

## Approach to novel design of CO<sub>2</sub> based centrifugal compressor

Tomasz Kura<sup>1,a</sup> and Elżbieta Fornalik-Wajs<sup>1</sup>

<sup>1</sup>AGH University of Science and Technology, Department of Fundamental Research in Energy Engineering, al. Mickiewicza 30, 30-059 Cracow, Poland

**Abstract.** Even though turbomachinery design issues have been investigated almost since the beginning of engineering, its optimization process is still important. With the development of refrigeration devices and ORC based distributed generation facilities, a need for efficient and low-energy compressors and turbines became even more demanding. Such machines working with typical fluid, like air, are well described, but there is a room regarding the fluids like CO<sub>2</sub>, vapour of organic fluids, etc. The main objective of present studies is to propose a numerical model of the centrifugal compressor, with CO<sub>2</sub> as the working fluid. Such unit may be a part of refrigeration cycle. Commonly, the scroll or piston compressors are used in such cases, however some discussed disadvantages show that the novel designs should be looked for. Properly designed centrifugal compressors can have higher efficiency than the presently used. Three dimensional analyses of proposed geometries were conducted – using a model including heat, mass and momentum conservation laws as well as ideal gas law. Verification of the proposed mesh and results was performed in the basis of values obtained using theoretical and empirical equations. With about 700 000 control volumes in the validated model, error of the results was no higher than 5%, with only about 1% in regards to the thermal parameters. Two design proposals were analysed, with performance maps as the main comparison factor. Apart from performance characteristics, the pressure and velocity fields were presented, showing the process of flow structure optimization. The main goal was to reduce negative effects of pressure and velocity gradients on the performance. Proposed precursory design might be a good starting point for further development of compressors. The results of numerical analysis were promising and shows the possibility of proposed design usage in practical applications, however to obtain deep understanding of the problem, the experimental analysis should be also done.

### 1 Introduction

One of the most important issues of modern world is to develop sustainable energy systems, that would work in spite of circumstances outside the system. Heat pumps, solar collectors, photovoltaic systems – they all are parts of such trials. More and more advanced usage of low-temperature heat sources became one of the breakthroughs that led to development of distributed energy generation. However it became clear for scientists and engineers, that water as the working medium, present in almost all big-scale energy systems, would not be suitable for many proposed solutions. Commonly, when it comes to small generation or refrigeration issues, there can be just one or only a few fluids, that suit particular design of machine. Best way is to construct operating maps for analysed devices, to achieve full image of the problem and the best solution [1]. Scroll or piston compressors are being used in such cases more often, but due to some slight drawbacks, such as worse steering sensibility or need to be oiled [2], researchers looked for the possibility of centrifugal compressors usage.

Not only type of machine, but also working fluid were a matter of investigation. For proper selection of the working fluid thermo-physical properties

are considered, but chemical stability in desired temperature range or safety should also be taken into account. These issues are important from the application point of view, because small-scale systems can be installed directly in households [3]. In ORC systems, organic fluids are used, namely isobutene, n-pentane aromatic hydrocarbons or similar. All of them have properties, that make them more efficient in low temperature ranges than water, especially in the superheating area of phase diagrams. These properties play major role in the design of expanders [4], but since ORC, heat pump and similar systems consist also of compressors, proper desing of them is also very important.

In the paper, analysis of radial compressor's rotor, with various design proposals, is presented. Carbon dioxide was chosen as the working fluid. Couple of articles or dissertations on the implementation of that fluid in turbomachinery were published in previous years, for example [5, 6]. Monje in [5] presented comprehensive analysis of particular model of CO<sub>2</sub> compressor designed for supercritical Brayton cycle. Only numerical results were included based on one case. For purposes of that work, Monje prepared literature review – and stated, basing on it, the necessity of further exploration, as CO<sub>2</sub> behaviour in turbomachinery is promising. Therefore there

<sup>a</sup> Corresponding author: kura@agh.edu.pl

is still a need for further compressor development and it becomes an aim of this paper.

