The Experimental Study of Diffuser Geometry for Low Pressure Exhaust Casing of Steam Turbine

Robert Kalista[1,2]; **Lukáš Kanta**[2]; **Lev Feldberg**[3]

[1] Doosan Skoda Power, Experimental Research of Flow,
Tylova 1/57, Pilsen, 301 28, Czech Republic

[2] University of West Bohemia, Department of Power System Engineering,
Univerzitní 8, Pilsen, 306 14, Czech Republic

[3] NPO CKTI, Joint-Stock Company, Polzunov Scientific & Development Association on Research and Design of Power Equipment,
St. Petersburg, Atamanskaya str., 3/6, 191167, Russia
e-mail: [1]robert.kalista@doosan.com

Abstract

The performance of the last stage of the LP part of a steam turbine is strongly influenced by the effectivity of the downstream exhaust casing. The 90° turning of the flow in a relatively short axial distance is a major cause of losses and the design of low-loss diffuser still remains a challenge for mechanical engineers. In this paper, results of studies on a several steam turbine exhaust diffuser designs of SKODA have been reported. Several experimental measurements were carried out in the special sector model air test rig. This unique test rig allows visual observation of the flow by the Schlieren method and evaluating the loss coefficient of static pressure in the diffuser. The test rig allows achieving very high (supersonic) speeds. The range of observed velocities was from 30 to 360 m/s. The experimental data from these measurements are very useful to be able to predict the exhaust casing losses during the real operation of steam turbine in non-designed states. The behaviour of individual diffuser designs has been discussed.

Keywords

Diffuser; Steam turbine; Exhaust casing; Pressure loss.

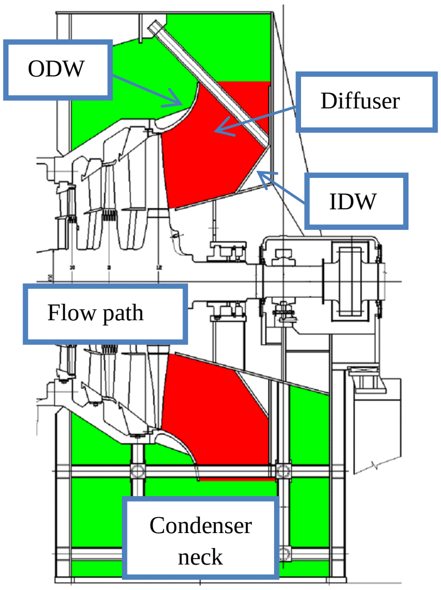
Introduction

As is well known, the performance of the last stage of the low pressure part of a steam turbine is strongly influenced by the effectivity of the downstream exhaust casing. The efficiency of the exhaust hood depends on many structural factors such as the design of the diffuser parts, dimensions of the outer casing or arrangement of internal supports. The fact that the influences on the last stage output can be quite important is dealt with in e.g. [1] by Hoznedl who provides as an example a 1090 MW turbine in the nuclear power plant with a pressurized water reactor where the increase of pressure loss between the last stage and the condenser by only 2000 Pa can cause output loss of about 4MW for one LP flow.

The low-pressure exhaust casing consisting of the diffuser, the exhaust hood and the condenser neck connects the last stage turbine and the condenser (see Fig. 1). The condenser is located below the turbine level and the steam flow leaving the last stage must change its direction about 90° from the axial direction to the radial direction before exhausting into the condenser. The function of the exhaust hood is to bring the steam into the condenser with minimal energy losses and to transform the output kinetic energy into potential energy. This mechanism leads to expansion line prolongation between the stage inlet and outlet and to higher turbine power output.

Previous studies and experimental measurements have shown that just the diffuser area has the most significant influence on the total loss of the exhaust casing, for example [2] and [3]. The 90° turning of the flow in a relatively short axial distance is a major cause of losses and the design of low-loss exhaust systems still remains a challenge for mechanical engineers.

