



SIMULATION OF DYNAMIC RESPONSE OF A VIBROISOLATED OBJECT

Jan TARGOSZ¹
Jarosław BEDNARZ¹
Wojciech LISOWSKI¹

¹Akademia Górniczo-Hutnicza im. St. Staszica, Wydział Inżynierii Mechanicznej i Robotyki, A. Mickiewicza 30, 30-059 Kraków, Polska

DOI: <https://doi.org/10.24136/jae.2021.003>

Abstract – This paper presents application of Finite Element Method in designing of a vibration isolated object. A model of power generator of 113 MW was chosen for the analysis. Such the generator is used in one of power plants in Silesia district. Several frequency domain dynamic analyses aimed at assessment of influence of vibration isolation on acceleration level of the generator foundation were performed. Three different models of the generator were considered. For the purposes of verification of the obtained results measured vibration accelerations were used.

Keywords – Finite Element Method, Limiting of industrial noise, Modal analysis, Vibration isolation, Vibration of Machinery

INTRODUCTION

The article concerns the structural dynamic analysis of a part of a housing of a power generator of a unit at a Combined Heat and Power Plant [1].

A preliminary survey of the considered structure resulted in detection of cracks at the bottom of the generator's housing (Figure 1). The cracks are indicated in Figure 1 by elliptical loops were located in the area of connection of flat bars and metal sheets as well as in channel sections reinforcing the structure.

The aim of the research was to elaborate a concept of modification of the considered part of the generator housing aiding elimination of damage of its bottom plate.

The reported research consisted in:

- vibration acceleration measurements;
- experimental modal analysis;
- numerical analysis with use of Finite Element Method (FEM);
- development of a technical concept of vibro-acoustic protection based on FEM (dynamic and strength analysis) of a generator taking into account air circulation inside the generator's housing.



Figure 1. General view of a part of a damaged structure of the considered generator's bottom plate

The following sections present results of experimental and numerical analyses carried out in the course of the reported research.

I. EXPERIMENTAL ANALYSIS

The experimental analysis was composed of measurements of vibration acceleration in 3 mutually perpendicular directions of a part of the generator's bottom plate and operational modal analysis [2].

During the modal experiment, a network of measurement points covering 95 points was used. The assumed distribution of points was composed of 19 lines distributed across the tested plate on its main structural elements (thin and thick plates, U-bars). Each

measurement line consisted of 5 measuring points. The measuring point net is presented in Figure 2.

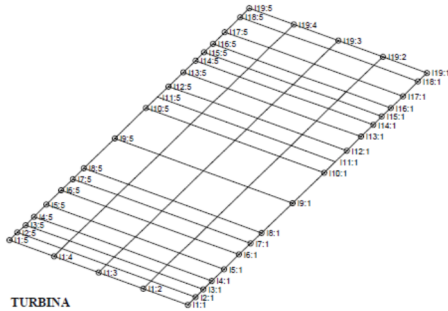


Figure 2. Measuring point net on the generator's bottom plate

The following coordinate system was used during the testing (directions assigned looking from the turbine side):

- the origin – edge of the plate at the outlet of the busbar on the right,
- X - horizontally along the plate in the direction from the busbar,
- Y - horizontally transversely to the right,
- Z - vertically downwards.

The measurement consisted of 19 partial experiments with a common reference measurement in each of them. The sampling frequency was set to 1024 Hz.

Figure 3 shows comparison of the amplitude spectra of the reference acceleration signal measured in each partial experiment. The graphs are dominated by a peak corresponding to the rotational speed of the generator rotor and its 2nd harmonic. The presented graphs proved that the reference direction acceleration spectrum amplitude did not vary substantially during the experiment, thus the recorded data could be used for modal analysis.

For modal parameter estimation LSCE method was used [2, 3] in frequency range up to 512 Hz. This system identification method is of output-only type and it does not require measurement of the excitation signal. There were operational mode shape identified of modal frequency very close to 50 Hz (corresponding to the rotational speed) and 100 Hz (the 2nd harmonic of the rotational speed) (Tab. 1). The two exemplary operational mode shapes estimated in the course of the analysis is presented in Figure 4. This picture indicates area of the bottom plate having the largest dynamic displacement during operation of the generator.