## 2 Mathematical model

Analysed flow was three-dimensional, stationary and compressible. The conservation laws of mass, momentum and energy coupled with ideal gas law were applied in the model. Continuity equation was considered in following form [7]:

$$\frac{\partial}{\partial x_j}(\rho U_j) = 0, \quad (1)$$

where  $\rho$  is the fluid density,  $\text{kg/m}^3$ ,  $U$  is its velocity,  $\text{m/s}$ . Momentum equation was represented by [7]:

$$\frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \overline{u_i u_j}) + S_{Cor} + S_{cfs} \quad (2)$$

where  $p$  is the pressure,  $\text{Pa}$ ,  $\tau$  is the molecular stress tensor,  $S_{Cor}$  is the source term accounting for the effect of Coriolis force and  $S_{cfs}$  is the centrifugal force source. Energy equation was applied in the form [7]:

$$\frac{\partial}{\partial x_j}(\rho i_{tot} U_j) = \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} - \rho \overline{u_j i} \right), \quad (3)$$

where  $T$  is the temperature,  $\text{K}$ ,  $\lambda$  is the fluid thermal conductivity,  $\text{W/(m}\cdot\text{K)}$ ,  $i_{tot}$  is the specific total enthalpy,  $\text{J/kg}$ ,  $i$  is the specific static enthalpy,  $\text{J/kg}$  and  $k$  is the turbulence kinetic energy,  $\text{J/kg}$ . ANSYS 14.5 software, particularly CFX and Turbogrid modules, were chosen to achieve results. CFX uses coupled solver to conduct the simulation, which main advantage is the computing speed, but significant memory resources are needed, because all equations are being determined at the same time. High resolution upwind advection scheme was applied, second order, to avoid numerical diffusion and therefore to get more certain results. Also, SST  $k-\omega$  turbulence model was chosen, due to some advantages [7] over other two-equations models. Comparing this model with typical  $k-\epsilon$  or  $k-\omega$  models, SST  $k-\omega$ , being the combination of both, gives more reliable results. It is suggested for simulation of high pressure gradients or flow separation areas [7] – those phenomena occur in turbomachinery flows, with high rotational speeds. Moreover SST  $k-\omega$  model allows obtaining feasible near-wall results. Research was based on the comparison of rotor's blade performance when compressing air and  $\text{CO}_2$ . Therefore, two working fluids were used in the model. The air had density value of  $1.225 \text{ kg/m}^3$ , the thermal conductivity –  $0.0242 \text{ W/m}\cdot\text{K}$ , the viscosity value of  $1.79\text{E-}05 \text{ kg/(m}\cdot\text{s)}$  and the specific heat –  $1006.4 \text{ J/(kg}\cdot\text{K)}$  in standard conditions, while  $\text{CO}_2$  had density of  $1.7878 \text{ kg/m}^3$ , the thermal conductivity –  $0.0145 \text{ W/m}\cdot\text{K}$ , the viscosity value of  $1.37\text{E-}05 \text{ kg/(m}\cdot\text{s)}$  and the specific heat –  $840.37 \text{ J/(kg}\cdot\text{K)}$  [7].

Performance maps were constructed for varying rotational speed of the rotor. Speeds were chosen in the basis of calculation procedure presented in [8], to avoid off-design conditions and non-reliable results. Rotation occurred around the axis shown in Figure 1, together with an indicated direction. Initial temperature in the whole passage as well as the blade was  $288 \text{ K}$ . Also inlet pressure

remained constant and was equal to  $1 \text{ bar}$ . For outlet boundary condition, “supersonic” type was chosen, as the proper one for simulations with possible compressibility effect [7]. As all analysed flows had the Mach number at the inlet above  $0.3$ , that effect was expected. That boundary condition demand no other parameters to be defined.

