Design and Flow Parameters Calculation of the Turbomachine Channels

Mykola Kalinkevych\textsuperscript{a}, Oleksandr Gusak\textsuperscript{b}, Andriy Skoryk\textsuperscript{c}, Oleg Shcherbakov\textsuperscript{d}, a\textsuperscript{*}

\textsuperscript{a,c,d}Sumy State University, Department of technical thermal physics, 2 Rimsky-Korsakov Str., Sumy 40007, Ukraine
\textsuperscript{b}Sumy State University, Department of applied hydro- and aeromechanics, 2 Rimsky-Korsakov Str., Sumy 40007, Ukraine

Abstract

The flow calculation method and design procedure of axisymmetric channels of turbomachines are presented in the paper. On the basis of the presented procedure, a special computer program has been developed. To confirm the adequacy of the technique, the results of the calculation were compared with the results of the numerical experiment.

© 2011 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Sumy State University

Keywords: Turbomachine; axisymmetric channel; flow parameters; channel design; CFD.

1. Introduction

The general elements of turbomachines are the channels of different type. Curvilinear axisymmetric channels have to be attended, in which the flow character strongly affect the compressor efficiency. Curvilinear axisymmetric channels are the part of input and output devices, return bend and return channel of turbomachine.

The channel curvature causes the strong flow nonuniformity in cross-section, which causes the initiation of separation zones and loss increasing. That’s why it is important to evaluate the flow character to create the shape of curvilinear axisymmetric channel which provides the low level of loss.

\textsuperscript{*} Corresponding author. Tel.: +38-098-439-5054.
\textit{E-mail address:} vikola58@ukr.net.
In the conceptual stage it is sensible to use programs, in which the simple mathematical flow models based on the firm physical foundation for the flow character are used. "Light" programs, as compared with “hard” programs like ANSYS CFX and FlowVision, allow to design channels faster and cheaper. It is convenient for channel design to apply the analytical form to define the channel shape. The efficiency of such approach confirms in the works of Puzyreowski and Flaszynski [1] and Stanitz [2].

2. Radial-axial axisymmetric channel flow calculation method

Developed calculation method is based on following assumptions: gas flow is compressible, stationary and adiabatic. Inlet flow parameters distribution is uniform. According to the accepted physical flow model, the flow core is nonviscous and the viscosity was taken into account for the boundary layer calculation. The curvilinear coordinate system, which is shown in Figure 1, consists of meridional projections of streamlines s and orthogonals to them n; \( c_m \) is the meridional projection of velocity; \( R_m \) is the radius of curvature of streamline in the meridional plane; \( c_m^2/R_m \) and \( c_u^2/r \) are the centrifugal forces applied to the unit mass of gas.

![Fig. 1. Scheme of the gas flow in axisymmetric channel](image)

The differential equations of motion for the particle of nonviscous gas are:

\[
\frac{1}{\rho} \frac{\partial p}{\partial s} + c_m \frac{\partial c_m}{\partial s} + \frac{c_u}{r} \sin \theta = 0, \quad (1)
\]

\[
-\frac{1}{\rho} \frac{\partial p}{\partial n} + \frac{c_u^2}{r} \cos \theta - \frac{c_m^2}{R_m} = 0. \quad (2)
\]

From the equations (1) and (2) with allowance for the Bernoulli equation for noncompressible gas:

\[
\frac{\partial c_m}{\partial n} - \frac{c_m}{R_m} = 0 \text{ and } \frac{\partial c_m}{c_m} - \frac{\partial n}{R_m} = 0. \quad (3)
\]
After the integration of the equation (3):

\[ c_m = c_{m0} \cdot e^{\frac{\int \frac{dn}{R_m}}{R_m}} = e^{A} \cdot c_{m0}, \]  \hspace{1cm} (4)

where \( c_{m0} \) is the velocity at the inner surface of the channel, which is defined from the continuity equation.

If the values of the integral \( A = \int \frac{dn}{R_m} \) are defined, the relation (4) allows to define the velocity distribution along the normal.