Table 1. Modal parameters estimation results

No.	Natural frequency [Hz]	Modal damping factor [%]
1	49.55	0.12
2	66.81	1.44
3	84.81	1.35
4	99.67	0.17
5	124.96	0.64

6	149.47	0.10
7	199.49	0.01
8	259.03	0.36
9	299.74	0.01
10	350.50	0.72
11	359.60	0.47
12	399.73	0.01
13	499.63	0.01

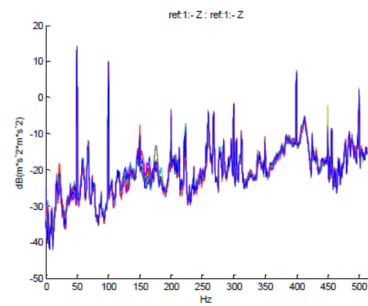
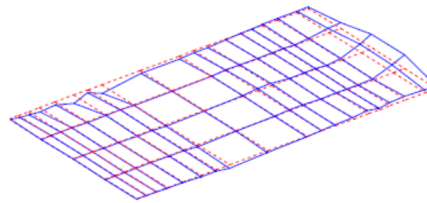
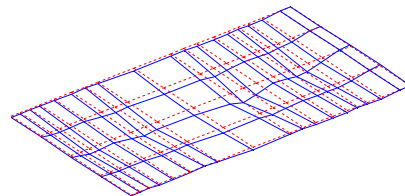


Figure 3. Comparison of amplitude of vibration acceleration measured at the reference point in 19 partial experiments



49.55 [Hz], 0.12 [%]



99.67 [Hz], 0.17 [%]

Figure 4. Identified operational vibration modes

The corresponding deflections of the structural elements of the bottom plate might be responsible for cracks, which in turn indicated direction of modification of the bottom plate aiming at elimination of source of damage. The results of the performed experiment were used as the starting point of the numerical simulation.

II. SIMULATION ANALYSIS

The purpose of this study was to modify stiffness [4] of the lower housing of the considered generator.

The carried-out analysis was based on the Finite Element Method (FEM), by means of which several dynamic analyses were performed in order to verify the influence of individual changes in the structure on the behavior of the shield forces for the critical flexural vibration form. A finite element model was adapted as the description of the model analyzed and the description of the analysis. A finite element model of the generator was made including the stator, rotor and the frame on which the generator is placed. Due to the lack of precise data on generator dimensions and mass distribution, a simplified model was made in which some dimensions were estimated on the basis of local inspection and measurements. The rotor was modeled with using beam elements connected to the frame with rigid elements with all degrees of freedom. The stator was modeled by increasing the density of the ribs on which it is supported. Such simplification takes into consideration only the mass effect of the winding and all stator elements. The lack of consideration of stiffness effects was caused by concentrating the forces on the generator shield and not the structure of the entire generator. The structure of the ribs was modeled with use of shell elements. The connecting tubes of the ribs were modeled using beam elements. The frame and the bottom housing were modeled using shell elements including the frame ribs and a simplified model of the generator current drain. The finite element grid was constructed taking into consideration the recommendations for the number of nodes per the longest wavelength in the range of interest, i.e., up to 200 Hz. This range resulted from the maximum range in which significant natural frequencies of the shield, identified in the result of experimental measurements, could occurred. The average size of the finite element was 50 mm, by which the condition of considering at least 10 nodes per wave period was met. Due to the simplifications adapted in creating the model, which also took into account the omission of the foundation structure and the influence of the turbine, special criteria were assumed for the finite element analysis. The optimization was based on the identification of the flexural vibration form of the model and the study of the effect of individual structural modifications on the value of modal displacements. The smaller value the more effective the particular modification was. The results were related to two reference models, i.e., the present structure and the original structure in which the stiffening channels were welded [4]. Modal displacements in the case of the analysis of the dynamic response of the system to the excitation (in this case the force coming from the unbalance of the rotor) linearly scale the value of the real displacements of the model for particular natural frequencies and therefore they can be adapted as the criterion of the influence of particular structural changes on the value of the vibration amplitude of the shield. The view of the generator model (Figure 5), and the coordinate system is shown in the following figures. Additionally, in the

figure below, the modified area of the generator's bottom shield is marked with red lines.