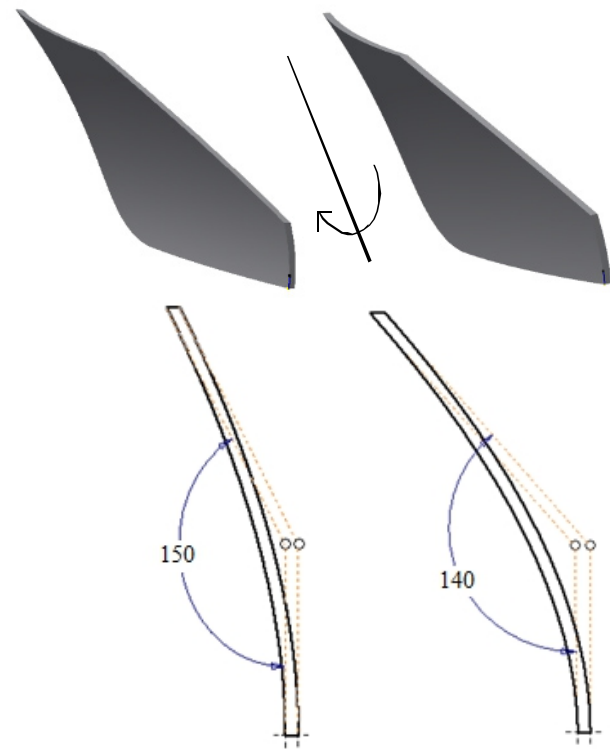
All implemented inlet boundary conditions are presented in Table 1.

**Table 1.** Boundary conditions at the inlet.

Rotational speed, $10^3 \text{ rpm}$	26	28	30	32	34	40
Temperature, K	288					
Pressure, bar	1					

## 3 Studied cases and applied solutions

Two different blades designs were proposed. Both of them based on the data from literature [8] but modified to fit the needs of presented approach.



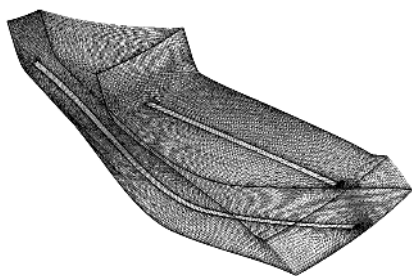
**Figure 1.** Blade 1 (B1) – up left, and blade 2 (B2) – up right. Visible angles of curvature and schematic view of blades shape, B1 – down left, B2 – down right.

One passage per simulation was discretized since the software is capable to interpolate the results for whole device. It led to reduction of computer resources necessary for analysis. Figure 1 presents schematic view of studied blades. Rotors consisted of 15 blades per machine. Both blades had the same length, around  $10 \text{ cm}$ , and height, varying from  $5$  to  $2 \text{ cm}$ . The only difference between two blades was the angle of curvature along the blade height. For blade 1 (B1) it was  $150^\circ$  and for blade 2 (B2)  $140^\circ$ . Mentioned angles for blades are also presented in Figure 1. Ansys software is based on finite elements method. Meshing and Turbogrid modules were used to generate mesh. Hexahedral elements were generated in the majority of the model. Their number was high enough to maintain

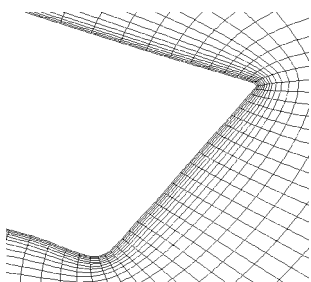
elements shape close to cube in areas of high curvature, which is important for gaining proper results and avoiding artificial gradients of calculated parameters. In Table 2, number of nodes and elements for both types of blades is listed. Figure 2 shows generated mesh, for blade B1. Refined mesh near the blade can be noticed, as shown in Figure 3. Setting an approximate Reynolds number value to be  $0.5E07$ ,  $y^+$  values did not exceed 2.

**Table 2.** Number of nodes and finite elements.

Blade	B1	B2
Nodes	739 860	699 825
Elements	694 235	693 676



**Figure 2.** Mesh for the passage with B1 blade. Direction of rotation.



**Figure 3.** Near-blade mesh refinement.

Verification of mesh was conducted through the comparison with theoretically obtained results, according to methods suggested in [8]. In Table 3, conformity of results is presented. Values in the second column are the differences between theory and numerical analysis with the proposed mesh. Coherence was higher than 96%, therefore the mesh was considered fine enough to continue the studies. Increase in the number of elements did not change the conformity significantly, therefore around 700 000 elements was kept for the whole analysis. Static and total pressure, used for verification purposes, came directly from Bernoulli's equation. Total value is the sum of its dynamic and static part. Temperature was evaluated basing on the same manner but also with use of equation of state.

