

Comparison of CFD-calculations of centrifugal compressor stages by NUMECA Fine Turbo and ANSYS CFX programs

Y B Galerkin¹, I B Voinov² and A A Drozdov^{1*}

¹ R&D Laboratory “Gas dynamics of turbo machines” Peter the Great St. Petersburg Polytechnic University, Polytechnical st. 29, St. Petersburg, Russia

² Engineering Center “Center of Computer-Aided Engineering” Peter the Great St. Petersburg Polytechnic University, Polytechnical st. 29, St. Petersburg, Russia

E-mail*: A_drozdi@mail.ru

Abstract. Computational Fluid Dynamics (CFD) methods are widely used for centrifugal compressors design and flow analysis. The calculation results are dependent on the chosen software, turbulence models and solver settings. Two of the most widely applicable programs are NUMECA Fine Turbo and ANSYS CFX.

The objects of the study were two different stages. CFD-calculations were made for a single blade channel and for full 360-degree flow paths.

Stage 1 with 3D impeller and vaneless diffuser was tested experimentally. Its flow coefficient is 0.08 and loading factor is 0.74. For stage 1 calculations were performed with different grid quality, a different number of cells and different models of turbulence. The best results have demonstrated the Spalart-Allmaras model and mesh with 1.854 million cells.

Stage 2 with return channel, vaneless diffuser and 3D impeller with flow coefficient 0.15 and loading factor 0.5 was designed by the known Universal Modeling Method. Its performances were calculated by the well identified Math model. Stage 2 performances by CFD calculations shift to higher flow rate in comparison with design performances. The same result was obtained for stage 1 in comparison with measured performances. Calculated loading factor is higher in both cases for a single blade channel. Loading factor performance calculated for full flow path (“360 degrees”) by ANSYS CFX is in satisfactory agreement with the stage 2 design performance. Maximum efficiency is predicted accurately by the ANSYS CFX “360 degrees” calculation. “Sector” calculation is less accurate. Further research is needed to solve the problem of performances mismatch.

1. Introduction

CFD-methods are applied to centrifugal compressors, both for research and design. Experience shows that results are highly dependent on turbulence model, computational grid quality and so on. To select the correct way of modeling is necessary to make a comparison of CFD- calculated and test data. R&D Laboratory “Gas dynamics of turbo machines” has extensive experience in CFD- calculations [1-6]. The stage 1 was tested in the Compressor problems R&D Laboratory [7]. Performances of 3D impeller and vaneless diffuser were compared with calculated performances by different turbulence models and grids. The stage 2 of a multistage industrial compressor consisting of 3D impeller, vaneless diffuser and return channel was designed and performances were calculated by the Universal modeling method [8]. Its performances were calculated by two different programs NUMECA Fine Turbo and ANSYS CFX. The fundamental equations that describe fluid flow behavior are the Navier Stokes equations.



This is a set of five partial differential equations that describe the conservation of mass, momentum and energy. Flow in gaps between an impeller and a stator were not modeled in some cases. Unstructured and structured computational meshes with different number of cells were constructed. Their number depends on the simulated elements of the flow part. The calculations were made in a stationary setting.

For the impeller and stator elements different computational meshes were constructed. At the interface between the rotating mesh of the impeller and the fixed mesh of the stator elements an interface defined to connect them. The stage averaging interface is used in both programs. The convergence criterion was the attainment of the value of quadratic discrepancies no higher than $1e-5$. A summary of various aspects and properties of the NUMECA Fine Turbo and ANSYS CFX flow solvers can be found in their online documentation.

Authors' experience [1-6] and known sources [9-11] show that CFD-calculation satisfactorily predicts stage maximum efficiency. Loading factor is overestimated by 6-12% at a design flow rate usually. Performances at flow rates below design flow rate are not predicted satisfactorily. The results in paper [11] show that numerical simulation in stationary and nonstationary calculation does not lead to any significant difference in the modeling of the right-hand side of the performances. Unsteady calculation allows more accurate calculation of the surge.

In the paper [12] it was shown that calculation of a centrifugal compressor stage in a "360 degrees" setting and with considering leakage losses, gives good agreement with the experiment. It is not possible to estimate the accuracy of the coincidence of the calculation and the experiment results on the given graphs. The polytrophic head performances are in good agreement with the experimental ones on the inclination angle and quantity. In the paper [13] the stage of the industrial centrifugal compressor is calculated by the ANSYS CFX program. Comparison of the calculation and experiment results had shown that the calculated efficiency performances are shifted towards higher flow rate compared to the measured one. The curves look similarly to each other from the qualitative viewpoint. The quantity of the maximum efficiency is well modelled. The maximum calculated pressure ratio is less than the experimental order of 6%.

2. Performance parameters

Calculated and tested performances are presented as functions $\eta^*, \psi_i, \psi_T = f(\Phi)$. Flow rate coefficient Φ is:

$$\Phi = \frac{4\bar{m}}{\rho_0^* \pi D_2^2 u_2}, \quad (1)$$

there $\rho_0^* = p_0^* / RT_0^*$ - gas density by total parameters.

