CFD study of leakage flows in shroud cavities of a compressor impeller

To cite this article: K Soldatova 2017 IOP Conf. Ser.: Mater. Sci. Eng. 232 012045

Related content

- Multi-objective optimization for impeller shroud contour, the width of vane diffuser and the number of blades of the centrifugal compressor stage based on the CFD calculation
  A M Danilishin, Y V Kozhukhov and V K Yun

- Optimization of a centrifugal compressor impeller using CFD: the choice of simulation model parameters
  V V Neverov, Y V Kozhukhov, A M Yablokov et al.

- Investigation on the electromagnetic centring technique in compressor with labyrinth seal structure
  W Zhang, C Feng, J Cheng et al.
CFD study of leakage flows in shroud cavities of a compressor impeller

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

Email: buck02@list.ru

Abstract: The flow character in a gap between shroud disc of an impeller and a stator surface (shroud cavity) influences disc friction loss, labyrinth seal loss (parasitic losses) and thrust force. Flow calculations inside the shroud cavity of a model of centrifugal compressor stage and its labyrinth seal in a range of flow rates and axial width and radial gap are presented. The results are presented in terms of non-dimensional coefficients of flow, disc friction and seal leakage losses coefficients and pressure coefficient. The distributions meridional and tangential flow velocities correspond to the continuity and equilibrium equations – flow radial circulation exists in wide cavity and is absent in narrow cavities. The radial pressure distributions as measured and calculated are not fully comparable. The possible reason is that CFD-calculated leakage coefficient is less than calculated by A.Stodola formula. The influence of a cavity width on the losses and the thrust force requires a balanced design.

Nomenclature

\[ \begin{align*}
    b & \quad \text{width of channel} \\
    \bar{r} & \quad \text{relative cavity width} \\
    c_m & \quad \text{meridional velocity} \\
    c_r & \quad \text{tangential component of absolute flow velocity} \\
    D & \quad \text{diameter} \\
    k & \quad \text{isentropic coefficient} \\
    k - \varepsilon & \quad \text{turbulence model} \\
    \bar{m} & \quad \text{mass flow rate} \\
    \bar{m}_{\text{leak}} & \quad \text{labyrinth seal leakage mass flow rate} \\
    M_a & \quad \text{blade Mach number} \\
    P_f & \quad \text{friction force moment} \\
    P_r & \quad \text{disk friction power} \\
    P_r & \quad \text{power transmitted to gas by blades} \\
    p & \quad \text{pressure} \\
    r & \quad \text{radius} \\
    Re & \quad \text{impeller diameter Reynolds number} \\
    \bar{R} & \quad \text{axial force} \\
    R_{zu} & \quad \text{force in a shroud cavity} \\
    \bar{R}_{zu} & \quad \text{non-dimensional mode by axial force coefficient} \\
    \bar{R}_f & \quad \text{friction force moment coefficient} \\
    \bar{R}_f & \quad \text{temperature} \\
    \bar{R}_u & \quad \text{peripheral velocity} \\
    \bar{R}_w & \quad \text{relative velocity} \\
    \bar{R}_f & \quad \text{flow circulation} \\
    \bar{R}_f & \quad \text{flow rate coefficient} \\
    \bar{R}_w & \quad \text{non-dimensional coefficient of disk friction} \\
    \bar{R}_{leak} & \quad \text{non-dimensional leakage coefficient} \\
    \bar{R}_{rot} & \quad \text{rotor/stator gaps} \\
    \bar{R}_{r} & \quad \text{relative radial gaps} \\
    \bar{R} & \quad \text{friction coefficient} \\
    \bar{R}_e & \quad \text{empirical coefficient for the seal} \\
    \bar{R}_\mu & \quad \text{density} \\
    \bar{R}_\tau & \quad \text{shear stress} \\
    \bar{R}_{\psi_r} & \quad \text{loading factor} \\
    \bar{R}_{\Delta p} & \quad \text{pressure differential} \\
\end{align*} \]

Subscripts

\[ \begin{align*}
    0 & \quad \text{impeller inlet} \\
    1 & \quad \text{impeller blade row inlet} \\
    2 & \quad \text{impeller outlet} \\
    \text{av} & \quad \text{average} \\
\end{align*} \]
1. **Object and goals**

The object of this CFD analysis is a shroud cavity, i.e. an axial gap between a shroud disc and a body of a centrifugal stage. The model stage cross section and typical configuration of a shroud cavity are presented in Figure 1.