**Table 3.** Verification of the proposed mesh.

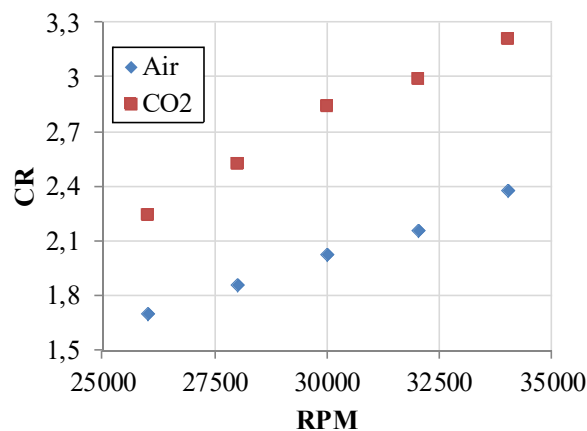
Variable	Coherence, %
Static pressure	95.8
Total pressure	94.4
Static temperature	99.6
Total temperature	99.1
Density	96.6

### 4 Results and discussion

The main objective of presented studies was to propose numerical model of CO<sub>2</sub> compressor, which would be competitive to other available devices. The compression ratio (CR) can be an indicating factor for comparison between particular units since it is one of the main parameters describing device performance. It is defined as:

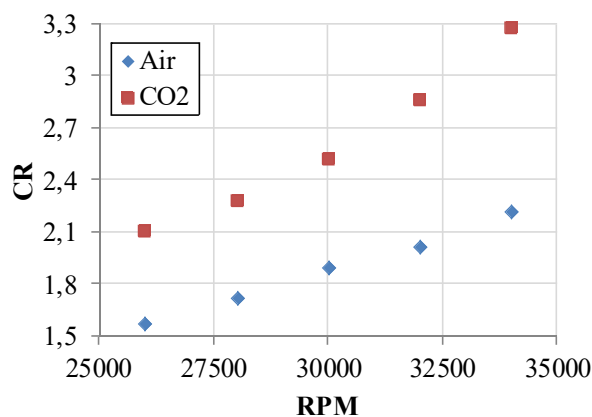
$$CR = \frac{p_{t,out}}{p_{t,in}} \tag{4}$$

where  $p_{t,in}$  is the total pressure at the inlet and  $p_{t,out}$  is the total pressure at the outlet. Both pressures were calculated using area integral. Total pressure is a sum of static and dynamic pressures and needs to be calculated in the cases with high speeds of the flow, where dynamic parameters have significant values.



**Figure 4.** Compression ratio vs rotational speed, B1 design.

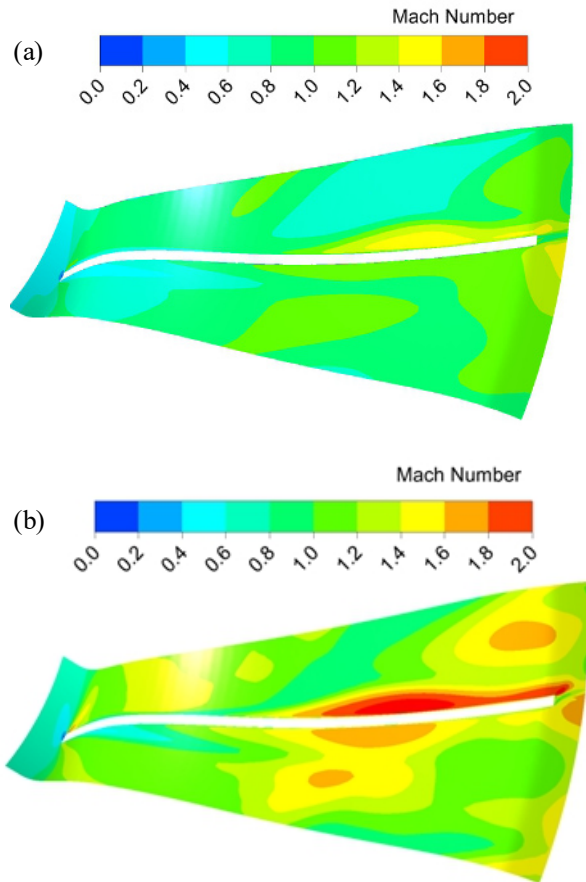
The comparison of CR versus rotational speed for two fluids and blades B1 and B2 is presented in Figures 4 and 5, respectively. In both blade cases CR was higher for CO<sub>2</sub> of about 25 to 30%. Moreover the difference was increasing with an increase in rotational speed. Higher values of CR could be noticed for B1 design in almost whole studied range, except the highest analysed rotational speed for which B2 could generate the highest value of CR.



