Applied and Computational Mechanics 10 (2016) 111–122

Enhancement of centrifugal compressor stable operation using an internal recirculation channel

M. Růžička[a,*]

[a] Aerospace Technology Division, Department of Engineering Analysis, První brněnská strojírna Velká Bíteš, a. s., Vlkovská 279, 595 12 Velká Bíteš, Czech Republic

Received 6 June 2016; received in revised form 6 October 2016

Abstract

Centrifugal compressors with a high pressure ratio are widely used in small aircraft turbine engines and turbocharges. At high rotational speeds they have a narrow stable operating region and the commonly used impellers with back swept blades are not able to ensure requested stability. In order to achieve a wider stable operating region, some other anti-surge measures can be used such as an Internal Recirculation Channel (IRC) located in the inlet area of the compressor impeller. This paper analyses the influence of various IRC inlet slot geometries on air flow parameters. The flow velocity profiles downstream of the IRC outlet slot are evaluated and the influence of various channel configurations on compressor performance map is shown. Results from experiments as well as CFD simulations enable us to analyse the airflow in IRC and find out its most suitable geometry.

Keywords: centrifugal compressor, recirculation channel, experiment, velocity profiles, CFD simulations

1. Introduction

The most important behaviour of a centrifugal compressor is the wide range of its stable operation. For the extension of the operational region, there is a number of active and passive methods. Active devices include especially variable guide vanes in the compressor inlet or variable diffuser vanes. As passive devices, one can use differently shaped channels in the compressor casing, called Internal Recirculation Channels (IRC), in which the air flows between the inducer and compressor inlet, Fig. 1. These passive devices also include a supply of air into the area of vane-less diffuser.

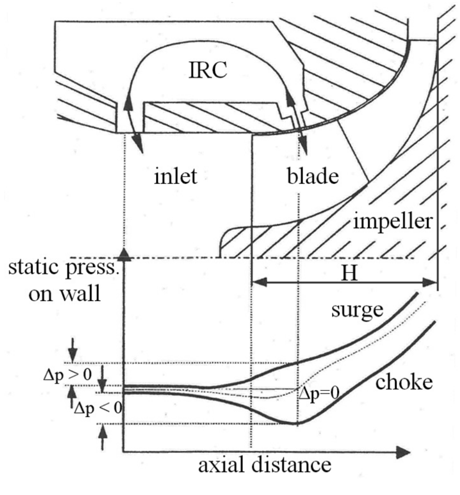


Fig. 1. IRC principle and layout, [6]

In the past, the active methods were more widely used than the passive ones, but complicated design, weight increase and additional losses in the case of vanes at compressor inlet led to the fact that the passive methods are currently more preferred. They do not significantly complicate the design and bring positive effect in the meaning of compressor stable operation enhancement.

*Corresponding author. Tel.: +420 566 822 949, e-mail: ruzicka.m@pbsvb.cz.

Nomenclature

c	velocity [$\text{m} \cdot \text{s}^{-1}$]	Indices, abbreviations and special signs:	
dp	pressure difference [Pa]	a	axial
H	axial impeller blade length [mm]	a, avg	axial, average
h_1/h_2	distance of inlet/outlet slot [mm]	cT	compressor total
n	revolution speed [%, RPM]	e	outer (radius)
p	pressure [Pa]	nom	nominal
Q	mass flow [$\text{kg} \cdot \text{s}^{-1}$]	u	tangential
r	radius at impeller inlet [mm]	$\overline{\text{"symbol"}}$	dimensionless value
s_1/s_2	inlet/outlet slot width [mm]	CFD	Computational Fluid Dynamics
z	axial coordinate [mm]	ch. e.	chamfer edge
η	isoetropic efficiency [%]	IRC	Internal Recirculation Channel
π	pressure ratio [–]	ISA	International Standard Atmosphere
		s. e.	sharp edge

The effort to extend operational margin of the compressor performance map is mainly carried out by forming compressor impellers [3,15]. Generally speaking, the performance maps of centrifugal compressors with impellers having backswept blades have wider operational region than the ones with straight blades impellers. However, often the backswept blades are not enough and it is necessary to look for further measures to increase the compressor stability. Among them, the design of internal recirculation channel appears as the most promising option.

When a compressor with IRC operates close to the surge, a part of the compressed air flows back from the inducer to the inlet duct. The air supplied from the IRC influences the flow field in front of the impeller in order to suppress stall on the inducer blades. Near to the compressor choking, the air in IRC flows in opposite direction, i.e. from inlet duct to the inducer, Fig. 1. The IRC does not complicate the compressor design, is easy to manufacture and can partially improve the compressor performance map.

