Influence of different design parameters on side channel compressor performance

Johanna Adam · Martin Heinrich · Rüdiger Schwarze

Received: 10 May 2022 / Accepted: 4 August 2022 / Published online: 5 September 2022 © The Author(s) 2022

Abstract
Only few design guidelines for side channel compressors are published in open literature. Among those the most common ones are from Grabow, Surek, and Raheel. In this publication we focus on the design guideline of Grabow, which includes, among several analytical equations, two empirical estimates for head coefficient and blade angle. These estimates lead to an uncertainty in the design process and could potentially effect the performance curve of the machine. Therefore, five side channel compressor geometries are designed based on Grabow with different estimates for head coefficient and blade angle in order to assess their influence onto the compressor performance. Those designs are the input parameters for transient CFD simulations to compute total pressure ratio and isentropic efficiency. The results indicate that higher head coefficient during the design process lead to a higher total pressure ratio, in particular for lower flow rates. Similarly, a higher blade angle also increases the total pressure ratio and furthermore reduces the flow circulation in the side channel. In general, influence onto isentropic efficiency is much weaker, though.

Einfluss verschiedener Auslegungsparameter auf die Kennlinie eines Seitenkanalverdichters

Zusammenfassung
Für Seitenkanalverdichter sind nur wenige Auslegungsvorschriften publiziert. Die geläufigsten dieser Auslegungsvorschriften sind von Grabow, Surek und Raheel. In dieser Veröffentlichung wird die Auslegungsvorschrift von Grabow untersucht. Diese beinhaltet, neben verschiedenen analytischen Gleichungen, zwei empirische Abschätzungen für die Druckzahl und der Schaufelwinkel. Diese Abschätzungen führen zu Unsicherheiten bei der Dimensionierung und beeinflussen potentiell die Kennlinie der Maschine. Daher werden fünf Seitenkanalverdichter-Geometrien erstellt, mit verschiedenen Abschätzungen der Druckzahl und des Schaufelwinkels um deren Einfluss auf die Performance des Verdichters zu bewerten. Diese Geometrien sind der Input für transiente CFD Simulationen, welche das Totaldruckverhältnis und den isentropen Wirkungsgrad berechnen. Die Ergebnisse zeigen, dass eine größere Druckzahl bei der Auslegung zu einem höheren Totaldruckverhältnis führt, insbesondere bei niedrigen Volumenströmen. Auch ein größerer Schaufelwinkel erhöht das Totaldruckverhältnis und verringert die Zirkulation im Seitenkanal. Im Allgemeinen ist der isentrope Wirkungsgrad ist nicht so stark von den Veränderungen betroffen.

1 Introduction
Side channel pumps and compressors have found a wide range of application in industry as an alternative to axial and radial turbomachines, and positive displacement pumps. They are capable of delivering high pressure ratios at low flow rates with just one stage, much higher than any other kind of turbomachinery with the same tip speed [2, 16]. They are also unaffected by surge and stall instability. Their main disadvantage is their low efficiency, which usually does not exceed 50% [9]. This might also be
the reason, why there are only few publications focusing on side channel pumps and compressors compared to more conventional machine types [8, 16].

A side channel compressor typically consists of an impeller, the side channel, inlet and discharge ports, and the stripper, see Fig. 1. The stripper between inlet and outlet interrupts the side channel and prevents significant backflow from outlet to inlet. Flow and energy transfer in impeller and side channel can be described by the circulation theory, first introduced by Wilson [27] in 1955 and since confirmed multiple times [15, 20, 23, 24, 26, 28]. According to this theory, the fluid has a helical trajectory from the outer radius of the impeller into the side channel and back into the impeller at the inner radius, as depicted in Fig. 2. This leads to several internal compression cycles during one rotation and thus a high pressure ratio. This flow pattern is typical for any type of side channel machine and highly turbulent [20].

The circulating flow strongly influences the performance of the side channel compressor and depends on different design parameters, such as blade angle, number of blades, side channel radius, and impeller radius. However, the number of publications regarding the design of side channel compressors is very limited. The most commonly known design guidelines are from Surek [25] and Grabow [7] for a semi-circular impeller design, and Raheel [22] for a radial impeller design.

