Compressor efficiency in the light of blade-fluid thermal interaction

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

E-mail: kura@agh.edu.pl

Abstract. Blade-fluid heat transfer effects in turbomachinery can have influence on its performance. With high rotational speeds, the dynamic parameters need to be considered, because they cause significant gradients of velocity, pressure and as a result – temperature. Three-dimensional numerical analysis of two radial compressor blades was conducted. Calculations were done for various rotational speeds. Air, treated as compressible ideal gas, was chosen as the working fluid. Results showed only a small change in heat absorbed by the blade due to an increase in rotational speed. Its value did not affected the efficiency and pressure ratio significantly. The main cause of this change was connected with turbulent eddies, occurring due to flow separation, as the result of second blade’s design drawbacks.

1. Introduction
Heat transfer effects in turbomachinery may have great impact on its performance. With high values of operating parameters like pressure and temperature, their gradients can cause high entropy generation and, as a result, energy losses. Moreover in extreme situations, they may lead to material deformations and destruction of device. Approach to reduction of such effects is focused on blade cooling proposals, because the moving parts are the most vulnerable to those deformations. Temperature gradients are significant especially in high speed machines. They can be reduced and there are two main possibilities of such reduction – an internal cooling of blades and generation of cooling film around the blade [2]. When considering fluid as the system, heat transferred by it to the blade is treated as its loss. Depending on the type of machine, different values of such losses may occur. For example in scroll compressors thermal energy dissipation might be less than 1% of power input [3]. On the other hand Wu and Hu stated, that in rotary compressors heat transfer cannot be neglected when calculating heat dissipation of device shell [4]. The working fluid may also play major role in the heat dissipation processes, due to various values of its heat capacity [4]. The influence of blades shape on the rotating machine performance was discussed in [5]. Analysis conducted for various cases showed its influence on temperature distribution, it means on the heat transfer processes. However it was not possible to find an example of research regarding similar blade-fluid thermal interaction. Related problem was discussed only in [4]. Presented studies were undertaken to understand the role of the fluid and blade in these processes, especially their impact on compressor performance, and to specify how significant that impact is. It was checked, if it is possible to control the process of heat transfer with accurate machine design. Two blade shapes were analyzed under typical operational conditions. Also, the flow structure and its influence on thermal processes and energy dissipation.
2. Mathematical model

In this paper, the flow was treated as three-dimensional, stationary and compressible. Analyses were conducted with ANSYS 14.5 software, particularly CFX and Turbogrid modules. High resolution, second order upwind advection scheme was applied to provide certain results. ANSYS uses coupled solver, assuring fast computing, yet as all variables are being determined at the same time, it needs significant memory resources. The conservation laws of mass, momentum and energy coupled with ideal gas law were discretized using Reynolds averaged equations. Continuity equation in the index notation was considered, as [6]:

$$\frac{\partial}{\partial x_j}(\rho U_j) = 0,$$

where \( \rho \) is the fluid density, kg/m\(^3\), \( u \) is its velocity, m/s. Momentum equation was represented by [6]:

$$\frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \vec{u}_i \vec{u}_j) + S_{Cor} + S_{cfg},$$

where \( p \) is the pressure, Pa, \( \tau \) is the molecular stress tensor, \( S_{Cor} \) is the source term accounting for the effect of Coriolis force and \( S_{cfg} \) is the centrifugal force source. Energy equation was in the form [6]:

$$\frac{\partial}{\partial x_j}(\rho \iota_{tot} U_j) = \frac{\partial}{\partial x_j}(\lambda \frac{\partial T}{\partial x_j} - \rho \vec{u}_j \vec{u}_j^2),$$

where \( T \) is the temperature, K, \( \lambda \) is the fluid thermal conductivity, W/(m·K) \( \iota_{tot} \) is the specific total enthalpy, J/kg \( i \) is the specific static enthalpy, J/kg and \( k \) is the turbulence kinetic energy, J/kg. Moreover SST k-\( \omega \) turbulence model was applied in following paper. It was chosen due to its widely known advantages [6] over other two-equations models, especially in the cases connected with high pressure gradients and flow separations – both of them were expected to occur in described analyses.

