NEW GENERATION OF UNIVERSAL MODELING FOR CENTRIFUGAL COMPRESSORS CALCULATION

Y. GALERKIN, A. DROZDOV
R&D Laboratory “Gas dynamics of turbo machines” Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia
Email:a_drozdi@mail.ru.

Abstract: The Universal Modeling method is in constant use from mid – 1990TH. Below is presented the newest 6th version of the Method. The flow path configuration of 3D impellers is presented in details. It is possible to optimize meridian configuration including hub/shroud curvatures, axial length, leading edge position, etc. The new model of vaned diffuser includes flow non-uniformity coefficient based on CFD calculations. The loss model was built from the results of 37 experiments with compressors stages of different flow rates and loading factors. One common set of empirical coefficients in the loss model guarantees the efficiency definition within an accuracy of 0.86 % at the design point and 1.22 % along the performance curve. The model verification was made. Four multistage compressors performances with vane and vaneless diffusers were calculated. As the model verification was made, four multistage compressors performances with vane and vaneless diffusers were calculated. Two of these compressors have quite unusual flow paths. The modeling results were quite satisfactory in spite of these peculiarities. One sample of the verification calculations is presented in the text. This 6th version of the developed computer program is being already applied successfully in the design practice.

Nomenclature

- $\bar{b}$ - blade non-dimensional height
- $c$ – absolute velocity, m/s
- $c_s$ – tangential velocity, m/s
- $c'$ –velocity in view of a blade blockage factor, m/s
- $c^*$ – velocity in a blade row throat, m/s
- $D$ - diameter, m
- $D_i$ - impeller diameter, m
- $\bar{D}_h$ - hub non-dimensional diameter
- $\bar{D}_b$ - impeller inlet non-dimensional diameter
- $\bar{i}_b$ - non-dimensional length of a blade
- $\bar{l}_{mdl}$ - non-dimensional length of a blade in meridian plane
- $\bar{l}_{ax}$ - non-dimensional axial length of impeller
- $\bar{l}_{disp}$ - non-dimensional length in meridian section
- $l$ - solidity of the vane cascade
- $l_{\text{opt}}$ - Mach number
- $M_u = \frac{u_2}{\sqrt{kRT_{\text{inil}}}}$
- $m$ - mass flow rate, kg/s
- $Re$ - Reynolds number
Re_{\text{inl}} = \frac{u_{\text{inl}}D_{2}}{\mu_{\text{inl}}} \frac{P_{\text{inl}}}{RT_{\text{inl}}}

\bar{R}_h - \text{hub non-dimensional radius of curvature}
\bar{R}_s - \text{shroud non-dimensional radius of curvature}
\bar{S}_b - \text{non-dimensional blade surface area}
\bar{S}_a - \text{non-dimensional average area of hub and shroud}
\bar{P}_{\text{inl}} - \text{inlet flow rate, m}^3/\text{min}
X(i) - empirical coefficient in the math model
z - \text{number of blades}
\alpha - \text{absolute flow angle}
\alpha_v - \text{vane angle (to tangential direction), }^0
\beta - \text{relative flow angle}
\beta_{\text{bl}} - \text{impellers’ blade angle to tangential direction, }^0
\bar{\delta}_b - \text{non-dimensional blade’s thickness}
\bar{\delta}_v - \text{non-dimensional vane’s thickness}
\phi - \text{flow coefficient}
\phi_2 - \text{angle of a shroud straight part}
\varepsilon' - \text{densities’ ratio at inlets of VD and of an impeller}
\varepsilon'' - \text{densities’ ratio at outlet of VD and of an impeller inlet}
\lambda_v - \text{velocity coefficient}
\lambda_v = \frac{u_z}{\sqrt{\frac{2k}{k+1}}RT_{\text{inl}}}
\pi = \frac{P_{\text{out}}}{P_{\text{inl}}} - \text{pressure ratio}
\tau - \text{blade blockage factor}
\psi_r = \frac{h_z}{u_z^2} - \text{polytrophic head coefficient}
\psi_w - \text{work coefficient}
\psi_w = \frac{c_{\text{u2}}}{u_z} - \text{loading factor}
\chi - \text{angle of blade generatrix inclination to radial direction, }^0
\eta = \frac{\psi_r}{\psi_w} - \text{polytropic efficiency}
\Phi - \text{flow rate coefficient}

**Subscripts**
1 - impeller blade row inlet
2 - impeller outlet
3 - vaned diffuser inlet
4 - vaned diffuser outlet
av - average
bl - blade
calc - calculated
corr - corrected
cr - critical
des - design
exp - experiment
1. INTRODUCTION