**Figure 5.** Compression ratio vs. rotational speed, B2 design.

As previously mentioned, compressibility effect was expected. To confirm that, Mach number, defined as the ratio of local fluid velocity to the local speed

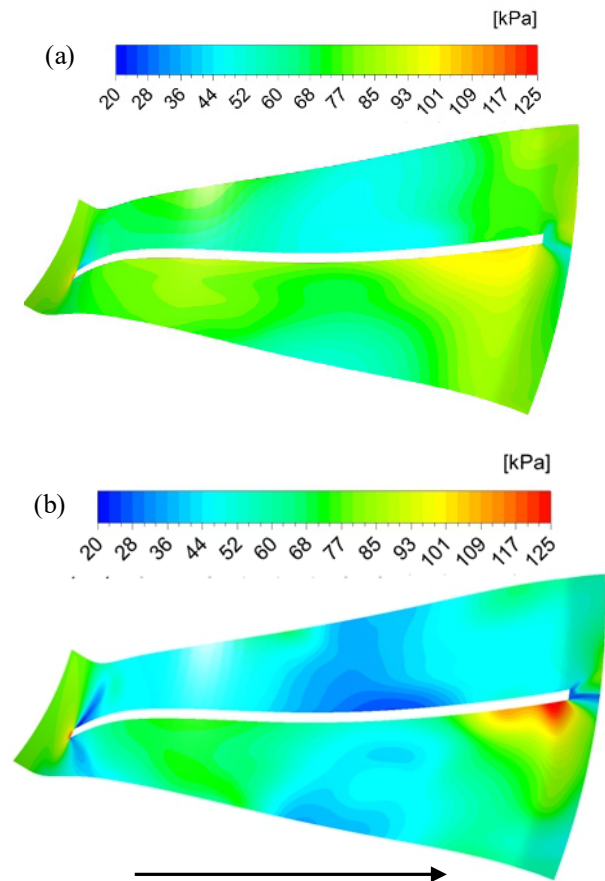
of sound, was calculated in the passages. Figure 6 presents its values for the case with 30 000 rpm and B1 design. Mach number was higher than 0.3 in almost whole passage, what confirmed the usage of supersonic boundary conditions, mentioned in previous section with mathematical model.



**Figure 6.** Mach number, B1 design, 30 000 rpm, (a) air, (b) CO<sub>2</sub>.

Moreover, its higher values for carbon dioxide could be noticed – flow of this fluid was more dynamic. To check, whether the blades were designed properly, pressure and velocity fields were analysed. In all Figures, the plane in the middle of the blade height was chosen to show variables distributions. Presented views are giving the whole passages view along the flow direction, indicated by the black arrow. Figures 7 and 8 present pressure and velocity fields of air and CO<sub>2</sub>, for B1 design, at 30000 rpm. The lowest pressure was observed for CO<sub>2</sub>, even though the flow of both fluids was highly irregular. Higher velocity could be noticed for CO<sub>2</sub> case. Figures 9 and 10 present the same variables, but for B2 design, and only for CO<sub>2</sub> fluid, for comparison purposes. Analysing B1 and B2 designs (Figures 7, 8, 9, 10), the biggest influence of the shape on the pressure could be observed. Red arrow in Figure 9 indicates low pressure area in B2 blade’s passage, that may be one of the reasons of lower CR values for that design. In Figures 11 and 12, CO<sub>2</sub> pressure and velocity fields for B1 and B2 case and 34000 rpm are presented. With visible irregularity of pressure distribution, it can be stated, that due to it, CR value for B1 is lower than in the case of B2. It can suggest, that B2 design is more suitable for higher rotational speed.