All those design guidelines have in common that they rely on empirical correlations or estimates for certain input parameters, such as head coefficient or blade tip Mach number. In order to investigate this influence onto the machine performance we first take the guidelines of Grabow and use it to create five different machine designs. Secondly, we develop a numerical CFD model to simulate the transient flow inside the side channel compressor and compute the performance curves of those designs. Thirdly, we compare the results and link those empirical correlations and estimates onto the final performance curve.

2 Design guidelines by Grabow

2.1 General methodology

The different geometric parameters for the design guidelines by Grabow are shown in Fig. 3. In the first step the design point is selected with in the process of designing the geometry is the selection of a design point with pressure difference between outlet and inlet $\Delta p$, suction density $\rho_i$, rotational speed $n$ and volumetric flow rate $Q$. Based on these operating conditions, the specific energy $Y$ and the specific rotational speed $n_{ETX}$ can be calculated by

$$Y = \frac{\Delta p}{\rho_i}, \quad n_{ETX} = n \cdot \frac{Q^{1/2}}{Y^{3/4}}. \quad (1)$$

Using $n_{ETX}$ the head coefficient $\Psi_n$ and the blade number $z$ can be determined from the diagram in Fig. 4. The head coefficient is defined as

$$\Psi_n = \frac{Y}{u_m^2}. \quad (2)$$
By using the value of $\Psi_n$ from Fig. 4, the circumferential velocity at radius $r_m$ can be calculated by

$$u_m = \sqrt{\frac{2 \cdot Y}{\Psi_n}}. \quad (3)$$

Applying the relation $u_m = r_m \cdot \omega$, the median radius $r_m$ of the impeller can be calculated as following:

$$r_m = \frac{1}{2\pi \cdot n} \sqrt{\frac{2 \cdot Y}{\Psi_n}}. \quad (4)$$

After this, the side channel radius $r_c$ can be calculated using an estimated maximal volumetric efficiency of $\phi_{\text{max}} = 0.9$, an estimation given by Grabow [7]:

$$r_c = \sqrt{\frac{3 \cdot Q}{2\pi \cdot \phi_{\text{max}} \cdot r_m \cdot n}}. \quad (5)$$

Finally, the inner and outer side channel radius can be derived:

$$r_o = r_m + r_c, \quad r_i = r_m - r_c. \quad (6)$$

Grabow states that the blades should be inclined towards the direction of rotation. However he gives different blade angles ranging from $90^\circ$ [6] to $130^\circ$ [7]. Following the approach by Grabow [7] results in two estimated parameters:

- head coefficient $\Psi_n$ determined by Fig. 4, and
- blade angle ranging from $\beta = 90^\circ$ to $\beta = 130^\circ$.

## 2.2 Dimensions of test geometry

In order to investigate the influence of these two design parameters on the characteristic curves, a side channel compressor based on the approach of Grabow is designed with the following operating point:

- $\Delta p = 40$ kPa,
- $\rho_{\text{in}} = 1.225$ kg/m$^3$,
- $Q = 0.1$ m$^3$/s, and
- $n = 50$ s$^{-1}$.

The design point is derived from typical values for industrial machines and a similar operation point has been chosen by [4]. This results in a specific rotational speed of $n_{q, D}^* = 7 \times 10^{-3}$. Based on these specifications, 5 different designs are derived. The first three are based on the mini-