The complex of computer programs for gas dynamic design and flow path optimization named Universal Modeling method is in constant use from mid – 1990TH [1, 2]. The results of application and the process of the Method improvement are presented at conferences since 1995 [3]. Math model is the set of algebraic equations for calculation of head loss and mechanical work transmitted to gas. By means of 4th version math model several dozens of successful gas dynamic designs were created and realized by compressor manufacturers of Russia, Ukraine and Poland. The disadvantage of the model was that to calculate a stage performance with good accuracy an individual set of empirical coefficients must be applied. Different sets for stages with different design flow rate and loading factors were necessary. The problem was overcome in 5th version of the math model [4, 5]. The presented 6th version extends 5th version advantages on stages with 3D impellers and vaned diffusers (VD). Below is presented the main features of the 6th version of the Method.

2. 3D IMPELLERS CALCULATION

The new, 6th version expands the 5th version model achievements [4, 6, 7, 5, 8] on 3D impellers and vaned diffusers. Calculation algorithm used in the 6th version suggests more detailed calculation of head losses in 3D impellers. An important part is the more accurate definition of the hub/shroud surfaces and blades’ surface area. Blades surface area is of the same order as hub/shroud surfaces’ area – unlike 2D impellers. It is important to calculate blades’ surface area precisely.

The previous, 4th version treated the problem in the very simplified way [9]. As it is shown at Figure 1 (left) only three geometry parameters specified a meridian shape: non-dimensional impeller inlet diameter $D_0$, non-dimensional hub diameter $D_h$, outlet height of blades $b_2$. Not universal empirical formulas were applied to surface area calculations. Hub and shroud non-dimensional area in 4th model:

$$ S_{hs} = \frac{\pi}{4} D_2^2 \left( 1 - \frac{D_h}{D_0} \right) \left( \tau_1 + \tau_2 \right) / 8 $$

Blades surface area is defined as the product of blade length $l_{bd}$ and average blades height of an impeller:

$$ S_{bd} = 0.5 l_{bd} \left( b_1 + b_2 \right) $$

The empirical formula for a blade length:

$$ l_{bd} = \frac{1.15 \pi}{4} \frac{\left( 1 + 0.5D_0 - 1.5D_h \right)}{\sin \beta_{bd} + \sin \beta_2} $$
This approach does not allow estimate accurately friction losses on all surfaces of an impeller. It also does not give an opportunity to optimize geometric dimensions of the 3D impeller by candidates’ comparison.

Figure 1 (right) demonstrates the set of geometry parameters that completely describe an impeller meridian shape in 6th version. This set is identical to one used for Q3D non-viscid calculations in the design process [1]. These parameters are: $D_0$, $D_h$, $b_2$, a non-dimensional axial length of the impeller $l_m$, hub and shroud curvature radius $R_h$ and $R_s$, the angle of a shroud straight part $\phi_2$. Position of the blade leading edge is determined too by the parameter $l_{inl}/l_{imp}$. The inlet blade height $b_1$ and diameter $D_0$ are calculated from the set of geometric quantities and is not user-defined.

The surface area of a blade in 6th version is:

$$S_{bl} = \sum_{0}^{\Delta l_{mi}} \frac{\Delta l_{ni}\cdot b_1}{\sin \beta_{bl-i} \cdot \cos \chi}$$  \hspace{1cm} (4)

Figure 2. Schematization of blade angle along blade meridian length

To calculate blade area, it is necessary to know the function $\beta_{bl-i} = f(l_{mi})$. The analysis of blade angle variation with the length for several versions of 3D impellers is made, and as a result the schematic linear approximation is proposed. The 3D impellers designed in accordance with [1], in most cases,
have firstly an increasing blade angle, a middle part with practically constant angles and descending angles near the outlet:
- the initial portion \( l_{\text{ini}} / l_m = 0 - 0.45 \), assuming a linear angle increase from \( \beta_{\text{bl1}} \) to \( \beta_{\text{blmax}} \);
- middle portion from \( l_{\text{me}} / l_m = 0.45 - 0.75 \), assuming constant blade angle \( \beta_{\text{blmax}} = 1.35 \beta_{\text{bl2}} \);
- outlet portion from \( l_{\text{out}} / l_m = 0.75 - 1.0 \), assuming a linear angle decrease from \( \beta_{\text{blmax}} \) to \( \beta_{\text{bl2}} \).

The graphical view of such schematization is shown in Figure 2.

To calculate the blade surface area an impeller meridian length is divided into 40 sections. Figure 3 demonstrates position of a blade surface generatrix on a conical surface a-a.