With development of CFD methods it is already a common practice to use numeric simulations to analyse and to optimize exhaust hood shapes. A lot of works dealt with this topic in the past, for example [4]. However, the great difficulty of these computations has always been the size and complexity of the computing area. From the point of view of computational and time capacities it is very difficult to fully capture very complex flowing through the entire output tract. For this reason CFD computations are simplified variously and this reduces credibility of results. From the above mentioned and other reasons it is always appropriate to complete CFD computations with relevant experimental measurements and thus to validate obtained results mutually.



ODW – Outer Diffuser Wall; IDW – Inner Diffuser Wall

Fig. 1: *Sketch of the axial-radial exhaust casing*

1 The Purpose of Experimental Measurement

The aim of this paper is the experimental study of flow in the sector model of low pressure axial-radial diffuser. The observed diffuser was designed by SKODA for an extra-long 54" last stage blade made from titanium. The sector model of the diffuser [7] at optical benches of the laboratory of physical and technical problems of power equipment of NPO CKTI JSC is investigated. This unique test rig allows visual observation of the flow by the Schlieren method and evaluation of the loss coefficient of static pressure in the diffuser. For one geometry of low pressure diffuser a few different variations of inlet velocities were tested. The test rig allows achieving very high (supersonic) speeds. The range of observed velocities was from 30 to 360 m/s. The effect of velocity change was observed. During such similar experiments, it is not usual for such high inlet velocities to be achieved. For this reason, the results obtained are of unique value.

The test facility also has the possibility to blow compressed air into the boundary layer of the ODW (Outer Diffuser Wall). The blown stream of air adds sufficient energy to the boundary layer, which may delay or prevent its separation from the surface of the diffuser and thus energy losses of the entire output tract should decrease essentially. Much attention to the influence of blowing into the wall of the diffuser together with the influence of the radial leakage of the steam over the bandage of the moving blade was already paid in the past. The air blowing into the boundary layer and simulation of the steam leakage over bandage sealing was not realised in this work. In this paper we concentrate to examine only the sensitivity of velocity on the loss coefficient of the low pressure diffuser.

2 Test Facility

The sketch of diffuser with the main dimensions is shown in Figure 2. The diffuser geometry is described by four basic dimensionless numbers (see Table 1). The sector model of low pressure diffuser is 1:10 scale. The dimensions of the model may differ slightly from the real state, depending on the layout options of the experimental test facility. The following table shows the values of the four basic characteristics for the tested diffuser model.

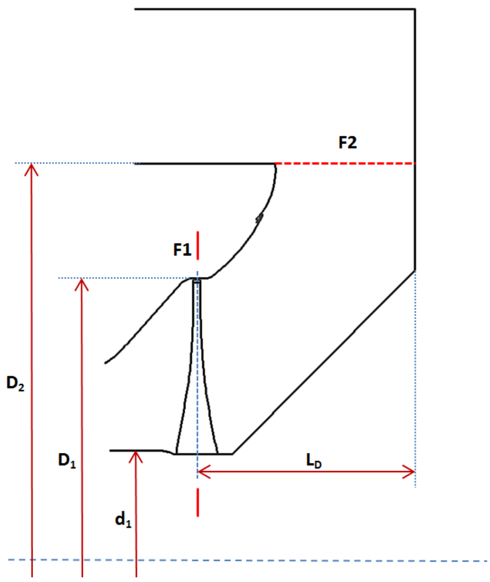


Fig. 2: *The sketch of diffuser with characteristics dimensions*

Tab. 1: *Diffuser geometry*

Hub-Tip ratio $d = \frac{d_1}{D_1} = 0.42$	Expansion degree $n = \frac{F_2}{F_1} = 1.50$
Diffuser radiality $D = \frac{D_2}{D_1} = 1.30$	Diffuser elongation $L = \frac{L_D}{D_1} = 0.36$

The illustrative photo of the test facility is shown in Figure 3.

