Near-blade flow structure modification

T Kura and E Fornalik-Wajs
AGH University of Science and Technology, Department of Fundamental Research in Energy Engineering, al. Mickiewicza 30, 30-059 Krakow, Poland

E-mail: kura@agh.edu.pl

Abstract. In this paper, the importance of near-blade flow structure influence on the performance of a centrifugal compressor was discussed. The negative effects of eddies and secondary flows appearance were described, together with the proposal of their reduction. Three-dimensional analyses were performed for the rotors. Focus was placed on the blade’s 3D curvature impact on the efficiency of compression, and the influence of blade-shroud tip existence. A few design proposals were investigated – their performance maps were the basis of further analysis. Proposed modification of blade shape changed the near-blade flow structure and improved the compressor performance.

1. Introduction
Turbomachinery has been a field of research since the beginning of modern engineering. Experimental data is widely available, but the researchers are still trying to improve designs of particular units and to reduce negative effects of the swirls or secondary flows – in modern world more efficient and, as a result, more ecological machines are desirable. The need for constant improvement is also connected with the fact, that each year, millions of new turbomachinery devices are being produced. Thus the main reason of following studies is a trial to develop appropriate model of centrifugal air compressor.

Progress in new design proposals is possible thanks to CFD methods. Due to them influence of geometry on the flow can be checked with many shape factors which help to obtain flow structure for any examined model. Author has taken the advantage of CFD in [1], where the impact of blades’ height, curvature and number on compressor’s efficiency was investigated. The results confirmed, that the leaks and backflows in compressor’s area can cause significant pressure losses. Those backflows were caused by some design issues, for example the blade-shroud clearance presence. Comprehensive analysis on mentioned flow behavior was done by Brun and Kurz [2]. They have presented the method to predict the direction and strength of secondary flows, as well as the pressure loss caused by them. In [3] Eum et al. have investigated the entropy generation in such areas. However only a few articles from the last couple of years cover the topic of losses caused by mentioned clearance [4, 5, 6]. While it is possible to avoid the issue by designing machines consisted of only one solid part, that process is more difficult and expensive. Apart from flow issues, such intervals between machine’s parts are the source of undesirable noises [7]. And, due to the high temperature gradients in that areas, shape deformations may occur, being an additional issue to consider during design process [4, 8].

Many of the equations proposed for evaluation of mentioned losses are empirical, and so they can be used only in particular cases. Therefore, analyses, such as these presented in the following paper, are necessary to verify them or become the basis of the alternative ones. To check, if it is possible to find any relation that would serve as foundation of those, characteristics of compression rate versus
mass flow rate depending on a few geometry types as well as overall performance of proposed rotors, together with pressure fields of compressors configurations will be presented in this publication.

2. Mathematical model
The analyses were based on the Reynolds-averaged (RANS) mass, momentum and energy conservation laws, coupled with the ideal gas law. Modeled flow was three-dimensional, stationary and compressible. Results were verified with those coming from theoretical considerations included in [9].

Continuity equation was considered in following form [10]:
\[
\frac{\partial}{\partial x_j}(\rho U_j) = 0, \quad (1)
\]
where \( \rho \) is the fluid density, kg/m\(^3\) and \( U \) is its velocity, m/s.

Momentum equation was represented by [10]:
\[
\frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \bar{u}_i \bar{u}_j) + S_{Cor} + S_{cfg},
\]

\[ S_{Cor} = -2\rho \omega \times U_r \]  
\[ S_{cfg} = -\rho \omega \times (\omega \times r) \]  
where \( p \) is the pressure, Pa, \( \tau \) is the molecular stress tensor, \( U_r \) is the relative frame velocity, \( u \) is the fluctuating velocity, \( S_{Cor} \) is the source term accounting for the effect of Coriolis force and \( S_{cfg} \) is the centrifugal force source. \( \omega \) is the angular velocity, rad/s and \( r \) is the location vector.

Energy equation was applied in the form of total energy [10]:
\[
\frac{\partial}{\partial x_j}(\rho_{i_{tot}} U_j) = \frac{\partial}{\partial x_j}(\lambda \frac{\partial T}{\partial x_j} - \rho \bar{u}_j \bar{u}_j),
\]

\[ i_{tot} = i + 0.5U_i^2 + k \]  
where \( T \) is the temperature, K, \( \lambda \) is the fluid thermal conductivity, W/(m·K) \( i_{tot} \) is the specific total enthalpy, J/kg \( i \) is the specific static enthalpy, J/kg and \( k \) is the turbulence kinetic energy, J/kg.