Total polytrophic efficiency η^* (adiabatic compression):

$$\eta^* = \frac{\ln\left(\frac{p_{ex}^*}{p_{inl}^*}\right)}{\ln\left(\frac{T_2^*}{T_{inl}^*}\right)} \frac{k-1}{k}. \quad (2)$$

Loading factor ψ_T calculated by CFD:

$$\psi_T = \frac{C_p (T_2^* - T_0^*)}{u_2^2}. \quad (3)$$

Work coefficient measured:

$$\psi_i = \frac{C_p (T_2^* - T_0^*)}{u_2^2}. \quad (4)$$

Impeller loss coefficient ζ_{imp} calculated by CFD:

$$\zeta_{imp} = \frac{2 \cdot \psi_T \cdot (1 - \eta_2^*)}{(w_1 / u_2)^2}. \quad (5)$$

Stator elements loss coefficient ζ_{stat} is calculated by the formula:

$$\zeta_{(VLD, Rch)} = \frac{P_{out}^* - P_{inl}^*}{(\rho_{inl} c_{inl}^2) / 2}. \quad (6)$$

Direct comparison of efficiency calculated by CFD and measured efficiency in case of the stage 1 is not quite correct. Parasitic losses, i.e. disc friction and labyrinth leakage losses were not taken into account by CFD-calculations. These losses lead to loss of efficiency not more than 1.5% in stage 1. I.e., if CFD -calculated efficiency is bigger than the measured efficiency inside 1.5% the prediction is quite correct.

3. Stage 1

The stage with 3D impeller, VLD and scroll was designed and tested in SPbPU [7]. The experimental 600-kW test rig was driven by an electric motor. Nominal motor speed was 18 000 rpm with the possibility of regulation. Test rig included a boosting mechanical transmission.

Its design parameters are: $\Phi_{des} = 0.08$, $\psi_{Tdes} = 0.74$, $M_u = 0.78$. The main part of blades is a radial plate. Its blade cascade is shown in figure 1. The VLD normalized radial length is $\bar{D}_4 = 1.6$. The computational domain was extended to $\bar{D}_4 = 2.4$. The gas flow in the gaps between the impeller and the stator elements is not modeled. Therefore, the loading factor was calculated – not a work coefficient.

Calculations were carried out by the software package NUMECA Fine Turbo. Unstructured computational grids with 1.083, 1.224 and 1.854 million elements were formatted by NUMECA Fine AutoGrid. The turbulence model Spalart-Allmaras was applied in accordance with NUMECA program guidelines. In order to evaluate the effect of turbulence models on the results the SST turbulence model and Spalart-Allmaras with extended wall function were used too. The surfaces of the hub, shroud and blades are modeled as a smooth wall with adhesion.

Experimental and calculated total polytrophic efficiency performances are presented in figures 2 and 3. Measured work coefficient performance and calculated loading factor performances are presented in figure 4.

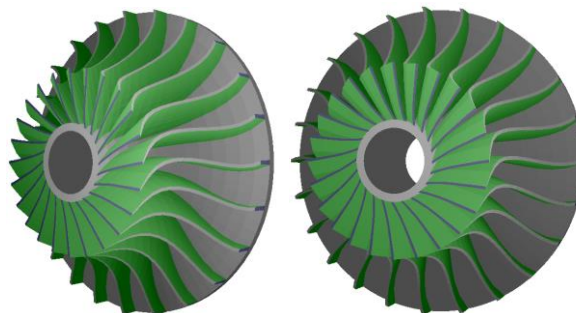


Figure 1. Stage 1 blade row views.

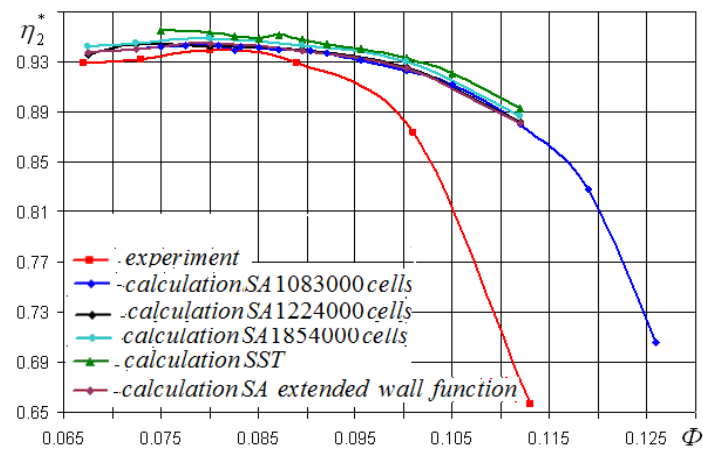


Figure 2. Measured and calculated total polytropic efficiency of the stage 1 impeller.