Table 4 presents isentropic efficiencies calculated for all simulations, which are counted as ratio of actual work in the machine to the corresponding work in reversible process (see Equation 5). Even though values for air were higher, it did not correspond with CR values. Explanation of this situation is connected with power needed to compress fluids.



**Figure 7.** Pressure fields, B1 design, 30 000 rpm, (a) air, (b) CO<sub>2</sub>.

**Table 4.** Isentropic efficiencies of all examined cases.

rpm		26000	28000	30000	32000	34000
B1	air	77	76	76	75	73
	CO <sub>2</sub>	75	72	70	68	66
B2	air	70	70	69	68	69
	CO <sub>2</sub>	69	68	66	65	64

For CO<sub>2</sub> cases more input power (calculated basing on Equation 6) was necessary, as shown in Figure 13, mostly due to its higher density. That impacted the values of isentropic efficiency, which is defined as:

$$\eta = \frac{H_{2s} - H_1}{H_2 - H_1}, \quad (5)$$

where  $H$  is the total enthalpy,  $J$ . Indexes mean: 1 – the inlet, 2 – the outlet,  $s$  – the total enthalpy value in isentropic process.

Input power,  $W$ , was defined for steady process, as:

$$IP = H_2 - H_1, \quad (6)$$

Work is connected with fluids enthalpy changes more than pressure, what explains the discrepancy between CR and efficiency changes.

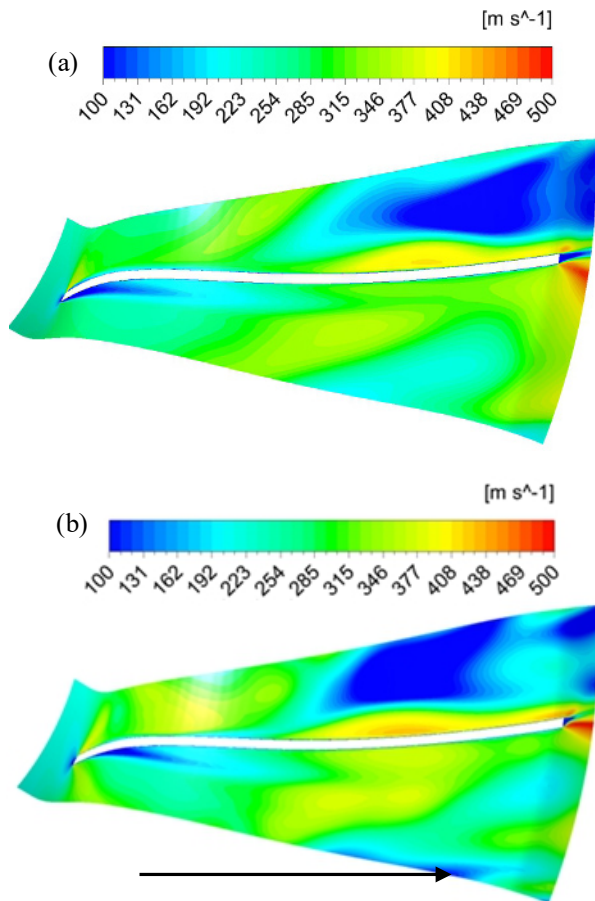


Figure 8. Velocity fields, B1 design, 30 000 rpm, (a) air, (b) CO<sub>2</sub>.