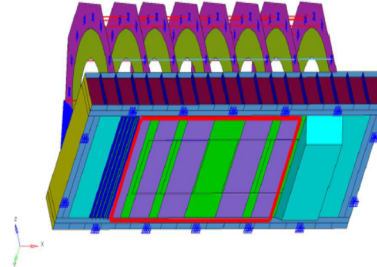


Fig. 5. A view of the finite element model of the generator with the marking of the optimized area of the bottom shield

The boundary conditions were assumed such that the model was restrained where the generator frame was bolted to the foundation. All degrees of freedom were restrained. Due to the simplification of the model, the bolts were not modeled with an accurate representation of the head and washer surfaces, but only a few nodes were restrained in the immediate vicinity of the bolt. A visualization of the boundary conditions is shown in Figure 6.

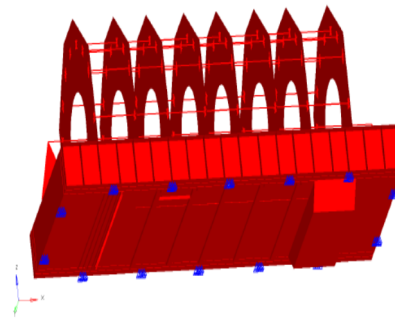


Fig. 6. View of restraints in the considered model

In the simulations, calculations were performed for eight models of finite elements [1], allowing to present both existing solutions and the best modifications of the structure. Figure 7 presents the final modification of the generator housing, and Figure 8 as well as Figure 9 show the results of its simulation.

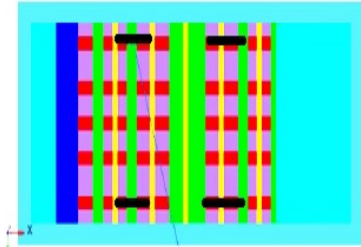
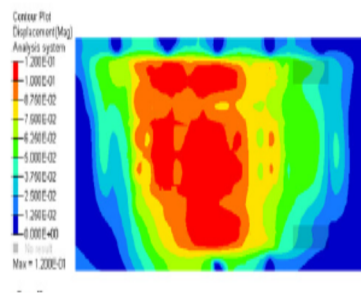


Fig. 7. Stiffness correction consisting of 5x 15 mm flat bars (red), 100x150 U bar on 18 mm thick flat bar and additional 50 x 20 mm flat bars (black)

Analyses based on the finite element method (FEM) indicate that the final modification assured the most favorable structural change of the bottom plate of the aggregate. The value of modal displacements was found to be 30% lower than the ones of existing design, which directly affected the reduction of the amplitude of the analyzed shield vibration.



Rys.8. Simulation result for the stiffness correction consisting of 5 x 15 mm flat bars, 100 x 150 U bar on 18 mm thick flat bar and additional 50 x 20 mm flat bars

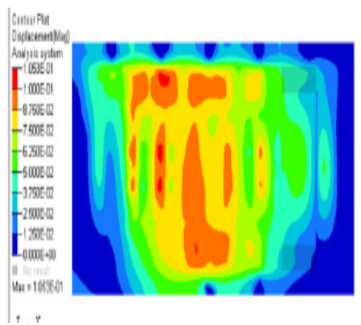


Figure 9. Result of a modification for the model in which only 5 I-sections of 100x100 were used as the stiffeners

III. CONCEPT OF VIBRATION ISOLATION OF THE GENERATOR FOUNDATION

Basic requirements concerning foundations of turbogenerators in comparison to other machines refer to the following issues [5-11]:

- The resonance operation is unacceptable by which the natural frequencies of the foundation cannot coincide with the operating frequencies of any of the machines. A foundation may be high tuned if its eigenfrequencies are above the operating frequency of the turbine generator or low tuned if its eigenfrequencies are below the operating frequency.
- The allowable displacement amplitudes at the turbogenerator operating frequencies shall not exceed the extreme values specified by the turbogenerator component manufacturer.
- The eccentricity of the base dimension, in relation to which the center of gravity is displaced, may not exceed 3%. The reason for such requirement is to minimize the moment of inertia, which may significantly affect the values of natural frequencies of the foundation.