The IRC was used in compressors with a higher pressure ratio (from 4.2 to 5.7), for example presented in [5,6,11,13,14], as well as in compressors with a relatively low pressure ratio (from 2.5 to 3.2), see [10] and [15]. All authors noted a positive effect of IRC near surge, where it shifts the surge line towards lower mass flows. On the other hand, there is also a negative effect of IRC in the form of a slight decrease of compressor total pressure ratio (π_{cT}) and decrease of efficiency (η_{cT}) in comparison with the same compressor stage but without IRC, Fig. 2. It is not a rule that the negative effect can be seen in the whole performance map, but can be noted only in its certain parts. The IRC efficiency is influenced by its design. The shape of the

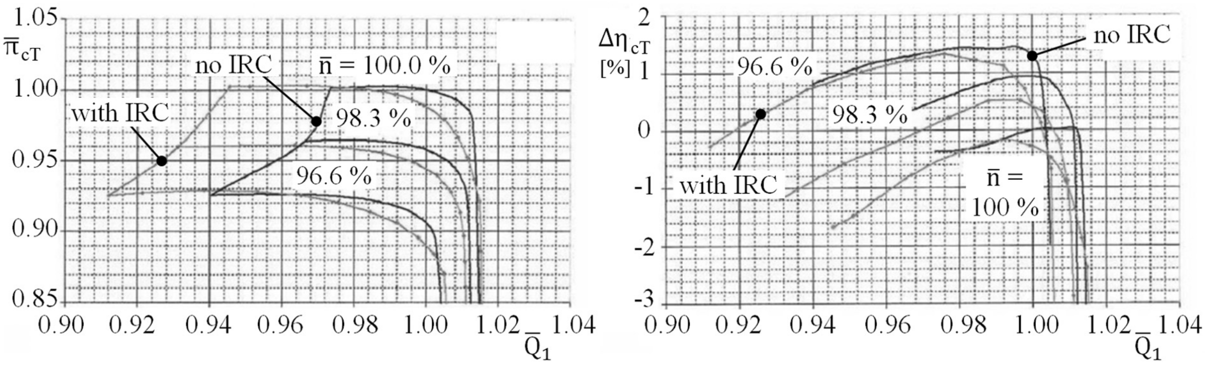


Fig. 2. Comparison of centrifugal compressor maps without and with IRC, [2], where: $\bar{\pi}_{cT} = \pi_{cT}/\pi_{cT,nom}$, $\bar{n} = n/n_{nom}$, $\bar{Q}_1 = Q_1/Q_{1,nom}$, and $\Delta\eta_{cT} = \eta_{cT} - \eta_{cT,nom}$

recirculation channel, the width of the inlet and outlet slots and their distance from impeller blades leading edges have to be considered carefully. To achieve maximal IRC air flow, the recirculation channel has to be designed as having minimal pressure losses.

From the above mentioned papers, it is possible to conclude how the IRC influences a performance maps; unfortunately, the papers do not provide any general analysis of flow inside the IRC. Pressure difference along a wall of compressor casing has a fundamental influence on the airflow in IRC; therefore its design is directly connected with a specific compressor stage.

2. Motivation

The present centrifugal compressors, designed and produced in the company První brněnská strojírna Velká Bíteš, a.s. (PBS) for auxiliary power units and turbine engines are able to achieve a pressure ratio of 6. The need to increase performance parameters of products requires an increase of compressor pressure ratio, but with the same stability range. It is not sufficient to do it only with backswept blades, but it is necessary to focus on IRC. This is the reason why extensive research is performed on theoretical and experimental level within the European project called LEMCOTEC (Low Emissions Core-Engine Technologies). In the project, co-funded by the European Commission within the Seventh Framework Programme, many engine producers are involved. PBS together with its strategic partner Výzkumný a zkušební letecký ústav, a.s. (VZLU) cooperates on a task dealing with compressor stable operation enhancement. The work follows the previous research presented for example in [1, 7, 9, 12].

3. Experimental research of the recirculation channel

3.1. Experimental test device

For better understanding of airflow through the recirculation channel, a special test device with IRC model was designed. The IRC was tested in two steps. In the first step, the IRC model was tested (calibrated) on a test stand and in the second step, the IRC will be tested in an experimental engine. The test stand arrangement is shown in Fig. 3.

The IRC model is located at the inlet of the test stand with the jet engine TJ100 at its outlet. The jet engine sucks the air through the IRC model and simulates the main flow. The external pressurised air is supplied into the inlet slot of the recirculation channel, Figs. 3 and 4.

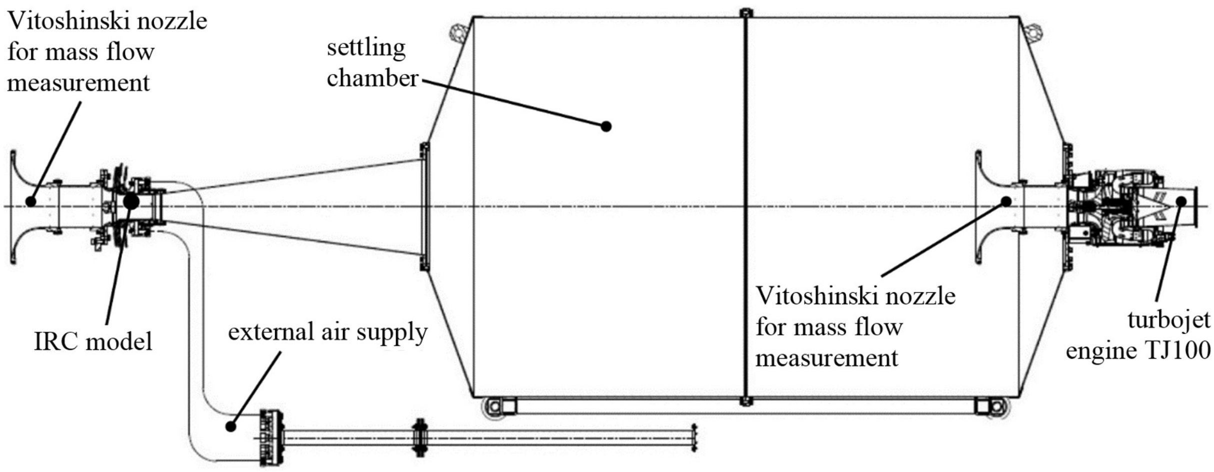


Fig. 3. Layout of the test device for recirculation channel testing, [4]