![Figure 3](image)

**Figure 3.** Blade meridian cross section (left) and blade generatrix position on a conical surface “a-a”

The angle \( \chi \) of an inclination of a generatrix to a meridian plane on a surface “a-a” is not the independent design parameter. The angle varies along blade length and its value at a leading edge is equal 45 - 55° sometimes. Big \( \chi \) increases blade surface area and a blade blockage factor. It is taken into account in a simplified way - the angle average value is assuming to be equal to 20° for all 3D impellers.

In accordance with loss model the friction force is proportional to average area of hub and shroud surfaces \( S_0 = 0.5 \left( \bar{S}_h + \overline{S}_{\text{me}} \right) \). The equation for calculations is:

\[
S_0 = \frac{S_0}{\pi D^2} = \frac{T_{\text{bl1}}}{4} \left( \frac{2 \left( 1 + \overline{D}_{\text{av}} \right)}{\pi \sin \beta_{\text{blav}}} \right) - z \frac{4 \overline{S}_h}{\pi} \tag{5}
\]

CFD calculations of stages with different 3D impellers have demonstrated that correct calculation of surface area is important. For instance, 3D impellers with slightly excessive number of blades are much less effective than could be expected.

**3. VANED DIFFUSER LOSS MODEL**

The sixth version uses a new algorithm for calculation the parameters in vaned diffusers. The vaneless part of greater or lesser extent is located before vaned diffuser. This is done to reduce the negative impact of uneven flow after impeller for vaned diffuser. Mechanical considerations are obvious. In this element, the approach presented in [4, 5, 6, 7, 8] is applied with some modifications in case of 3D impeller. Some equations of the mathematical modeling for the vane part of the diffuser are presented below.

Flow coefficient at vanes’ inlet is:

\[
\phi_i' = \frac{\Phi}{4 \varepsilon_i' \overline{D}_i \left[ \bar{b}_i - X(i) \right] \tau_i} \tag{6}
\]
The empirical coefficient \( \chi(i) \) takes into account a boundary layer blockage factor. The blade blockage factor is a geometry parameter:

\[
\tau_3 = 1 - K_{\tau_3} \frac{z_{VD} \delta_{VD}}{\pi D_3 \sin \alpha_v} \tag{7}
\]

The coefficient \( K_{\tau_3} \) is different for vanes of different shape – airfoil type or other.

The non-incidence inlet condition and incidence losses are determined with taking into account vanes’ load. The crucial factor is a direction of a streamline that goes to a critical point on a blade surface. Near a vane cascade a critical streamline deviates to a suction side of vanes where pressure is less. The scheme of this streamline behavior is presented in Figure 4.

![Figure 4. Velocity triangles at VD inlet](image)

The critical streamline increases its velocity tangential component on \( \bar{u}_{\alpha_3} \) under an influence of vanes’ load. Non incidence condition is:

\[
\alpha_v = \alpha_{3\text{cr}} \tag{8}
\]

The angle of a critical streamline is defined by the formula:

\[
\alpha_{3\text{cr}} = \arctg \frac{\phi_3'}{\bar{c}_v + \Delta \bar{c}_v} \tag{9}
\]

To calculate a critical streamline deviation the scheme from [1] is applied:

\[
\Delta \bar{c}_v = 2 \frac{(\bar{c}_v \bar{D}_3 - \bar{c}_3 \bar{D}_4)}{z_{VD}(\bar{D}_4 - \bar{D}_3)} \tag{10}
\]

If a critical streamline direction does not correspond to vane inlet angle then incidence losses take place. The analogy with sudden expansion losses is used. The losses are proportional to difference of vectors \( \bar{c}_{3\text{inc}} \) and \( \bar{c}_v \) (velocity of inlet flow turned to direction of vanes) - Figure 4:

\[
\Delta \bar{c}_v = \bar{c}_v + \Delta \bar{c}_v - \frac{\phi_3'}{\tan \alpha_{3\text{cr}}} \tag{11}
\]
Formula for calculating efficiency losses $\Delta \eta_{incVD}$ due to incidence inlet is similar to the corresponding formula in 4th version [10]:

$$\Delta \eta_{incVD} = X(i)(1 + X(i)(\lambda_{inc}c_i^2))^{X(i)} \frac{\Delta c_{inc}^2}{2\nu T}$$  \hspace{1cm} (12)

The exit angle $\alpha'_4$ of the flow differs from the blade angle $\alpha_{v4}$ on a deviation angle $\Delta \alpha_4$:

$$\alpha'_4 = \alpha_{v4} - \Delta \alpha_4$$  \hspace{1cm} (13)

