ZÁPADOČESKÁ UNIVERZITA V PLZNI

FAKULTA STROJNÍ

KATEDRA ENERGETICKÝCH STROJŮ A ZAŘÍZENÍ

ZÁPADOČESKÁ UNIVERZITA V PLZNI

POWER SYSTEM ENGINEERING, THERMODYNAMICS & FLUID FLOW

2013

JEDNOTLIVÝ PŘÍSPĚVEK ZE SBORNÍKU

evropský sociální fond v ČR

EVROPSKÁ UNIE

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

OP Vzdělávání pro konkurenceschopnost

INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

CONCEPTION FOR INVESTIGATION THE INLET DAMPING PHENOMENON IN A LOW SPEED SINGLE STAGE CENTRIFUGAL BLOWER

KABALYK Kirill, KRYŁŁOWICZ Władysław

Single stage low-speed centrifugal blower with unshroded impeller is chosen as an object of investigation. The blower is equipped with inlet cassette filter followed by S-shaped inlet-guide apparatus. The test rig will be used in a series of experimental measurements to investigate the pressure fluctuations damping effect that is thought to appear as a result of interaction between inlet elements and stage members. Additionally, unsteady RANS computations are planned to be carried out to find out whether their results might show good correspondence with the measurements. ANSYS CFX 13.0 code will be applied for this purpose. This paper, focuses on the aspects of experimental tests preparation and shows the results of initial steady-state numerical calculations. The scheme of steady and unsteady static pressure measurement stations and mass flow measurement is given and discussed. Numerically predicted stationary stage performance maps were found to show good qualitative agreement with the ones obtained experimentally by the earlier researchers. Due to this, the numerical model of the blower is identified to be applicable for URANS simulations, which are to be started in the nearest future. As the process of test rig building and preparation for tests requires reasonable amounts of time, the experimental results also stay the subject for later publications.

Keywords: centrifugal blower, inlet filter, inlet guide, RANS-based numerical simulation

Introduction

Centrifugal compressors are utilized in various fields of industry. Their main advantage over axial flow compressors might be considered higher pressure ratio obtainable at one stage. On the other hand, this type of machinery allows processing larger volumes of gas when compared to reciprocating or screw type machines. In addition, centrifugal units usually have rather simple design. The above features make centrifugal compressors being applied in different fields of industry from aerospace to ecology.

Nevertheless, all type of turbocompressors tend to have one common disadvantage. Their operational mass flow range is limited due to design scheme they usually utilize. The interaction of continuous fluid flow with rotating and stationary constructive elements on one hand provokes the lack of pressure rise at flows higher than operational. Oppositely, when the mass flow is lower than the design one, phenomena known as rotating stall and surge are likely to occur. The latter cases are thought to be the most dangerous for machine operation and its operational staff. They usually lead to the excitation of forced vibrations in impeller and other elements in case of rotating stall and might lead to a total unit failure in case of surge [6]. Besides, authors [1, 4, 5, 8, 11] admit the importance of investigating other unsteady processes that exist within the range of mass flows considered as safe for a certain machine (steady operational range). Usually, these are those initiated by angular flow disturbance at the outlet of rotating or stationary blade cascades [5, 11]. This disturbance will obviously appear at every operating regime and will provoke blades to vibrate, which at certain conditions might also cause mechanical problems or damage (lattice flutter, flexural-torsional flutter, etc. [9]).

In conclusion, it is possible to admit that unsteady phenomena play a crucial role in the field of mechanical reliability of centrifugal compressors. Due to this, they have been a subject of scientific investigation over the last sixty years [1]. D Eckardt was one of the first researchers who explored the “jet-wake” flow at the outlet of centrifugal compressor impeller using semi-conductor pressure transducers [1].

Hathaway et al [3] performed series of measurements inside NASA low speed centrifugal compressor test rig (LSCC) using rotating frame total pressure and total temperature rakes. This brought to a better understanding of flow disturbance inside centrifugal impeller.

Trebinjac et al [11] studied rotor-stator interaction as well as the inside-impeller flow in a small-size supersonic centrifugal stage using the L2F technique. The study concludes that impeller outlet velocity fluctuations reach their maximum when the splitter blade wake crosses the vicinity of vane diffuser leading edge.