---

**Table 1** Side channel compressor designs considered in this study for different $\Psi_n$ and $\beta$ and the resulting impeller and side channel radius, $r_m$ and $r_c$, respectively

| No. | $\Psi_n$ | $\beta$ | $r_m$ | $r_c$ |
|-----|----------|---------|-------|-------|
| 1   | 7.4      | $90^\circ$ | 290 mm | 34 mm |
| 2   | 8.9      | $90^\circ$ | 270 mm | 35 mm |
| 3   | 10.0     | $90^\circ$ | 260 mm | 36 mm |
| 4   | 8.9      | $110^\circ$ | 270 mm | 35 mm |
| 5   | 8.9      | $130^\circ$ | 270 mm | 35 mm |
maximum, median, and maximum head coefficient $\Psi_{n}$ specified as shown in Fig. 5 (denotation is ‘min’, ‘med’ and ‘max’). Blade angle is kept constant for them at $\beta = 90^\circ$. Two additional designs are created based on two different blade angles of $\beta = 110^\circ$ and $\beta = 130^\circ$ at a constant head coefficient of $\Psi_{n,med}$. Table 1 summarizes the five different designs.

### 3 Numerical model

The open source CFD library OpenFOAM version v2106 [21] was employed to solve the three-dimensional, compressible, transient, and Reynolds- and Favre-averaged Navier-Stokes equations. These equations are given as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 ,
\]

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (\mathbf{r} + \mathbf{r}^{RS}) ,
\]

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \mathbf{u} E) = -\nabla \cdot \mathbf{q} - \nabla \cdot (\rho \mathbf{u} p) + \nabla \cdot ((\mathbf{r} + \mathbf{r}^{RS}) \cdot \mathbf{u})
\]

with Reynolds-averaged density $\rho$, pressure $p$, viscous and Reynolds stress tensor $\tau + \tau^{RS}$, velocity $\mathbf{u}$, and the specific total energy $E$ calculated as

\[
E = e + \frac{1}{2} |\mathbf{u}|^2
\]

with the specific internal energy $e$ and the heat flux density $\mathbf{q}$ calculated by Fourier’s law. The $k-\omega$ shear-stress transport (SST) turbulence model by Menter and Esch [18] with updated coefficients from Menter et al. [17] is employed for modeling $\tau^{RS}$. The PIMPLE algorithm, a combination of SIMPLE [5] and PISO [14], performs the pressure-velocity coupling. The CFD software has been thoroughly validated to accurately depict the real flow structure in the past, e.g. for compressible turbomachinery flows [10, 11] and incompressible turbomachinery flows [12, 13].

The convective terms in the transport equations are discretized using second order upwind schemes for velocity, energy, and turbulent quantities. Interpolation is performed with the central differencing scheme and temporal accuracy is first order with an Euler scheme for all variables. This is accurate because of the small time step which is set to $5 \times 10^{-7}$ s. This results in a maximum Courant number of 2.6 throughout the runs. The physical time simulated is 0.2 s, which resolves ten rotations of the rotor. Using a steady-state solution as an initial condition reduces the computational cost of the transient simulations. After five rotations the flow is assumed to be quasi-steady state. Therefore, the flow field is averaged over the remaining five rotations for postprocessing. The total computing time for each operating point is around 120 h on 24 cores on the HPC Cluster at the Tu Freiberg.

The geometry of the flow domain is shown in Fig. 6. It consists of a stationary and a rotating part. The stationary domain is composed of the side channel and an inlet and outlet pipe, whose lengths are extended to reduce the influence of the boundary conditions. The rotating domain resembles the compressor wheel and its tip gap at the stripper. Both domains are connected using arbitrary coupled mesh interfaces (ACMI) in order to apply sliding mesh approach. Thus, a full $360^\circ$ model for the rotor is used to capture all blade interactions. Using ACMI has the advantage that the tip gap has to be modelled only in the rotating domain. In regions where the ACMI interfaces overlap, the flow field is interpolated from one side to the other. In regions where there is only one interface, such as in the stripper, the ACMI interface simply acts as a wall. The gap to the outside at

---

**Fig. 6** Numerical model of the side channel compressor with static side channel (grey) and moving rotor (blue)

**Fig. 7** Unstructured mesh in the static domain in white (top), structured mesh in the dynamic domain in blue (bottom)
The fluid considered is air with a suction density of $\rho_{in} = 1.225 \text{ kg/m}^3$. The inlet is set to ambient conditions with $p_{tot} = 101325 \text{ Pa}$ and $T_{stat} = 293 \text{ K}$ and the turbulent intensity and turbulent length scale of $I_t = 5\%$ and $L_t = 0.01 \text{ m}$, respectively. A varying static pressure at the outlet allows for different operating points while other variables are treated as zero gradient. The stationary and moving walls are defined as smooth and no-slip. Air as medium is defined as perfect gas with a constant kinematic viscosity of $\mu = 1.716 \times 10^{-5} \text{ Pas}$. The Sutherland-model for temperature coupling of viscosity is not used, due to the fact that the temperature in the numerical model changes only moderately.