Analyses were conducted with Ansys 14.5 software, particularly CFX and Turbogrid modules. ANSYS CFX solves all equations with coupled solver method. It is useful due to fast computing speed, yet powerful hardware is needed, with high memory resources. Following simulations were conducted in Academic Computer Centre CYFRONET AGH. High resolution second order upwind advection scheme was utilized, to avoid numerical diffusion and obtain more certain results. To solve turbulence, SST k-\( \omega \) turbulence model was chosen, since as the combination of typical k-\( \epsilon \) and k-\( \omega \) models, is considered one of the best for simulations [10], because it is stable when high pressure gradients or flow separation areas occur – such as in centrifugal compressors. Most important feature of that model, however, is the ability to generate appropriate results in the boundary layers – which was one of the main goals for Authors.

The air was chosen as the working fluid. It had the density value of 1.225 kg/m\(^3\), the thermal conductivity –value of 0.0242 W/(m·K), the viscosity value of 1.79E-05 kg/(m·s) and the specific heat – 1006.4 J/(kg·K).

A few design proposals were investigated, with rotational speed of the rotor being changed for each case. Its particular values were chosen in the basis of previous experience [1]. The initial temperature in the whole control volume was equal to 288 K. At the inlet, pressure value of 100 kPa was chosen. Due to high rotational speeds in all cases, supersonic conditions were expected throughout the model – therefore adequate outlet boundary condition type was chosen [10]. Inlet turbulence intensity value was set at the low level equal to 5%, as the main reason of the turbulence was the rotation, not the inflow. Table 1 presents boundary conditions implemented in all simulations.
### Table 1. Boundary conditions.

|                          |        |        |        |
|--------------------------|--------|--------|--------|
| Initial temperature [K]  | 288    |        |        |
| Inlet pressure [kPa]     |        |        | 100    |
| Rotational speed [rpm]   | 28 000 | 30 000 | 32 000 |

### 3. Studied cases and applied solutions

Three cases were investigated for the purposes of the article. Starting from the typical radial blade and finishing on two examples of centrifugal blades. No diffusers were analysed, as the output of the studies was only the rotor performance. Therefore only they were discretized. Software was capable of interpolating results from just one compressor’s passage into the whole machine, so that more accurate mesh was generated. In figure 1, first blade, B1, is presented. It was around 10 cm long and its height varied from 3 cm on the end to 2 cm close to the rotor axis. Its shape was developed basing on data from [9] and was used mainly for purposes of mesh verification. In figure 2 second blade proposal, B2 is shown. While its curvature along the length was the same as for B1 case, the curvature along the height was added as the second shape function. It was shorter than the B1 blade, and, at the same time, higher in the inlet zone. In figure 3 third blade design, B3, is shown. Its main dimensions are the same as for the B2 case, apart from the curvature along height. B3 blade is more straight. In figures 1, 2 and 3 the axis of the view is slightly different, to show changes of the shape. Figure 4 shows B2 and B3 blades cut planes to show mentioned difference in that curvature. Considered shapes and the number of blades per rotor, 15, were chosen due to previous experience, supported by research, such as [4,9]. Rotation of all impellers occurred around the axis visible in figure 1. One of the article purposes was to investigate the impact of blade-shroud clearance on the machine’s efficiency. Typically, they are no bigger than a few millimetres. For the following publication, two lengths were taken into consideration – 1 mm and 2mm, constant along the blade. Blade-shroud tip position is shown in figure 5.

![Figure 1. Design of B1 blade.](image1)

![Figure 2. Design of B2 blade.](image2)

![Figure 3. Design of B3 blade.](image3)

![Figure 4. Blades curvature along the height, (a) B2 and (b) B3.](image4)

![Figure 5. Blade-shroud tip.](image5)
Discretization of geometries was performed with Ansys Turbogrid, in the basis of finite volume method. For the quantitative numerics, regular, mostly hexahedral elements were generated. As investigation of near wall flow behaviour was of great importance, refined mesh was necessary in the boundary layer around the blade. Also, due to dimensions of the blade-shroud tip, refining was obligatory in that area. Basing on boundary conditions and therefore approximate Reynolds number, \( y^+ \) parameter was calculated, as the main factor that influences boundary layer shape. For the Reynolds number value equal to approximately \( 0.5 \times 10^7 \), \( y^+ \) did not exceeded 2 – value considered maximal for proper calculations [10]. Figure 6 presents overall look of the mesh, with visible refining around the blade. Figure 7 represents the close-up of blade-shroud tip mesh. The view is located above the blade, and due to the software characteristics such shape of the mesh is necessary for reliable computing.

