Development of the Universal Modelling Method mathematical model and the practice of its application

Y.B. Galerkin¹, O.A. Solovyeva², A.A. Drozdov³ and V.B. Semenovskiy³

1. Leading research center "Digital Design and Modeling (Smart Design)" Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia
2. Higher School of Hydraulic and Power Engineering Construction Engineering Institute Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia
3. Higher School of Power Engineering Institute of Power Engineering Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia

Email: A_drozdi@mail.ru.

Abstract. We present the latest improvements to the Universal Modelling Method for optimal gas-dynamic design of centrifugal compressors. The calculation of the gas velocity at the inlet to the impeller blade cascade was corrected. The flow restructuring scheme at the inlet to the impeller blade cascade has been refined. In accordance with this, the mathematical model of the non-incidence gas inlet to an impeller has been improved. A new approach to modelling the velocity diagram on the blade surfaces is proposed, based on the flow parameters corresponding to the non-incidence flow inlet. A mathematical model with quasi-three-dimensional approach to calculating losses in 3D impellers has been developed. The method of gas-dynamic functions is applied to the calculation of flow parameters in control sections. The new mathematical model was identified based on the test results of model stages and compressors. The mathematical model showed a 0.525% error in modelling efficiency of stages with 2D impellers for the design flow rate. For stages with 3D impellers, the modelling error is 0.889%. To verify the new model, the characteristics of 11 low-flow-rate model stages were calculated. The efficiency modelling error for the design flow rate is 1.08%. A series of model stages with a range of design flow rates coefficient 0.15–0.015 and design loading factor of 0.5 was designed. Four of these model stages were tested on the Gas dynamics of turbomachines laboratory ECC-55 test rig. The measured gas-dynamic characteristics confirmed those declared in the design.

1. Introduction
Centrifugal compressors are used in almost all key industries. The beginning of their use dates back to the end of the 19th century. Initially, one of the main areas of application was ferrous and then non-ferrous metallurgy. Further industrial development has led to the emergence of areas requiring the use of centrifugal compressors, such as chemical production, pneumatics, refrigeration, etc. [1].

Economic development leads to an increase in demand for compressor equipment and a corresponding increase in their production volume. This is associated both with the construction of new industrial enterprises utilizing compressor equipment or with construction of new gas pipelines and with the need to renew the existing compressor park due to wear or changes in their operating
conditions. At the same time, the unit capacity of individual compressors also increases (up to 64 MW inclusive). The global market for centrifugal refrigeration compressors is estimated to have reached 18,000 units in 2017, up 3.5% compared to the same period in 2016 due to the rapid development of the magnetic bearing market [2]. The compressor drive consume up to 20% of the generated electricity [3], [4]. In total, more than 100 million compressors are produced annually in the world [5]. Although centrifugal compressors are not the most numerous, their total capacity is very high. In this regard, an important task is to ensure the maximum energy efficiency of centrifugal compressors.

In general, the industry requires design methods that will ensure high compressor efficiency while obtaining the specified design parameters (mass flow rate and pressure ratio) and at the same time do not require experimental verification of the designed compressor or its stages. For this, both commercial CFD programs and proprietary software systems can be used. However, the experience of using CFD calculations shows that CFD programs do not allow simulating the gas-dynamic characteristics of a centrifugal compressor with the required accuracy. Thus, it is necessary to create specialized engineering programs.

For this, compressor equipment manufacturers develop their own design methods and mathematical models based on them. In Russia, such work and research is presented in [6] - [10]. Similar researches are carried out by foreign experts [11] - [18].

A well-known foreign model for calculating centrifugal compressors is used in the Concepts NREC software package, authored by Dr. D. Japikse [12], [13]. The calculation of head losses in the mathematical model uses the techniques of the boundary layer theory and is based on a two-zone scheme: friction loss and mixing loss. When designing a centrifugal stage, one-dimensional optimization of the stage dimensions is initially performed, then an inviscid quasi-three-dimensional calculation of the impeller is performed, and at the final step, gas-dynamic characteristics are calculated using the CFD program. The mathematical model contains empirical formulas for calculating losses in individual elements of the flow path of the stage, which implies identification.

In paper [16], the authors compare the experimental gas-dynamic characteristics of an internal combustion engine turbocharger at three speeds with calculations using a proprietary computer program Compressor Off-design Radial Analysis (CORA). The CORA program is based on mathematical models for calculating flow parameters previously obtained by various authors.