Two blade designs were proposed. For both of them, the same inlet boundary conditions, listed in table 1, were applied, to make the comparison available. Moreover, inlet turbulence intensity was set to 5%. Its values range, advised as convenient for high-turbulence cases, such as turbomachinery flows, is between 5 and 20% [7]. Yet as the inlet was in the zone before the rotor, where flow is largely stabilized, minimal value was chosen. The only difference between both cases was the rotational speed. It was variable for the purpose of evaluation, how big influence it had on the dynamic parameter and then on the heat transfer. Its higher values were assigned to the centrifugal type compressor, as those are typical for this kind of units. Different outlet boundary condition types were used – which was necessary due to software specification. For the first design, outlet was set as typical subsonic type, with defined outlet mass flow rate: 1.6722 kg/s, which was taken from [8]. Second design had the supersonic boundary type – as higher velocities were expected. For the second case, there was no need to define mass flow rate. Steel was chosen as the blades material, with the density, thermal conductivity and specific heat values equal to default values for ANSYS software [6]. Air was chosen as the working fluid, treated with ideal gas model. Its density had value of 1.225 kg/m\(^3\), thermal conductivity – 0.0242 W/(m·K) and specific heat – 1006.4 J/(kg·K), in standard conditions [6]. Initial temperature in the whole passage as well as the blade was 288 K. Rotation occurred around the axis presented in figures 1, 2, 3 and 4, in the indicated direction.

| Table 1. Boundary conditions. |
|-------------------------------|
| **First design** | **Second design** |
| Temperature [K] | 288 | 16 000 |
| Pressure [bar] | 1 | 32 000 |
| Rotational speed [rpm] | 16 000 | 16 000 |
3. Studied cases and applied solutions
Two different blade geometries were investigated. Their shape was based on data available in
literature [8]. Due to the features of software, only one blade passage was discretized per design, to
reduce computer resources needed for simulation. No diffusers were included in the project, only short
extensions of passages, to check the outflow from blades. In figure 1 the first proposed design, B1, is
shown. It represents the typical radial blade, that theoretically causes two-dimensional flow. Figure 2
presents second design, B2, the centrifugal blade, that even theoretically should generate three-
dimensional flow. Second design corresponds with typical blades used in modern high-speed
compressors. Arrows indicate the flow direction. B1 was about 10 cm long and of variable height,
from almost 3 cm at inlet side, to 2 cm at the outlet side. B2 was 50% shorter and at the same time
higher at the inlet side, to maintain similar passage’s volume size. As the first design was taken
straight from book [8], it served also as a model for verification of proposed model. To maintain better
flow behaviour, hub, shroud and blades were treated as one part, without clearances – such designs are
feasible in reality. Whole passages are shown in figures 3 and 4.

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

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

![Figure 3. Mesh of the first design passage.](image3)

![Figure 4. Mesh of the second design passage.](image4)

Conducted analysis was based on finite elements method. Discretization of the computing space
was done using two available modules – Meshing and Turbogrid. Mostly hexahedral elements were
chosen. Their number was high enough to maintain shapes close to cubes in areas of high curvature. In
blades mesh along the thickness of blade 10 elements were kept to obtain realistic solution. The most
important issue was to refine the fluid mesh near blades, to achieve proper heat transfer between solid
and fluid. In Turbogrid module, it can be achieved basing on $\gamma^+$ parameter. An approximate Reynolds
number value was set to be $0.5\times10^7$, therefore $\gamma^+$ parameter in both models was not higher, than 2. That
value was considered small enough for proper calculations [6].

In the beginning, mesh independence test was conducted on the model and various numbers of
elements were investigated to find the most accurate compromise between the time of simulation and
conformity. Global size factor was being changed with each step, and elements number ranged from
3·10⁵ to 1.3·10⁶. However, starting from value of about 7·10⁵, negligible changes of results were being obtained. Therefore that number of elements was chosen for further analysis. The number of nodes and elements generated for each design is listed in table 2. Disparity of values comes from the fact, that the first design’s more dense mesh was generated for the verifications process.

| Table 2. Number of nodes and finite elements. |
|-----------------------------------------------|
| **First design**                     | **Second design**                   |
|----------------------------------------|--------------------------------------|
| Blade B1                               | Blade B2                             |
| Nodes                                  | Passage                              |
| 118 163                                | 1 251 952                            |
| Elements                               | Passage                              |
| 26 500                                 | 1 190 544                            |
| **Blade B2**                           | **Passage**                          |
| Nodes                                  | 61 892                               |
| 693 676                                |                                      |

Afterwards, to verify the results obtained numerically, they were compared with those obtained theoretically, mostly in the basis of [8], where empirical relations, dependent particularly on velocity triangles concept, were presented.

Elementary energy balance had the following form:

\[ dl_i - dq = d_i_{tot}, \] (5)

where \( l_i \) is the internal work done on gas J/kg, \( q \) is the heat transferred to casing J/kg.