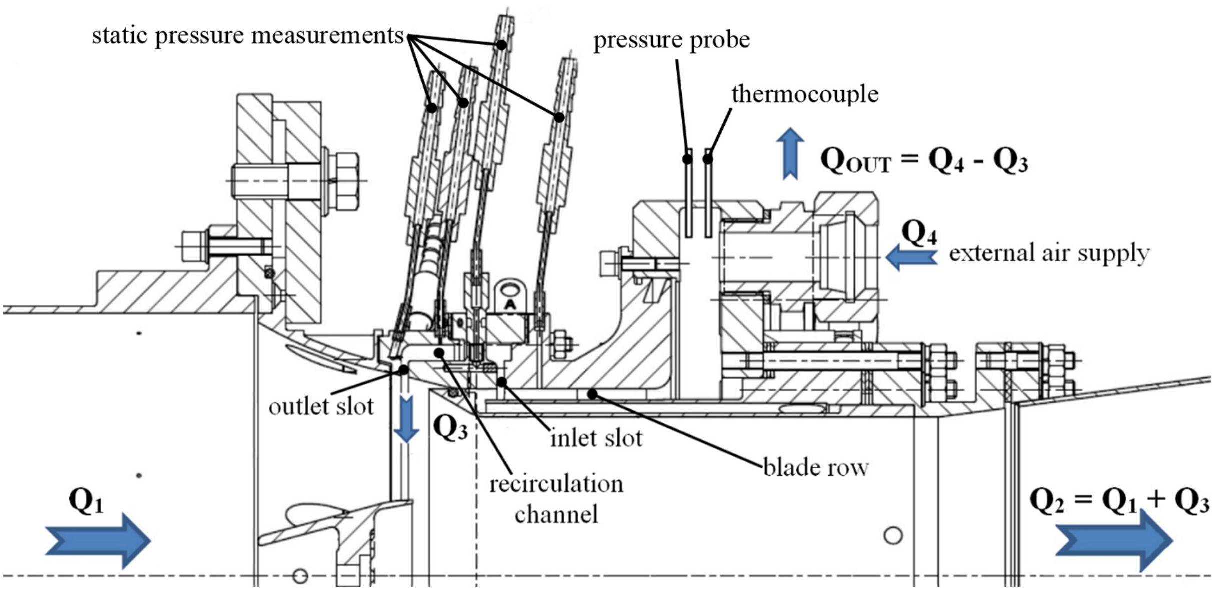


Fig. 4. Detailed view at the tested IRC model with mass flows designation and probes positioning, [4]

The mass flow rate measurement is performed for different Mach numbers at the inlet slot and for various pressure differences between the inlet and outlet slots of the recirculation channel. The mass flow rate is measured by the Vitoshinski nozzle at the test stand inlet and at its outlet (at the turbojet engine TJ100 inlet in the settling chamber).

Static pressure as well as temperature were measured at the IRC inlet and outlet slots and in the recirculation channel. Velocity profiles and flow angles behind the outlet slot were measured by means of a three-hole cylindrical pressure probe. The test device was designed in VZLU according to PBS instructions.

3.2. Configurations of the experimental test device

Two variants of the IRC model test device, which differ in the outlet slots shape, were designed. The first one is inclined in angle $40°$ to the main flow direction, whereas the second is radial (perpendicular) to the main stream. Both variants of the test device enable to change the width of both inlet and outlet slots within a range from 1 to 3 mm. The inlet slot can be used with sharp or chamfer edges.

The layout of IRC model including installed pressure probes and thermocouples is shown in Fig. 4. The data recorded during the measurement were used for the evaluation of pressure losses and Mach numbers inside the channel. The mass flow in IRC (Q_3) was calculated as a difference between the mass flows exiting (Q_2) and entering (Q_1) the model. Because the measurements of all possible configurations would be very time consuming, it was decided to select only some configurations for investigation: inlet slot widths 1, 2 and 3 mm. Outlet slot width 3 mm was constant for all the measured points. The measurements were performed with both vertical and inclined outlet slot. Using the three-hole calibrated pressure probe placed 4 mm downstream of the perpendicular outlet slot, the parameters of flow penetrating from IRC channel into the main flow were measured. The three-hole probe was shifted from shroud to hub of the inlet duct.

