Applied and Computational Mechanics 3 (2009) 213–220

Static and modal analysis of the brake stand

M. Švantner[a,*], J. Klepáček[a,b], V. Lang[a]

[a] Department Thermomechanics of Technological Processes, NTC, University of West Bohemia, Univerzitní 22, 306 14 Plzeň, Czech Republic
[b] Department of Machine Design, Faculty of Mechanical Engineering, University of West Bohemia, Univerzitní 22, 306 14 Plzeň, Czech Republic

Received 5 September 2008; received in revised form 2 December 2008

Abstract

Testing and development of braking systems belong to significant technical problems solved in automotive industry. The braking systems testing could be carried out directly on a car or using a special testing device — a brake stand. The brake stand developed at the workplace of the University of West Bohemia is introduced in this paper. Its function and usage characterisation are described. Static and modal analyses, which were performed for adjustments of the final design of the brake stand, are presented. The results of the computer modelling are commented and discussed.

Keywords: automotive brakes, computer modelling, static analysis, modal analysis, finite element method

1. Introduction

Brake systems are one of the most important components in a car. A proper function of brakes is crucial for transport safety not only for personal cars. Brake systems are also exposed to a strong mechanical and thermal load that can cause their intensive wear, material and shape changes or damage. An experimental and numerical testing of braking systems is therefore an important technical problem.

Function (stopping power) of brake systems or wear rate of brake system components are the most often tested parameters. However, a presence of thermo-mechanical instabilities during braking [1, 2] is also an important problem. The thermo-mechanical instabilities manifest themselves in hot-spot origination. The hot-spots are regularly distributed places on a brake disc or drum surface of significantly higher temperature. Their presence causes instable contact between the brake disc and a pad and it is connected with undesirable vibrations and noise produced during braking. Therefore, brake discs and cars producers try to prevent the hot-spots origination or to reduce their consequences.

An experimental testing of brake systems is fundamental in regard of a complexity of the problem [3, 4]. There are two ways of a brake systems testing. The testing can be carried out on a car directly or using a special experimental equipment - a brake stand [5]. The testing on a car is closer to reality and it is very useful for a “final” testing. However, using the brake stand makes more detailed analysis of brake systems parts possible. It allows an exact definition of different braking regimes and testing of different designs and materials of brake system parts on one experimental device. The brake stand is therefore very important for brake systems experimental testing.

*Corresponding author. Tel.: +420 377 634 721, e-mail: msvantne@ntc.zcu.cz.

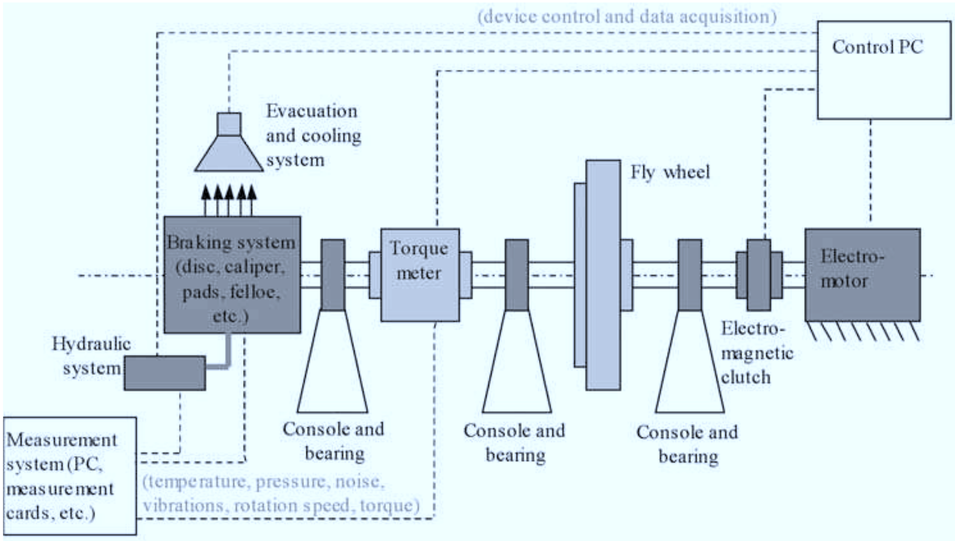

Fig. 1. Brake stand simplified scheme

2. Brake stand

2.1. Function description

The brake stand developed at the UWB (University of West Bohemia) workplace [5] consists of five basic parts — braking system, torque meter, fly-wheel, electromagnetic clutch and electromotor. A central shaft connects all the parts. Three consoles support the shaft. The motor drives the shaft with the fly-wheel, torque meter and brake disc through the electromagnetic clutch. The flying wheel weight is variable so that it can simulate inertial moment of different cars. When a required speed is achieved the electromagnetic clutch disconnects the motor from the shaft and a braking test starts according to chosen regime. In some special cases, the motor can drive the shaft also during the braking. The brake stand scheme is shown in fig. 1.