Three different models were used to calculate the losses in the CORA program in the impeller: Auinger [14], Oh [19], and Galvas [20]. These three models take into account a different number of loss components in the impeller, so the Auinger model is the most detailed and takes into account most of the loss components, and the Galvas model is the least detailed (it does not take into account mixing losses, leakage losses, etc.). The model for calculating losses in the vaneless diffuser is taken from Herbert [21], and the model of losses in a scroll from Weber and Koronovski [22]. According to the authors, a combination of the Galvas and Auinger loss models for the impeller matches experimental data well. The authors emphasize the relevance of one-dimensional models for calculating the gas-dynamic characteristics of centrifugal compressors and the importance of their further development.

The authors of papers [23], [24] compared the gas-dynamic characteristics of a stage with a 3D impeller, a vaneless diffuser and a return channel using the one-dimensional calculation program of their own production KSTK and the CFD program NUMECA. The authors used the one-dimensional KSTK model to design a series of centrifugal compressor stages. This demonstrates the wide practical application of one-dimensional models.

The paper [25] presents the identification and verification of a one-dimensional model for calculating turbocharger centrifugal compressors, a long-term project of the Dept. of Industrial Engineering of the Naples University. The creators of this mathematical model pay special attention to the system for specifying the shape of the flow path of the stage, and in the process of verification, to the exact repetition of the 3D impeller shape. The mathematical model takes into account the external heat exchange with the casing. Losses in the impeller are divided into incidence losses in the flow around the leading edges of the blades, losses in shock waves, friction losses on surfaces, and
recirculation losses. Scroll losses are divided into tangential and radial losses. The authors used the results of gas dynamic tests for 5 different compressors in the model identification process.

In the paper [26], the application of the quasi-three-dimensional method to the calculation of a turbocharger centrifugal compressor is presented. The applied model is based on the equations of conservation of mass, momentum, and energy for unsteady and compressible flows. The foundations of the model are taken from the Navier-Stokes equations, but it uses the inviscid flow assumption, neglecting the terms of the second-order partial differential equation. Also, the effect of thermal diffusivity was not taken into account. The model allows calculating the parameters of a single-stage centrifugal compressor consisting of 3D impeller, vaneless diffuser and scroll.

The mathematical model was verified based on the results of turbocharger tests carried out at the University of Genoa. The comparison was made for tests with four different Mach numbers. The results showed good agreement between the calculated and experimental data and the importance of applying quasi-three-dimensional approaches to modeling centrifugal compressors.

For decades, research on centrifugal compressors has been carried out at LPI-SPbPU [27]. Analysis and generalization of the results obtained allowed the prof. Yu.B. Galerkin to create a method for gas-dynamic design of centrifugal compressors and a mathematical model based on it. It allowed to determine the characteristics of the compressor by dimensions of its flow path and similarity criteria (isentropic exponent, conventional Mach and Reynolds numbers). The mathematical model was implemented in the form of computer programs called the Universal Modeling Method (further referred to as the Method) [28] - [31].

Mathematical models of the Universal Modeling Method use simplified one-dimensional, two-dimensional and quasi-three-dimensional models, the calculation of which implies the use of empirical equations and the presence of simplifying assumptions. The values of empirical coefficients are determined in the process of identifying a mathematical model based on the results of experimental researches. The empirical equations themselves are implemented in the form of iterative processes. All this imposes a number of restrictions on the capabilities of programs.

The accumulation of new calculated and experimental data made it possible to make the mathematical models of the 9th version of the Universal Modeling Method more detailed and complex, to move away from the organization of iterative processes to the use of gas-dynamic functions. To transfer information to designers and carry out checks using CFD calculations, the Universal Modeling Method was supplemented with a program for constructing meridional and radial schematic images of the flow path and a program for constructing a three-dimensional impeller model. The main improvements of the 9th version of the Universal Modeling Method mathematical model, its identification and application practice are shown below.

2. Innovations of the mathematical model

Impeller inlet velocity. 1-D approach to calculate inlet velocity rate in the 8th and earlier versions of the Method [32]:

\[ \bar{w}_1^* = \frac{\Phi_0}{4r_1D_1 b_{im} \sin \beta_{sl} \epsilon_1^*}, \]

where \( r_1 \) is blade blockade factor at the inlet to the impeller; \( \epsilon_1^* \) is coefficient of compressibility at the inlet; \( \Phi_0 \) is flow rate coefficient of an impeller; \( \beta_{sl} \) is inlet blade angle; \( D_1 \) is inlet diameter of the impeller relative to \( D_2 \); \( b_{im} \) is height of impeller blades at the inlet relative to \( D_2 \).