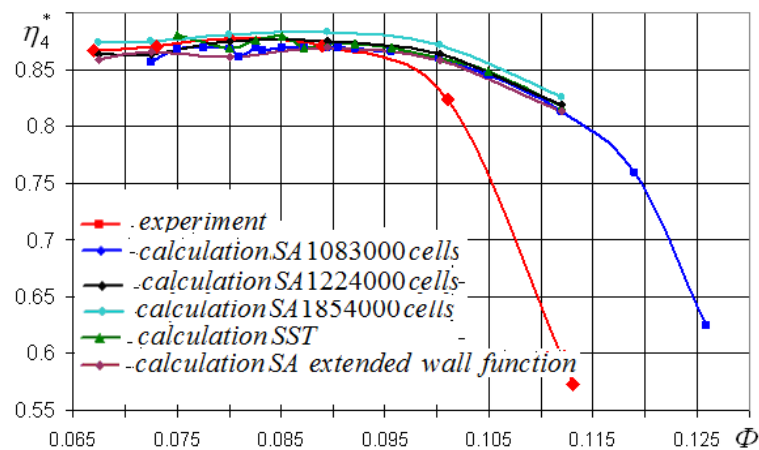


Figure 3. Measured and calculated total polytropic efficiency of the stage 1 impeller and vaneless diffuser.

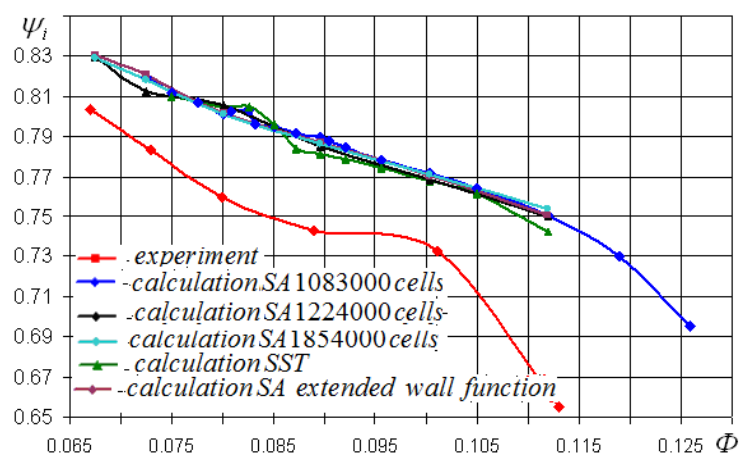


Figure 4. Measured performance of the work coefficient and calculated performances of the loading factor.

The visible result is that all calculations cannot predict efficiency of the impeller if $\Phi > 1.08\Phi_{des}$. The same is true for the stage “Impeller and VLD” if $\Phi > 1.19\Phi_{des}$ (that is not bad). The calculated efficiency of the impeller in the flow rate zone $\Phi < 1.08\Phi_{des}$ is bigger than the measured efficiency. The parasitic losses are about 1.5% for this impeller. These losses are neglected by the calculations. Meaning it the result of modeling is satisfactory in all cases. The performance calculated by the Spalart-Allmaras model with 1.224 million elements is more logical. The stage “impeller and VLD” performance is modeled the best of all by the Spalart-Allmaras model with 1.854 million elements. Calculation by different turbulence models with 1.084 million elements has not demonstrated significant difference. For further calculations the turbulence model Spalart-Allmaras was selected according to NUMECA recommendations. In all cases calculations with flow rate coefficients less than 0.070 have demonstrated problems with iterative processes.

Loading factor performances modeling is incorrect in all calculations. Calculated loading factor exceeds the measured work coefficient by 7.5% at $\Phi_{des} = 0.080$. The calculated and measured efficiency at the design flow rate is quite satisfactory.

4. Design of the stage 2

The stage “impeller, VLD and return channel” has the following design parameters: $\Phi_{des} = 0.15$, $\psi_{Tdes} = 0.50$, $M_u = 0.55$. The stage was designed by the method presented in [14]. Inviscid velocity diagrams on three blade to blade surfaces and normalized meridional velocities on eight three blade to blade surfaces are presented in figure 5.

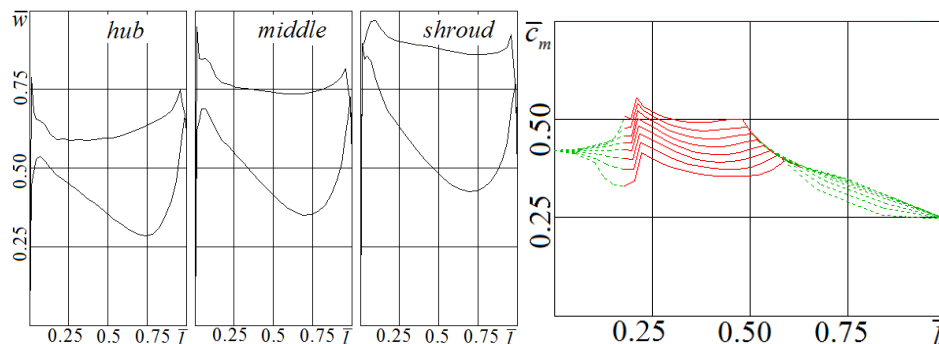


Figure 5. Stage 2. Inviscid velocity diagrams on three blade to blade surfaces and normalized meridional velocities on eight blade to blade surfaces.