The electromotor, electromagnetic clutch, braking system (brake hydraulic system) and evacuation/cooling system are fully controled by a control computer. The control computer also collects and displays the basic operation parameters (temperature, rotation speed etc.). A special measurement system is used for high sampling rate measurement of temperature (contact and non-contact), brake system pressure, noise, vibrations, rotation speed and torque.

2.2. Usage characterization

The brake stand allows testing of disc and drum brakes for cars from about 800 to 1 800 kg. The electric motor power is 18.5 kW and it enables the maximum rotation speed 1 450 rpm that corresponds to a speed 160 km/h approximately (for a wheel of diameter 60 cm). The fly-wheel weight is 175 kg and diameter is 80 cm. A "car weight" can be changed by additional felloe-weights mounted to a fly-wheel. The brake stand allows static (constant speed — downhill journey) as well as dynamic (variable speed — car stopping) tests. A brake stand control system enables different regimes — constant pressure in a brake system, constant braking moment, constant speed or defined curves of the mentioned parameters. Additionally, a cooling air flow on the braking system can be controlled. The brake stand model is shown in fig. 2.

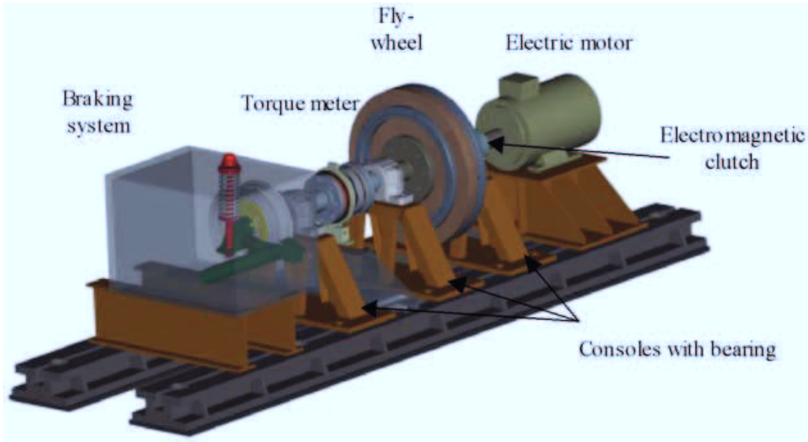

Fig. 2. Brake stand final version drawing

3. Numerical analysis of the brake stand

The shaft and the consoles are exposed to high static and dynamic loads during testing. A proper design and dimensions of the components are very important for stability and safety of the measurement system operation. Therefore, the final system design was proposed with numerical simulation support. The numerical simulation was focused on the shaft part with the fly-wheel, which seems to be the most critical. Only this part between the two consoles (including these consoles) was analyses by the numerical computation. Three basic problems were solved:

- Static and modal analysis of the shaft with the fly-wheel of the maximum weight.
- Static and modal analysis of the console.
- Static and modal analysis of the shaft-console assembly.

Some preliminary computations were performed using the specialized programs *Mechsoft* and *Prev*. The complete numerical analysis was carried out using the finite elements system *Cosmos DesignStar* (Structural Research & Analysis corp.). The material of all components is standard steel type AISI 1020 (Elastic modulus 2e11 Pa, Poisson's ratio 0.29) and linear-elastic material model is used in all cases.

3.1. Numerical analysis of the shaft

The shaft with the fly-wheel is one of the most important and also one of the most loaded parts of the system. Therefore, its stiffness and shape stability has to be guaranteed. The maximum rotation speed is about 1 450 rpm, that is about 24 Hz, and the weight of the shaft central part with the fly-wheels is about 370 kg. The shaft central part length is 840 mm, the maximum diameter is 100 mm and the shaft flange diameter is 350 mm. The basic dimensions and constraint positions are marked in fig. 3. The fly-wheels of the maximum diameter 800 mm are mounted to the shaft flange. The bearings are about 150 and/or 250 mm from the shaft flange.