Fig. 3: *Photo of sector model of diffuser (illustrative photo)*

2.1 Advantages and Disadvantages of the Sector Model

The principle of sectorial modelling has a number of advantages. First, it makes it possible to visually observed the flow by using a large arsenal of optical methods. Second, the sector model makes it possible to carry out studies at full-scale flow velocities in a diffuser at lower energy costs. (Air consumption is 10 - 15 times less than in the 3D model of the same scale). Third, this type of construction is much simpler, cheaper, and allows easy variation of the basic geometric parameters of the circular diffuser (elongation, radiality, area ratio, internal and external contour configuration) at a constant bushing ratio, which provides mobility in carrying out the experiment.

The studies on the sector model allows significant energy saving (both in terms of the power of the blowers and the duration of the experiments), which becomes particularly important due to a sharp increase in the cost of energy today. In addition, the sector model allows investigation of the effect of inlet conditions (such as radial leakage of the last stage) on the efficiency of the diffuser.

The disadvantages of the sector model in comparison with the 3D model are:

- the complexity of modelling such an important phenomenon as the twist of the flow at the entrance to the diffuser,
- the possibility of studying only an isolated diffuser without an exhaust hood.

In addition, it is important to note that there is some deviation from the geometric similarity in the sector model associated with the technological possibilities. On the other hand, the comparison between the use of a sectional and a full 3D model of the exhaust hood was already tested. No important difference was found using both approaches [6].

3 Description of Measurement

Evaluation of diffuser power performance values assumed for our researches involves flow parameters measurement at inlet and outlet reference cross-sections. Location of pressure measurement points is shown at Fig. 4. The basic element of experimental test rig is the compressed air source (fan, compressor – different in dependence on the required inlet velocities – pressure ratio). In front of the diffuser model is a stabilisation vessel. A large volume of the stabilisation vessel causes a zero flow velocity (with respect to the Bernoulli equation). Due to the minimum airflow rate in these parts, the stagnation pressure and temperature is measured here.

Design of diffuser model does not allow measurement in the plane corresponding to axis of last stage blades as side walls of the channel in this point are limited with protection glasses. For this reason the inlet reference cross-section (A-A) is located 100 mm upstream of the blade axis. Pressure is measured at nine points from both sides of trapezoidal channel. Measurement points in symmetrical locations on opposite sides are paralleled (seen in Figure 4). Outlet reference cross-section (B-B) is located 200 mm downstream of the outlet edge. Pressure is measured at 20 points distributed at the perimeter from both sides of rectangle channel. Measurement points in symmetrical locations on opposite longer sides are paralleled again. The main flow-rate has been measured by means of an orifice located at the exhaust pipe from the test rig.