Blade configuration is result of candidates' comparison with different inviscid velocity diagrams. The aim is to minimize velocity peaks and blade load near leading edges. Flow deceleration on a blade suction side must be minimized too. Its blade cascade is shown in figure 6.

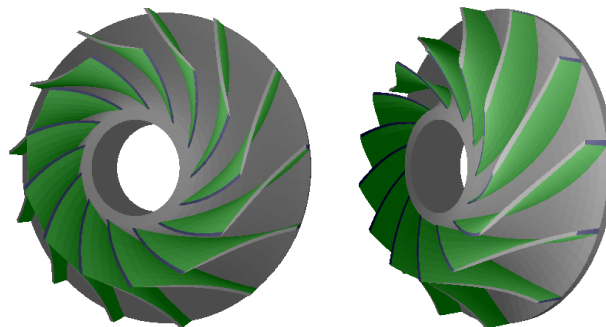


Figure 6. Stage 2 blade row views.

The calculations by the CFD-program NUMECA Fine Turbo were performed to optimize the stator part. The turbulence model Spallart-Almarras and a mesh with 2.5 million cells were used. Calculated flow visualization was applied to detect problematic zones in the stage's flow path. The calculation results have allowed correcting the shape of the stator elements. The meridional streamlines in original and corrected stages are shown in figure 7.

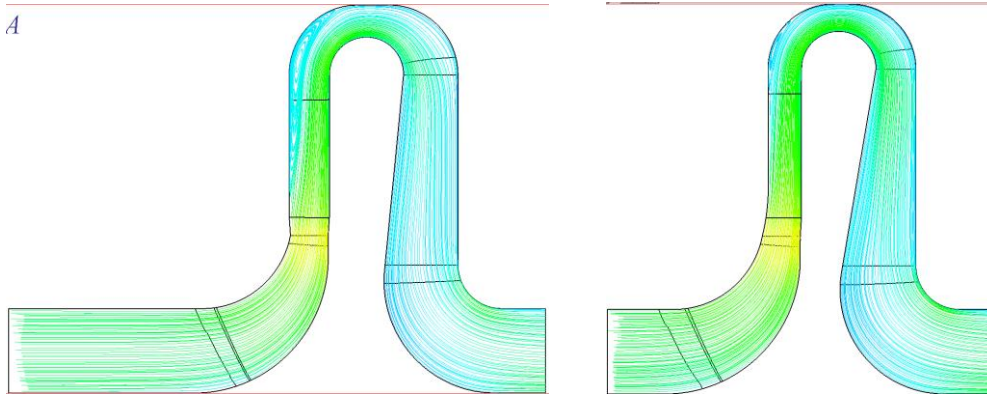


Figure 7. The meridional streamlines in two stage 2 candidates. Primary design - left, corrected by the results of CFD-calculations – right (NUMECA Fine Turbo).

VLD width was reduced to eliminate separation on the shroud wall of VLD. The RCH blades height at the entrance has been respectively reduced. It leads to the efficiency rise by 0.7%.

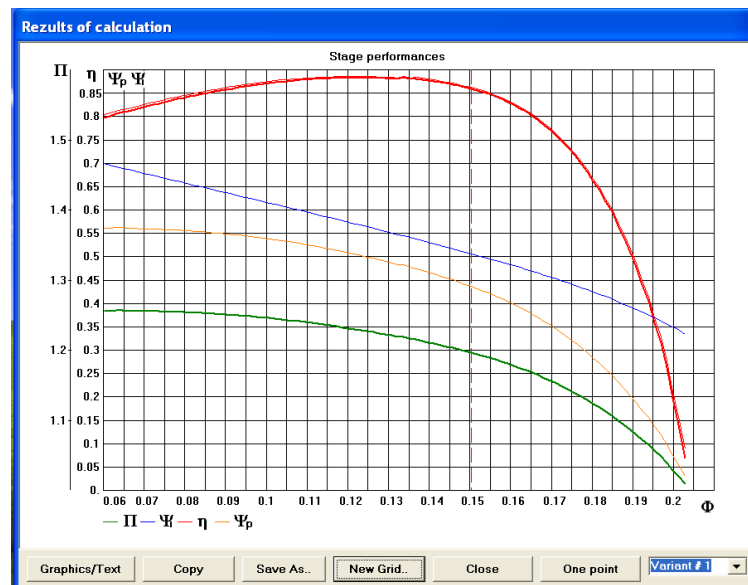


Figure 8. Stage 2 performances calculated by Universal modeling method.