Preliminary computation in Mechsoft showed, that the maximum deflection of the shaft is about 21 μm and the maximum stress is 67 MPa. The FEM model in Cosmos DesignStar was

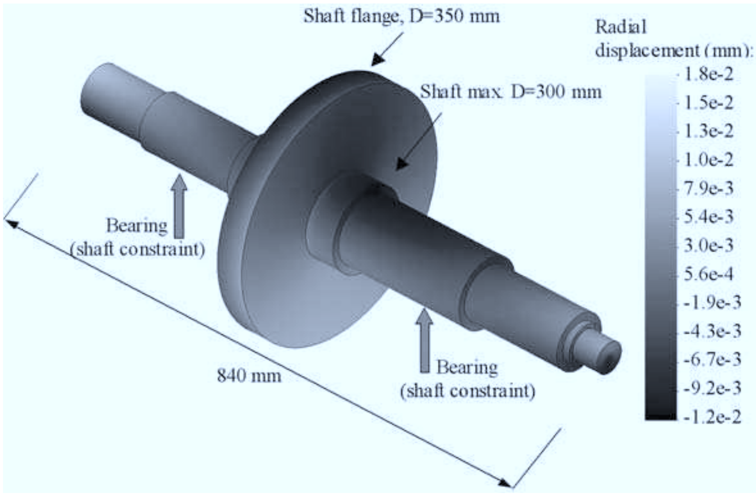

Fig. 3. The shaft central parts radial displacements

created based on proposed shaft design. The gravitational acceleration ($10\,\mathrm{m}\cdot\mathrm{s}^{-2}$), centrifugal forces and braking inertial forces (rotational deceleration) for the maximum fly-wheels weight were assumed in the computation. The shaft was constrained (radial and tangential directions) at the places of the bearings. Additional "elastic support" constraint was used at the end of the shaft — it did not influence the results but it is necessary for the to numerical stability. The model was simplified — small holes and other geometrical shapes were omitted, because it was assumed that it does not influence static or modal analysis results. FEM mesh was generated by DesignStar and consists of about 73 thousand nodes and 45 thousand parabolic tetrahedral solid elements.

The shaft radial displacement results are shown in fig. 3 (the fly-wheels are assumed in the analyses but they are hidden in the figure). The maximum radial displacement of the shaft is about $18\,\mu$m. The maximum computed local Von Mises stress intensity was about 44 MPa. These results are similar to that determined by Mechsoft software. The modal analysis showed that the first mode shape value of the shaft is 31 Hz (fly-wheels expansion) and the second is 85 Hz (shaft deflection). The first mode in not assumed as critical and the second mode is considered far enough from the maximum shaft rotational frequency 24 Hz.

The both static and modal analysis results are dependent on a constraint method. The constraint used in the model fixes the shaft to radial and tangential direction at one position. Nevertheless, the bearing supposed should fix the shaft at a small area along the shaft axis and therefore prevents its bending partially. Moreover, the shaft is also fixed partially at its ends by next shaft parts. The deflection and maximum stress can be therefore assumed lower then the FEM analysis showed. Therefore, the displacement (deflection), stress and modal analysis results were evaluated as satisfactory.

3.2. Numerical analysis of console

The consoles support the shaft by the bearing. They have to resist both static and dynamic loading also with respect to possible fatigue damage. The mechanical properties of the consoles are therefore crucial. However, a design of the consoles was proposed also with respect to complexity and costs of their manufacturing.