Correct value of \( \bar{w}_1^* \) is important to calculate loss efficiency:

\[ \Delta \eta_{imp} = 0.5 \frac{\zeta_{imp}}{\psi_T} \bar{w}_1^*, \]

where \( \zeta_{imp} \) is impeller loss coefficient, \( \psi_T \) is loading factor.
The problem of impeller efficiency simulation of stages in a wide range of \( \Phi_{\text{des}} \) and \( \psi_{T_{\text{des}}} \) stressed the necessity to investigate inlet velocity calculation. The PC program 3DM-023 [28] executed inviscid Q3D calculations in impellers with flow rate coefficients \( \Phi_{\text{des}} = 0.015–0.15 \), loading factors \( \psi_{T_{\text{des}}} = 0.5–0.65 \), relative hub ratios \( \beta_{\text{hub}} = 0.25–0.45 \). Measurements inside impellers [28] demonstrated good simulations of inlet velocity by Q3D inviscid calculations. Figure 1 demonstrates a sample of calculation.

![Figure 1. Sample of Q3D inviscid flow calculation.](image)

Mean Q3D velocities are less than the velocity calculated using the eq. (1). The coefficient \( K_{w_1} = \frac{W_{\text{IMM}}}{W_{\text{IQ3D}}} > 1 \) is inserted into the eq. (1) in the 9th version of the mathematical model:

\[
W_{1}^{*} = \frac{1}{K_{w_1}} \frac{\Phi_{0}}{4\tau_{\text{D}} B_{\text{in}} \sin \beta_{\text{bl}} c_{1}'},
\]

(3)

Values of \( K_{w_1} \) lie in range 1.2–1.5 for different impellers. Formula (3) is a satisfactory approximation of the calculation experiment:

\[
K_{w_1} = 2.3226 \cdot 10^{-6} \beta_{\text{bl}}^{0.8} - 0.000273 \beta_{\text{bl}}^{1.0} + 0.012052 \beta_{\text{bl}}^{2.0} - 0.24047 \beta_{\text{bl}}^{3.0} + 2.9458.
\]

(4)

A graphic representation on Figure 2 demonstrates accuracy of this approximation. The introduction of the empirical coefficient made it possible to increase the accuracy of calculations for the 9th version of the mathematical model compared to the 8th version.

**Inlet flow conditions.** The Method calculates characteristics as 110–150 points at different flow rates \( \bar{m} < \bar{m}_{\text{des}} < \bar{m} \). For correct characteristics’ simulation it is important to start with properly defined design flow rate. The design flow rate corresponds to the impeller non-incidence inlet condition, Figure 3. The flow pattern was obtained as a result of calculations using the NUMECA Fine/Turbo program.
Figure 2. Calculated ratio $\frac{W_{1MM}}{W_{1Q3D}}$ and approximation by the formula (4)

Figure 3. Velocity vectors around a blade leading edge. Non-incidence inlet.

The condition of a non-incidence inlet: stagnation point is located at the center of the leading edge. Blade load influences critical streamline direction. The scheme in Figure 4 (where $\bar{W}_{ini}$ is relative flow velocity at the inlet to the impeller, taking into account constraint in non-incidence mode) replaces blade load by a vortex with the same circulation.
Figure 4. Scheme of the effect of a vortex (non-incidence inlet flow rate)

The vortex position is \( r_{pc} \). Non-incidence condition: a critical streamline angle is equal to the inlet blade angle \( \beta_{icrit} = \beta_{hil} \). Average relative inlet flow velocity \( \bar{w}_{ini}' = \bar{w}_{ini} / u_2 \) differs from the velocity of a critical streamline by a component \( \Delta c_{ini} = \Delta c_{i} / u_2 \). The vortex creates this component:

\[
\Delta c_i = \frac{X \psi_T}{z} \left( \frac{1}{1 - K_{pc}} \right) \sin \beta_{hil},
\]

where \( z \) is number of blades, \( K_{pc} = \frac{r_{pc} - r_1}{r_2 - r_1} \) is the coefficient that represents blade load distribution along the blade radius, \( X \) is the empirical coefficient to be identified. The viscosity of the fluid is taken into account when calculating the loading factor using the empirical coefficient \( K_{il} \).

The value of the velocity \( \Delta c_i \) that determines the restructuring of the flow is involved in the calculation of incidence losses. The formulas for calculating incidence losses presented in [31] include empirical coefficients determined in the process of identifying a mathematical model.