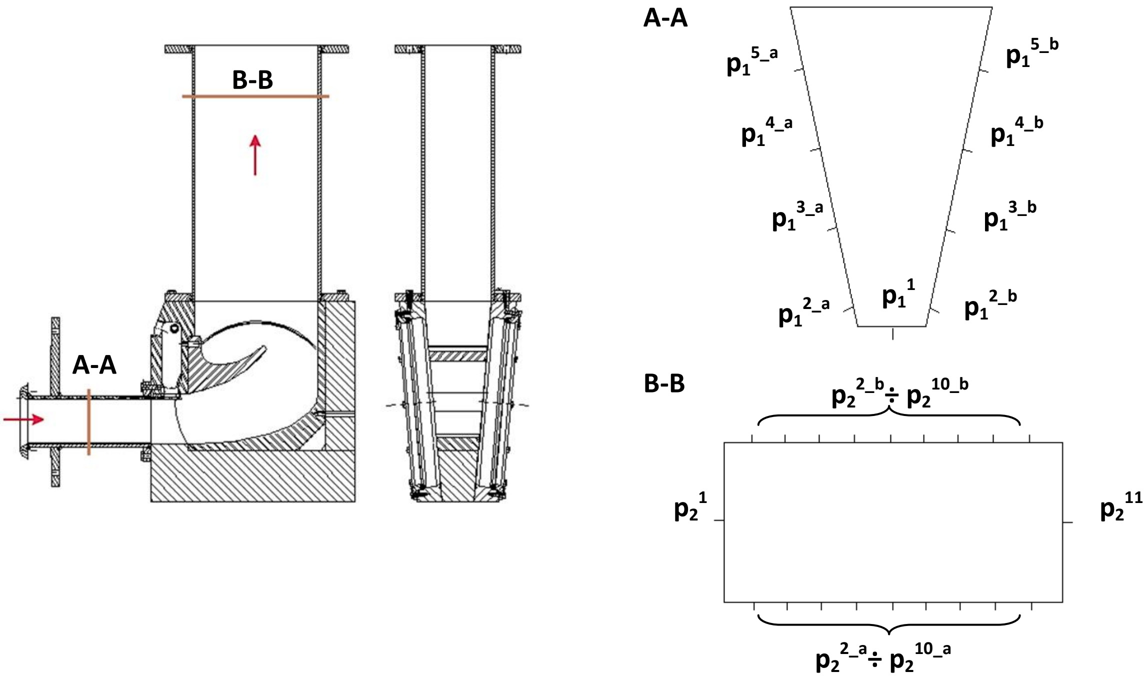


Fig. 4: *Sketch of measurement*

Figure 4 shows the designation of the individual quantities in the respective measuring planes.

Flow stagnation temperature has been measured with the help of a chromel-cupel thermocouple with analog transducer to voltage. Ambient pressure is measured by a stationary calibrated barometer. The total pressure in the stabilisation vessel is measured by one absolute pressure transducer. All other pressures (static pressures at diffuser inlet/outlet, static pressure before/behind the orifice) are measured by a second pressure transducer of the same type (measurement range 0 ÷ 0.15 Mpa). Measurement of individual pressures takes place step by step using a rotary pneumatic switch. The output from the pressure transmitter (4-20 mA) is displayed on the measuring computer.

3.1 Evaluation of Experimental Data

Processing of experiment results was based on results of direct measurement of temperature, static pressure and total pressure at inlet cross-section and static pressure at outlet cross-section (conventionally named “isoentropic” approach).

Diffuser energy characteristics are estimated by total loss coefficient, which was calculated by the following formula

$$\zeta_T = \frac{1-\left(\frac{\overline{P}_2}{\overline{P}_{01}}\right)^{\frac{k-1}{k}}}{1-\left(\frac{\overline{P}_1}{\overline{P}_{01}}\right)^{\frac{k-1}{k}}} \tag{1}$$

where

$\overline{P}_{01}$	–	averaged total pressure in inlet control section,
$\overline{P}_1$	–	averaged pressure in inlet control section
$\overline{P}_2$	–	averaged pressure in outlet control section

The mean pressure value in the inlet section is determined by numerical integration of the individual pressure values along the trapezoidal channel height, see equation 2 and Figure 5.

$$\overline{P}_1 = \frac{1}{F_{P1}} \sum_{j=1}^{N-2} P_{1j} \left\{ \frac{y_{j+1}-y_{j-1}}{2} \left[a + tg\alpha \left(\frac{y_{j+1}-y_{j-1}}{2} + y_j \right) \right] \right\} \tag{2}$$

where:

FP1	–	integration area
y	–	height coordinates of static pressure tap
N	–	number of static pressure taps
α	–	the sector diffuser opening angle

It is important to note the fact that the area FP1 is not equal to F1.

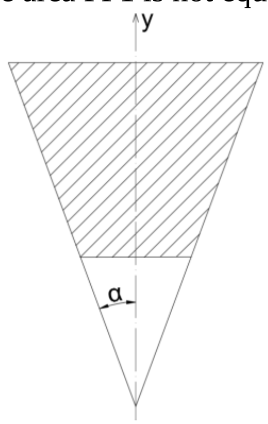


Fig. 5: *The inlet control section*

The dependence of the total loss coefficient on the inlet isoentropic Mach number was evaluated.

