Actual stator elements «vaneless part + vane diffuser + return channel» have been designed under recommendations [1]. Specific conditions of a supersonic flow were taken into account. The program ANSYS CFX was applied for calculations.

On Fig. 8 flow fields on the mean blade height are shown at two inlet flow angles.

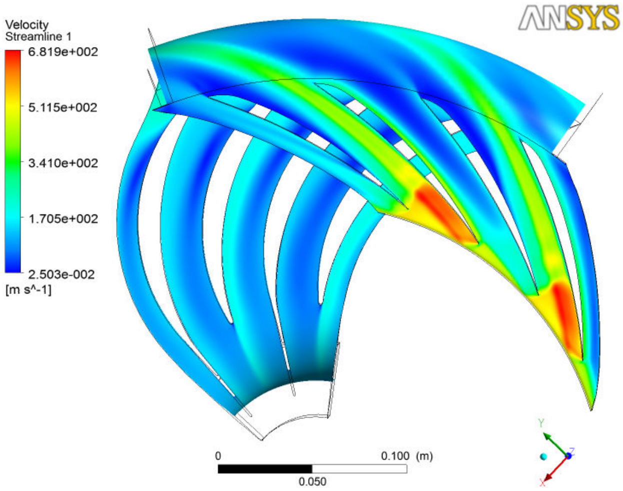
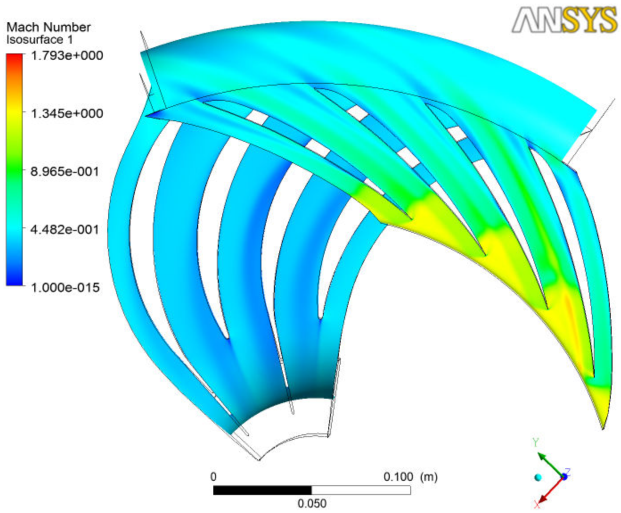


Fig. 8 Flow fields on mean blade to blade surfaces in stator elements of the stage. Inlet flow angle 22^0 - Mach numbers (left), angle 18^0 - flow velocities (right)

Despite high level of supersonic speeds on the inlet, flow character in the vane diffuser and return channel at design flow rate angle 22^0 is favorable. Low intensity shock takes place at the inlet. No separation takes place in the vane diffuser. Low energy zones are visible at suction sides of the return channel vanes.

The situation is worse at the inlet flow angle 18^0 ($\bar{m} = 0,82\bar{m}_{des}$). Positive incidence leads to flow acceleration up to 680 m/s on suction side of blades, to intensive shocks and flow separation followed with negative influence on flow conditions in the return channel. The loss coefficient of the stator part $\zeta = h_w / 0,5c_2^2$ as function of inlet flow angle is presented at Fig. 9.



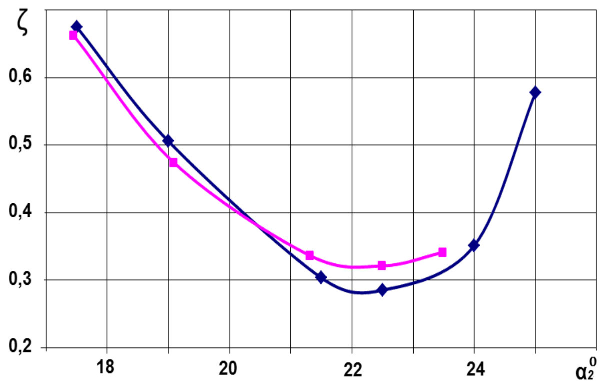


Fig. 9 Loss coefficient of the stage stator part performance

◆ – ANSYS CFX, ■ - Universal modeling, 5th generation [2], [3]

The ANSYS CFX calculation gives value ζ_{des} = 0,28 at the design regime (α_2 = 18^0) and ζ = 0,60 at the inlet flow angle 18^0 ($\bar{m} = 0,82\bar{m}_{des}$).The Universal modeling method [1], [5] computer program predicts the character of the performance $\zeta = f(\alpha_2)$ well. The value of ζ_{des} = 0,315 is 12,5% higher than calculated by ANSYS CFX.

V. Conclusion

The Authors recon that design principles and computer technology well proven for industrial compressor stages can lead to sound solutions for high flow rate supersonic stages too. Unsolved is the problem of the inducer inlet where blade load leads to flow acceleration with intensive shock and excessive losses at the shroud blade to blade surface. The problem deserves further study.

References

[1] Galerkin, Y.B., Soldatova, K.V., Drozdov, A.A. Turbo compressor efficiency application and calculation. (text) // Compressors & Pneumatics. – 2011. – No. 8. – Page 2-11. (in Russian).
[2] Galerkin Y.B., Drozdov A.A., Soldatova K.V. (SPbSTU). Improvement of method to calculate industrial centrifugal compressors performance. (text) // Compressors & Pneumatics.. – № 8. – 2013. – C. 24-32. (in Russian).
[3] Soldatova K.V. Universal modeling method application for development centrifugal compressor model stages. Conference «Compressors and their Systems». – London. – 2013. - P. 477-487. (in English).
[4] Transactions of the 2nd Open Conference of Users of NUMECA in Russia. – 2013.
[5] Galerkin, Y.B Turbocompressors. - Moscow. - 2010. – C. 596. (in Russian).