Twenty impellers with different design parameters were the object of the analysis. Non-incidence flow rates were obtained by iterative calculations of inviscid Q3D flow. Inlet velocity triangles demonstrated non-incidence inlet with different \( X \) values for different impellers. The geometric parameter, relative throat width \( \bar{a}_i = a_i / D_2 \), influences the coefficient \( X \) the most. Approximating formulae for different types of impellers are:

- 3D impellers (two different series):

\[
X = 1 + 9.75 \Delta \bar{w} - 31.9 \bar{a}_i;
\]

- 2D impeller with profiled blades (controlled blade load distribution along radius):

\[
X = 1 + 5 \Delta \bar{w} - 24.9 \bar{a}_i.
\]

- 2D impellers with arc blades:

\[
X = 1.37 = \text{const}.
\]

Figure 5 demonstrates the accuracy of the formulae (6–8). Non-incidence flow rate coefficient \( \Phi_{n \text{MM}} \) obtained using the formulae (6–8) is related to the \( \Phi_{n \text{3DM}} \) calculated by the 3DM-023 PC program.
Figure 5. Comparison of non-incidence flow rates. Dependence $\Phi_{\text{niMM}} / \Phi_{\text{ni3DM}} = f(\Phi_{\text{ni3DM}})$ for stages with 3D and 2D impellers with arc and profiled blades when calculating new formulas for calculating flow restructuring.

The accuracy of formulae (6–8) is sufficient for a preliminary design. To calculate characteristics of an existing compressor, better simulation is necessary. New test results will be approximated in the future.

**Q3D approach for 3D impellers.** A significant non-uniformity of the flow along the height of the 3D impeller blades leads to the fact that using the one-dimensional approach used in the 8th version of the mathematical model is not the best solution. In the 9th version of the Method, quasi-three-dimensional approach is applied. The flow in the blade cascade is divided by seven blade-to-blade surfaces. The mathematical model calculates flow parameters in eight elementary blade cascades. Friction losses on the hub and shroud surfaces occur on the 1st and 8th blade elements. Mixing losses in the case when the VLD width is higher than the height of the impeller blades at the outlet $\zeta_{\text{mix b}b/2}$ are calculated using the 1-D formulation.

At the entrance to the blade cascade, it is assumed that the flow coefficient $\phi_{li}' = c_{li} / u_2$ is the same on each streamline. Relative velocity at the entrance to the blade cascade:

$$\bar{w}_{li} = \sqrt{\bar{D}_{li}^2 + \phi_{li}'^2}.$$ (9)

After determining all the components of the head losses in the blade cascade, the polytropic coefficient of the impeller is calculated. Its value determines the polytrophic process: $pV^n = \text{const}$. The sum of losses of all blade elements is in the square brackets:

$$\frac{n_{i-2}}{n_{i-2} - 1} = \frac{k}{k - 1} - \frac{k}{k + 1} \left[ \sum_{i=1}^{k} (\zeta_{fr bli} + \zeta_{mix}) \bar{W}_1^{r2} + \zeta_{mix b} \bar{W}_1^{r2} + \zeta_{lim-i} \bar{W}_{i-1}^{r2} + \zeta_{mix} \bar{W}_{i-1}^{r2} + \zeta_{inc} \bar{W}_{i-1}^{r2} + 2 \nu_f (\beta_{\text{lax}} + \beta_p) \right] \left[ \frac{T_{i-1}^* T_1^*}{T_{i-1}^* (T_1^*)} \right].$$ (10)

where $\beta_{\text{lax}}$ is labyrinth seal leakage coefficient; $\beta_p$ is disc friction coefficient; $k$ is isentropic coefficient; $\zeta_{fr bli}$ is coefficient of friction loss on blade surfaces; $\zeta_{mix}$ is coefficient of mixing losses; $\zeta_{inc}$ is coefficient of incidence losses; $T_{i-1}^*$ is static temperature at the impeller inlet; $T_{i-1}^*$ is total temperature at the impeller inlet; $T_2$ is static temperature at the impeller outlet; $\lambda_{\infty}$ is velocity coefficient at the impeller inlet.
Q3D approach more precisely calculates blade surface area. It is important for the friction loss coefficient calculation. The PC program 3DM.023 calculates blade area quite precisely. However, these calculations are too time-consuming to be directly used in the Method. Figure 6 shows a comparison of the blade areas of the 3D impellers, calculated by the 3DM.023 program with 1-D approach of the 8th and Q3D approach of the 9th Method versions.

![Figure 6. The blade area ratio S_{MM} / S_{3DM} of 3D impellers with different \( \Phi_{des} \)](image)

The average error is 31% in the 1-D approach and 7% in the Q3D approach.