Gas-dynamic performances were calculated by the Universal modeling method (figure 8). The high efficiency is expected because the design parameters $\Phi_{des} = 0.15$, $\psi_{Tdes} = 0.50$, $M_u = 0.55$, $\bar{D}_4 = 1.72$ are close to optimal.

5. Results of stage 2 performance modeling

The stage 2 performances were calculated by ANSYS CFX and by NUMECA Fine Turbo in four different variants. Results are presented in figures 9 and 10.

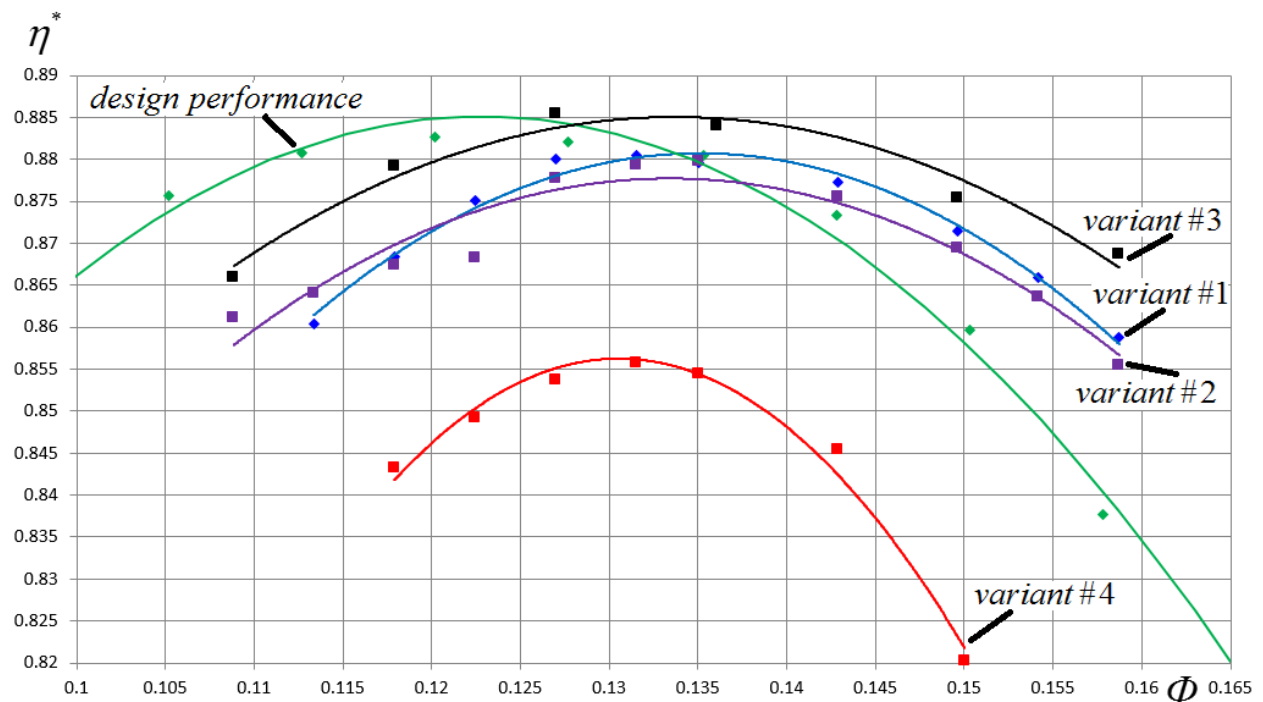


Figure 9. Stage 2 calculated efficiency performances.