As might be seen, usually the author’s attention is focused on certain stage elements such as impeller or diffuser. Typically, the design scheme utilized for research includes inlet pipe or plenum, impeller, vaneless or vaned diffuser and outlet plenum connected to outlet pipeline. However, it is known that industrial compressors very often are equipped with such systems as inlet filter, inlet guide apparatus, intermediate gas coolers etc. These elements, if considered as members of oscillatory system implemented by centrifugal compressor installation, might influence its behavior within whole operational range. Henceforth, the main scope of this work, is to evaluate the effect that inlet filter and inlet guide have on pressure fluctuations inside stage setting.

2. Experimental test rig.

The research is being carried out at a single stage centrifugal blower named DP1.12. The test rig is situated at fluid mechanics laboratory of Lodz University of Technology and was designed and built by R. Magiera [10]. The blower was firstly used for general performance tests and later was rebuilt for unsteady pressure measurements [4]. The configuration utilized in this work is shown in fig. 1.

The stand consists of inlet filter (pos. 1) installed into a suction pipe with rectangular cross section. The latter is followed by a rectangle-circle connector (pos. 2) and S-shaped inlet guide (IG) (pos. 3) with 18 vanes. The vanes might be fixed at five positions that correspond to the following vane angles: $\alpha_{IGV} = (-30°;0°;30°;45°;60°)$. The unshrouded impeller (pos. 4) blade number is $z_{imp} = 23$ and outlet blade angle equals to $\beta_{bl2} = 90°$. The tests are planned to be run at rotor rotational speed of $n = 6000 rpm$, which corresponds to specific Mach number $M_u = 0{,}33$.

Afterwards goes vaneless diffuser (pos. 5) with constant width and an overhung constant internal radius volute (pos. 6), which is connected to an outlet pipe (pos. 7). The stage is then completed by a flow straightener (pos. 8), mass flow measurement orifice (pos. 9) and an outlet regulation valve (pos. 10).

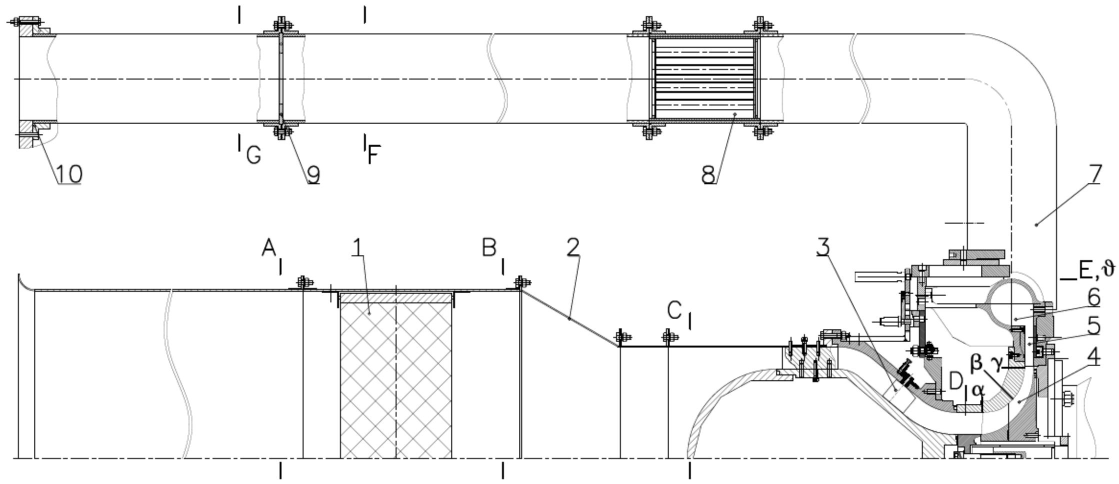

Fig. 1 Experimental test rig and measurement stations location scheme

The filter consists of a rectangular shaped metal cassette casing with fiberglass bars inside it. The bars are covered by metal grids that give them the proper shape and don't let them fall out of the casing. According to manufacturer, for typical industrial applications a single bar will be enough to install to provide required degree of filtering. However, during the measurements it is planned to change the material quantity from total lack to full load to see the effect of filter resistance on pressure fluctuations behavior.