**CFD mathematical model for vaneless diffusers.** The CFD model substitutes a rather precise but complicated model that calculates friction losses with a number of empirical coefficients. The ANSYS CFX program was used to calculate efficiency characteristics and outlet parameters of diffusers with their relative width \( b / D_2 = 0.006–0.10 \) and with radial length \( D_4 / D_2 = 1.6–2.0 \) in the range of velocity coefficients \( \lambda_c = 0.39–0.82 \) and Reynolds numbers \( Re_{b2} = 36 \, 800–73 \, 600 \), inlet flow angles \( \alpha_z = 10^\circ–90^\circ \). The 8th version of the mathematical model guaranteed the adequacy of the application for the relative width \( b / D_2 = 0.014–0.10 \). Loss coefficient and flow outlet angle are functions of geometry and gas dynamic parameters:

\[
\zeta_{diff}, \alpha_4 = f(\vec{b}_2, \vec{D}_4, \alpha_2, \lambda_{c2}, Re, \vec{k}_{Re}) \tag{11}
\]

where \( \vec{b}_2 \) is relative width, \( \vec{D}_4 \) is relative length, \( \alpha_2 \) is inlet flow angle, \( \lambda_{c2} \) is inlet velocity coefficient (compressibility criteria), \( Re_{b2} \) is Reynolds number, \( \vec{k}_{Re} \) is relative roughness.

The data of calculations were approximated by sets of algebraic equations. A sample for a loss coefficient:

\[
\zeta_{diff} = \frac{A \cdot \alpha^{b} \cdot K_{D4}}{K_{Re \cdot \vec{k}_{Re}}} \tag{12}
\]

where \( A, B, \alpha, K_{D4}, K_{Re \cdot \vec{k}_{Re}} \) are sets of algebraic equations.

**Mathematical model of tangential exit nozzles.** The model establishes nozzle dimensions and calculates loss coefficient. The continuity equation for a scroll:

\[
\bar{m}_{\Theta} = \frac{\Theta}{360^\circ} \bar{m}_{des} \approx \rho c_{u4} R_4 \int_{\tau_1}^{\tau_2} \frac{\rho r}{c} \, dr \tag{13}
\]

An average flow velocity at the outlet and an area of the exit follow from eq. (13):
where the empirical coefficient \( K_s = 1.25 \) \cite{33}. Equations to calculate sizes of scrolls in Figure 7 and others are formulated assuming proper functions \( b = f(r) \).

\[
\bar{c}_{des}360 = \frac{1}{K_s} c_{av} \bar{D}_i \bar{D}_{60\text{avr}},
\]

Figure 7. Dimensions of scrolls of different types

The model of a loss coefficient summarizes losses in different parts of a scroll and losses of different nature: friction, separation, secondary, and incidence. The new mathematical model contains 10 empirical coefficients, which is 1.5 times less than in the previous model.

Flow parameters in control planes calculation using the velocity coefficient \( \lambda \). The engineering 1-D model calculates static and total pressures and temperatures, density and velocity vectors in control planes throughout the flow path: from its inlet to the outlet. The continuity equation \( \dot{m} = cf \frac{P}{RT} = \text{const} \) solves the problem iteratively. It creates numeric problems in some special cases. To avoid it, the continuity equation is presented in the form that does not require iterations:

\[
\frac{\lambda_{i+1}}{\lambda_i} \sqrt{\frac{\dot{m}_{i+1}^*}{\dot{m}_i^*}} \left( \frac{1 - \frac{k - 1}{k + 1} \lambda_{i+1}^2}{1 - \frac{k - 1}{k + 1} \lambda_i^2} \right)^{\frac{1}{k-1}} = \frac{\Phi_i}{\Phi_{i+1}} \frac{\bar{f}_i}{\bar{f}_{i+1}},
\]

and

\[
\Phi = \left( \frac{k + 1}{2} \right)^{\frac{j}{x-1}} \left( 1 - \frac{k - 1}{k + 1} \right)^{\frac{j}{x-1}} \frac{\dot{m}}{\lambda_u} \frac{\dot{f}}{\lambda (1 - \lambda^2)^{\frac{j}{x-1}}},
\]

where: \( \lambda = \frac{c}{\sqrt{k + 1} RT} \), \( \lambda_u = \frac{u}{\sqrt{k + 1} RT_{in}} \), \( \Phi = \frac{4\dot{m}}{\rho \pi D^2} \), \( n \) is polytrophic coefficient.