Proper selection of the vibration isolation parameters is one of the most important aspects of the generator foundation design process. The purpose of vibration isolation is to isolate the foundation from the vibrations coming from rotating machines by appropriately selecting the eigenfrequencies of vibration isolators, which are usually in the very low range of 1-5 Hz, and to provide proper support for the static mass of the generator with the foundation [5, 6, 12]. The applied vibration isolation in form of helical spring packages is one of the most popular solutions used in industry (Fig. 10). In the tested model, different load values were applied to the outer and middle columns, i.e., 1030 kN for the boundary columns and 1200 kN for the middle columns. Each battery consists of 18 springs. The measured static deflection is approximately 15 mm.



Figure 10. View of vibration isolation package for one of the columns

A finite element model of the generator was made including the structure of stator, rotor and the frame on which the generator is placed as well as the foundation

structure including the vibration isolation and the columns on which the whole structure is placed (Figure 11).

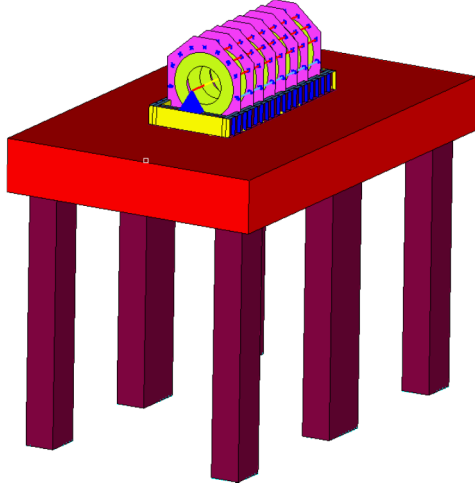


Figure 11. A view of finite element model of the generator with foundation and vibration isolation system

The rotor was modelled by means of beam elements connected to the frame structure by rigid elements in all degrees of freedom. The stator was modeled by increasing the density of the ribs on which it is supported. Such simplification takes into consideration only the effect of mass winding and all stator elements. The structure of the ribs was modelled with the use of shell elements. The connecting tubes of the ribs were modeled using beam elements. The frame and the bottom housing of the generator were modelled using shell elements including the frame ribs and a simplified model of the generator current drain. The foundation was constructed with eight-node solid elements without including reinforcement due to lack of data. In addition, all external surfaces of the concrete were covered with 5mm thick sheet made of shell elements. The vibration isolation was modeled by BUSH elements allowing for the definition of stiffness in all directions as well as damping for a single element.

The finite element analyses were performed in MSC. Nastran software. This program is appreciated standard for dynamic analyses. The dynamic analyses were based on the principle of modal superposition based on performing simulation in two steps. In the first step, a modal analysis is performed to determine eigenfrequencies and modal forms in a certain frequency range. The range should be selected so that it is about twice higher than the highest frequency for which the analysis is performed in the second step, i.e., examination of response of the system to dynamic excitation in frequency domain. The excitation is the value of unbalance force, which can change as a function of frequency. Due to the constant value of rotor speed i.e., 3000 rpm, the frequency of the forcing force had a constant value.

The advantage of using the modal superposition approach consist in the single analysis of a first step i.e., modal analysis is done once and the analysis can be restarted multiple times for different excitations and frequency ranges. This significantly reduces the analysis time. Due to vibration measurements indicating 100 Hz as the most critical frequency, analyses were performed in the range of 0-130 Hz. The unbalance force was applied on the rotor and consisted of two components offset by 90 degrees to model the rotational motion of the vibrator. The results of the analyses of the dynamic effects of the object are values of accelerations, displacements and, among others, stresses, which allow to determine the fatigue life of components exposed to excessive vibration. The computational model was restrained in six degrees of freedom in place of foundation of columns on which the entire structure of the foundation and the generator was mounted.

Two different models were prepared:

- a model with a rigid connection between the columns and the generator foundation,
- a model for vibration isolation with nominal values.

Table 2 summarizes the most important material data of the components of the FEM model. These data were adapted from the available literature.