![Figure 1. Compressor stage meridional cross-section with the shroud cavity and labyrinth seal [1-2](image)](image)

Cavity flow results in so-called parasitic or winding losses. Static pressure difference in shroud and hub cavities influences on thrust force. Recent publications [3-4] point out the importance of cavity modeling for precise CFD calculation of centrifugal compressor performances. This work was done in 2004 – 2005 but has never been presented at international level. The Author is aware of recent study of the problem presented in several ASME and ATI publications and have decided to add that contribution to their results. It is very encouraging that recent numerical study and test verification presented in [4] are in good agreement with flow structure calculated by the Author.

The numerically studied shroud cavity belongs to one of tested model stages as shown in Fig. 1, above, left. The stage was tested in the TU SPb Federal Laboratory of Compressor problems in 1990th [5]. Boundary conditions at calculations correspond to measured flow parameters – pressures and temperatures at the impeller inlet and exit – 6 points with different flow coefficient inside a performance range. Similarity criteria are $M_{s} = 0.60$, $Re_{u} = 6E6$, $k = 1.4$. Three relative cavity widths were tested: $b_1 / D_2 = 0.0145$ (as of the model stage), 0.0073 and 0.029. Relative radial gaps of the labyrinth seal were $\delta_1 / D_2 = 6.1E-4, 12.2E-4$ (as of the model stage) and 24.4E-4. The minimal value $\delta_1 / D_2 = 6.1E-4$ was increased twice to study seal wearing influence.

Pressure in five points along radius of the shroud cavity was measured altogether with overall performances of the model stage with $\Phi_{des} = 0.028 – four tests. Goals of the numerical research were:

- To analyze the flow structures
- To compare the pressure distribution along the radius with the measured one,
- To estimate the influence of axial gap between the impeller disk and the stator wall,
- To estimate the influence of the radial gap in the labyrinth seal,
• To estimate the possible variation in the thrust force.

2. Cavity modeling
Initially calculations were performed with software package called FLUENT and later using package ANSYS CFX. The necessity to change software package was caused by access problems. Grids were created by the programs called TurboGrid. Turbulence model applied is $k-\varepsilon$. Flow is steady. The casing wall is stationary; the impeller wall is rotating with the velocity corresponding to $M_u = 0.60$. Calculation results that are presented below, they are related to three aspects:

- Flow pattern in a cavity,
- Flow pattern in a labyrinth seal,
- Integral parameters as leakage coefficient, disk friction coefficient, thrust force coefficient.

3. Flow structure in the cavity
Streamlines on meridional plane are presented in Fig. 2, while meridional velocity vectors at inlet and outlet of the shroud cavity are shown in Fig. 3. Radial circulation appears due to different centrifugal forces of gas particles near rotor and stator surfaces: $dp_{cent} = \rho \omega^2 r$. Peripheral velocity is close to zero near a stator surface. Peripheral velocity is close to $u = \omega \times r$ near an impeller disk surface.

Radial circulation is significantly active in the cavity with $b_r/D_z = 0.029$ (it is two times higher than standard value $b_r/D_z = 0.0145$). Streamlines are of the same character in the cavity with $b_r/D_z = 0.0145$.

Flow radial circulation is suppressed to some extent in the cavity with $b_r/D_z = 0.0073$. This width is too small to be in a real stage due to danger of rotor-stator contact.

As a result - meridional velocities are positive near an impeller and are negative near a stator wall. Fig. 4 demonstrates meridional velocities at different radii at design flow coefficient.

![Figure 2. Meridional projections of streamlines. Design flow coefficient. Left - $b_r/D_z = 0.029$, right - $0.0073$. FLUENT [2]](image-url)
Meridional centripetal flow is also a result of a leakage in a labyrinth seal. Labyrinth seal leakage mass flow rate $\bar{m}_{\text{leak}}$ enters a shroud cavity with peripheral velocity $c_{u2} = \psi_u u_2$. Flow circulation $\Gamma = 2 \pi r c_{u2} = \text{const}$ would take place in a cavity of an ideal impeller without viscosity. “Inviscid” peripheral velocity increases in a cavity at smaller radii in accordance with $c_u = c_u^2 \frac{r_2}{r}$. In reality the friction forces are diminishing flow circulation and peripheral velocity: $d (c_r r) / dr = -dM_z$, where

$$dM_z = 2\pi r dr \left( \tau_{\text{body}} + \tau_{\text{imp}} \right), \quad \tau_{\text{body}} = \lambda \rho \frac{\dot{c}_{m}^2}{2}, \quad \tau_{\text{imp}} = \lambda \rho \frac{\psi_u \dot{c}_{m}^2}{2}.$$