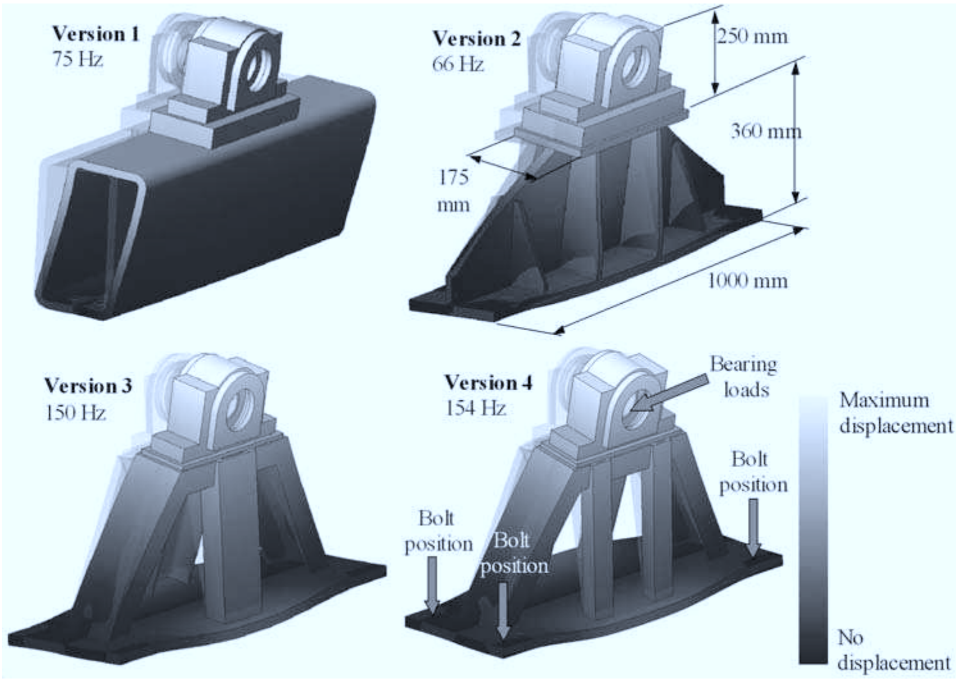

Fig. 4. The results-displacements of modal analyses of the consoles versions — the first natural frequencies

Subsequently, four versions shown in fig. 4 were proposed. All the versions consist of a main support part and simplified bearing part. The support part is mounted to base rails as can be seen in fig. 2. The mounting is approximated as a constraint at bolts places (from the bottom side). A load caused by a shaft weight (at the bearing) and a gravitational load caused by a weight of the console itself is assumed. The main dimensions of all the versions shown for version 2 in fig. 4 are the same.

The version 1 is the first version proposed. The advantage of this version is very simple design and minimum manufacturing requirement. The static analysis results revealed a maximum displacement about 4 μm and stress intensity (Von Mises) about 20 MPa. However, the modal analysis results were not satisfactory. The first natural frequency 57 Hz was considered too low. Therefore, the second console version based on one central plate supported by cross-ribs was proposed. The maximum displacement and stress intensity values were 6 μm and 5 MPa. However, the modal analysis results were still not good as shown in fig. 4 — the first natural frequency is 66 Hz.

The numerical analyses showed that simple shaped design based on long plates is not suitable enough for this purpose. Although the static analyses results were good, the modal analyses results were not optimal. Therefore, the third and fourth versions based on profiled beams were proposed. These two versions are similar. The only one difference is that the version 3 has two crossbeams and the version 4 has 4 crossbeams. The static analyses results showed maximum values of displacement and stress intensity about 6–7 μm and 6 MPa for the both versions. These results were fully acceptable even if the weight of these is much lower than the weight of the versions 1 and 2 (see tab. 1). The results of modal analyses showed the first natural frequency 150 and 154 Hz for the version 3 and 4. These first natural frequencies were better than

Table 1. The summary of the static and modal analysis of the consoles versions

	Version 1	Version 2	Version 3	Version 4
Max. displacement (μm)	4	6	7	6
Max. stress intensity (MPa)	20	5	6	6
First natural frequency (Hz)	75	66	150	154
Weight (kg)	211	171	135	133
Nodes	280 K	229 K	204 K	206 K
Elements	174 K	146 K	122 K	123 K

these for console versions 1 and 2. The numerical analysis results for versions 3 and 4 were considered as satisfactory. The version 4 was realized because the load distribution is supposed to be better with respect to mounting of the consoles to the base rails. The first mode shape results for all the versions are shown in fig. 4. The results summary for all the versions is in tab. 1.

The maximum displacement results from 4 to 6 μm for all the versions are considered as acceptable and it should not affect the brake stand usage. The maximum stress intensity is very low for all the versions considering static loading (the yield stress of the assumed steel is more than 350 MPa). However, the maximum stress values should be low as possible with regard to supposed intensive dynamic and long-term loading. The version 4 is the most complicated from a manufacturing process point of view. However, the modal analysis results are the best for this version and also a material consumption is much lower compared to version 1 or 2. It is supposed that smaller surface areas of parts of the console version 4 can also influence a vibration-noise generation favourably.