3.2 Flow Visualization – The Optical Method

Flow visualization is essential for exploring and understanding fluid behaviour and can be both qualitative and quantitative. The main methods for visualization of these flows are optical methods. The three basic principal optical methods are: shadow, Schlieren and interferometry. In this case, the optical Schlieren method was used.

In general, flow field is transparent environment with light refraction index n. The light refraction index in each flow field point is the function of air density in that point, which is function of velocity, pressure and air temperature. The relation between air density and the refraction index is called the Gladstone-Dale equation n = 1 + Kρ. The constant K is different for each gas. A light ray, passing through a nonhomogeneous refracted field, is deflected from its original direction and a light path is different from that of an undisturbed ray. The equation (3) shows the dependence of the deflection of light rays on the value of the gradient of the refractive index of the medium, integrated over the thickness of the optical inhomogeneity.

$$\varepsilon(x, y, z) = \frac{1}{n}\int_{0}^{D} \frac{\partial}{\partial s} n(x, y, z)\, dz \qquad (3)$$

where:

D – thickness of optical inhomogeneity (width of working area).

4 Obtained Results

All measurements for the examined diffuser were performed under similar conditions over a wide range of inlet velocities from Ma = 0.15 to “aerodynamic blockage” (Ma ~ 1.10). Reynolds number Re varied from 2.35 x 105 to 4.6 x 106.

Figure 6 shows the dependence of total energy losses on inlet velocity into diffuser. Up to Ma = 0.7 is the negligible dependence of loss on inlet velocity. Slight differences in measured values of total losses do not reach levels of measurement uncertainty. Based on this finding, it can be said that the diffuser losses are independent of inlet velocity until the reaching of Ma = 0.7. With a further increase of inlet velocity to the transonic region, the behaviour of losses is difficult to explain. The sharp drop in losses is over with the aerodynamic blockage of the diffuser channel at Ma = 0.96 which is accompanied by a sharp increase in losses.

The usual velocity in the diffuser inlet during operation is within the range Ma = 0.4 ÷ 0.7. In some turbine limit states these values may be higher. For the purpose of proper design is very useful to know that in the range of these axial velocities, the total losses of the diffuser are practically unchanged. However, the unchanged value of the diffuser losses, which is higher than 100%, suggests that the shape of the LP diffuser is not optimal for the purely axial character of flow.

Interestingly, similar results were obtained in the previous experimental measurement of several shape variants of the LP diffuser type Skoda mentioned in the report [5]. A sample of the obtained results is shown in Figure 7. These are the results of the measurement for the diffuser 1-3 and 0-4 shape version, which are compared with the results for the basic shape of diffuser (marked: basic variant) for 54 last stage blade, see Figure 7. Even in this case, there was an aerodynamic blockage of the diffuser and a sharp increase in total losses around the area of the sound velocity. It is interesting to find that the basic variant of diffuser achieves the lowest level of total losses even though it was designed for inlet flow with a significant radial component of velocity that is not simulated during the measurement.

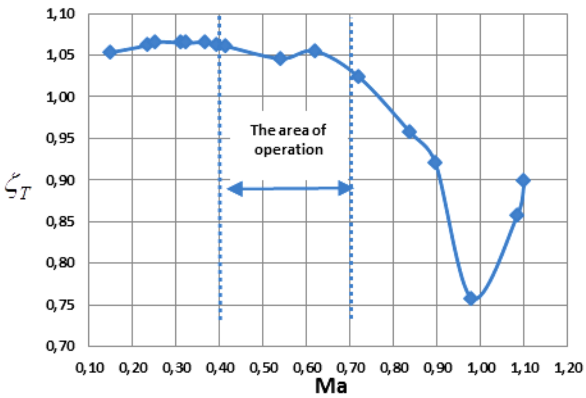


Fig. 6: *Dependence of total loss coefficient on the inlet isoentropic Mach number*