As loading factor $\psi_f$ is larger at smaller flow coefficients, tangential velocities grow as well. Tangential velocities in rotational coordinates $\tilde{w}_u$ are less than rotation velocity of an impeller disk. Disc surface transmits mechanical energy to a gas at all radii. Eventually tangential velocities sufficiently decrease from inlet to outlet of a cavity as result of friction forces on a stator wall.
4. Flow structure in the labyrinth seal

Velocity vectors in one element of the seal and meridional velocities in the seal are presented in Figure 6. The flow structures depict well the principle of a labyrinth seal operation [2].

![Flow structure in the labyrinth seal](image)

**Figure 6.** Flow velocity vectors in an element of labyrinth seal by ANSYS CFX [2] and meridional flow velocities in the labyrinth seal by FLUENT

5. Static pressure: measured and calculated

Calculation has demonstrated that static pressure in the cavity at the given radius is practically constant in axial direction. Its variation along the radius is due to flow rotation with peripheral velocity $c_a$:

$$p(r) = p_2 - \int r \frac{c_a^2}{u_c} dr .$$

The measurements and calculations results are presented as non-dimensional pressure coefficients:

- $\Delta \bar{\eta}(r)$ is a function of radius. It demonstrates pressure change in a cavity at given flow rate:

$$\Delta \bar{\eta}(r) = \frac{p_2 - p(r)}{\rho u_c^2} .$$

- $\Delta \bar{\eta}_{in}$ demonstrates the pressure change at the inlet of labyrinth seal, which is a function of the flow rate coefficient or loading factor $\psi_F$ (a loading factor $\psi_F$ is a function of a flow rate coefficient $\Phi$). It represents pressure drop in all cavity in non-dimensional mode:

$$\Delta \bar{\eta}_{in} = \frac{p_2 - p_{in}}{\rho u_c^2} .$$
Results of measurements from four tests are presented in Fig. 7 as \( \Delta \rho (\psi_T) = f(\Phi) \). Measurements were made during four tests. Results are presented as three groups of graphics with \( \psi_T = 0.33-0.44 \), 0.48-0.55 and 0.58-0.64.

The stage in question was tested several times with different purposes with proper assembling/disassembling. The seal radial gap was not controlled properly. It was approximately \( 0.5 \pm 0.15 \) mm. Nevertheless, the general trend is evident: the higher the loading factor is – the more pressure drops in the cavity. Calculations also demonstrated this trend. Comparison of measurements and calculations is presented in Fig. 8.

![Figure 7. Pressure coefficient as function of radius at different loading factors.](image)

![Figure 8. Measured and calculated pressure coefficient as function of radius. Dash – measurement, solid – calculation (ANSYS CFX).](image)

The comparison is made at loading factors close to design regime. Matching is better at higher loading factors in this figure and in general.

Pressure coefficient \( \Delta \rho_{lz} = \frac{p_z - p_{lz}}{\rho_m u_c^2} \) represents pressure drop in a cavity. Measured and calculated coefficients are compared in Fig. 9.

Calculated function \( \Delta \rho_{lz} = f(\psi_T) \) is based on averaged values measured at four different tests. Calculated coefficients are visibly lower. It means that loss of flow circulation \( c_u \times r \) is more than in reality. It is shown below that CFD calculated seal leakage is less than calculated by well proven A.Stodola formula. It also influences \( \Delta \rho_{lz} = f(\psi_T) \) mismatch.
Figure 9. Measured and calculated pressure coefficient $\Delta P_{\text{sc}}$ as function of loading factor. Red – measurement, black – calculation (ANSYS CFX)

6. Disk friction losses
Non-dimensional coefficient of disk friction is a ratio of disk friction power and power transmitted to gas by blades. This coefficient for the shroud cavity is:

$$
\beta_{f_{sc}} = \frac{\omega \int \tau r 2\pi r dr \cdot r}{\psi \cdot \rho u \cdot \Phi \frac{\pi}{4} D_z^2 u_2^3}.
$$

Parameters in the denominator are parameters of the tested model stage. Shear stress $\tau$ is CFD calculated. Fig. 10 presents calculation results.