Table 2. Material parameters

Steel	
Density [t/mm ³]	7.86E-09
Damping factor	0.02
Poisson factor	0.3
Young's modulus [MPa]	2.10E+05
Concrete	
Density [t/mm ³]	2.40E-09
Damping factor	0.02
Poisson factor	0.2
Young's modulus [MPa]	30000

IV. ASSEMENT OF EFFECTIVENET OF THE CONCEPT OF THE VIBRATION INSULATION

The obtained simulation results proved the effect of vibration isolation on the level of acceleration in both vertical and transverse directions. The amplitude in the vertical direction for the measurement point below the vibration isolation is about 12 times smaller, which is consistent with the industrial practice in this type of structures.

Dynamic analyses with the vibration isolation showed, similarly to the experimental studies, the excellent efficiency of such solution, which is necessary for proper functioning of rotating machinery of significant masses. Similar to the experimental studies, for columns in the Z direction, the dominating frequency is 100 Hz. As an example, simulation results are presented for the outermost and middle columns (Figure 12) before and after

introduction of vibration isolation (Figure 13 and Figure 14)

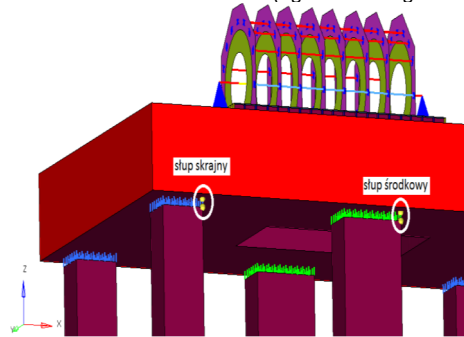
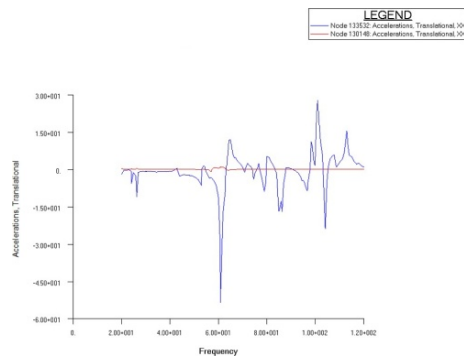
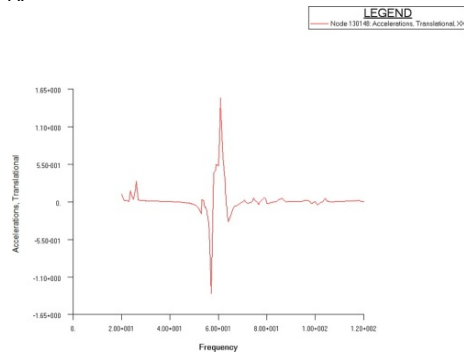


Figure 12. View of measurement points for the outermost and middle columns upstream and downstream the vibration isolation and for the coordinate system



A.

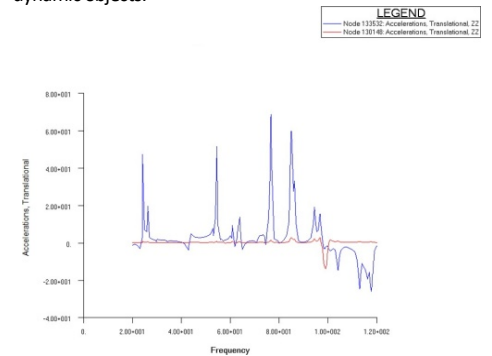


B.

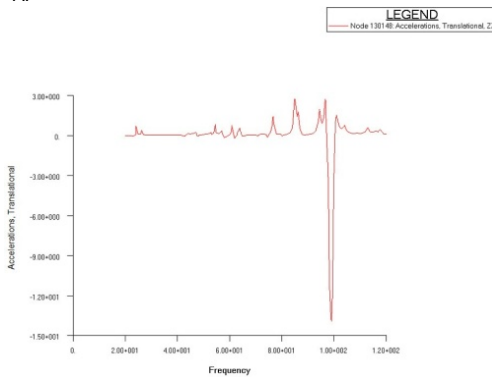
Figure 13. Acceleration course for the outermost column in X direction: A. – upstream and downstream the vibration isolation, B. – course downstream the vibration isolation. Red color (column), blue color (foundation)