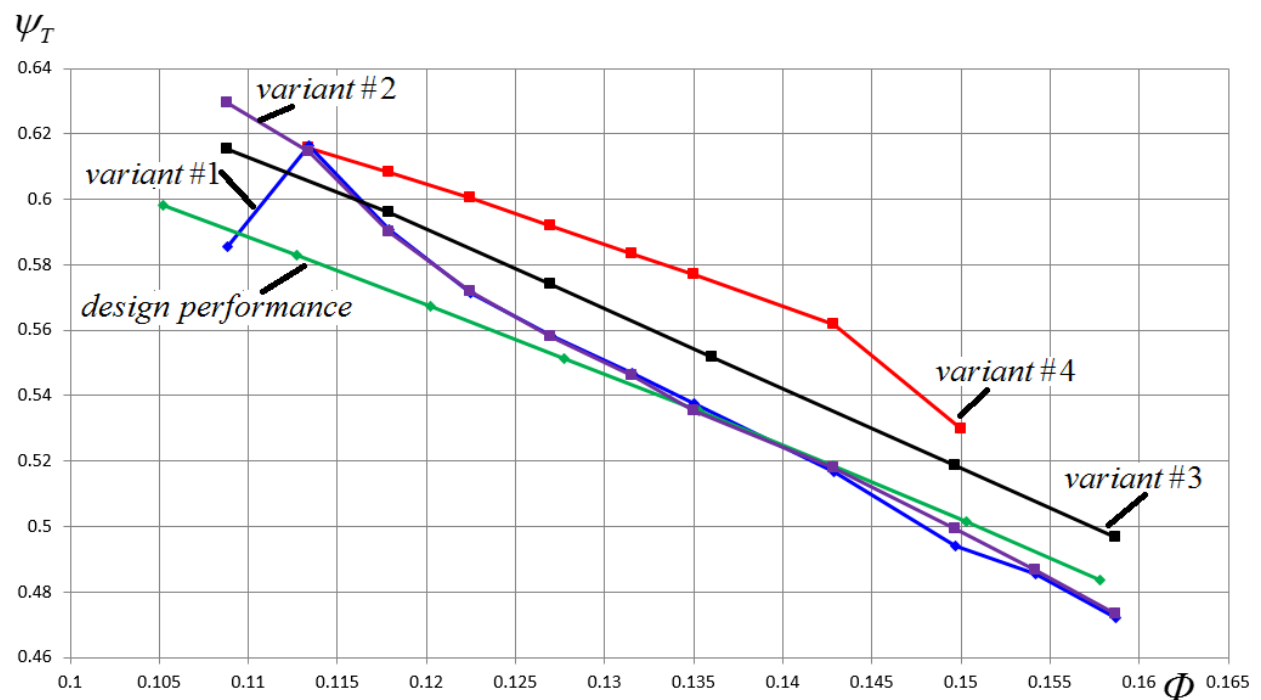


Figure 10. Stage 2 calculated loading factor performances.

The variants 1 includes radial inlet nozzle. The variant 2 has an axial inlet. Full flow path was calculated in both cases ("360 degrees"). Flow in "impeller – body" gaps was modeled and parasitic losses were calculated. The program ANSYS CFX uses the turbulence model SST. The design mesh for variants 1 contains 72 million cells, for variants 2 mesh contains 70 million cells.

Maximum efficiency is the same at $\Phi = 0.13$ that contradicts to practical experience. Tests demonstrate that the same stage with radial nozzle is less effective than with an axial nozzle by 2 - 4% [15]. The variant 3 was calculated without “impeller – body” gaps and one channel of the impellers and the RCH were modeled. The grids of different variants are presented in figure 11.

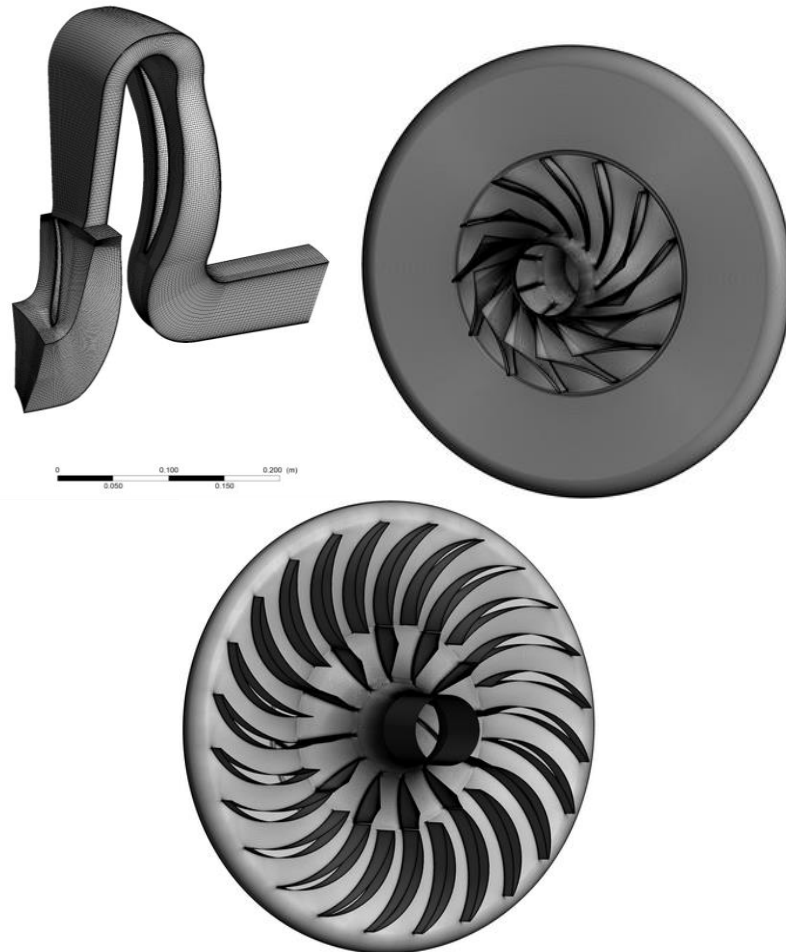


Figure 11. Computational grid for stage 2, variant 3 (left), variant 2 (amid and right).