If the distribution of the streamline curvature in cross-section along the normal is linear, the integral becomes

\[ A_j = \frac{n_j}{R_{in}} + \frac{n_j^2}{2b} \left( \frac{1}{R_{in}} - \frac{1}{R_{out}} \right), \]  \hspace{1cm} (5)

where \( R_{in} \) and \( R_{out} \) are the radiiuses of curvature for the generatrix of the inner and outer surfaces, respectively; \( b \) is the distance from the inner surface to the outer one along the normal; \( n_j \) - the distance from the inner surface to the point where velocity \( c_{m0} \) is being estimated.

The mass flow is estimated from the average flow velocity at the inlet of the channel. Each orthogonal is divided into a certain number of equal parts. For each part the average velocity is estimated such that the total mass flow will be equal to the given mass flow for the channel.

It should be noted that the total pressure and the total temperature are assumed as a constant along the channel and equal to the value at the channel inlet.

3. Radial-axial channel meridional contour design technique

The channel meridional contour design includes next steps:
- the center line construction;
- the center orthogonal coordinates definition according to the given law of cross section area variation along the channel;
- the inner and outer surface generatrix construction;
- construction of the orthogonals to the center streamline and definition of the radiuses of curvature at the cross points of inner and outer surfaces and orthogonals.

The source data for channel design is geometrical characteristics shown in Figure 2 (a).
The channel length can be specified in range of \( 0.15 \leq L/D_0 \leq 0.35 \).

It is recommended to turn the flow in the meridional plane with maximal radius of curvature. This condition is satisfied by circular arc [3]. The center line is the combination of circular arc 04 and linear segment 42 (Figure 2 (b)).

The equation of straight line which is crossed the point 0 at an angle of \( \gamma_{0cl} \) to the axis \( z \):

\[ r - r_0 = (z - z_0) \cdot \tan \gamma_{0cl}. \]  

(6)

The equation of straight line which is crossed the point 2 at an angle of \( \gamma_{2cl} \) to the axis \( z \):

\[ r - r_2 = (z - z_2) \cdot \tan \gamma_{2cl}. \]  

(7)

The cross point of those straight lines (point 3) is defined from the solution of the set of equations (6) and (7):

\[
z_3 = \frac{r_2 - r_0 + z_0 \tan \gamma_{0cl} - z_2 \tan \gamma_{2cl}}{\tan \gamma_{0cl} - \tan \gamma_{2cl}}, \quad r_3 = \frac{(z_2 - z_0) \tan \gamma_{0cl} + r_0 \tan \gamma_{2cl} - r_2 \tan \gamma_{0cl}}{\tan \gamma_{2cl} - \tan \gamma_{0cl}}. \]  

(8)

Coordinates of point 4 are estimated from the equality of straight lines 03 and 34. Coordinates of the center of the circle (point 5) are estimated by the cross point of straight lines which are perpendicular to the straight lines 03 and 34.

The radius of curvature of the circular arc with a point 5 as a center

\[ R = \sqrt{(z_5 - z_4)^2 - (r_5 - r_4)^2}. \]  

(9)

The coordinates of the center orthogonal are estimated according to the given law of cross section area variation along the channel. Thereafter coordinates of the cross points of inner and outer surfaces and center orthogonal are used to estimate unknown coefficients of those generatrixes.
The law of cross section area variation along the radial–axial channel is given by the quadratic dependence

\[ F_i = f_0 + f_1 l_i + f_2 l_i^2. \]  

(10)

where \( l_i \) is the center line length current value; \( f_0, f_1, f_2 \) are channel area coefficients. Coefficients \( f_0, f_1 \) are estimated from the boundary conditions. The \( f_2 \) value is given by the initial data. The inner and outer surface generatrixes are given by the second-order curve:

\[ f = r^2 + A \cdot z^2 + 2B \cdot z \cdot r + 2C \cdot r + 2D \cdot z + E = 0. \]  

(11)

To define unknown coefficients A,B,C,D,E the set of five equations was compiled. The sets of equations for inner and outer surfaces are resolved by Kramer’s method [4].