The performed analyses show the great possibilities of using the FEM in the design process of vibration-isolated structures [6]. Due to the enormous cost of such objects as well as the one-time production, the use of FEM allows to

accurately verify the entire structure not only for the dynamic behavior of the object, but also for the structural strength and resistance to earthquake-type excitations. The complexity of such an object requires, unfortunately, a very accurate reconstruction of the whole structure together with taking into account the correct material parameters or at least damping. The research presented in this paper was based on a limited amount of data and therefore cannot be expected to be very consistent with experimental measurements. Particularly important is the consideration of the soil on which the columns are placed. Nonetheless, the idea of vibration isolation and its influence on the behavior of the object were effectively presented. The performed analyses can be used to determine, among others, the fatigue life at harmonic excitations, tests on propagation of vibrations in the ground from all types of dynamic objects.



A.



B.

Figure 14. Acceleration course for the outermost column in Z direction. A. – upstream and downstream the vibration isolation. B. – course downstream the vibration isolation. Red color (column), blue color (foundation)

V. A CONCEPT OF LIMITING NOISE EMISSION THROUGH THE BOTTOM PLATE OF THE GENERATOR K

Statistical and deterministic methods are used to evaluate the sound field distribution. The first group of methods is simple in mathematical description, allowing to

obtain the given result easily. However, the obtained results are usually burdened with a considerable error. The second group (most often, these are radial methods, FEM, etc.) allows one to obtain satisfactory results only with a huge effort (not commensurate with the benefits obtained). In some cases, the results obtained are even subject to greater error than in the case of the statistical methods. A different approach to the sound field distribution in an industrial hall is also possible by combining the statistical method with empirical results of sound pressure distribution.

The value of the sound level in a given room (industrial hall) depends not only on the sources located in it but also on their distribution, shape and acoustic properties of the room itself. The volume of the air surrounded by walls, ceiling and floor (room) and inside arranged with various types of machines creates a specific acoustic system. This acoustic system may be studied using various methods. Using the statistical method, it is assumed that the acoustic field inside the system is homogeneous, i.e., that in all places of the investigated area the equilibrium state has been established. This method gives the most accurate results when the average absorption coefficient is the lowest (i.e., the most significant part of acoustic energy is the energy of the reflected waves). On the other hand, when the sound sources have large dimensions (i.e., they are acoustic screens for direct waves from other sources) or the room is substantially non-convex the errors made with the theoretical methods are very large.

For computer simulation of the sound field distribution A special computer software was developed in order to determine the sound field distribution in the Generator Hall. Acoustic field distribution was determined on the assumption that the acoustic field inside the hall has a partially dispersed character. The diffuse part of acoustic field was determined on the basis of statistical relations. In the vicinity of sound sources, the sound field was specified as determined. Moreover, corrections were made to take into account the specific shape of the Generator Hall. The data for the computer software have been chosen on the basis of performed measurements [10].

As a result of computer simulation, the distribution of equilibrium level of A-weighted noise at the level of 1.6 m above the hall floor without and after realization of acoustic shielding has been obtained [11]. Figure 15 shows the predicted distribution of the level without acoustic shielding. Figure 16 presents the predicted distribution of the A level with acoustic shield.