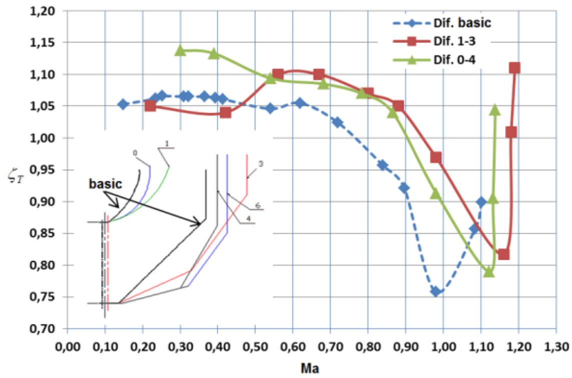
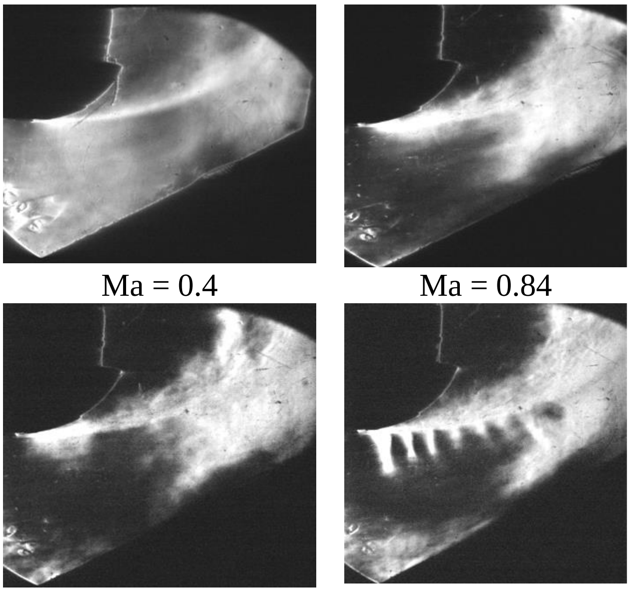


Fig. 7: *Comparison of results with previous measurements*

The explanation of the obtained results can be partially found in the flow visualization by using the Schlieren method. Figure 8 shows four images for different inlet velocities in the diffuser. The flow direction is, in this case, from left to right. For Ma = 0.40 the character of flow corresponds with high loss at level. Flow-rate in diffuser is so small that practically the diffuser effect does not occur. A significant separation of the boundary layer on the ODW is due to the absence of a radial component of velocity and a small flow compression in the IDW region that would help to direct the flow towards the ODW. A significant improvement in diffuser filling is evident with increasing velocity. More visible is also the more significant bending of the flow in the radial direction. The fourth picture captures the occurrence of a series of shock waves in flowing in the transonic region. This corresponds to a significant increase in losses in the diffuser.



Ma = 0.4 Ma = 0.84

Ma = 0.98 Ma = 1.08

Fig. 8: *Visualization of flowing obtained results*

Conclusion

Several experimental measurements were carried out at NPO CKTI. The most extensive measurement was concerned with monitoring the loss coefficient of the diffuser for an extra-long 54 last stage blade in dependence on the inlet isoentropic Mach number. Experimental measurements were performed on the sector model of the monitored diffuser. The Schlieren method was used to visualize the flow. From the obtained results, it is evident that up to Ma ≈ 0.7 the pressure losses are practically independent of the velocity. This demonstrates the ability of the diffuser to operate over a wide range of inlet flow rates without the significant variable of diffuser losses.

As the inlet velocity increases, the total diffuser losses are sharply decreasing. This phenomenon is manifested up to the area of the transonic velocities, where aerodynamic blockage of the diffuser and formation of the shock waves occurs, resulting in a severe degradation of the diffuser function.