3. Identification and Verification of the 9th Version of the mathematical model

The main objects of modeling are loss coefficients in the eq. (10) and in similar equations for other elements of the flow path. The model treats the loss coefficient as a sum of coefficients of friction (secondary flow is included), mixing (separation) and incidence (for blades and vanes only) losses. Calculation of losses friction loss coefficient \( \xi_s \) of a blade suction side is shown as an example:
\[
\zeta_s = c_{ws} z \frac{4b_{avr}}{\pi \Phi} \bar{e}_{avr} \bar{w}_{avr} \left( \frac{w_{avr}}{w_1} \right)^2,
\]

where \( b_{avr} = b_{avr} / u_2 \) is average blade height, \( \bar{T} = l / D_2 \) is blade length, \( \bar{w}_{avr} = w_{avr} / u_2 \).

In the eq. (17), the mathematical model associates the coefficient of friction force of a blade \( c_w \) with the coefficient of a thin plate friction force \( c_f = 0.0306 Re^{-1/7}[34] \):

\[
c_{ws} = c_f \left( 1 + X_i \left( 1 - \frac{w_{z2}}{w_{x1}} \right)^{X_{x1}} \right) \left( 1 + X_{x2} Ro^{X_{x3}} \right).
\]

In the eq. (18), the empirical coefficient \( X \) of the mathematical model needs to be identified. The expression in the first bracket represents the influence of a velocity gradient along a blade surface. The expression in the second bracket represents the influence of a velocity gradient normal to a blade surface. The normalized normal gradient for an impeller is \( Ro = 4 - \bar{w}_{avr} / \bar{R}_{blav} \) (\( \bar{R}_{blav} = R_{blav} / D_2 \) is the average radius of a blade curvature).

Figure 8. Identification results. Comparison of calculated (red lines) and experimental (blue and green lines) gas dynamic characteristics.
The identification of the mathematical model is carried out in the specialized program IDENT [32]. To identify the loss model, the characteristics of the model stages [27] were used. In total, 70 stage characteristics were used with 6 values of the flow coefficient for each test. Figure 8 demonstrates comparison of measured and calculated characteristics.

Average inaccuracy of efficiency calculation after the identification is ±1.0 % [35].

To verify the mathematical model, the IDENT program calculated characteristics of the stages that did not participate in identification. Low flow rate model intermediate type stages in Figure 9 (left) have very short VLD and 2D impellers with blades of constant height. The model 1:2 of a single-stage 32 MW pipeline compressor in Figure 9 (right) has the axial inlet nozzle, long VLD, and a scroll of design according to the part 2 of this text [36].

![Figure 9. Verification results. Left, intermediate type low flow rate model stages: measured (dashed lines) and calculated (solid lines); right, the model of a single-stage 32 MW pipeline compressor: experiment (blue and green lines) and calculation (red lines).](image)

The accuracy of verification calculations is the same as in identification.

4. The 9th Version Mathematical Model design results
The programs of the 9th version of the Universal Modeling Method in varying degrees of readiness have been used since 2014 in the design practice of the Laboratory “Gas Dynamics of Turbomachines”.

Twenty designed model stages have the following design parameters: flow rate coefficient \( \Phi_{des} = 0.015–0.15 \) and impeller diameter of 0.35 m. Two stages with \( \Phi_{des} = 0.025 \) and 0.035 were tested on the test rig ECC-55 of the Laboratory [37]. Design and test characteristics are in Figure 10.

The mathematical model accurately determines the flow rate range of both stages. Near the design flow rate, the mathematical model overestimates the efficiency by 0.8–1.3%. In case of the stage with \( \Phi_{des} = 0.025 \) at maximum flow rate the measured efficiency is even higher. The measured characteristic of the stage with \( \Phi_{des} = 0.035 \) shifts to the left in comparison with the calculated one. The test data of all new model stages will be the material for future improvements of the model stage.

The turbocharger compressor was jointly designed by the Laboratory and NPO Turbotekhnika JSC. The industrial partner manufactured and tested the compressor at his test rig [38]. The turbocharger is
intended for a low speed diesel converted for natural gas. The pressure ratio is not high. Gas-dynamic characteristics of the TKR 140E compressor are presented on Figure 11.

![Figure 1](image1.png)

**Figure 10.** Gas dynamic characteristics of a stage with $\Phi_{des} = 0.035$ at 8700 rpm, $M_u = 0.464$ (left) and stage with $\Phi_{des} = 0.025$ at 5300 rpm, $M_u = 0.282$ (right): calculation (solid line) and experiment (dots)
The industrial partner is satisfied with the results of cooperation.