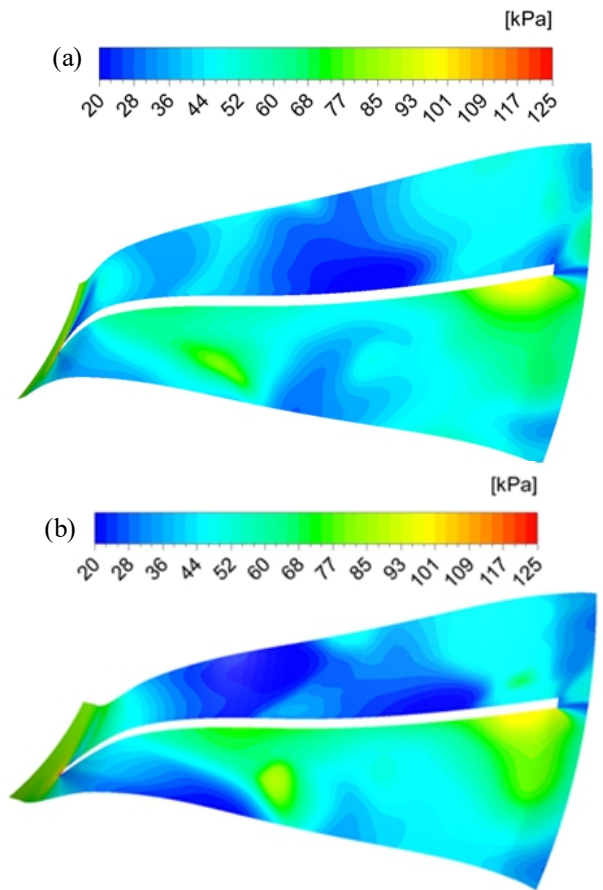


Figure 11. Pressure fields, 34 000 rpm, CO<sub>2</sub>, (a) B1, (b) B2.

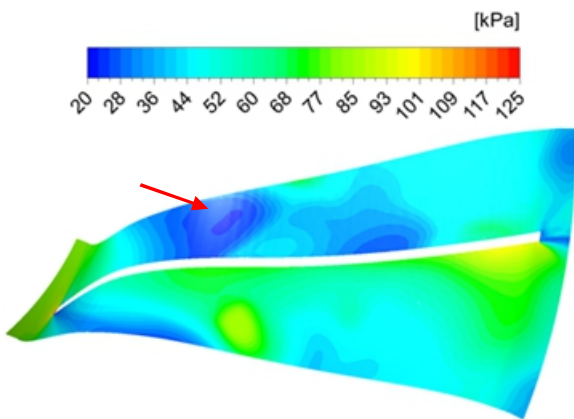


Figure 9. Pressure field, B2 design, 30 000 rpm, CO<sub>2</sub>.

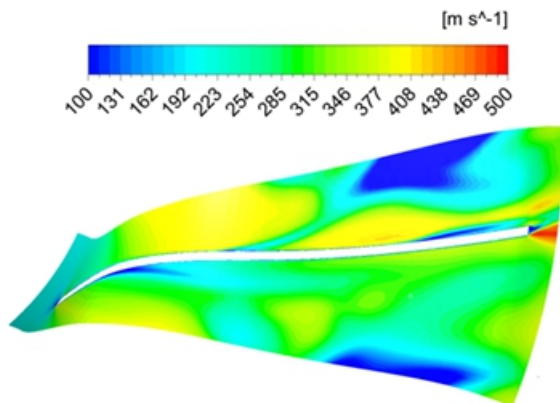


Figure 10. Velocity field, B2 design, 30 000 rpm, CO<sub>2</sub>.

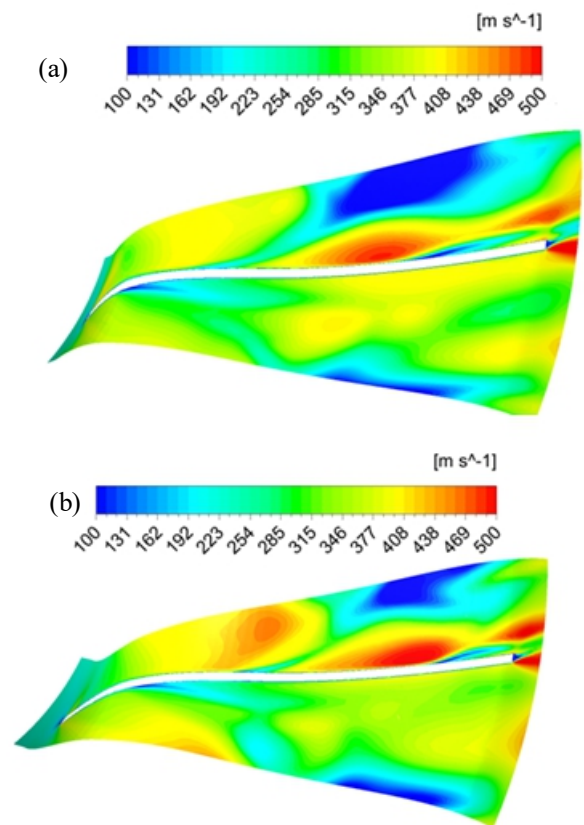


Figure 12. Velocity fields, 34 000 rpm, CO<sub>2</sub>, (a) B1, (b) B2.