ANSYS ICEM CFD [1] is used for meshing side channel, inlet and outlet domain resulting in an unstructured, tetrahedral mesh with prism layers. Refinement is employed towards the interface and the walls. The rotor consists of a structured hexahedral mesh also created with ANSYS ICEM CFD. The tip gap of 0.5 mm size is resolved using 4 cells. Fig. 7 shows a cross-sectional view of the mesh in the side channel and the rotor.

A mesh dependency study is conducted with a total cell count ranging from $(0.5–6.0) \times 10^6$ cells with a maximum skewness of 2.3, non-orthogonality of less than 67, and aspect ratio of less than 22. The average $y^+$ value for the mesh with $2 \times 10^6$ cells is 100 at the compressor wheel and about 50 at the side channel and inlet and outlet pipes. Therefore, an all-$y^+$ wall function is used to model near-wall turbulence. This mesh is a compromise between accuracy and computational cost. Completely resolving the boundary layer up to $y^+ = 1$ requires a significant number of additional cells and therefore computing time for a fully transient simulation.

Compressor performance is evaluated using the total pressure ratio $\Pi_{tt}$ and the isentropic efficiency $\eta_{is}$ calculated as following:

$$\Pi_{tt} = \frac{p_{tot,\text{out}}}{p_{tot,\text{in}}}, \quad \eta_{is} = \frac{\Pi_{tt}^{(\gamma-1)/\gamma} - 1}{T_{tot,\text{out}}/T_{tot,\text{in}} - 1},$$

where $p_{tot}$ is the total pressure and $T_{tot}$ the total temperature at the compressor inlet (subscript: in) and outlet (subscript: out), respectively. A similar numerical investigation of an entire side channel compressor has been carried out by Beilke [4]. He showed, that the results yielded by the simulation are similar to his experimental results.
4 Results

4.1 Side channel compressor flow structure

As mentioned in Chapt. 1 the flow in a side channel compressor follows a highly turbulent helical path. The circulation can be measured by the flow velocity and the re-entry length, the distance a fluid element travels in the side channel before re-entering the impeller [4]. The helical flow path and the re-entry length are visualised via streamlines in the static part of the numerical model on the left hand side in Fig. 8. On the right hand side the circulation of the fluid in the cross section opposite of the inlet is depicted. All examples in this chapter are the configuration 2 in table 1 at a pressure ratio of 1.4.

As the fluid is moved though the machine, its pressure increases continuously from inlet to outlet. The development of the pressure in the side channel compressor is depicted in Fig. 9. As stated by [4], the circulation in the machine increases with increasing pressure. This is also visible in Fig. 8: the re-entry length shortens with increasing pressure.

4.2 Mesh dependency study

The mesh dependency study is based on configuration 2 as shown in Table 1 ($\Psi_{\text{med}} = 8.9$ and $\beta = 90^\circ$) with a total cell count ranging from $(0.5-6.0) \times 10^6$ cells. Fig. 10 and Fig. 11 show the total pressure ratio and isentropic efficiency as function of mass flow rate, respectively. The results for $(2-6) \times 10^6$ cells are almost identical for pressure ratio and isentropic efficiency. The results for $(0.5-1.0) \times 10^6$ cells deviate from the others. For this reason a mesh with $2 \times 10^6$ cells was selected for the remaining simulations.

These simulations as well as all the following ones are expected to result in lower characteristic curves in reality since the model neglects the gap losses to the ambient environment.