In the next part of the paper, the obtained results are compared with the results from previous years, which were carried out under similar conditions. Unfortunately, compared variants varied more or less from each other and it was not possible to compare the absolute values of the obtained loss coefficients. On the other hand, from the trends of the obtained results, a similar behaviour is visible.

This is a useful finding that demonstrates the correctness and suitability of the experimental methods used.

The individual results presented in this report point to the interesting behaviour of the diffuser, especially for higher inlet velocities. To explain this issue, it is necessary to do more experiments and numerical simulations.

Acknowledgment

This work was realized with financial contribution of TA ČR within the project TJ01000048 – Lowering outlet losses in steam turbines of modern construction.

Nomenclature

F	Area [m^2]
p	Pressure [Pa]
Ma	Mach number [-]
Re	Reynolds number [-]
ζT	Total loss coefficient [-]
ODW	Outer diffuser wall
IDW	Inner diffuser wall

Subscripts

0	Total pressure
1	Diffuser inlet
2	Diffuser outlet

Literature

[1] M. HOZNEDL et al.: The Pressure Field at the Output From a Low Pressure Exhaust Hood and Condenser Neck of the 1090 MW Steam Turbine: Experimental and Numerical Research. In: *Proceedings of ASME Turbo Expo 2018*. Oslo, Norway. 2018, Paper No. GT2018-75248. ISBN 978-0-7918-5117-3. DOI: 10.1115/GT2018-75248

[2] KALISTA, R.; MRÓZEK, L.; HOZNEDL, M.: The Experimental Investigation of the Internal Support Effects on Exhaust Casing Pressure Recovery. In: *Proceedings of ASME 2017 International Mechanical Engineering Congress and Exposition*. 2017, Paper No. IMECE2017-70279. ISBN 978-0-7918-5841-7. DOI: 10.1115/IMECE2017-70279

[3] HOZNEDL, M.; PACÁK, A.; TAJČ, L.: Effect of internal elements of the steam turbine exhaust hood on losses. *EPJ Web of Conferences*. 2012, Vol. 25, Paper No. 01024. DOI: 10.1051/epjconf/20122501024

[4] FU, Jing-Lun; LIU, Jian-Jun: Investigation of Influential Factors on the Aerodynamic Performance of a Steam Turbine Exhaust System. In: *Proceedings of ASME Turbo Expo 2010*. Glasgow, UK. 2010. Paper No. GT2010-22316. ISBN 978-0-7918-4402-1. eISBN 978-0-7918-3872-3. DOI: 10.1115/GT2010-22316

[5] ФЕЛЬДБЕРГ, Л. А.: *Аэродинамическое исследование диффузора цнд турбины Шкода на секторной модели. внутренний отчет для SKODA*. Контракт 203/290912/11. 2009.

[6] TAJČ, L. et al.: The experimental investigation of the influence of the flow swirl and the tip clearance on aerodynamics characteristics of exhaust hoods. In: *Turbomachinery 2007*. Athens, Greece. 2007, pp. 395–404.

[7] ДОБКЕС, А. П.; НИШНЕВИЧ, В. А.; ФЕЛЬДБЕРГ, Л. А.; ЮШКЕВИЧ, Ю. Э.: *Устройство для исследования газовых потоков оптическими методами*. Авторское свидетельство СССР 811116.