3.3. Measurement on the experimental test device

The measurements, performed in VZLU, were focused on the assessment of relative mass flow Q_3/Q_1 as a function of pressure difference dp between the inlet and outlet slots. The results

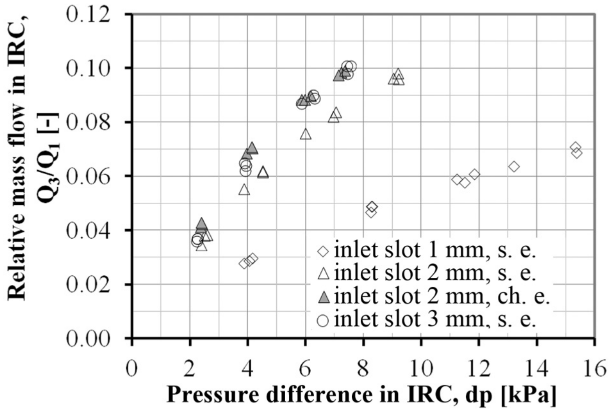


Fig. 5. Dependence of mass flow in IRC on pressure difference measured on the test device for various configurations, s. e. – sharp edge, ch. e. – chamfer edge, [4]

plotted in Fig. 5 show that the inlet slot with 1 mm width gives low mass flow in IRC even at high pressure ratios, whereas in the case of slots with 2 and 3 mm width, the mass flow increased significantly. It should be further noted that the chamfer edges of the inlet slot had an indisputable positive effect on the relative mass flow. The performance of the 2 mm inlet slot with chamfer edges was almost the same as the performance of the 3 mm slot with sharp edges.

The axial and tangential components of absolute velocity measured by the three-hole probe are shown in Fig. 6. The tangential component significantly influences the incidence angle on impeller blades and the work transferred by impeller blades on corresponding radius.

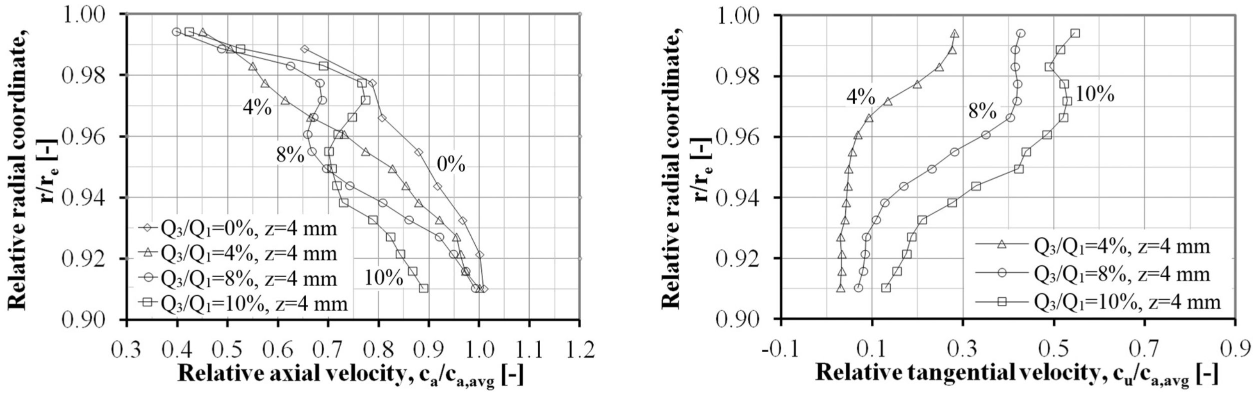


Fig. 6. Profiles of relative axial and tangential velocities measured by the three-hole probe in the IRC model on a plane placed $z = 4$ mm downstream of the outlet slot; inlet slot width was 1.8 mm, [8]

After completing the measurement on the IRC model, the bypass channel of this model was inserted into the compressor of experimental engine. The tests will be focused on demonstrating the influence of IRC on the performance map of the compressor stage.

4. CFD simulation of flow in the recirculation channel

4.1. Geometrical model preparation and mesh generation

The requirement on deeper theoretical understanding of recirculation channels, not only on the experimental level, led to CFD analysis on a three-dimensional geometry. The simulation of direct interaction of rotating impeller blades with the recirculation channel inlets gives us the opportunity to assess the impact of IRC on the work of the compressor. Considerable contribution can be also seen in the possibility to investigate the influence of flow from the outlet slot on velocity profiles and flow angles upstream of the compressor impeller leading edges. The computational geometry consisted of compressor impeller, supplemented by the recirculation channel model. Due to computational time reduction, only an impeller without diffusers was used for simulations. Of course, this simplification will affect the pressure ratio and efficiency, which will be significantly higher compared to the compressor including diffusers. In our study, the omission of diffusers is acceptable as the main objective is not to investigate the exact performance maps of the whole compressor, but just the flow structure inside the IRC and downstream of the outlet slot. The performance maps were evaluated only for the necessity to assess the surge line shift. For this purpose, the simplification is fully sufficient.

The model of the recirculation channel, used for simulation, was designed on the basis of drawings provided by VZLU in order to maintain the same geometry as in the case of the IRC model. Moreover, it is useful to have the possibility to easily and quickly change the inlet and outlet slots configurations between particular simulations. Therefore the geometry of the recirculation channel was designed as variable and enabled a modification of key dimensions in the following ranges:

- inlet slot width from 0 to 3 mm with 0.5 mm step,
- inlet slot position against impeller blade leading edge from 6 to 8.5 mm (depending on actual slot width) with 0.5 mm step,
- either sharp or chamfer edges of the inlet slot,
- outlet slot width from 0 to 19 mm with 1 mm step (in practice, it is usual to use slots with 3 or 4 mm width),
- outlet slot position against impeller blade leading edge from 6 to 23 mm (depending on the actual slot width) with 1 mm step,
- there are only sharp edges of outlet slot.