The equation of the orthogonal to the center line defined implicitly

\[ \left( \frac{\partial f_c}{\partial r_{c_{li}}} \right)_i (z - z_{c_{li}}) - \left( \frac{\partial f_c}{\partial z_{c_{li}}} \right)_i (r - r_{c_{li}}) = 0. \]  

(12)

The quadratic equation with respect to \( r \) is estimated by the substitution of derivative \( \frac{\partial f_{c_{li}}}{\partial r_{c_{li}}} \) and \( \frac{\partial f_{c_{li}}}{\partial z_{c_{li}}} \) into equation (12):

\[ r^2 \cdot p_i + 2 \cdot q_i \cdot r + s_i = 0, \]  

(13)

where constants \( p, q, s \) are functions of A,B,C,D,E.

Cross points of orthogonals and inner and outer surface generatrixes are obtained in kind. The radius of curvature at the arbitrary point of inner and outer surface generatrix:

\[ R_s = \left[ \left( \frac{\partial f}{\partial z} \right)^2 + \left( \frac{\partial f}{\partial r} \right)^2 \right]^{1/2} \left\{ \frac{\partial^2 f}{\partial z^2} \left( \frac{\partial f}{\partial r} \right)^2 - 2 \frac{\partial^2 f}{\partial z \partial r} \cdot \frac{\partial f}{\partial z} \cdot \frac{\partial f}{\partial r} + \frac{\partial^2 f}{\partial r^2} \left( \frac{\partial f}{\partial z} \right)^2 \right\}. \]  

(14)

4. Boundary layer influence accounting

Boundary layer thickness is calculated in the first approximation by the flat plate formula [5]

\[ \frac{\delta}{l} = 0.37 \cdot \left( \frac{u_{c_{li}}}{V} \right)^{1/5}, \]  

(15)

where: \( \delta \) - boundary layer thickness at length \( l \), which is counted from the channel inlet along the inner and outer surfaces; \( V \) - kinematic viscosity coefficient; \( u_{c_{li}} \) - flow velocity at the boundary layer limit along the generatrix.
The power law distribution of the boundary layer velocity is assumed. For $Re \approx 10^5$ the exponent is $n = 1/7$. If one takes into account that the density change is negligible, displacement thickness can be obtained:

$$\delta = \int_0^\delta \left(1 - \frac{u}{u_\infty}\right) dy = \int_0^\delta \left(1 - \left(\frac{y}{\delta}\right)^{1/7}\right) dy = \delta \left(\frac{y}{\delta}\right)^{1/7} \cdot \frac{7}{8} \delta^{7/8} = \frac{1}{8} \delta. \quad (16)$$

To define the real velocities value, the calculation is implemented for the restricted channel by the corresponding displacement thicknesses.

The inner and outer surface generatrixes are obtained using displacement thicknesses for the center and outlet cross-sections.

5. “R-A Channel” software for radial-axial channel design

According to the presented method the “R-A Channel” software was created. It allows to design and calculate the flow of the radial-axial channels automatically.

The inlet and outlet geometrics, channel length, thermophysical parameters and velocity at the channel inlet are the initial data. Also the law of cross section area variation along the channel is given (the value of coefficient $f_2$ is given).

The software provides the channel meridional contour design and construction of orthogonals to the center line automatically. The results of velocity and pressure calculation are presented graphically by the superficial meridional velocity distribution $\lambda = \frac{c_m}{a_{cr}}$ ($a_{cr}$ is critical velocity) and superficial pressure distribution $\pi(\lambda) = \frac{p}{p^*}$ ($p^*$ is stagnation pressure) along the channel (Figure 3).

The superficial meridional velocity distribution graphics along the inner and outer surface generatrixes are presented in Figure 3 by red and brown colour respectively.

The superficial meridional velocity distribution and superficial pressure distribution along the inner and outer surface generatrixes are used for the calculating of form-parameter value, which allows to define the boundary layer separation will be or not.
As is known, it is recommended to design turbomachine channels with the minimal axial dimensions. The developed software allows to design the channel relatively fast with the minimal axial dimensions for the gas flow without separation. It is obvious that loss will be minimal in that case, which favour the increasing of turbomachine efficiency.