4.3 Influence of head coefficient

The influence of $\Psi_{\text{in}}$ on the compressor characteristics is depicted in Fig. 12 with the design point indicated as black pentagon.

All the tested geometries have in common that their characteristic curves pass below the design point and that their graphs are converge for lower pressure ratios. An explanation for the missing of the design point is the blade angle. For this study a constant blade angle of $\beta = 90^\circ$ has been chosen. This value lies beneath the suggested value of $\beta = 130^\circ$ in the design specification of [7].

Decreasing $\Psi_{\text{in}}$ and thus increasing medium radius $r_m$ and decreasing channel radius $r_c$ leads to a steeper performance curve. Therefore, a larger mass flow can be transported at the same pressure ratio. For example, the difference between $\Psi_{\text{in, min}}$ to $\Psi_{\text{in, max}}$ at $\eta = 1.4$ leads to 34% difference in mass flow rate. Badami [3], Senoo [24] and Wilson [27] had similar results for pumps, stating that a larger side channel area reduces the slope of a characteristic curve. With decreasing compressor radius $r_m$ the maximum pressure

![Fig. 10](image1)

![Fig. 11](image2)

![Fig. 12](image3)
ratio declines: For the minimal sized side channel $r_c$ the maximal pressure ratio is $\Pi_{\text{max}} = 1.6$, for the compressors with greater channel radius it is only $\Pi_{\text{max}} = 1.5$. This is different from the findings of Badami [3], Senoo [24] and Wilson [27] who concluded that a larger side channel area would increase the pressure ratio of a pump.

The isentropic efficiency of the compressor as a function of mass flow rate is shown in Fig. 13. The influence of the head coefficient on the isentropic efficiency shows an increase of 5% in $\eta_s$ from $\Pi_{\text{min}}$ to $\Pi_{\text{max}}$ at the maximum. The compressor designed with $\Psi_{\text{in, max}}$ displays overall better values: at a low mass flow rate the isentropic efficiency for $\Pi_{\text{max}}$ lies 3% above the compared designs and at the maximal mass flow rate even 5% above the value for $\Pi_{\text{min}}$. It is notable that the highest efficiency is achieved for a pressure ratio of $\Pi_{\text{it}} = 1.2$ and not at the design point of $\Pi_{\text{it}} = 1.4$. Why this is the case can not be answered now and is still matter of research.

### 4.4 Influence of Blade Angle

The results for the variation of the blade angle from $\beta = 90^\circ$ to $\beta = 130^\circ$ is given in Fig. 14. For all these configurations, the pressure coefficient is kept constant at $\Psi_{\text{in, med}} = 8.9$. The design point is indicated with the black pentagon.

As the blade angle increases, the slope of the design curve increases. The steepest blade angle of $\beta = 130^\circ$ displays the maximum slope, reaching the highest pressure ratios at a given mass flow rate. This confirms the findings of Müller [19] and Badami [3], which can be applied for compressors as well. They stated that a forward inclination of blades improves the performance of a pump. At high flow rates the blade angle has no strong effect on the total pressure ratio. The difference between the blade angles can be detected at small flow rates where a steeper blade angle delivers a higher total pressure ratio: An impeller with steeper blades can reach higher pressure ratios for the same mass flow rate: the blades with $\beta = 130^\circ$ achieve a value of $\Pi = 1.8$ at $\dot{m} = 0.08 \text{kg s}^{-1}$ whilst the blades with $\beta = 110^\circ$ reach a value of $\Pi = 1.67$ and the perpendicular blades only reach a value of $\Pi = 1.46$.

Fig. 15 depicts the isentropic efficiency for the varying blade angles. Like with the variation of the head coefficient in Fig. 13, $\eta_s$ has the highest values at a pressure ratio of $\Pi_{\text{it}} = 1.2$. The isentropic efficiency has the highest values...
for the perpendicular blades, which are about 2–5% above the angled blades.