3.3. Numerical analysis of console-shaft assembly

The assembly shaft-consoles model shown in fig. 5 was build based on previously achieved results. Static and modal analysis were carried out. These analyses could confirm the previous computations and it could bring more accurate results.

The model consists of two consoles (the version 4 consoles) and the shaft with the flywheels. The shaft is supported on bearings. A bearing-shaft contact is simplified. The bearing

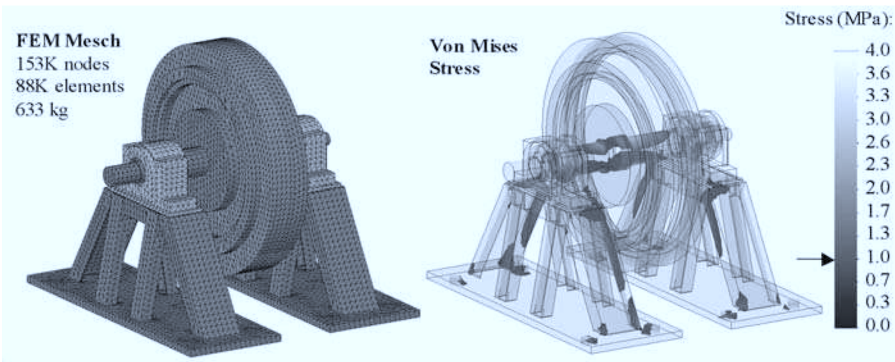

Fig. 5. The consoles-shaft assembly FEM mesh and von Mises stress intensity results

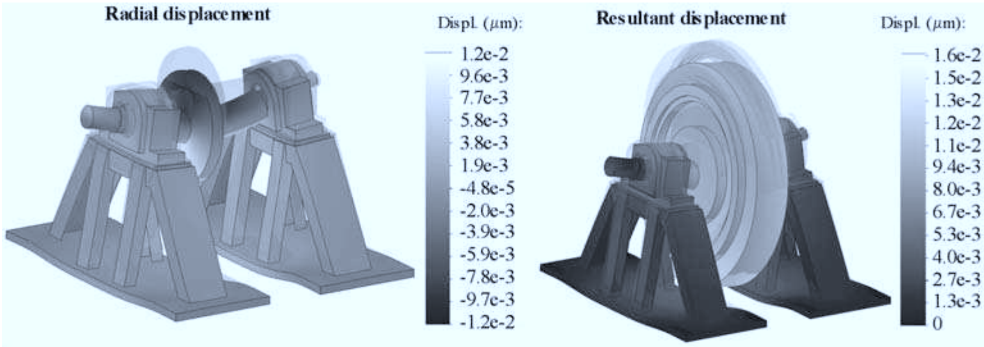

Fig. 6. The radial displacements (with respect to shaft axis) and resultant displacement of the consoles-shaft assembly

is assumed as a compact and homogeneous part with a circular hole of a diameter that equals to the diameter of the shaft. The contact condition between the shaft face and bearing hole face ensures that the shaft can move in the bearing, the parts can interact but they cannot penetrate to themselves. Additionally, an elastic support constraint is set to the end of the shaft — it does not affect results but it is necessary for a computation numerical stability. The gravitational loads of all the components are assumed for static and modal analysis. The overall weight of the parts including fly-wheels is about 630 kg and the FEM mesh contains 153 thousand nodes and 88 thousand parabolic tetrahedral solid elements. The solution is non-linear in this case due to contact boundary condition between the shaft and the bearings.

The results of the static analysis are shown in fig. 5 and fig. 6. One can see that the maximum stress intensity in all the components does not exceed 4 MPa. These results correspond to these determined for the parts computed separately for a stationary case (centrifugal loads are not included in this analysis). The radial displacements (with respect to shaft axis, the fly-wheels are hidden but included in the analysis) shown in fig. 6 are lower than for the shaft computed separately including centrifugal loading. However, the results are very similar, if the centrifugal forces are not considered in the shaft stationary analysis. The maximum resultant displacement of the assembly is about 16 μm — this value is higher than the radial displacement due to a misalignment of the consoles and/or fly-wheel in the axial direction. The displacements (radial) obtained by the Prev software were about 7 μm. However, all shaft parts (as displayed in fig. 2 — including torque meter, electromagnetic clutch etc.) were included to a simplified Prev model. Lower displacement values were therefore expected.