The geometrical model, used for flow analysis, was prepared in the ProEngineer CAD system and subsequently exported into the GAMBIT software, which was used for mesh generation. The axis-symmetric character of the geometry enabled us to use only one periodic segment (one full blade and one splitter) for the analysis. This had a positive effect on computational time reduction. Another way how to reduce the computational time was the use of a structured mesh, Fig. 7, allowing better cell size and distribution control in comparison with an unstructured

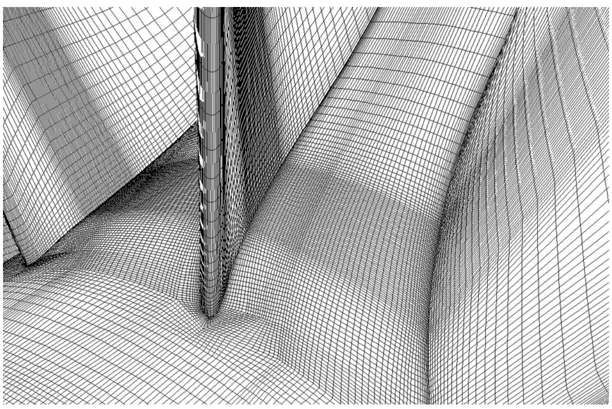


Fig. 7. Computational mesh in a passage and with refinement in the area of the IRC inlet

mesh. In the vicinity of walls and in areas with excessive geometrical change the mesh was refined, whereas in passages, the mesh was generated using larger cells.

4.2. Computational analysis execution

The simulations were performed in ANSYS FLUENT postprocessor. Because of significant density change during the air compression, the density-based solver was used. At the compressor inlet, the International Standard Atmosphere (ISA) conditions were defined. For the capability to simulate the whole constant rotational speed line, i.e. from maximal mass flow up to surge line, the loss coefficient boundary condition on impeller outlet was used. For every speed line, several operating points were simulated (usually from 8 to 10), which is fully sufficient for our purposes. If the point on surge line is reached, the computation becomes instable and the pressure ratio and mass flow decrease suddenly.

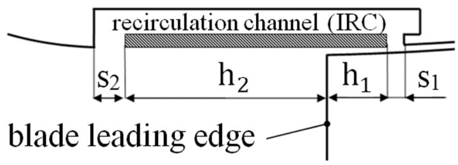
During the calculation, the achievement of stable values of outlet pressure and temperature and mass flow through the compressor and recirculation channel were monitored continuously. They gave good survey about the convergence history and achievement of stable values. In case that the monitored parameters changed only in a defined minimal tolerance, the currently calculated point was ended and evaluated.

4.3. Computational analysis evaluation

The CFD simulations were performed for three impeller rotational speeds and three basic configurations of the recirculation channel presented in Table 1. Altogether more than 70 compressor operational points were calculated and evaluated.

Table 1. Overview of recirculation channel configurations analysed within CFD simulations

Speed [% RPM]	**Inlet slot** s_1 [mm]	$\bar{h}_1$ [–]	Edges	**Outlet slot** s_2 [mm]	$\bar{h}_2$ [–]
91	1	0.092	sharp	3	0.292
95	2	0.092	sharp	3	0.292
100	2	0.092	chamfer	3	0.292
	3	0.092	sharp	3	0.292



$\bar{h} = \frac{h}{H}$, H [mm] — see Fig. 1

Because the inlet slot width is the most important parameter from the IRC performance and efficiency point of view, all the evaluated data were sorted using this parameter. With respect to the information mentioned above, there were three groups with inlet slot width 1, 2 and 3 mm, which were investigated separately. The evaluation method respects physical principles of flow dynamics and at the same time allow a comparison of calculated results with the measured ones. The recirculation channel performance was described by the dependency of relative mass flow (Q_3/Q_1) on pressure difference on slots ($\mathrm{d}p$) for particular slot widths, Fig. 8.

Relatively good correspondence between experimental results and results of CFD simulations is evident from the charts in Fig. 8. This correspondence is significant, because the mass flow in the recirculation channel is driven by actual static pressure distribution and local losses on compressor casing. It is not set as a boundary condition, as it was during the experimental simulation. From the charts it is also evident that wider slots reach higher relative mass flow for the same pressure difference. Chamfered edges have unquestionably positive effect on the flow, as already concluded from our measurements.