The flow straightener consists of fifty five plastic pipes with external diameter of 18 mm which are packed inside a 160x4,7 mm PCV sewage outlet pipe with internal diameter of D_{in}=150mm. The straightener length is chosen to equal to 1,5D_{in}, which corresponds to PN-EN ISO 5167-1 national standard. The unit is mounted in between two metal flanges and is located 12D_{in} downstream of the compressor outlet knee.

Mass flow measurement orifice contraction value equals to $\beta = 0.6$. The orifice is designed in accordance with PN-EN ISO 5167-1 norm and is located 10.5D_{in} downstream of the straightener.

Regulation valve represents a circular shaped flap driven by a screw-and-nut mechanism in a direction perpendicular to the outlet pipe axis.

3. Measurement equipment.

At the moment of writing this paper some parts of the test rig were at manufacturing stage. That's why it was only possible to describe the conception of measurements.

It is decided to divide experimental tests into two sessions. Initially, the measurements are to be held with unloaded filter. This is necessary to find out if the inlet guide vanes have any impact on pressure fluctuations in impeller and other stage elements. The vanes will be set at three positions: $\alpha_{IGV} = (-30°;0°;30°)$. After that the filter will be uploaded and major attention will be focused on the relationship between filter static pressure drop Δp_{filter} and static pressure fluctuations. The IG is planned to be held opened ($\alpha_{\text{IGV}} = 0°$).

The measurement methodology utilized in this work, requires a division into steady and unsteady test series. The former implies the use of equipment for steady value measuring while the latter demands implementing unsteady pressure transducers. Fig. 1 shows the location of steady (A-G) and unsteady (α-θ) pressure measurement stations. Stations A,B; C,D; F,G are necessary to know the pressure drop correspondingly on the filter, inlet guide and orifice while D and E stations are to give the pressure rise obtained within the stage. All the unsteady stations will be used to obtain a rough signal data for further mathematical transformation (FFT) and identification the rotating stall and surge pressure frequencies and amplitudes. Besides, a temperature station will be installed downstream the orifice for fluid density definition.

The tests are going to be carried out within the whole operational range from the fully opened valve position to a fully closed one. This comes possible due to rotor design, which was especially prepared to allow measuring even in deep surge condition.

Fig. 2 represents the devices planned to be used in the tests. Honeywell (C) static transducers (a) would be connected to a 10 channel Keythley (C) scanner while Kulite (C) unsteady sensors (b) would be linked to a 12 channel Wavebook (C) scanner. Transformed in scanners into an electric one, the signal then will be treated by a corresponding software inside PC.

a.

b.

Fig. 2 Pressure gauges chosen for experimental measurements:
Honeywell © stationary pressure gauge; b. Kulite © unsteady pressure trancducer.

4. Numerical simulations

Experimental measurements results are planned to serve as a verification database for numerical calculations. However, the test rig preparation required more time than the process of its numerical model development. Due to this, at the moment of writing this article only the numerical results were ready to be presented.
The final three dimensional CAD model is shown in fig. 3. The model was generated using different commercial CAD applications. Creo 1.0 was used for the inlet filter casing and rib region; ANSYS Bladegen 12.0 for inlet guide domain and Autodesk Mechanical Desktop for the rest of the model (impeller, VLD, volute, outlet pipe).