The modal analysis showed that natural frequencies are lower than for the separate parts. The first natural frequency is 60 Hz, however it concerns the fly-wheels and it is not so important. The second natural frequency is 79 Hz and it is related to all parts of the assembly. This value is similar to the second natural frequency of the shaft computed separately. Next natural frequencies exceed 120 Hz. The first and the second natural frequencies are displayed in fig. 7.

4. Conclusion

The contribution shows a practical support of the computer modelling in a technical problems solving. The FEM analyses were used for a confirmation of simple numerical model results and for designing of the brake stand parts.

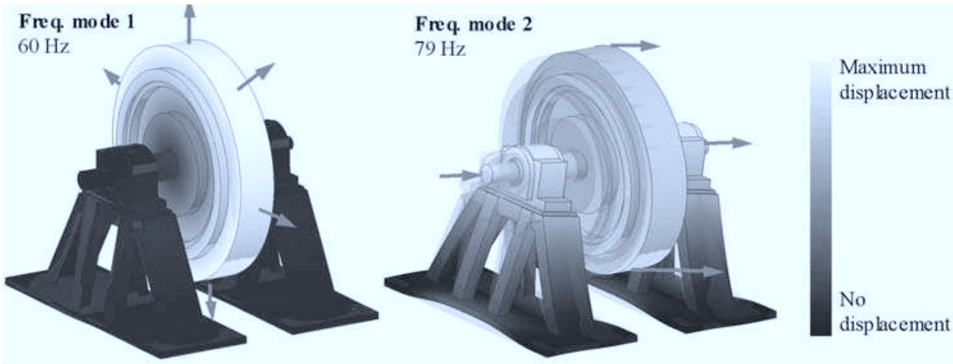

Fig. 7. The assembly modal analysis results (displacements) — first and second natural frequencies

The optimal brake stand consoles design was supposed using the numerical analysis. The "profiled-beam" design proved to be the most suitable from the point of view of modal analysis results, static analysis results and overall weight of the part in spite of the manufacturing process of this version is the most complicated. The proposed consoles design is rigid enough and has good natural frequency characteristic.

The static analyses of the shaft with fly-wheels and of the consoles showed that the maximum displacements of the supposed parts does not exceed 20 μm, that is considered as satisfactory. The maximum stress intensity of all parts is under 10 MPa in a stationary case and about 45 MPa (shaft) if the centrifugal loadings are included. These values are assumed sufficient with regard to static, dynamic and also possible fatigue loading of the brake stand components.

Acknowledgements

This work is the result of the project No. 2A-1TP1/080. This project has been solved with financial support from the state budget by the Ministry of Industry and Trade of the Czech Republic.

References

[1] O. Altuzarra, E. Amezua, R. Aviles, A. Hernandes, Judder vibration in disc brakes excited by thermoelastic instability, Engineering Computations 19 (2002), pp. 411–430.

[2] Yi. Yun-Bo, J. R. Barber, D. L. Hartsock, Thermoelastic instabilities in automotive disc Brakes — Finite element analysis and experimental verification, in J. A. C. Martins and Manuel D. P. Monteiro Marques eds., Contact Mechanics, Kluwer, Dordrecht (2002), pp. 187–202.

[3] P. Litoš, M. Honner, J. Šroub, V. Lang, The system for the measurement of thermo-mechanical instabilities of disk brakes, Proc. of 46-th Int. Sci. Conf. Experimental Stress Analysis 2008, VŠB Technická univerzita Ostrava, Ostrava, 2008, pp. 151–154.

[4] D. Hartsock, R. Dinwiddie, J. Fash, T. Dalka, G. Smith, Y. Yi, R. Hecht, Development of high speed system for temperature mapping of rotating target, Thermosense XXII (2000), pp. 2–9.

[5] V. Lang, J. Klepáček, J. Mertl, Honner, Laboratory for the experimental research of thermo-mechanical instabilities of disc and drum brakes, Proc. of 46-th Int. Sci. Conf. Experimental Stress Analysis 2008, VŠB Technická univerzita Ostrava, Ostrava, 2008, pp. 147–150.