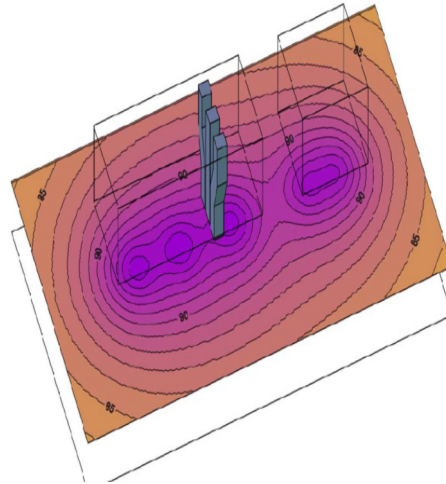


Figure 15. Predicted distribution of A sound level without acoustic shield

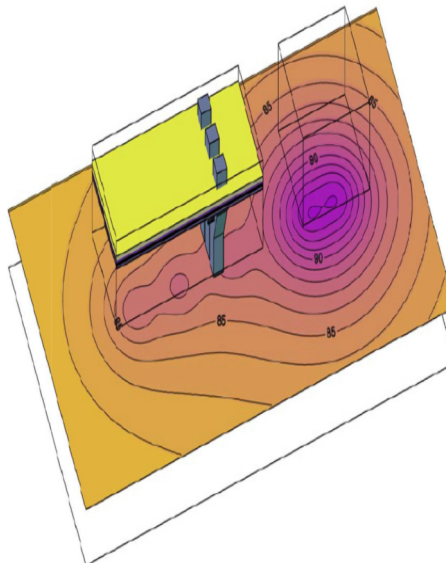


Figure 16. Predicted distribution of A sound level with acoustic shield

VI. CONCLUSIONS

The paper presents a sequence of activities aiming at formulation of a concept of modification of a part of a power generator structure with the aim of decreasing intensity of development of its damage resulting from its operation.

The performed modal experiment and its results showed that the studied system vibrates mainly at frequencies $n \times 50\text{Hz}$ ($n=1,2,\dots$), which indicated that the dominant excitation is related to the rotational speed of the aggregate shaft.

The results of the modal tests have shown that the dominant vibration mode was around 100 Hz. Therefore, it was reasonable to modify the structure of the bottom plate of the aggregate so as to reduce the amplitudes of the modal displacements.

Analyses based on the Finite Element Method (FEM) indicated the most favorable structural modification of the bottom plate of the aggregate. The value of modal displacement became 30% lower than that of the existing design, which directly reduced the amplitude of the analyzed shield vibration.

The vibration isolation system in z-axis direction was selected.

The use of sound insulating enclosure was suggested. Its effectiveness was confirmed by numerical simulation, which showed significant reduction of the level of sound emitted by the generator to the surroundings

SYMULACJA ODPOWIEDZI DYNAMICZNEJ WIBROIZOLOWANEGO OBIEKTU

W artykule przedstawiono zastosowanie metody elementów skończonych w projektowaniu obiektów wymagających zastosowania wibroizolacji. Jako model testowy wybrano generator o mocy 113 MW. Generator ten jest eksploatowany w jednej z elektrowni na Śląsku. Przeprowadzono kilka analiz dynamicznych w dziedzinie częstotliwości dla trzech różnych modeli, aby zweryfikować wpływ wibroizolacji na poziom przyspieszenia na podłożu. W przeprowadzonych badaniach wykorzystano również wyniki pomiarów przyspieszenia drgań.

Keywords: Analiza modalna, Drgania maszyn, Metoda elementów skończonych, Ograniczanie hałasu, Wibroizolacja

REFERENCES

- [1] Ekspertyza generator bloku nr 1 w Elektrociepłowni Chorzów "ELCHO" Sp. z o.o.
- [2] Uhl T., Lisowski W. (2000) Identification of modal models of building structures based on in-operation measurements, Archives of Civil Engineering, XLVI, 1, pp. 183-206
- [3] Hermans L., Van der Auweraer H., Coppens P., Mathieu L. (1996) Modal parameter extraction from in-operation data, XV IMAC, Orlando, pp. 531-539
- [4] VDI 2056 (1964) Evaluation of Mechanical Vibration of Machines
- [5] Goliński J.A.: Wibroizolacja maszyn i urządzeń, WNT warszawa 1979.
- [6] Bowels, J.E., "Foundation Analysis and Design", 5th Edition, McGraw-Hill, 1996
- [7] ISO 1940-1 (1993) Mechanical Vibration – Balance Quality Requirements of Rigid Rotors – Part 1: Determination of Permissible Residual Unbalance
- [8] ISO 10816-1 (1995) Mechanical Vibration – Evaluation of Machine Vibration by Measurements on Non-Rotating Parts – Part 1: General Guidelines
- [9] ISO 10816-2 (2001) Mechanical Vibration – Evaluation of Machine Vibration by Measurements on Non-Rotating Parts – Part 2: Land-Based Steam Turbines and Generators in Excess of 50 MW
- [10] DIN 4024 Part 1 (1988) Machine Foundations: Flexible Structures Supporting Machines with Rotating Masses
- [11] ACI 351.3R (2004) Foundations for Dynamic Equipment
- [12] Norma PN-85/B-02170 pt: Ocena szkodliwości drgań przekazywanych przez podłoże na budynki.