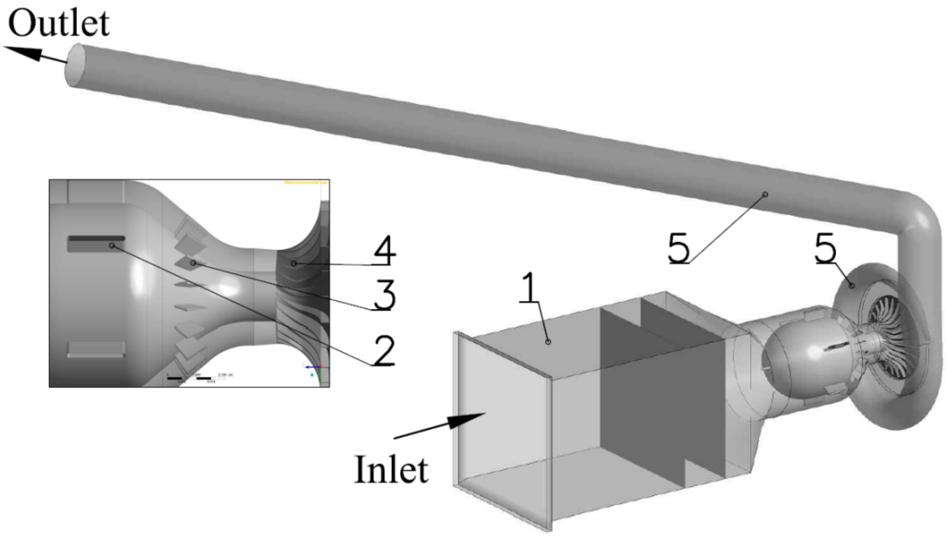

Fig. 3 Computational domain used in CFD computations

The whole computational region might be divided into five large blocks: inlet filter (pos. 1), ribs (pos. 2), inlet guide (pos. 3), rotor (pos. 4) and outlet (pos. 5).

The geometry was meshed using ICEM CFD for inlet filter, rib and outlet. CFX Turbogrid generator was chosen for inlet guide and rotor parts. The ultimate computational grid comprises 7 million nodes and is built from hexahedral elements. The highest $y+$ values are situated close to impeller leading edge and don't seem to overcome the value of 20. The mesh around rib is a typical H-grid with a slight density increase near the rib ($y+ < 10$). IG and rotor meshes employ H-O-H grids ($y+ < 20$).

The results of steady flow simulations will be presented. The filter is unloaded and the inlet guide angle is $\alpha_{IGV} = 0°$. The calculations are run within the range between the maximum mass flow and surge margin. Total pressure, total temperature and straight flow direction are set at the inlet boundary. Outlet condition depends on the machine operating point: area-averaged integral static pressure is set when the mass flow is between $0{,}94 \le \overline{m} \le 1{,}39\, kg/s$. When the mass flow is lower $(\overline{m} < 0{,}94\, kg/s)$ it is being set up itself up to the surge margin.

The data exchange between rotating mesh of impeller and stationary grids of IG and VLD is performed by means of "frozen rotor" interface model (parameters are transferred without circumferential averaging). The Reynolds stress tensor is defined using 2-equation $k-\omega SST$ turbulence model. According to observations of authors [2, 7] this model usually shows good agreement (at least qualitative) with experimental measurements in centrifugal compressors. As the average velocity levels in impeller and volute differ by about an order, a proper timescale definition was necessary. To do this, the impeller and volute domains were simulated individually, which allowed to obtain average flow velocity levels. After that, domain lengths (meridional middle span line length - impeller; radial middle scroll line length - volute) were divided by the corresponding values of velocity scale, which finally gave physical timescales for each part. The mean average of $\Delta\tau_{\text{mean}} = 0{,}0125\,\text{s}$ of the two cases was chosen as a final timescale for simulations.

Fig. 4 shows theoretic head coefficient ψ_T [1] (a) and static isentropic efficiency η_{is} (b) versus mass flow rate $\dot{m}$ relations simulated for rotor rotating speed of $n = 5500\,rpm$. The continious line represents numerical results obtained with test rig model shown in fig. 3 while dashed line corresponds to the experiments carried out by Magiera, 2006 [10] with axial inlet guide configuration. Due to this, a qualitative comparison seems to be proper only.

According to fig. 4.a experimentally measured values comprise a steeper curve if compared to the one corresponding to CFD computations. Kinematically it means that numerically predicted impeller outlet flow delay angle $\delta\beta_2 = \beta_{bl2} - \beta_2$ values are lower than the real ones. However, the difference in slope is slight, which identifies a good qualitative agreement of the results.

Comparing the efficiency curves (fig. 4.b) it might be noted that the graphs discrepancy propagates with the growth of the mass flow rate. Numerical solution gives averagely 5% higher values than the experiment.