### 4.5 Re-entry length

Besides the differences in characteristic curves, another parameter that highlights the differences between the compressor configurations is the re-entry length $l$. In Fig. 16 the relative re-entry lengths are compared. For this comparison, the average length of a stream line in the side channel, excluding the inlet and outlet pipes, is set in relation to the reference re-entry length $l_r$. The reference value is marked in the diagram. At this point the re-entry length is $l = 7.8 \text{ mm}$ which is close to the circumference length of the side channel with 7.7 mm.

As expected, the re-entry length decreases for higher values of $\Pi_{nt}$. The geometric features have a great influence on the re-entry length. The inclined blades lead to longer streamlines in the side channel compared to perpendicular blades. For a compressor with perpendicular blades and the minimal value of $\psi_n$, the re-entry length at $\Pi_{nt} = 1.6$ is 0.3, only a fraction of the side channel circumference. From this point the flow can not twist any more, limiting the maximum pressure ratio.

From Fig. 16 it can be deducted that this re-entry length is increased by the inclined blades. To stress this, the re-entry lengths for the different blade angles are compared at $\Pi_{nt} = 1.4$ in Fig. 17.

For all three pressure ratios it is visible, how the re-entry length decreases from inlet to outlet. This is expected, as the fluid has been exposed to the impeller multiple times at the outlet. Also the re-entry length is visibly shorter for the perpendicular blades, as derived from Fig. 16.

### 5 Conclusion

With the design specification by Grabow [7], five different compressor geometries were determined. For this process, three values for the head coefficient and three blade angles were chosen. The derived compressor geometries have varying median and channel radii as well as blade angles.

The different compressor geometries were with the open source CFD library OpenFOAM by performing transient RANS simulations. Future simulations should include the losses through the gaps to the outside.

The numerical investigation of these cases indicate that a smaller value of the head coefficient $\psi_n$ leads to an increase in mass flow rate at a given pressure ratio. The isentropic efficiency shows the highest values for the maximum chosen head coefficient.

A blade inclination heavily influences the characteristic curve. With increasing blade angle, the pressure ratio in the compressor grows. A greater blade angle also enables the compressor to achieve higher values of $\Pi_{nt}$. The perpendicular blades display the highest isentropic efficiency.

The re-entry length for perpendicular blades never exceeds the circumferential length of the side channel. With an inclination of the blades the re-entry length is increased.

As a conclusion it can be stated that the specification by Grabow [7] can be used to design a side channel compressors for a given design point. To further improve the results, a future investigation should include experimental investigations as a comparison. In this case it also possible to include a calculation of the gap losses, to further improve the accuracy of the model for the real machine.

The influence of the design parameters and geometric dimensions on the side channel compressor performance is still the matter of ongoing research. This paper could not fully explain the reason behind the highest isentropic efficiency being situated at a higher mass flow rate than the design point of the investigated machine.
Funding  Open Access funding enabled and organized by Projekt DEAL.

Conflict of interest  The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Open Access  This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article’s Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article’s Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit http://creativecommons.org/licenses/by/4.0/.