Figure 10. Friction coefficient of the shroud disk. Red – FLUENT, black – ANSYS CFX

All members of the equation for $\beta_{f_{sc}}$ calculation are the same for both program packages, except the shear stress coefficient $\tau$ calculated by both packages. The difference near design point $\Phi_{\text{des}} = 0.028$ is approximately 8%. The author cannot explain the difference of results presented by FLUENT and ANSYS CFX packages.

The coefficient $\beta_{f_{sc}}$ is higher at high flow coefficients because of the loading factor diminishing.

Calculations in Fig. 11 are made for the shroud cavity with an axial width $b_z/D_z = 0.0145$. For cavities with $b_z/D_z = 0.0073$ and 0.029 the more the width the more is friction coefficient. In wide gaps radial circulation of flow is more intensive that leads to higher losses. The difference at the design flow rate is 9–10%. It is about 0.15% of a loading factor that is not insufficient. But minimization of friction losses by applying $b_z/D_z < 0.0145$ is impossible because of hazard of contacting rotor and stator.
Calculations have demonstrated negative correlation between radial gaps in a labyrinth seal and disk friction coefficient. But this results in stronger flow leakage and higher tangential velocities in the cavity. This is not the way to increase efficiency as higher leakage losses cannot be compensated.

7. Leakage losses

Non-dimensional leakage coefficient is:

\[ \beta_{\text{leak}} = \frac{m_{\text{leak}}}{m} = \frac{m_{\text{leak}}}{\Phi \cdot \rho_0 \cdot \frac{\pi}{4} D_j^2 u_2}. \]

The parameters in the denominator are obtained from the tested stage, while the flow rate of leakage is directly from CFD calculations. Leakage coefficient was calculated by ANSYS and FLUENT and compared with the result of Stodola. The formula for a shroud cavity is recommended in its simplified variant due to low compressibility effects in a cavity [7]:

\[ m_{\text{leak}} = \mu \cdot 2\pi D_{zh} \delta_{zh} \rho_0 \sqrt{\frac{2(p_{zh} - p_0)}{z_{zh} \rho_0}}. \] (1)

The empirical coefficient for the seal of the tested stage recommended value is \( \mu = 0.7 \). Results of calculations are presented in Fig. 11.

**Figure 11.** Calculated leakage loss coefficients. Red – Stodola, green – ANSYS CFX, black – FLUENT

CFD calculations predict character of \( \beta_{\text{leak}} = f(\Phi) \) well enough. Calculated leakage flow is less than calculated by the A.Stodola formula. For a compressible flow it is recommended in [7-8]:

\[ \bar{m}_{\text{leak}} = \mu \cdot 2\pi D_{zh} \delta_{zh} \rho_0 \sqrt{\frac{P_{zh} - P_0}{z_{zh} RT_{zh}}}. \] (2)

Flow rate according to eq. (2) is 5% smaller than calculated by the eq. (1) for incompressible flow. Calculations by eq. (2) and by ANSYS CFX at different radial gaps are presented in Fig. 12.

Calculations eq. (2) were made with constant values of \( \mu = 0.7 \). CFD calculations produced results not quite good. At \( \delta_j = 0.25 \) the calculated coefficient is not far, but for \( \delta_j = 0.5 \) mm it is less by about 30%, while for \( \delta_j = 1.0 \) mm this coefficient is more by 15 – 20%.

The possible reason of Stodola and CFD calculation mismatch must be the following. CFD packages calculate flow with peripheral velocity up to 125 m/s in the labyrinth seal. Most of all that experiments for definition of coefficient \( \mu \) in the Stodola formula were made for flow without peripheral velocity.
8. Thrust force coefficient

Gas pressure difference on impeller surfaces creates an axial force that is balanced by a thrust bearing. Precise calculation and minimization of the thrust force are important. The total force is directed to impeller inlet, while in the shroud cavity this force is opposite. A force in a shroud cavity $R_{sc}$ is directed against total force. Drop of pressure in a shroud cavity due to tangential velocities increases an axial force.

Results of calculation are presented in non-dimensional mode and are presented by axial force coefficient:

$$R_{sc} = \int_{\omega}^{\Phi} \frac{\Delta P}{1 - (D_b/D_z)^2} \rho \, dr.$$  

Axial force in a shroud cavity is:

$$R_{sc} = R_{sc} \cdot \rho u_z^2 \pi (D_z^2 - D_b^2).$$

The more is $R_{sc}$ the less is total axial force. For axial force minimization bigger $R_{sc}$ are necessary. Calculation results by ANSYS CFX are presented in Fig. 13.