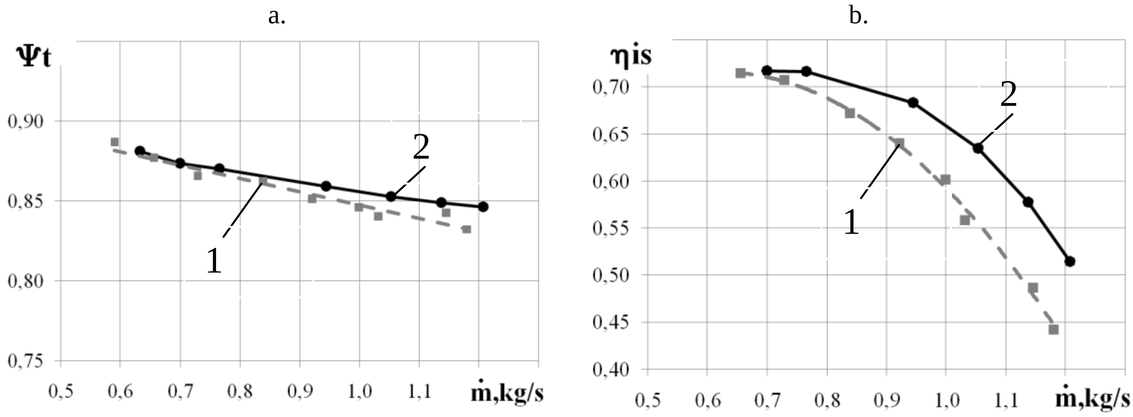

Fig. 4 Theoretic head coefficient ψ_T (a) and stage isentropic efficiency η_{is} (b) versus mass flow rate $\dot{m}$ graphs:
1 – Experiment [Magiera], 2 – Numerical simulation

Fig. 5 illustrates the Inlet-to-Outlet history of total temperature T* and total pressure p* change for three operating regimes. The points chosen for the analysis correspond to the mass flow rate values of $\dot{m} = 1.208\,\text{kg/s}; 0.945\,\text{kg/s}; 0.700\,\text{kg/s}$. As might be visible, all the noticable trends do not confront with physically expected ones. Total temperature rises in impeller and stays constant inside stationary stage elements. The temperature rise grows with the decrease of mass flow. Total pressure drops in all the elements but impeller, which is also theoretically expected.

[1] $\psi_T = \dfrac{h_T}{u_2^2}$, where h_T - total enthalpy rise in impeller, J.kg, u_2 - impeller tip speed, m/s, [2].

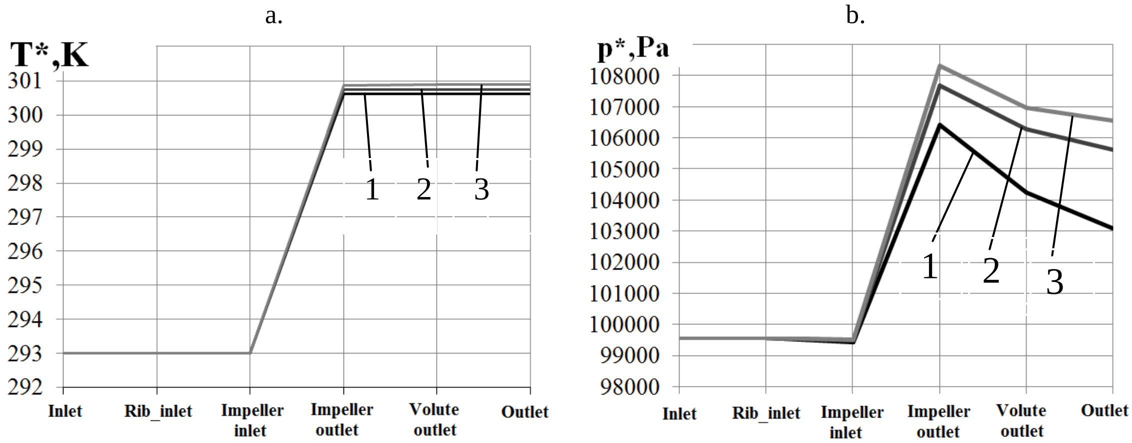

Fig. 5 Inlet-to-Outlet total temperature T* (a) and total pressure p* (b) history:
1. $\dot{m} = 1.208\,\text{kg/s}$ **; 2.** $0.945\,\text{kg/s}$ **; 3.** $0.700\,\text{kg/s}$