Deviation angle $\Delta \alpha_4$, according to [4, 11] is defined by the formula:

$$\Delta \alpha_4 = \frac{\alpha_{v4} - \alpha_{v3}}{\frac{\sqrt{1/t_{av}}}{0.346} + 1}$$  \hspace{1cm} (14)

The optimum solidity of the vane cascade according to [12] is defined by the formula:

$$\left( \frac{l}{t_{av}} \right)_{opt} = \frac{\log \frac{D_s}{D_t}}{2.73 \sin \left( \frac{\alpha_{v3} + \alpha_{v4}}{2} \right)}$$  \hspace{1cm} (15)

Measurements of flow structure at diffuser exit demonstrated sufficient non-uniformity. Kinetic energy of flow is bigger than by 1D calculation (1D representation corresponds to uniform flow). Kinetic energy influences head loss in the following element – return channel or scroll. Calculations by ANSYS CFX programs of a performance of the stator of the medium specific speed centrifugal stage have presented information on flow structure [9, 13]. The values of $c_{43D}$ were averaged by energy $\bar{m} \cdot 0.5c_{43D}^2 = \sum \bar{m} \cdot 0.5c_{43D}^2$ at nine flow rates, i.e., at nine incidence angles $i_3 = \alpha_{v3} - \alpha_3$. The same element was calculated by 6th version computer program and $c_{41D}$ values at the same conditions were defined. The ratio of the calculated by CFD velocity $c_{43D}$ to velocity by 1D mode $c_{41D}$ is presented in Figure 5 – black square. Coefficient $K_{nu}$ takes into account flow non-uniformity at a vane cascade outlet:

$$K_{nu} = \frac{c_{43D}}{c_{41D}}$$  \hspace{1cm} (16)

Minimum of $K_{nu}$ corresponds to zero incidence angle $i_3$.

The corrected value of non-dimensional velocity $\bar{c}_{4corr}$ is used in formula for head loss calculation in the element after vaned diffuser:

$$\bar{c}_{4corr} = \left( \varphi_4 / \sin \alpha_4 \right) \cdot K_{nu}$$  \hspace{1cm} (17)

Presented in Figure 5 calculations were approximated by the formula:
The Eq. (18) approximates results of the numerical experiment data satisfactory in the practically important part of the performance.

\[ K_{nn} = 1.16 + 20 \sin^3 \left( \frac{i_3}{3} \right) \]  

(18)

![Graph showing non-uniformity coefficient versus the incidence angle](image)

**Figure 5.** Non-uniformity coefficient versus the incidence angle \( i_3 \)

Black square – CFD/6\(^{th}\) version calculations, solid line – simulation by eq. (18)

### 4. MATHEMATICAL MODELLING VERIFICATION

Test results of 38 model stages were used for 6\(^{th}\) version loss model identification (9 basic stages and their variants). The range of the main parameters of the model stages: \( \Phi_{des} = 0.028 – 0.080 \), \( \psi_{T,des} = 0.52 – 0.65 \), \( \bar{D}_d = 0.25 – 0.373 \), \( \bar{D}_t = 1.428 – 1.615 \), \( M_s = 0.60 – 0.86 \), \( \text{Re}_{u} = 4.8 \times 10^6 – 6.9 \times 10^6 \). All stages consist of an impeller, vaneless or vaned diffuser and return channel. The basic stages were tested with varied hub ratio, vaneless diffusers (VLD) width, blade trailing edge configuration. Each test consisted of six different flow rate regimes. Thus, the identification process was made on the base of 228 empirical efficiency values.

The identification process consists of search of empirical coefficients values that satisfy the condition:

\[ d \eta_{av} = \frac{\sum Z | \eta_{exp} - \eta_{calc} | }{Z} \rightarrow 0 \]  

(19)

After identifying the average error in calculating the maximum efficiency for all stages with vaned and vaneless diffusers which is equal to 0.007. Average calculation error for six points on efficiency performance curves is 0.028. Average calculation error for five points (except the highest flow rate regime) is 0.013. Several model stages test performance curves and results of their simulation by the 6\(^{th}\) version model are presented at Figure 6.

Verification of a mathematical model was made by comparison of calculated efficiency performance curves with measured ones for several industrial compressors designed by the Universal modeling method and by other designer – design procedure is unknown. Test data are presented by manufacturers as results of official plant tests. There is one sample below.