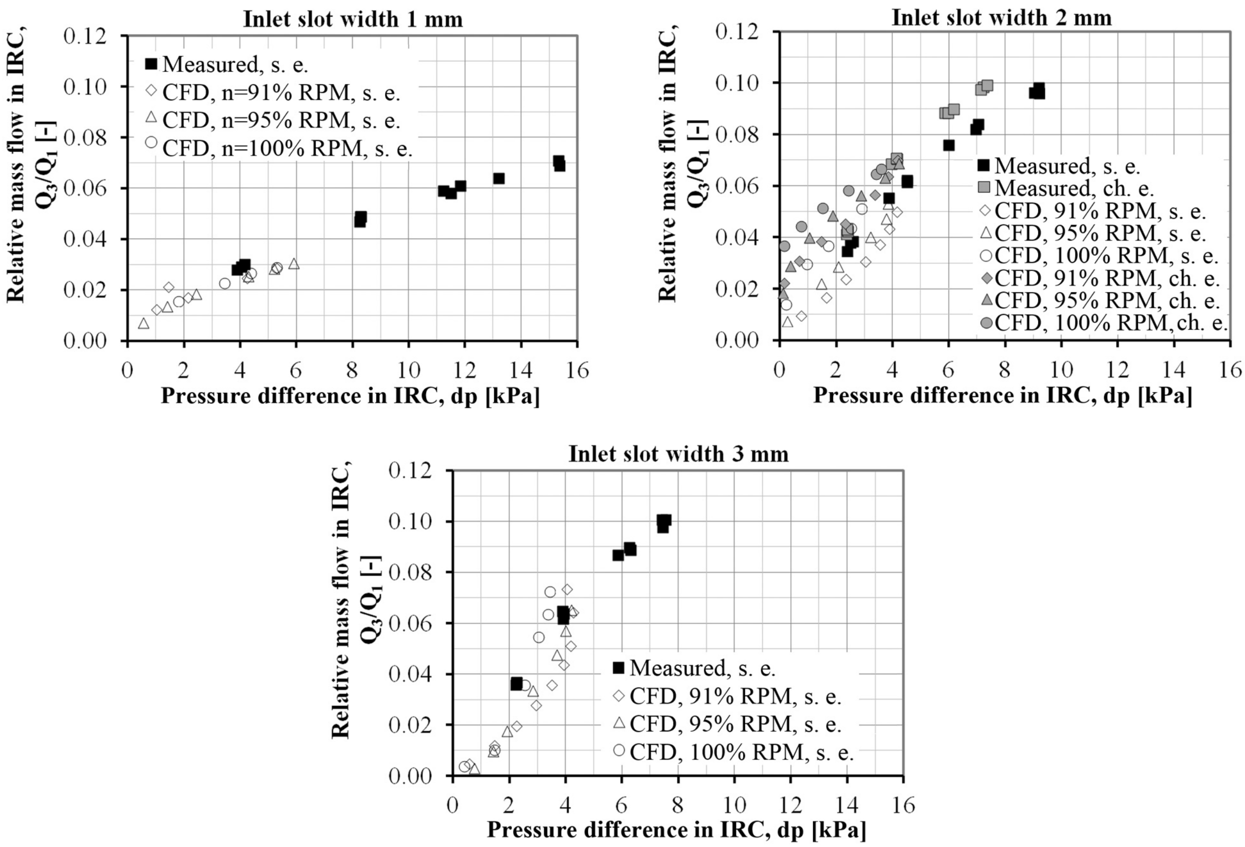


Fig. 8. Dependency of relative mass flow on pressure difference for various IRC inlet slot widths and edges, s. e. – sharp edge, ch. e. – chamfer edge

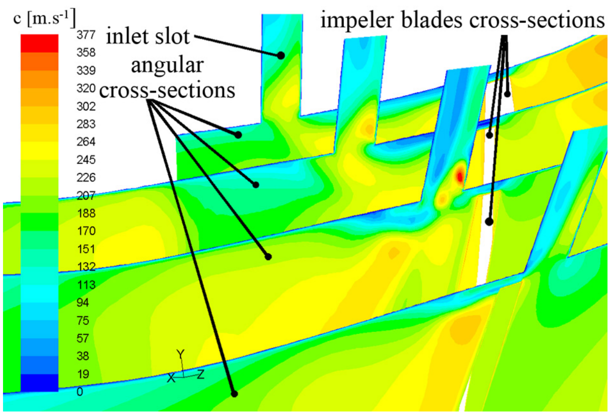


Fig. 9. Velocity flow field in the region of impeller blades and inlet slots

Generally speaking, the simulations describe the complicated interaction between the rotating blades and inlet slot better than the measurements carried out on the model, Fig. 9. The figure shows velocity distribution near the inlet slot at several angular cross-sections through the impeller passage. The aforementioned interaction was not considered in the experimental model. This is also the reason why simulations indicate lower pressure difference connected with lower mass flow in recirculation channel than it was expected during the measurements.

The main flow velocity profiles entering the compressor were plotted on planes located 4 and 19 mm downstream of the outlet slot (plane of impeller leading edges). The modification of

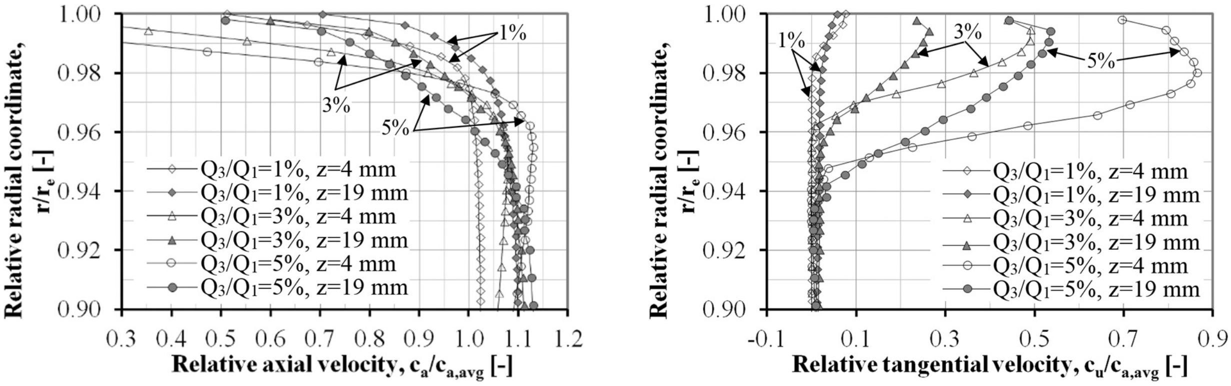


Fig. 10. Profiles of relative axial and tangential velocities at the compressor inlet for 100 % RPM and for different relative mass flows in IRC in planes placed 4 and 19 mm downstream of the outlet slot, inlet slot width 2 mm