Discrepancy between theory and presented results was no higher than 5%, as shown in table 3. It includes the values of such parameters as the static and total pressure, static and total temperature and density at the outlet area.

| Table 3. Discrepancy of results for purposes of verification, theoretical calculation based on [8]. |
|-----------------------------------------------|-----------------------------------------------|
| Static pressure, Pa                        | Theory | Numerical analysis | Conformity [%] |
| 133 264                                    | 127 707 | 95.8               |
| Total pressure, Pa                         | 158 730 | 149 804            | 94.4           |
| Static temperature, K                      | 316.2   | 314.9              | 99.6           |
| Total temperature, K                       | 332.4   | 329.4              | 99.1           |
| Density, kg/m³                             | 1.46    | 1.41               | 96.6           |

As it can be seen, the pressure values were slightly less consistent than thermal ones, however since the attention was paid to the heat transfer processes, such conformity was acceptable. Moreover theory was mostly connected with ideal processes.

4. Results and discussion

4.1. Temperature distribution along blades

Three calculations were done, at first B1 and B2 were checked with the same rotational speeds, then that speed was doubled for the second design. Results are presented in table 4.

| Table 4. Results of blade temperature and absorbed heat calculations. |
|---------------------------------------------------------------|
| Outlet temperature in passage, K                            | B1, 16000 rpm | B2, 16000 rpm | B2, 32000 rpm |
| 329.4                                                       | 311.2         | 388.9         |
| Mean temperature of blade, K                                | 307.3         | 293.3         | 307.8         |
| Absorbed heat, J                                            | 262.2         | 85.9          | 320.8         |

In the first analyzed passage, with inlet temperature value of 288 K, blade temperature varied from 297.9 K to 319.6 K, as it is shown in figures 5 and 6. Distribution of temperature along the blade was almost linear. Both sides of the blade had very similar temperature field. It was caused by the fact, that the blade shape made the flow streams acting close to two dimensional movement – without noticeable eddies and secondary patterns. For the second case, with centrifugal blade and 16 000 rotations per minute, blades temperature varied from 288 K to 301 K. Temperature distribution was again nearly linear, as it is shown in figures 7 and 8. Temperature rise in the blade had value of only
5.3 K (difference between the initial and the mean temperature of blade). In the second analysed passage rotating with doubled speed, inlet temperature remained the same, 288 K. However, blade temperature varied from 287 K to 347.7 K (see figures 9 and 10). Due to three-dimensional shape of the blade, temperature distribution was not regular. Especially in the lower part of the blade, the visible changes of temperature were mostly caused by the turbulence occurrence. This effect would be explained in the next subsection.

Figure 5. B1 temperature field, left side, 16000 rpm.

Figure 6. B1 temperature field, right side, 16000 rpm.

Figure 7. B2 temperature field, left side, 16000 rpm.

Figure 8. B2 temperature field, right side, 16000 rpm.

Figure 9. B2 temperature field, left side, 32000 rpm.

Figure 10. B2 temperature field, right side, 32000 rpm.
4.2. Heat transfer between fluid and blades

The main purpose of research was to define the gas heat losses that occur in the passages due to the blade heating. Authors wanted to investigate, what would be the enthalpy loss in the outlet area when blade heating was included in the model, with doubled rotational speed. Amount of heat absorbed by the blade was calculated using standard equations for such case [9], that is an integral:

\[ Q = \int \rho V c dT, \]

where \( Q \) is the absorbed heat, \( J \), \( \rho \) is the blade density, \( \text{kg/m}^3 \), \( V \) is the volume of the blade, \( \text{m}^3 \) and \( c \) is its specific heat, \( J/(\text{kg} \cdot \text{K}) \). Mean temperature of both designed blades was necessary to determine the absorbed heat. Table 4 presents the calculated values. They are very low, because only small blade temperature rise occurred for all analyzed cases. Temperature distribution along passages had to be investigated to check the main reason of such situation. Figures 11, 12 and 13 demonstrate temperature fields within passages, in which the blades are shown. Position of the fields corresponds to the centre of all investigated passages (0.5 of relative height).

![Figure 11. Temperature distribution in B1, 16000 rpm.](image1)

![Figure 12. Temperature distribution in B2, 16000 rpm.](image2)

![Figure 13. Temperature distribution in B2, 32000 rpm.](image3)

![Figure 14. Velocity field in B2, 32000 rpm.](image4)

When comparing temperature distributions in figures 11, 12 and 13, it can be noticed, that B1 gave almost linear temperature field, without any disturbances. The most clear image of non-uniform distribution is shown in figure 13, where the zones of various temperatures appeared in whole passage.
To understand the origin of such complex temperature field, the gas flow structure and velocity field were examined. Figure 14 shows velocity field in the passage with B2 design and rotational speed equal to 32000 rpm. The areas of low and high velocity values corresponds to the areas of lower and higher temperature values shown in figure 13. Resulting velocity gradients suggested occurrence of the vortices. To verify such suggestion, turbulence kinetic energy was calculated.