Conclusions

A single stage centrifugal blower with cassette type inlet filter and S-shaped inlet guide was chosen to investigate the pressure fluctuation damping phenomenon. The phenomenon is expexted to be initiated by the inlet machine elements. Up to the moment of publishing the present paper, the following conclusions might be pronounced:

- the experimental measurements strategy and principal scheme are completed and ready to be introduced into life in the nearest future;
- first series of steady state numerical simulations showed good qualitative correspondence to the available experimental data in case of theoretic head coefficient and a satisfactory one in case of isentropic efficiency prediction;
- the completed CFD simulations were found to physically soundly predict the total temperature and total pressure change within compressor stage elements;
- nevertheless the numerical model showed good performance for a series of steady state runs, it still needs to be checked for mesh and timestep independence;
- the liability of applying a porous body model to predict the flow inside the filter stays the subject for further research.

Literature

[1] ECKARDT, D. Investigation of the jet-wake flow of a highly-loaded centrifugal compressor impeller / NASA Tech. Mem. 72232, 1978 – 182 p.

[2] GAMBURGER, D. M. Numerical viscous gas flow simulation within the centrifugal compressor stage: methodology and results, PhD Thesis, Compressor, vacuum and refrigeration technologies department, SPbSTU (in Russian), pp.100 – 108., 2009

[3] HATHAWAY, M.D., WOOD, J.R., WASSERBAUER, C.A., NASA Low-Speed Centrifugal Compressor for 3-D Viscous Code Assessment and Fundamental Flow Physics Research/ NASA Tech Mem. 103710, 1991 – 14 p.

[4] HORODKO, L. Badania Eksperimentalne Zjawisk Poprzedzajacych Pompowanie Sprezarki Promieniowej i Ich Identyfikacja za Pomoca Falkowej Analizy Sygnalow Cisnien. Raport Nr 4 T07B 02827 (in Polish), Politechnika Lodzka, 2008

[5] IZMAILOV, R. A., SELEZNEV, K. P. Nonsteady state processes in the flow-through section of centrifugal compressors, Chemical and Petroleum Engineering, 1972

[6] IZMAILOV, R. A., SELEZNEV, K. P. Unsteady Processes in Centrifugal Compressors, Chemical and Petroleum Engineering, 1995

[7] KABALYK, K., KRYLLOWICZ, W. Verification of the numerical model of the viscous gas flow within a centrifugal compressor stage (in Russian), Nauchno-Technicheskie Vedomosti SPbGPU, 2011

[8] KRAIN, H., HOFFMAN, W. Verification of an impeller design by laser measurements and 3D viscous flow calculations, ASME Paper No. 89-GT-159, 1989

[9] LASKIN, A. S., ZUEV, A. V., STRIZHAK, L. Ya., Power machines durability (in Russian), Leningrad, Leningrad Polytechnic Institute Publishing, 1987 – 180p.

[10] MAGIERA, R. Eksperimentalne i numericzne badania struktury przeplywu za kierownica wlotowa dmuchawy promieniowej (in Polish), Praca Doktorska, Politechnika Lodzka, 2006

[11] BULOT ,N., OTTAVY, X., TREBINJAC, I. Unsteady Pressure Measurements in a High-Speed Centrifugal Compressor, Journal of Thermal Science, 2010

MSc, KABALYK Kirill, Institute of Turbomachinery, Lodz University of Technology, 219/223 Wolczanska St, 90-924 Lodz, Poland, +4842 6312466, kirill.kabalyk@p.lodz.pl
Dr hab. prof., KRYŁŁOWICZ Władysław, Institute of Turbomachinery, Lodz University of Technology, 219/223 Wolczanska St, 90-924 Lodz, Poland, +4842 6312383, wladyslaw.kryllowicz@p.lodz.pl