EXPERIMENTÁLNÍ ZKOUŠKA GEOMETRIE DIFUZORU PRO NÍZKOTLAKOVÉ VÝFUKOVÉ PLÁŠTĚ PARNÍ TURBÍNY

Výkon posledního stupně LP části parní turbíny je silně ovlivněn efektivitou spodního výfukového pláště. Otáčení 90° v relativně krátké axiální vzdálenosti je hlavní příčinou ztrát a konstrukce difuzoru s nízkou ztrátou stále zůstává výzvou pro strojní inženýry. V tomto článku byly publikovány výsledky studií několika konstrukcí difuzoru výfukových plynů parodontu ŠKODA. Několik experimentálních měření bylo provedeno ve speciálním sektorovém modelu vzduchotechniky. Toto jedinečné zkušební zařízení umožňuje vizuální pozorování průtoku metodou Schlieren a vyhodnocení ztrátového koeficientu statického tlaku v difuzoru. Testovací zařízení umožňuje dosažení velmi vysokých (nadzvukových) rychlostí. Rozsah pozorovaných rychlostí byl od 30 do 360 m/s. Experimentální data z těchto měření jsou velmi užitečná, aby bylo možné předvídat ztráty výfukových plášťů během reálné činnosti parní turbíny v neupravených stavech. Bylo popsáno chování jednotlivých návrhů difuzoru.

EXPERIMENTALE PRÜFUNG DER DIFFUSERGEOMETRIE FÜR NIEDRIGDRUCKAUSPUFFMÄNTEL DER DAMPFTURBINE

Die Leistung der letzten Stufe des LP-Teils der Dampfturbine wird stark von der Effektivität des unteren Auspuffmantels beeinflusst. Eine Drehung um 90° in relativ kurzer Axialentfernung ist der Hauptgrund der Verluste und die Konstruktion eines Diffusors mit niedrigem Verlust bleibt weiterhin eine Herausforderung für die Maschinenbauingenieure. In diesem Artikel werden die Ergebnisse der Studien einiger Konstruktionen des Diffusors von Auspuffgasen des Parodonts ŠKODA publiziert. Es wurden einige experimentelle Messungen in einem speziellen Sektorenmodell der Lufttechnik durchgeführt. Diese einzigartige Versuchseinrichtung ermöglicht ein visuelles Beobachten der Durchflussmenge mit der Schlieren-Methode und eine Auswertung des Verlustkoeffizienten des statischen Drucks im Diffusor. Die Testeinrichtung ermöglicht das Erreichen sehr hoher Geschwindigkeiten (Überschallgeschwindigkeit). Der Umfang der beobachteten Geschwindigkeiten betrug zwischen 30 und 360 m/s. Die aus dieser Messung hervorgegangenen experimentellen Daten sind sehr nützlich dazu, die Verluste der Auspuffummantelungen während der realen Tätigkeit der Dampfturbine in ungeregelten Zuständen voraussagen zu können. Es wird das Verhalten der einzelnen Entwürfe des Diffusors beschrieben.

EKSPERYMENTALNA PRÓBA GEOMETRII DYFUZORA DLA NISKOPRĘŻNYCH OBUDÓW WYLOTOWYCH TURBINY PAROWEJ

Na wydajność ostatniego stopnia niskoprężnej części turbiny parowej w dużym stopniu wpływa skuteczność dolnej obudowy wylotowej. Obroty 90° przy stosunkowo małej odległości osiowej stanowią podstawową przyczynę strat a konstrukcja dyfuzora o niskich stratach pozostaje wciąż wyzwaniem dla inżynierów maszyn. W niniejszym artykule przedstawiono wyniki badań obejmujących kilka konstrukcji dyfuzora wylotowego w turbinach marki SKODA. Kilka eksperymentalnych pomiarów przeprowadzono w specjalnym modelu sektorowym systemu wentylacyjnego. To wyjątkowe urządzenie testowe umożliwia wizualną obserwację przepływów przy wykorzystaniu metody Schlierena oraz ocenę współczynnika strat ciśnienia statycznego w dyfuzorze. Urządzenie testowe pozwala na osiągnięcie bardzo dużych (naddźwiękowych) prędkości. Zakres obserwowanych prędkości wynosił od 30 do 360 m/s. Dane eksperymentalne z tych pomiarów są bardzo przydatne do prognozowania strat w obudowach wylotowych w trakcie realnej pracy turbiny parowej także o innej konstrukcji. Opisano także zachowania poszczególnych konstrukcji dyfuzora.