References

1. ANSYS ICEM CFD (2019) Version 19. http://www.ansys.com. Accessed 13.04.2022
2. Appiah D, Zhang F, Yuan S, Osman M (2017) Effects of the geometrical conditions on the performance of a side channel pump: A review. Int J Energy Res 42(2):416–428. https://doi.org/10.1002/er.3803
3. Badami M (1997) Theoretical and experimental analysis of traditional and new periphery pumps. SAE Technical paper series
4. Beilke J (2005) Numerische Untersuchung der instationären Strömung in Seitenkanalverdichtern. PhD thesis
5. Caretto L, Gosman A, Patankar S, Spalart D (1972) Two calculation procedures for steady, three-dimensional flows with recirculation. In: Proceedings of the Third International Conference on Numerical Methods in Fluid Mechanics, vol 19, pp 60–68
6. Grabow G (1999) Berechnungsverfahren für Radial-Seitenkanalmassen. Forsch Ing-Wes 64(11):307–316
7. Grabow G (1999) Ermittlung der Hauptabmessungen für Seitenkanalmassen. Forsch Ing-Wes 65:41–47
8. Grabow G, Suong N (1992) Berechnungsmodell zur Auslegung von Seitenkanalmassen. Forsch Ingenieurwesen 58(11–12):273–282. https://doi.org/10.1007/BF02560552
9. Griffin D, Salvadori S, Carnevale M, Cappelletti A, Ottanelli L, Martelli F (2015) On the development of an efficient regenerative compressor. Energy Procedia 82:252–257. https://doi.org/10.1016/j.egypro.2015.12.030
10. Heinrich M, Schwarze R (2013) Simulation and validation of the compressor stage of a turbocharger using openfoam. In: 8th Institution of Mechanical Engineers – 8th International Conference on Compressors and their Systems, pp 659–667 https://doi.org/10.1533/9781782421702.11.659
11. Heinrich M, Schwarze R (2013) Simulation of the compressor stage of a turbocharger – validation of the open source library openfoam. Proc ASME Turbo Expo. https://doi.org/10.1115/GT2013-94511
12. Heinrich MFC, Khaleghi H (2020) Effect of circumferential casing treatment on low-speed contra-rotating fans. J Appl Fluid Mech 13(6):1719–1726. https://doi.org/10.47176/jafm.13.06.31492
13. Heinrich MSR, Friebe C (2018) Experimental and numerical investigation of a gearless one-motor contra-rotating fan. In: 7th European Conference on Computational Fluid Dynamics (ECCM-ECFD 2018), Glasgow, Schottland, 11 - 15 Juni
14. Issa RI (1986) Solution of implicitly discretized fluid flow equations by operator-splitting. J Comput Phys 62(1):40–65
15. Krake A, Fiedler K (2000) Flow conditions in a side channel compressor. In: The XVth Bi-Annual Symposium on Measuring Techniques in Transonic and Supersonic Flow in Cascades and Turbo-machines, pp 1–8
16. Meakhlai T, Park SO (2005) An improved theory for regenerative pump performance. Proc Inst Mech Eng Part A J Power Energy 219(3):213–222. https://doi.org/10.1243/095765005X7565
17. Menter F, Kuntz M, Langtry R (2003) Ten years of industrial experience with the sst turbulence model. In: Turbulence, heat and mass transfer 4, vol 32, pp 625–632
18. Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J 32:1598–1605
19. Müller T (2012) Anwendung von algebraischen Berechnungsmodellen und CFD-Ergebnissen zur Bewertung der hydraulischen Güte von Seitenkanalmassen. PhD thesis
20. Münsterjohann S, Heinemann T, Becker S (2017) Circulatory flow characteristics in side channel blowers. Appl Mech Mater 871:237–243. https://doi.org/10.4028/www.scientific.net/amm.871.237
21. OpenFOAM (2021) Version v2106. http://www.openfoam.com. Accessed 26.04.2022
22. Raheel MM, Engeda A (2005) Systematic design approach for radial blade regenerative turbomachines. J Propuls Power 21(5):884–892
23. Schroll M (2003) Untersuchungen der instationären Strömungsvorgänge in Seitenkanalverdichtern mit Hilfe der Particle-Image Velocimetry. PhD thesis
24. Senoo Y (1956) A comparison of regenerative pump theories supported by new performance data. Transcr ASME 78:1091–1102
25. Surek D (1996) Geometrische dimensionierung von Seitenkanalmassen. Forsch Ingenieurwesen 62(9):229–238. https://doi.org/10.1007/BF02601429
26. Surek D (2004) Strömung mit Energiezufluhr im Seitenkanal von Verdichtern. Forsch Ingenieurwesen 68(4):213–226. https://doi.org/10.1007/s10010-004-0129-8
27. Wilson W, Santalo M, Oelrich J (1955) A theory of the fluid-dynamic mechanism of regenerative pumps. Transcr ASME 77:1303–1316
28. Zhang F, Chen K, Appiah D, Hu B, Yuan S, Asomani S (2019) Numerical delineation of 3d unsteady flow fields in side channel pumps for engineering processes. Energies. https://doi.org/10.3390/en12071287