4.3. Turbulence kinetic energy (TKE)

TKE, which is the kinetic energy per unit mass of the turbulent fluctuations can be defined as [7]:

$$k = \frac{1}{2}(u_x'^2 + u_y'^2 + u_z'^2)$$  \hspace{1cm} (7)

where $u_x'$, $u_y'$, $u_z'$ are the velocity fluctuations of the flow in $x$, $y$, $z$ directions, respectively.

Figure 15. TKE, B2, 16000 rpm.

Figure 16. TKE, B2, 32000 rpm.

Figure 17. Streamlines along the B2 passage with rotational speed value of 32000 rpm.

Figure 15 presents distribution of turbulence kinetic energy in the second design passage for lower value of rotational speed, figure 16 for higher values. Significant increase of TKE value can be seen. For higher speed (figure 16), very irregular flow can be noticed. The shape of B2 was not optimized yet, because the main goal was to investigate heat absorption in the blades for different rotational velocities and constant inlet parameters. In the consequence such effects were expected and possible. With centrifugal design B2, two main curvature changes were proposed – one along the whole blade, second closer to the inlet area – that might be the reason of flow separation and existence of eddies, visible in the lower right part of figures 14 and 16. Corresponding streamlines distribution is exhibited.
in figure 17. Flow disturbances are clearly visible in highly turbulent areas. Further research on the shape of blades needs to be undertaken to avoid such effects. However, due to existence of such disturbances, actual temperature of the flow in the near blade area was lower in comparison to maximal temperature (see table 4). Flow separation led to decrease in absolute velocity values and as result in temperature values, which affected the blade temperature.

4.4. Impact on efficiency

Result of heat transfer to the blades in all presented cases led to the conclusion, that it can be omitted. With computed outlet total enthalpy (sum of static enthalpy and kinetic energy of the flow, with inclusion of TKE) values of $10^4$ J order, calculated losses did not exceed 0.7% (table 5). Isentropic efficiency of compressor, which is the relation of its work in reversible process to the work in actual process [8], was also calculated. Values in third row are the ratios of isentropic efficiency with and without heat transfer to the blade included.

Table 5. Results of blade temperature and absorbed heat calculations.

|                  | B1, 16000 rpm | B2, 16000 rpm | B2, 32000 rpm |
|------------------|---------------|---------------|---------------|
| Heat absorbed in blade, J | 262.2         | 85.9          | 320.8         |
| Outlet total enthalpy, J     | 36713.3       | 13136.3       | 91096.6       |
| Relative difference in isentropic efficiency, % | 0.61          | 0.52          | 0.23          |

5. Summary

Radial compressor blade heating with different rotational velocities and constant inlet conditions was investigated. Even though the problem is widely known, the variety of possible blade designs make it still an important issue. Problem was solved using three dimensional, compressible solver, with advanced turbulence modeling. The results revealed almost negligible values of heat dissipation in all examined cases. It was mostly caused by not ideal velocity distribution along the second passage – existence of eddies led to reduction of temperature around the blade. Higher outflow temperatures would make the heat absorption rate higher as well. Summarizing the results described above – proper blade design is crucial for maintaining appropriate flow, especially in the light of outlet enthalpy changes. On one hand the heat losses did not increase significantly – what can be treated as the advantage, on the other hand flow properties did not fulfilled optimal conditions for compressors. Further research, however, can lead to the development of models, in which flow structure would be satisfactory and at the same time would not cause considerable heat losses to the blades.

Acknowledgements

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. 11.11.210.312).

References

[1] Vinayaka N, Niloptal B and Ajay Kumar B S 2015 Procedia Engineering 127 947-954
[2] Han J, Dutta S, Ekkad S 2012 Gas Turbine Heat Transfer and Cooling Technology (CRC Press)
[3] Stosic N 2015 International Journal of Heat and Fluid Flow 51 285-297
[4] Wu J, Hu J, Chen A, Mei P, Zhou X and Chen Z 2016 Appl Thermal Engineering 95 365-373
[5] Kura T 2015 Numerical analysis of velocity and temperature fields of compressible fluid flowing through a rotor (MSc Thesis, in Polish)
[6] ANSYS® CFX-Solver Theory Guide 2013 (ANSYS, Inc.: Cecil Township, PA, USA)
[7] Wilcox D 2006 Turbulence Modeling for CFD (DCW Industries, Inc.)
[8] Witkowski A 2013 Sprężarki wirmikowe Teoria, konstrukcja, eksploatacja (Wyd. Pol. Śląskiej) (in Polish)
[9] Cengel J and Ghajar A 2015 Heat and Mass Transfer: Fundamentals and Applications (McGraw-Hill)