To compare efficiency with the variant 2 parasitic losses were calculated by the Universal modeling method and calculated efficiency was diminished by about 1%. Maximum efficiency of the variant 3 is by 0.5% higher than that of the variant 2. It is equal to maximum efficiency calculated by the design program [14].

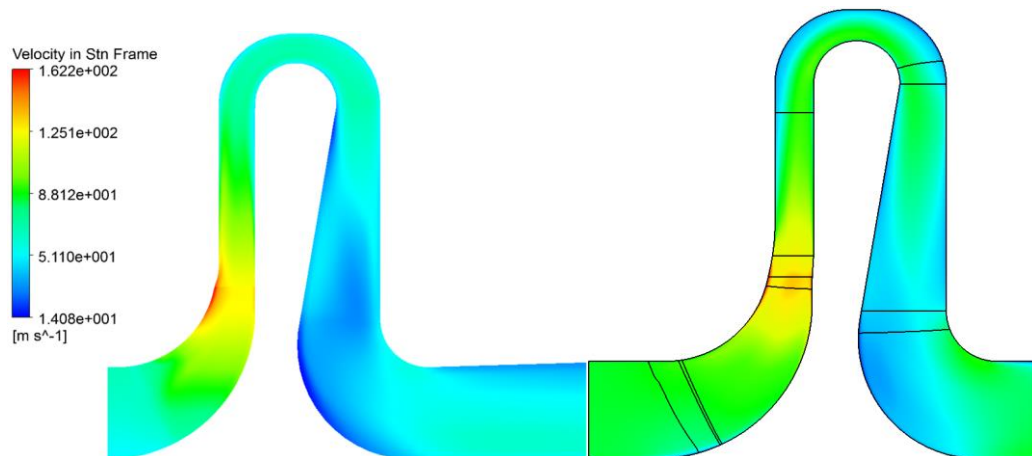
Variant 4 is the equivalent of the variant 3 but is calculated by NUMECA Fine Turbo. The efficiency is less by about 3% in comparison with ANSYS CFX calculation. The positive moment is that the loading factor performance calculated for full flow path (“360 degrees”) is in satisfactory agreement with the design performance. For other variants, the difference between the calculated and design loading factor is achieved at 3.5% for variant 3 and at 8.7% for variant 4. The angle of inclination of loading factor performances is good match in all cases.

Several calculated parameters of the stage 2 variants are presented in table 1.

Table 1. Gas-dynamic parameters of stage on design flow rate and on the maximum efficiency.

	Universal modeling Method (design performance)	Variant 2 ANSYS «360 degrees»	Variant 3 ANSYS "sector"	Variant 4 NUMECA "sector"
η_{des}^*	0.860	0.869	0.875	0.820
$\psi_{T des}$	0.502	0.504	0.519	0.53
$\zeta_{imp des}$	0.0986	0.07	0.121	0.123
$\zeta_{VLD des}$	0.0600	0.127	0.03	0.09
$\zeta_{RCH des}$	0.366	0.275	0.29	0.322
η_{max}^*	0.883	0.88	0.885	0.856

There is no visible regular tendency of the presented parameters variation which once more demonstrates problems of CFD-methods. Analysis of flow visualization leads also to contradictory results in some cases. As an instance figure 12 presents meridian velocity field in the variants 3 and 4 (design flow rate).

**Figure 12.** Meridian velocity field. Left – variant 3 (ANSYS CFX), right – variant 4 (NUMECA Fine Turbo).

Meridian flow seems more uniform in the variant 4, but impeller loss coefficients are practically equal in both cases. Flow in the VLD is more uniform in the variant 3 and its loss coefficient is much lower. On the contrary, flow is more uniform in the return channel of the variant 4 but its loss coefficient is higher.

6. Conclusions

1. The general tendency is that CFD-calculated performances shift to the right of real stage performances.
2. The connecting problem is that both programs cannot calculate flow rates far away of the design point. The modern stages with $\psi_{T des} \leq 0.50$ can operate with flow rate coefficients $\Phi = 0.5\Phi_{des}$ [16]. But CFD- calculation stops at $\Phi = 0.75\Phi_{des}$.
3. Maximum efficiency and design loading factor are predicted accurately by ANSYS CFX “360 degrees” calculation. “Sector” calculation is less accurate.

4. The program NUMECA Fine Turbo overestimates loading factor by 7–8% and underestimates efficiency by 3% in case of the stage with high flow rate coefficient $\Phi_{des} = 0.15$.