To verify, whether generated mesh is correct, its independence test was obligatory to conduct. Various number of elements and nodes was taken into account, from about \( 3 \times 10^5 \) to \( 1.5 \times 10^6 \). However, after reaching \( 1.3 \times 10^6 \), further increase did not visibly changed the obtained results – and that number was chosen as the proper one. Quantity of elements and nodes for each case is presented in table 2. All designs are also divided depending on the blade-shroud tip size. Afterwards, to validate the results, comparison was performed on the basis of empirical formulas taken from [9]. The results are shown in table 3. Difference of numerical and analytical results did not overstep 5% and, as the purpose of the research was to check general tendency and behaviour of flow in described rotors, such discrepancy was considered sufficient and the chosen mesh was decided to be fine enough to continue the studies.

\[
dl i - dq = dl_{tot},
\]

(7)

where \( l_i \) is the internal work done on gas J/kg, \( q \) is the energy loss J/kg. Based on it, the static and total pressures were used for the verification, and were calculated using Bernoulli’s equation. Total parameter was the sum of its dynamic and static part. Total temperature was evaluated in the same manner but the equation of state had to be included as well.

**Table 2.** Elements and nodes number for each case.

| Case | B1 | B2 | B3 |
|------|----|----|----|
|      | 1mm | 2mm | 1mm | 2mm | 1mm | 2mm |
| Elements | 1 251 166 | 1 258 965 | 1 238 344 | 1 245 666 | 1 224 274 | 1 233 906 |
| Nodes | 1 311 860 | 1 319 916 | 1 305 557 | 1 313 399 | 1 290 864 | 1 301 248 |

**Table 3.** Validation of proposed mesh. Compatibility of analytical and numerical results.

| Variable | Static pressure | Total pressure | Static temperature | Total temperature | Density |
|----------|-----------------|----------------|-------------------|-------------------|---------|
| Accordance, % | 96.1 | 94.6 | 99.6 | 99.2 | 96.7 |
4. Results and discussion

For the purposes of presented work, all investigated rotors were numerically tested with varying rotational speeds. One of the goals of such behaviour was to examine, if the compression ratio of any machine would change with the change of blade’s shape. Compression ratio (CR) is one of the main factors that indicates the performance of the machine. It is being defined as [10]:

\[ CR = \frac{p_{\text{out}}}{p_{\text{in}}} \]

(8)

where \( p_{\text{in}} \) is the total pressure at the inlet and \( p_{\text{out}} \) is the total pressure at the outlet – they were calculated by software as the area integral in two mentioned zones. Figures 8 and 9 present values of compression ratio for each case, apart from B1. That design turned out to be inaccurate for proposed boundary conditions and did not give any compression. Its shape was however designed for lower rotational speeds [9] and that situation was expected. Therefore, B1 blade was not included in further analyses. B2 and B3 rotors gave quite similar CR values, up to almost 2.2 for 5 out of 6 situations, however, B3 blades turned out to be slightly better, with higher CR numbers. Moreover, increase of rotational speed led to better compression in all calculations – connected though with higher input energy needs. The noticeable fact is, that it is hard to define any pattern connected with the size of blade-shroud tip, even though it reached 10% of the blade’s height in the most narrow part of blade – which is rather significant value. To compare the proposed designs another parameters were chosen – the relative head coefficient, the relative flow coefficient and the relative isentropic efficiency. The head coefficient is a dimensionless factor which relates the adiabatic head capability of a wheel with its peripheral velocity requirement and the flow coefficient is a non-dimensional unit which takes flow into consideration for a given tip speed and impeller diameter [11]. Isentropic efficiency measures the irreversibility of the process. They are defined, as:

\[ \Psi = \frac{3600gh}{(N\pi D)^2}, \]

(9)

\[ \Phi = \frac{240\bar{V}}{\pi^2D^3N}, \]

(10)

\[ \eta = \frac{I_{2s}-I_{1i}}{I_{2o}-I_{1i}} \]

(11)

where \( \Psi \) is the head coefficient, \( \Phi \) is the flow coefficient, \( \eta \) is the isentropic efficiency, \( g \) is the gravity constant, 9.91 m/s\(^2\), \( H \) is the generated head per impeller, m, \( N \) is the rotational speed, rpm, \( D \) is the diameter of the rotor, m, \( \bar{V} \) is the volume flow rate, m\(^3\)/s and \( I \) is the total enthalpy, J. Indexes mean: 1 – the inlet, 2 – the outlet, s – the value in isentropic process. Relative amounts of those non-dimensional parameters were obtained by dividing their values for each case by the maximal value.