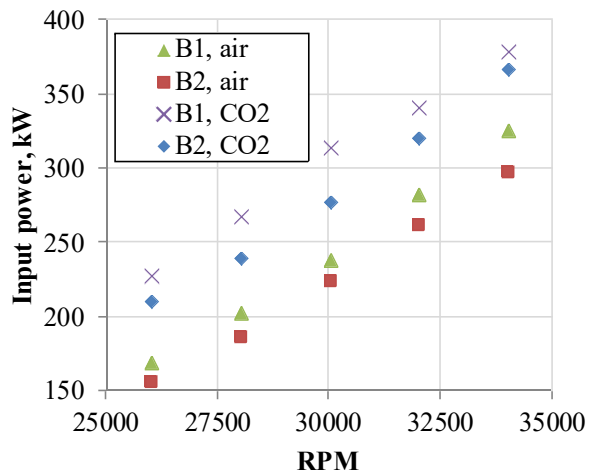


Figure 13. Input power comparison.

## 5 Summary

Radial compressor's blade designs were investigated, with various rotational speeds, to check the difference in the flow behaviour between two fluids – CO<sub>2</sub> and air. As CO<sub>2</sub> is thought to be one of the possible fluids to be used in ORC or refrigeration systems, its behaviour should be precisely investigated. For proposed compressors, CO<sub>2</sub> turned out to give more effective pressure output in whole examined range.

In overall, B1 design turned out to be better, with higher CR, which was the main factor for comparison. The only disagreement with that relation occurred for CO<sub>2</sub> compression at 34000 rpm – possible answer for that situation is, that the required power input for blade B2 is lower than for B1. Although it has to be investigated, it did not impacted main conclusion and was left for future research.

Further examination needs to be undertaken to develop adequate model of CO<sub>2</sub> compressors. The experimental data would be necessary for the proper image. However it may be concluded, that presented results construct the basis for further development as some important issues characterizing the flow have been described and analysed.

## Acknowledgements

The present work was financially supported by the Polish Ministry of Science and Higher Education (Grant AGH No. 15.11.210.344) and by AGH UST Vice-Rector for Student Affairs, Professor Anna Siwik. Present work was also supported by Academic Computer Centre CYFRONET AGH, under award no. MNiSW/IBM\_BC\_HS21/AGH/021/2015.

## References

1. S. Quilini, S. Declaye, A. Legros, L. Guillaume, V. Lemort, *Int. Compressor Eng. Conf. at Purdue*, **2012**, 1546 (2012)
2. T. Turunen-Saaresti, P. Roytta, J. Honkatukia, J. Backman, *Int. J. of Refrigeration*, **33**, 1152 (2010)
3. H. Chen, D. Yogi Goswami, E. K. Stefanakos, *Ren. and Sust. En. Rev*, **14**, 3059 (2010)
4. L. Moroz, C. Kuo, O. Guriev, Y. Li, B. Frolov, *Proceedings of the ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*, **5A** (2013)

5. B. Monje, *Design of supercritical carbon dioxide centrifugal compressors* (PhD thesis, Universidad de Sevilla, 2014)
6. B. Monje, D. Sanchez, R. Chacartegui, *Proceedings of the ASME Turbo Expo*, **3**, 405 (2012)
7. *ANSYS® CFX-Solver Theory Guide 2013* (ANSYS Inc., Cecil Township, 2013)
8. A. Witkowski, *Sprezarki wirnikowe Teoria, konstrukcja, eksploatacja* (wyd. Politechniki Slaskiej, Gliwice, 2013)