Nomenclature

C_p	heat capacity, J/kg*K;
c	absolute velocity, m/s;
D	diameter, m;
$\bar{D} = \frac{D}{D_2}$	relative diameter;
\bar{D}_h	hub relative diameter;
k	Isentropic coefficient;
\bar{m}	mass flow rate, kg/s;
M	Mach number;
$M_u = \frac{u_2}{\sqrt{kRT_{int tot}}}$	
p	pressure, Pa
R	gas constant, J/(kg*K);
T	temperature, K
u_2	impeller periphery speed, m/s;
$\bar{w} = \frac{w}{u_2}$	relative velocity
β_{fr}	disc friction coefficient;
β_{leac}	shroud disc labyrinth seal leakage coefficient;

ρ	gas density, kg/m ³ ;
ζ	loss coefficient;
$\psi_T = c_{u2} / u_2$	- loading factor;
ψ_i	work coefficient;
η^*	total polytropic efficiency;
Φ	flow rate coefficient;

Subscripts

0, 1, 2, 3, 4	indices of control sections
des	design flow rate
inl	inlet
ex	exit
max	maximum
m	meridian
stat	stator

Abbreviations

CFD	Computational Fluid Dynamics
ST	stator part of a stage
RCH	return channel
VLD	vaneless diffuser

Acknowledgments

The information about science work that is presented in the paper as result of the work of the R&D Laboratory "Gas dynamics of turbomachines", in science, research, technology, and development: "The creation of modern hi-tech production of design, production and installations tests, has compressing gaseous products for effective use in transport systems and technologies". (With financial support of the Government of the Russian Federation, Ministry of Education and Science of the Russian Federation. The contract 02.G25.31.0140 from 12/1/2015).

References

- [1] Galerkin Y, Borovkov A I, Voinov I B, Gaev A V, Gamburder D M, Sofronova A A and Lozovaja N S 2007 Results of calculation of a viscous flow in motionless elements of centrifugal compressor stages by means of program ANSYS/CFX system *Compressors and pneumatics* **2** pp 10–16 [Russian]
- [2] Galerkin Y B and Soldatova K V 2007 The analysis of gas flow in a gap "the covering disk – the case" a centrifugal compressor stage according to the FLUENT and CFX programs *Chemical and oil and gas mechanical engineering* **5** pp 27–29
- [3] Galerkin Y B and Marenina L A 2014 Research and improvement of centrifugal stages fixed elements by methods of computational dynamics Part 1 *Compressors and pneumatics* **1** pp 30–36 [Russian]
- [4] Galerkin Y B and Marenina L A 2014 Research and improvement of centrifugal stages fixed elements by methods of computational dynamics Part 2 *Compressors and pneumatics* **2** pp

10–15 [Russian]

- [5] Galerkin Y, Soldatova K and Solovieva O 2015 Numerical study of centrifugal compressor stage vaneless diffusers “*Compressors and their Systems*” Conference (London)
- [6] Galerkin Y and Drozdov A 2015 Sample of CFD optimization of a centrifugal compressor stage “*Compressors and their Systems*” Conference (London)
- [7] Simonov A M 2000 Research and development of centrifugal compressor stages with increased loading factor *Proceedings of the scientific compressor school* (SPbPU) pp 189–212 [Russian]
- [8] Galerkin Y 2010 *Transactions of the TU SPb compressor school* [Russian]
- [9] Melnicov V and Proscurov A 2005 Using the software package FlowVision for flow calculation of the turbochargers elements in *SKBT CAD and graphics* **4** pp 92–96 [Russian]
- [10] Gamburger D, Epifanov A and Gaev A 2009 Numerical investigation of three-dimensional viscous compressible gas in the centrifugal compressor impeller *Scientific and technical transactions of the TU SPb* **2**(78) pp 76–82 [Russian]
- [11] Lurie E, Van Slooten P, Medic G, Mulugeta J, Holley B, Feng J and Sharma O 2011 Design of a high efficiency compact centrifugal compressor for rotorcraft applications *American Helicopter Society 67th Annual Forum* (Virginia Beach May 2011)
- [12] Hazby H, Casey M, Robinson C and Spataro R 2017 The design of a family of process compressor stages *Proceedings of 12th European Conference on Turbomachinery Fluid dynamics & Thermodynamics (ETC12, April 3-7, 2017, Stockholm, Sweden)* Paper ID: ETC2017-134
- [13] Kryłłowicz W, Kozanecki Z, and Kabalyk K 2017 Technical and aerodynamical aspects of a high pressure synthesis gas turbocompressor modernization *Proceedings of 12th European Conference on Turbomachinery Fluid dynamics & Thermodynamics (ETC12, April 3-7, 2017, Stockholm, Sweden)* Paper ID: ETC2017-171
- [14] Galerkin Y 2010 *Tubrocompressors* (SPbSTU) 2010 [Russian]
- [15] Seleznev K P and Galerkin Y B 1982 *Centrifugal compressors* (Leningrad) p 271 [Russian]
- [16] Soldatova K 2016 Bank of model stages, with performances calculated based on the results of factory centrifugal compressors experiments *Compressors and pneumatics* **8** pp 20–24 [Russian]