The flow path scheme drawings of two pipeline booster compressors 16 MW with a delivery pressure of 7.45 MPa and pressure ratio \( \pi = 1.7 \). The designs of both compressors were made in due time by 4\(^{th}\) version of the Method. Two-stage compressor flow path was places in the body of the standard pipeline compressor with \( \pi = 1.44 \). The elevated pressure ratio was achieved by increasing of impeller diameters and loading factor \( \psi_{T,des} = 0.82 \). Low flow rate coefficient and big loading factor limited the efficiency by level of 80% - that is good for given non-dimensional parameters – Figure 7.
The four-stage compressor with the same dimensional parameters has optimal non-dimensional parameters. The loading factor at the design flow rate is slightly less than 0.5. The optimal design flow rate coefficient of stages with 2D impellers lies in range 0.050 – 0.080. Kinetic energy and surface area corresponds to minimum loss of a head in this case. Design flow rate coefficients of the compressor stages lie in this range. According to the official test maximum efficiency is 87.6%. Calculated and measured performance curves are compared at Figure 8.

Simulation results are good in main part of performances. Maximum flow rate corresponds to a compressor operation with low pressure ratio. Compressors do not operate at these regimes in practice. Poor simulation result at the maximum flow rate has no practical significance.
The analogy results were obtained for three more centrifugal compressors with significantly different parameters.

CONCLUSION

The new 6th version of the math model demonstrated ability to predict centrifugal stages and compressors gas dynamic performances with good accuracy. Precise calculations of stages and compressors with quite different design flow rate coefficients and loading factors are made by one common set of the empirical coefficients. Elevated power of modern PC opens way to further improvements of math modeling. One of evident ways is Q3D approach for modeling of 3D impellers that is inside visible future of the authors.

ACKNOWLEDGMENT

Work is performed with support of a Grant of the President of the Russian Federation for young PhD MK-7066.2015.8.

REFERENCE

[1] Galerkin Y. Turbo compressors. [text] / Galerkin Y. // LTD information and publishing center. - Moscow. – 2010 [Russian].
[2] Galerkin, Y. B., Soldatova, K.V. Operational process modeling of industrial centrifugal compressors. Scientific bases, development stages, current state. Monograph. [text] / Galerkin Y., Soldatova K. // Sankt-Peterburg. - SPbTU. – 2011. [Russian].
[3] Galerkin Y.B., Popova E.Y., Danilov K.A., Mitrofanov V.P. Quasi-3d Calculations in Centrifugal Impeller Design. VDI Berichte. № 1425. Hannover. – 1998.
[4] Galerkin, Y. B., Soldatova, K.V. Operational process modeling of industrial centrifugal compressors. Scientific bases, development stages, current state. Monograph. [text] / Galerkin Y., Soldatova K. // Sankt-Peterburg. - SPbTU. – 2011. [Russian].
[5] Galerkin, Y. B., Soldatova, K.V. The application of the Universal Modeling Method to development of centrifugal compressor model stages. [text] / Galerkin Y., Soldatova K. // International Conference “Compressors and their Systems”.– London. – 2013. – P. – 477-487.
[6] Galerkin Y., Drozdov A., Gas dynamic performances simulation of centrifugal compressor stages with 3D impeller. [text] / Galerkin Y., Rekstion A. // Scientific and technical transactions of the TU SPb. – 2014. – No. 3. – Page – 45-53. [Russian].
[7] Y. Galerkin, A. Drozdov, K.Soldatova Centrifugal compressor efficiency types and rational application. [text] / Galerkin Y., Drozdov A., Soldatova K. // International Conference on Compressors and their systems. – London: City University, UK. – 2013.
[8] Galerkin, Y. B., Soldatova K.V., Drozdov A.A. New version of the Universal modeling for centrifugal compressor gas dynamic design. [text] / Galerkin Y., Drozdov A., Soldatova K. // Purdue Conference 2014. USA.
[9] Galerkin, Y. B., Marenina L.A. Research and improvement of centrifugal stages fixed elements by methods of computational dynamics. Part 1. / Galerkin Y., Marenina L. // Compressors and pneumatics, 2014. – №1. – P. 30-36. [Russian].
[10] Galerkin Y. Danilov K., Popova E. Design philosophy for industrial centrifugal compressor. [text] / Galerkin Y., Danilov K., Popova E. // International Conference on Compressors and their systems. – London: City University – 1999.
[11] Den G.N. Mechanic of a stream in centrifugal compressors. [text] / Den G. // – Leningrad. – Mechanical engineering. – 1973. [Russian].
[12] Ris V. Centrifugal compressors. [text] / Ris V. // - Leningrad. – 1981. [Russian].
[13] Galerkin, Y. B., Marenina L.A. Research and improvement of centrifugal stages fixed elements by methods of computational dynamics. Part 2. / Galerkin Y., Marenina L. // Compressors and pneumatics, 2014. – №2. – P. 10-15. [Russian].