![Figure 8. B2 rotor compression ratio.](image)

![Figure 9. B3 rotor compression ratio.](image)
Figures 10 and 11 present relative head coefficient values, for B2 and B3 case, respectively. Figures 12 and 13 present relative flow coefficient values, for B2 and B3 case, respectively. Figures 14 and 15 present relative isentropic efficiency values, for B2 and B3 case, respectively. Analysis of obtained Φ values does not indicate clear pattern. However, there is a clear decrease of Ψ and η parameters with the increase in rotational speed, which means, that further acceleration would move each machine from the design conditions. The next step in comparison process needed analysis of pressure fields. Figures 16 and 17 represent the pressure distribution on cross-section in the middle of device passage. For all images, constant rotational speed, 30000 rpm, was chosen, as representative one. In such planes, slightly higher values of outlet pressure can be noticed for impellers with 2 mm blade-shroud tip, but the overall compression ratio, already mentioned, was higher only for B2 case. For both designs pressure distribution along the blade seemed similar, with visible increase of that parameter along it. Two areas, where pressure increase occurred, can be noticed – closer to inlet and closer to outlet. Black arrows above the figures indicate the flow direction.
Figure 16. Pressure fields, B2 case, 30000 rpm, blade-shroud tips: (a) 1 mm and (b) 2 mm.

Figure 17. Pressure fields, B3 case, 30000 rpm, blade-shroud tips: (a) 1 mm and (b) 2 mm.

Figure 18. TKE distribution, 30000 rpm, (a) B2 case and (b) B3 case.
Another possibly important parameter, that could have helped receive the whole image of examined impellers was the turbulent kinetic energy, TKE, which is defined as the kinetic energy per unit mass of the turbulent fluctuations and is written as [12]:

$$
k = \frac{1}{2} \left( \overline{u_x'^2} + \overline{u_y'^2} + \overline{u_z'^2} \right),
$$

(12)

where \(u_x', u_y', u_z'\) are the velocity fluctuations of the flow in \(x, y, z\) directions, respectively. Its values for B2 and B3 cases, again for rotational speed of the rotor set to 30000 rpm, are presented in figure 18. While in general both distributions of TKE were similar, differences can be seen near the outlet zone. TKE corresponded with the velocities fields in both cases – and with its values increasing along the flow direction, it indicated increasingly turbulent and possibly irregular flow closer to the outlet.

5. Summary

Various impellers of centrifugal compressor were investigated to examine the impact of blades shape and shroud-tip size on their performance. All boundary conditions remained the same, apart from rotational speeds. Air was being compressed in each case. While B1 design turned out to be absolutely inaccurate for as high rotational speeds as in simulations, B2 and B3 rotors gave very similar results, mainly compression ratios of about 2. Both blades, B2 and B3, depending on speed of rotation, had slightly different parameters: head and flow coefficients and isentropic efficiencies, yet disparities of their values were not very high. It was impossible to deduce any convincing relation between them.

For both cases, B2 and B3, pressure distribution along the passages was not ideal. Due to blades shape, two areas of more significant pressure increase appeared. That fact may have influenced outlet pressure values. Analysis of TKE distribution led to opinion, that the biggest turbulence was noticed in the outlet area, corresponding with the highest velocities.

It was hard to conclude any impact of different blade-shroud tip size on devices performance. It was surprising, considering the fact, that in the most narrow area, it was around 10% of the blade’s height. Analyzing the obtained results, it is impossible to find any correlation. Presented results can be the basis of further research. While blade-shroud tip size impact was not confirmed, it is considered a fact in other publications [3]. However, as they reported the various height of it along the blade, more insight investigation should be performed. One of possible explanations of different results can come from the shape of the clearance.

Acknowledgments

The present work was supported by Academic Computer Centre CYFRONET AGH, under award no. MNiSW/IBM_BC_HS21/AGH/021/2015 and by the Polish Ministry of Science (Grant AGH No. 15.11.210.344).

References

[1] Kura T 2015 Numerical analysis of velocity and temperature fields of compressible fluid flowing through a rotor (MSc Thesis, in Polish)
[2] Brun K, Kurz R 2005 Int. J. of Rotating Machinery 1 45-52
[3] Hark-Jin E, Young-Seok K, Shin-Hyoung K 2004 KSME Int. J. 18 979-998
[4] Kim C, Lee H, Yang J, Son C, Hwang Y 2016 Int. J. of Refrigeration 65 92-102
[5] Swamy S M, Pandurangadu V 2013 Int. J. of Research in Engineering and Technology 2(9) 445
[6] You D, Wang M, Moin P, Mittal R 2006 Physics of Fluids 18
[7] Galindo J, Tiseira A, Navarro R, López M A 2015 Int. J. of Heat and Fluid Flow 52 129-139
[8] Jung Y, Choi M, Oh S, Baekl J 2012 Int. J. of Rotating Machinery 2012
[9] Witkowski A 2013 Sprężarki wirnikowe Teoria, konstrukcja, eksploatacja (Wyd. Pol. Słaskiej) (in Polish)
[10] ANSYS® CFX-Solver Theory Guide 2013 (ANSYS Inc., Cecil Township, 2013)
[11] Khan M O 1984 Int. Compressor Engineering Conference 509
[12] Wilcox D 2006 Turbulence Modeling for CFD (DCW Industries, Inc.)