**Conclusion**

Starting from the 4th version (1990s), all versions of the Universal Modeling Method execute optimal design and calculate gas-dynamic characteristics of centrifugal compressors. However, in the 4th version, it was necessary to use several sets of empirical coefficients X for stages with different design parameters. Due to the significant detailing of the mathematical model, the 5th version (2010s) correctly calculates the efficiency at the design point. To calculate the entire range of characteristics, individual sets of empirical coefficients are required. Further improvements allowed the 6-8th versions to operate with single sets of X for all stages to calculate the entire characteristics. In the 9th version presented above, the characteristics of very low flow rate stages are correctly modeled by introducing the vaneless diffuser model based on CFD calculations. To calculate high-flow rate 3D impellers, a quasi-three-dimensional calculation is introduced. The new model of exit nozzles has increased the accuracy of calculating of related stages. The Method, like other engineering techniques, cannot guarantee 100% accuracy in calculating the characteristics of any compressor. There are directions for further development. The introduction of a quasi-three-dimensional inviscid calculation into the mathematical model is very promising. This calculation will provide the most realistic information about the local flow velocities and dimensions of the flow path. It is necessary to bear in mind the opposite trend in the development of computational gas dynamics. As CFD calculations of centrifugal compressor performance become reliable, engineering techniques will play a modest role in preliminary design. The future will show for how long the further development of the Method will be relevant.

**Acknowledgments**

The research was performed by a Grant of the President of the Russian Federation for young PhD MK-1893.2020.8.

**References**

[1] Khisameev, I.G., Maximov, V.A., Batkis, G.S., Guzelbaev, Ya.Z. *Design and operation of industrial centrifugal compressors*. Kazan. P. 671. 2012 [in Russian]

[2] https://www.c-o-k.ru/market_news/mirovoy-rynok-kompressorov-2017 access 11.11.2020 [Electronic resource].

[3] Maksimov, V.A., Miftakhov, A.A., Khisameev, I.G. Compressor and refrigeration engineering at the present stage. *Bulletin of KSTU*. 1. p. 104-113. 1998 [in Russian]

[4] Safin, A.Kh. Trends in the technical and economic structure of production and development of compressor equipment. *Compressors and pneumatic*. 2. p.4-9. 2002 [in Russian]

[5] Maximov, T.V., Maximov, V.A., Egorov, A.G. Trends in the development and production of compressor technology at the present stage. *Bulletin of Kazan Technological University*. 5. p. 176-179. 2013 [in Russian]

[6] Afanasyev, B.V., Lunev, A.T., Mustafin, N.G., Potasheva, E.V. Designing a compressor impeller using an inverse problem for a rotating lattice of profiles on an axisymmetric current surface. *Compressors and pneumatic*. 1-2. p. 33-37. 1996 [in Russian]

[7] Den, G.D. *Design of the flow path of centrifugal compressors*. Publishing house "Mechanical engineering". - Moscow. p. 232. 1980 [in Russian]

[8] Lyubimov, A.N. Improvement of methods for calculating the gas-dynamic characteristics of the flow path of stationary centrifugal compressors: *PhD Thesys*. Saint Petersburg, 2016. p. 138. [in Russian]

[9] Selyanskaya, E.L., Zhenikhov, S.V., Goldobin, A.S. Compressors NPO Iskra for underground gas storage stations. *Compressors and pneumatic*. 4. p. 2-7. 2015 [in Russian]
[10] Riess, V.F. *Centrifugal compressor machines*. L.: Mechanical engineering. p. 336. 1964 [in Russian]

[11] Guidotti, G.E. Towards Centrifugal Compressor Stages Virtual Testing. Ph.D. thesis Università degli Studi di Bologna, 2014.

[12] Japikse, D. Agile design system in the age of concurrent engineering. JANNAF Conference. Albuquerque. 1996.

[13] Japikse, D. Design system development for turbomachinery (turbopump) designs - 1998 and a decade beyond. JANNAF Conference. Cleveland. Ohio. 1998.

[14] Aungier, R.H. *Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis*. ASME Press, ISBN 0-7918-0093-8, USA, 2000

[15] Brebenel, M. An analytical method for simulation of centrifugal compressors. INCAS BULLETIN, 12, Issue 1 2020, p. 35 – 49.

[16] Harley, P., Spence, S., Filsinger, D., Dietrich, M., Early, J. Assessing 1D loss models for the off-design performance prediction of automotive turbocharger compressors. Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition GT2013 June 3-7, 2013, San Antonio, Texas, USA GT2013-94262

[17] Monje, B. 1D and 3D Tools to Design Supercritical CO2 Radial Compressors: A Comparison. Proceedings of the 2nd International Seminar on ORC Power Systems, Rotterdam, 2013.

[18] Swain, E. Improving a one-dimensional centrifugal compressor performance prediction method. Part A: J. Power and Energy IMechE. 219. p. 653-659. 2005

[19] Oh, H.W., Yoon, E.S., Chung, M.K. An optimum set of loss models for performance prediction of centrifugal compressors. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 1997, 211, London, England: 331. DOI: 10.1243/0957650971537231.