axial and tangential velocity profiles, caused by flow penetrating from the outlet slot, is shown in Fig. 10. From both profiles it is evident that the circulating air ifluences only approximately 5–10 % of the diameter near the inlet duct outer wall. When the Q_3 increases, the mean axial velocity in central part of inlet duct increases slightly, as well. With increasing distance downstream of the outlet slot, the effect of IRC mass flow decreases significantly.

For similar IRC relative mass flows and the same distance from outlet slot, it is possible to compare velocity profiles calculated by CFD simulations with those measured by the three-hole probe on the IRC model, Fig. 6. The calculated and measured velocity profiles are comparable. However, the CFD simulation shows lower influence of recirculating flow on the main flow velocity profile. The computed layer close to the wall is thicker than the one measured by the three-hole probe in the IRC model. It is important to note that it is not possible to achieve the same absolute values of velocity components, because the conditions at the IRC inlet slot were different for the measurement and computation.

Fig. 11 shows the part of compressor performance map for particular configurations of IRC at 91, 95 and 100 % RPM. From the chart we can see only small differences of maximal

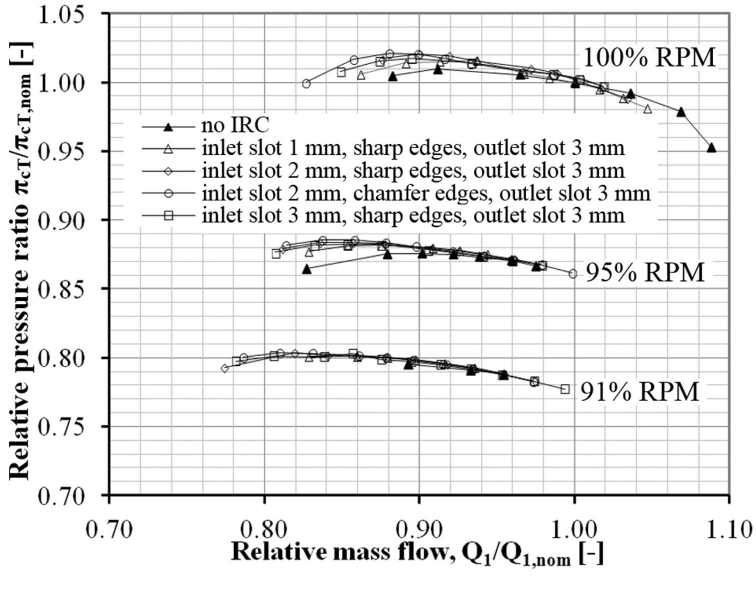


Fig. 11. Compressor performance map for various IRC

pressure ratio, which can be denoted as insignificant. All the IRC configurations shift the surge line towards lower mass flows. The effect of recirculation channel on compressor stability also depends on impeller rotational speed. The predicted performance map and the results of performed CFD simulations in general can be used for preliminary design of new centrifugal compressors equipped with IRC.

The CFD simulations enable a demonstration of one assumption from basic IRC theory. If the loss coefficient of compressor impeller increases, the operational point moves from aerodynamic choking to surge line. At the choking, air in the recirculation channel flows in the same direction as the main flow. As the operational point at the constant speed moves towards lower mass flow, the flow direction in the recirculation channel is changed. In this case, the air flows in IRC in the opposite direction against the main flow entering the compressor impeller, Fig. 12.

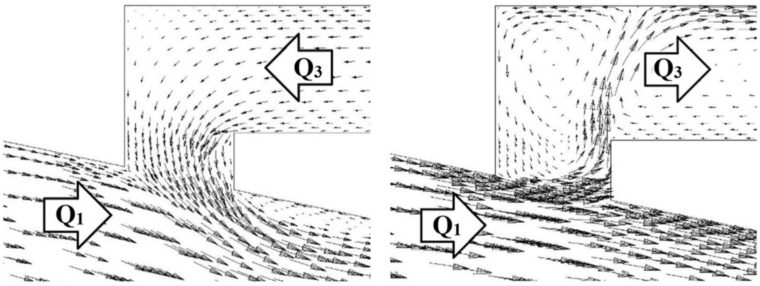


Fig. 12. Flow direction in IRC close to compressor operation at high speeds at surge (left) and choke (right)

5. Conclusion

Centrifugal compressors with a high pressure ratio have often a narrow region of stable operation. The implementation of an Internal Recirculation Channel (IRC) located at compressor inlet is one of possibilities how to shift the surge line towards lower mass flows and thus to increase the stability margin. This is the reason why extensive research is performed on both levels, experimental as well as computational.

The research indicates a significant influence of inlet slot width on IRC performance and efficiency. A narrow inlet slot with a 1 mm width generates higher losses and therefore lower air flows through the IRC. Thus, increasing the slot width, the flow in IRC will rise.