6. Radial-axial channel flow analysis by means of ANSYS CFX software

For the calculation of the flow, the solid model of the radial-axial channel has been designed. Because the flow is axisymmetric, not the whole area of the model, but a particular sector with the angle between the planes of the flat sides equal 50° was calculated to be. For the flat sides an interface boundary condition was given.

Geometrics of the model (according to the Figure 2) are $D_0=300$ mm, $b_0=20$ mm, $D_2=200$ mm, $d_2=100$ mm, $\gamma_{0i}=90,1^\circ$, $\gamma_{2i}=178^\circ$, $\gamma_{0o}=90,1^\circ$, $\gamma_{2o}=178^\circ$, $L=98$ mm, $f_2=0$, generatrices of the meridional contour of the channel were given as the second-order curves.

Inlet flow parameters were specified according to the initial data given for the calculation by means of “R-A Channel”. These parameters are $c_{m0}=60$ m/s, $p_0=0,101$ MPa, $T_0 = 273$ K.

The SST turbulence model and Total Energy heat transfer model were used for the calculation. Air, ideal gas was specified as working fluid. Static pressure, static temperature and flow velocity were specified at the inlet boundary and the mass flow rate was specified as the outlet condition.

For display purposes the results of the calculation are presented as velocity and pressure fields (Figure 4). Velocity distribution and pressure distribution in a channel correspond to theoretical conception of flow character in axisymmetric curvilinear channels. As you can see, in the flow core velocity increases...
from the inner surface to the outer one. Also there is a velocity peak at the initial part of the outer surface. It is caused by significant curvature in this area.

The meridional velocity distributions along the inner and outer surfaces are shown in Figure 5. This diagrams show the distribution of the flow velocity at the boundary layer limit.

Comparison analysis of the velocities calculated by “R-A Channel” and by ANSYS CFX v.11 shows good agreement, with the exception of the short section at the inlet, where discrepancy is about 10%. It occurs because uniform velocity distribution at the inlet was specified for the calculation by “R-A Channel”. While using ANSYS CFX an influence of the streamline curvature on the inlet flow parameters was taken into account (i.e. the flow was calculated as for nonuniform velocity and pressure distribution at the inlet). For the rest of the channel (\(L_{rel}=0,1\ldots1\)) discrepancy is insignificant.

Pressure distributions along the inner and outer surfaces are shown in Figure 6. Discrepancy does not exceed 2%.

Fig. 5. Meridional velocity distribution along the inner and outer surfaces
7. Radial-axial channel flow analysis by means of FlowVision software

The calculation was provided using the SST turbulence model and totally compressible liquid model of fluid. The boundary conditions are total pressure and static temperature at the inlet and average velocity and average pressure at the outlet of the channel.

For the clearness the results of the calculation are shown as a velocity field and pressure field (Figure 7).

The pressure values in Figure 7 are excessive values of pressure relatively to the reference value 101000 Pa.

It should be noted that the results of calculation by FlowVision software considerably depend on boundary conditions. The conducted numerical experiment showed the best correspondence of the calculation results and theoretical conception of the flow character if the boundary conditions were average velocity and average pressure at the outlet of radial-axial channel.

The meridional velocity distributions along the inner and outer surfaces, which are estimated by “R-A Channel” and FlowVision v.2.05, are shown in Figure 8, the pressure distributions are shown in Figure 9.
As shown in Figure 8, there is a good comparison between velocity distributions calculated by FlowVision and “R-A Channel” software. The quantitative comparison shows acceptable accuracy. Maximal discrepancy is not exceeding 14%. The pressure distributions have a high agreement (Figure 9). Maximal discrepancy is not exceeding 1%.

The radial-axial channel calculation by ANSYS CFX and FlowVision software shows a good qualitative and quantitative agreement with the developed “R-A Channel” calculation.

8. Conclusion

The method of flow calculation and design method of radial-axial channels of turbomachines are presented in the paper. The method was used in the computer program “R-A Channel”. The comparison between the results of “R-A Channel” and ANSYS CFX and FlowVision calculation was conducted. The comparison shows a good agreement, so the developed “R-A Channel” software can be used for the radial-axial channel design.

References