[20] Galvas, M.R., “Fortran Program for Predicting Offdesign Performance of Centrifugal Compressors”. 1974 NASA Lewis Research Center, Cleveland, Ohio, USA, NASATN-D-7487.

[21] Herbert, M.V. “A Method of performance prediction for centrifugal compressors”. ARC R&M No. 3843, 1980, H.M. Stationery Office, London, England.

[22] Weber, C.R., Koronowski, M.E. “Meanline Performance Prediction of Volutes in Centrifugal Compressors”. ASME Conference Proceedings, New York, USA, 1986, 86-GT-216.

[23] Syka, T., Luňáček O. Numerical simulation of radial compressor stage. EPJ Web of Conferences 45, 01088 (2013) DOI: 10.1051/epjconf20134501088.

[24] Syka, T., Luňaček, O., Khoure J. Numerical simulation of radial compressor stages with seals and technological holes. EPJ Web of Conferences 67, 02115 (2014) DOI: 10.1051/epjconf/20146702115.

[25] De Bellis, F., Bozza, F., Bevilacqua, M., Bonamassa, G., et al. "Validation of a 1D compressor model for performance prediction". SAE Int. J. Engines 6(3):2013, doi:10.4271/2013-24-0120.

[26] Montenegro, G.A., Tamborski M., Marinoni A., Della A. Torre, A.Onorati, S.Marelli Quasi 3D approach for the modelling of an automotive turbocharger’s compressor. AIP Conference Proceedings 2191, 020113 (2019); https://doi.org/10.1063/1.5138846.

[27] Proceedings of the scientific school of compressor engineering SPbSPU. / Edited by Galerkin Yu.B. // - M.: Ed. SPbSPU - 2010. [in Russian]

[28] Galerkin, Yu.B., Danilov, K.A., Mitrofanov, V.P., Popova, E.Yu. To the use of numerical methods in the design of the flow path of centrifugal compressors. SPb. SPbSTU. p. 68. 1996 [in Russian]

[29] Galerkin, Yu.B., Danilov, K.A., Popova, E.Yu. Development of the method of universal modeling of the workflow of the central control center, software systems of the first level (third generation), experience in the development and practical use of the complex of the third level. SPb. SPbSTU. 1995. [in Russian]
[30] Galerkin, Yu.B., Danilov, K.A., Popova, E.Yu. Numerical modeling of centrifugal compressor stages (physical foundations, state of the art). Compressors and pneumatic. 2. 1993 [in Russian]

[31] Danilov, K.A. Creation of a mathematical model and software systems for optimal gas-dynamic design of refrigeration centrifugal compressors. PhD Thesis; SPbSTU. SPb., 1999 p. 176 [in Russian]

[32] Galerkin, Yu.B., Soldatova, K.V., Modeling the workflow of industrial centrifugal compressors. Scientific foundations, stages of development, current state. Monograph. Publishing House of the Polytechnic University, 2011. - p. 327. [in Russian]

[33] Miftakhov A.A., Zykov V.I. Inlet and outlet nozzles of centrifugal compressors. - "Fan". - Kazan. - 1996. [in Russian]

[34] Loytsansky, L. G. Mechanics of liquid and gas. M. Nauka., 1978. - P. 736.

[35] Galerkin, Yu.B., Rekstin, A.F. Research methods of centrifugal compressor machines. L. : Mechanical Engineering. p. 303. 1969 [in Russian]

[36] Galerkin, Yu.B., Rekstin, A.F., Soldatova, K.V., Drozdov, A.A. Highly efficient single-stage full-pressure compressor for gas compressor units (gas-dynamic design, result of model tests). Compressors and pneumatic. 8. p. 19-25. 2014 [in Russian]

[37] Borovkov, A.I., Galerkin, Yu.B., Drozdov, A.A., Rekstin, A.F., Semenovsky, V.B., Yadykin V.K. Stand ECC-55 with direct high-frequency drive for gas-dynamic research of industrial centrifugal compressors. Compressors and pneumatic. 4. p. 32-43. 2020 [in Russian]

[38] Galerkin, Yu.B., Rekstin, A.F., Drozdov, A.A., Kaminsky, R.V., Sibiryakov, S.V., Turgenev, T.I., Usenko, A.E. Experience in creating a low-pressure turbocharger for pressurizing an internal combustion engine using a modern version of the Universal Modeling Method. Compressors and pneumatic. 2. p. 2-10. 2019 [in Russian]