The axial and tangential velocity profiles downstream of the outlet slot show that the influence of recirculation is first of all at the outer inlet duct diameter, more than at the remaining 90 % of its value. Further effect can be also noted in the central part of the inlet channel, where the axial velocity in front of the impeller is slightly increased.

The CFD analyses using 3D geometry of compressor impeller and IRC were performed simultaneously with the measurement. The results of analyses were verified with data acquired from the measurement on the model test device. The evaluation was focused on trends of particular parameters displacement and not on absolute values of pressures, mass flows and losses. Good correspondence between analysis and measurement was found. This is valuable with regard to the fact that the definition of boundary conditions is different. During the measurements, the pressure difference on the recirculation channel was strictly set as a boundary condition, whereas during the CFD analyses it was given by an actual pressure distribution on compressor casing.

The CFD simulations enabled an investigation of the IRC influence on compressor performance map. All analysed configurations shifted the surge line towards lower mass flow. Only an insignificant drop of impeller pressure ratio at some compressor operating modes was obtained.

Within another research, the CFD analyses will be compared with results of compressor measurement on the experimental engine with IRC situated in compressor intake. The objective is to calibrate calculation methods for future design of IRC for similar compressors without the necessity to perform extensive experiments. CFD calculations will be also focused on the investigation of velocity profiles and angles of relative velocity on impeller blades leading edges, influencing stable operation enhancement.

Acknowledgements

This research has been financially supported by "LEMCOTEC – Low Emissions Core-Engine Technologies", a Collaborative Project co-funded by the European Commission within the Seventh Framework Programme (2007–2013) under the Grant Agreement No. 283216.

References

[1] Babák, M., Compressor IBW6 – anti surge measures, Technical Report No. VDT-344/2010, První brněnská strojírna Velká Bíteš, a.s., Velká Bíteš, 2011. (in Czech)

[2] Babák, M., Optimised design of radial compressor with axial diffuser, NEWAC Deliverable No. D2.2.5D, Technical Report No. NEWAC-PBS-DEL-D2.2.5D-R1.0, První brněnská strojírna Velká Bíteš, a.s., Velká Bíteš, 2010.

[3] Barton, M. T., Mansour, M. L., Liu, J. S., Palmer, D. L., Numerical optimization of vaned shroud design for increased operability in modern centrifugal compressor, Journal of Turbomachinery 128(4) (2004) 627–631.

[4] Dobřichovský, J., Kmoch, P., Slanec, J., Measurement on IRC test device, Technical Report No. R-6236, rev. 1, Výzkumný a zkušební letecký ústav, a.s., Prague, 2015. (in Czech)

[5] Hideaki, T., Xinqian, Z., Yangjun, Z., Experimental investigation of high pressure ratio centrifugal compressor with axisymmetric and nonaxisymetric recirculation device, Journal of Turbomachinery 135(3) (2013) 1–12, doi: 10.1115/1.4007579.

[6] Hunziker, R., Dickman, H. P., Emmrich, R., Numerical and experimental investigation of a centrifugal compressor with an inducer casing bleed system, Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 215(6) (2001) 783–791.

[7] Jílek, A., The flow in centrifugal compressor inlet, Ph.D. thesis, University of Defence, Brno, 2008. (in Czech)

[8] Kmoch, P., Prachař, A., Comparison of results of calculations and measurement on IRC model — rev. 1, Technical Report No. R6317, Výzkumný a zkušební letecký ústav, a.s., Prague, 2015. (in Czech)

[9] LEMCOTEC participants, IRC principles definition, Deliverable No. D2.2.3, Technical Report No. VZLU-DEL-D2.2.3-R1-0, Výzkumný a zkušební letecký ústav, a.s., Prague, 2013.

[10] Numakura, R., Tamaki, H., Hazby, H., Casey, M., Effect of a recirculation device on the performance of transonic mixed flow compressors, ASME Proceedings, paper No. GT2014-25365, Turbo Expo 2014, Düsseldorf, Germany, 2014, doi: 10.1115/GT2014-25365.

[11] Park, Ch.-Y., Choi, Y.-S., Lee, K.-Y., Yoon, J.-Y., Numerical study on the range enhancement of a centrifugal compressor with a ring groove system, Journal of Mechanical Science and Technology 26 (5) (2012) 1371–1378.

[12] Poledno, M., Compressor stable work enhancement, Ph.D. thesis, University of Defence, Brno, 2013. (in Czech)

[13] Sivagnanasundaram, S., Spence, S., Early, J., Nikpour, B., An investigation of compressor map width enhancement and the inducer flow field using various configurations of shroud bleed slot, Proceedings of ASME Turbo Expo 2010: Power for Land, Sea, and Air, Glasgow, The American Society of Mechanical Engineers (ASME), 2010, pp. 1701–1710, doi: 10.1115/GT2010-22154.

[14] Sivagnanasundaram, S., Spence, S., Early, J., Nikpour, B., Experimental and numerical analysis of a classical bleed slot system for a turbocharger compressor, Proceedings of IMechE 10th International Conference on Turbochargers and Turbocharging, London, UK, 2012, pp. 325–341.

[15] Yamaguchi, S., Yamaguchi, H., Goto, S., Nakao, H., Nakamura, F., The development of effective casing treatment for turbocharger compressors, Proceedings of the 7th International Conference on Turbomachinery and Turbocharging, London, UK, 2002, pp. 23–32.