Low leakage flows lead to higher pressure in the shroud cavity and lower thrust force. Minimization of leakage flow is better for the compressor efficiency and for a thrust bearing.

Intense radial circulations in the wide shroud cavity ($b_z/D_z = 0.029$) leads to a loss in flow momentum. Comparison of pressure coefficients for cavities of elevated $b_z/D_z = 0.029$ and the recommended $b_z/D_z = 0.0145$ is presented in Fig.14.

![Figure 12. Leakage coefficient at different radial gaps. Above – eq. (2), below – ANSYS CXF. Green - $\delta_r = 0.25$ mm, red - $\delta_r = 0.5$ mm, blue - $\delta_r = 1.0$ mm ($D_2 = 410$ mm)](image)

**Figure 12.** Leakage coefficient at different radial gaps. Above – eq. (2), below – ANSYS CXF. Green - $\delta_r = 0.25$ mm, red - $\delta_r = 0.5$ mm, blue - $\delta_r = 1.0$ mm ($D_2 = 410$ mm)

![Figure 13. Thrust force coefficient as function of loading factor. 1 – $\delta_r = 0.25$ mm; 2 – $\delta_r = 0.50$ mm; 3 – $\delta_r = 1.0$ mm)](image)

**Figure 13.** Thrust force coefficient as function of loading factor. 1 – $\delta_r = 0.25$ mm; 2 – $\delta_r = 0.50$ mm; 3 – $\delta_r = 1.0$ mm
The strong influence of the relative width on pressure distribution is evident. Low level of tangential velocities leads to higher disc friction, while larger pressure at the seal inlet leads to higher leakage. The disk friction and leakage coefficients are about 1% bigger in wide shroud cavity at design flow coefficient $\Phi_{des} = 0.028$ (ANSYS CFX), however, the loss in flow momentum is good for thrust force minimization. Thrust force coefficient versus loading factor for two shroud cavities are shown in Fig. 14. In case of necessity the designer can choose a wider shroud cavity for the thrust force minimization.

Figure 14. ANSYS CFX. Pressure coefficients in two shroud cavities. (Solid - $b_2 / D_2 = 0.0145$, dashed - $b_2 / D_2 = 0.029$. Data for three loading factors $\psi_T = 0.46, 0.53, 0.62$) and thrust force coefficient versus loading factor (green - $b_1 / D_2 = 0.0076$, red - $b_1 / D_2 = 0.0145$, blue - $b_1 / D_2 = 0.029$)

Conclusion

CFD calculations present detailed information on flow character and important integral parameters as disc friction coefficient, seal leakage coefficient, thrust force coefficient. These parameters are essential for a compressor design and performance calculations. Calculated flow behavior at different seal gaps and cavity width corresponds to described already in [1] as result of 1-D analysis. Measured and calculated pressure distribution along the cavity is close by character but differ in values. CFD-calculated leakage coefficient is less than calculated by known Stodola formula. The problems require further study. The Author’s recommendations to pay attention to axial gaps for loss minimization or the thrust force minimization are indirectly supported by results presented in [4]. Better understanding of flow behavior and following ideas influenced author’s design practice positively. There is a hope that presented information and considerations will be useful for colleagues who are active in this area now.

References

[1] Seleznev, K.P., Galerkin, Y.B. Centrifugal compressors. Leningrad. – 1982. – P. 271. [Russian].
[2] Soldatova, K.V. Analysis of gas flow in a gap «shroud - casing» of a centrifugal compressor stage by numerical methods and design recommendations. Diss. Cand.Tech. sciences. SPbTU. – SPb. – 2007.
[3] Guidotti, E., Tapinassi, L., and Naldi, G., 2012. Influence of seal leakages modeling on low flow coefficient centrifugal compressor stages performance. In ATI Congress 2012.
[4] Guidotti, E. Towards centrifugal compressor stages virtual testing. Universita delgi studi di Bologna. – 2014.
[5] Transactions of the TU SPb compressor school. –TU SPb. – 2000. [Russian].
[6] Soldatova, K.V. Calculation of leakages in a shroud – casing gap and in a labyrinth seal of a centrifugal impeller by FLUENT package. Master. diss. - SPbTU. – SPb. – 2004.
[7] Ris V. Centrifugal compressors. Leningrad. – 1981. [Russian].
[8] Transactions of the TU SPb compressor school. Under Y.B. Galerkin’s edition. - 2005 [Russian].