Sławomir JURKOWSKI[1], **Edmund KULAWIK**[2]

[1] State University of Applied Sciences in Nowy Sącz, Institute of Technology, 33-300 Nowy Sącz, Zamenhoffa 1A street, e-mail: sjurkowski@pwsz-ns.edu.pl
[2] State University of Applied Sciences in Nowy Sącz, Institute of Technology, 33-300 Nowy Sącz, Zamenhoffa 1A street, e-mail: kedmund@wp.pl

Possibilities of using the FEM optimization package to improve the structure of self-supporting stairs in a building

Abstract

The article presents the results of the analyzes on the influence of the optimization self-supporting staircase geometry on its strength and dynamic properties. The primary goal was to achieve a structure with a lower mass. The Topology Optimization tool from the ANSYS package allowed to the elimination of the finite elements carrying the smallest stresses from the structure. The iterative approach resulted in a systematic reduction of the model mass up to decrease by 40 (%). The reduction in the structure led to a decrease in its strength – the safety factor decreased from 5.95 to 4.92. The influence of the optimization of the structure on its dynamic properties was determined as an additional aim of the study.

Key words: FEM, optimization, mode shape, resonance, staircase, ANSYS.

Możliwości zastosowania pakietu optymalizacyjnego FEM do poprawy konstrukcji samonośnych schodów w budynku

Streszczenie

Artykuł przedstawia wyniki analiz dotyczące wpływu optymalizacji geometrii klatki schodowej o konstrukcji samonośnej na jej właściwości wytrzymałościowe oraz dynamiczne. Podstawowym celem było uzyskanie konstrukcji o mniejszej masie. Wykorzystanie narzędzia Topology Optimization z pakietu ANSYS pozwoliło na wyeliminowanie z konstrukcji elementów skończonych przenoszących najmniejsze naprężenia. Iteracyjne podejście skutkowało systematyczną redukcją masy modelu aż do założonego jej spadku o 40 (%). Redukcja ta doprowadziła do nieznacznego zmniejszenia wytrzymałości konstrukcji – współczynnik bezpieczeństwa zmniejszył się z poziomu 5,95 do 4,92. Jako dodatkowy cel badania założono ustalenie wpływu przeprowadzenia optymalizacji konstrukcji na jej cechy dynamiczne.

Słowa kluczowe: MES, optymalizacja, postać drgań własnych, rezonans drgań, klatka schodowa, ANSYS.

1. Introduction

Currently analyses based on the finite elements method (*FEM*) are gaining increasing popularity in numerous branches of science and technology. Application of computer simulations markedly reduced the time between preparing a conceptual design and creating a finished product. Simulations are used not only to perform strength and fatigue tests or dynamic analysis of an ultimately finished and shaped structure of an object but, even more importantly, also to utilize the collected data in order to determine the most advantageous geometry for the purpose of optimization.

The *ANSYS* software provides users with *Topology Optimization* module which enables modeling an object of study through iterative method and determining the optimal geometry of an object on the basis of the collected data. The basic comparison of the results achieved prior and post re-design is based on the strength analysis; however, for the purpose of this studies the approach taken has been expanded with *Modal Analysis* and *Harmonic Response* modules.

The goal of the study is to demonstrate the applicability of the optimization package implemented within the FEM package for modifying structure of a staircase which is to achieve lower mass with simultaneous preservation of safety conditions in terms of strength. The optimization criterion was reducing mass of stairs through redesigning supporting structure for steps. Furthermore, the article analyzes influence of changes introduced into the construction on its dynamic properties (lowest forms of intrinsic vibrations and harmonic response).

The finite elements method utilized to optimize construction structures has been presented by Fai et al. (2002). The goal of the study is to answer the question how strength properties of a remodeled structure and its dynamic parameters (frequency and form of intrinsic vibrations) change after optimization. This issue is crucial for safety and comfort of using a staircase.

2. The studied structure and its FEM model

A geometry of a self-supporting staircase based on standard steel, cold formed structural sections has been designed as a starting point. In turn, steps of the stairs have been designed using walnut wood (Figure 1, designation 4) whereas compilation of the essential material properties is presented in Chart 1. Dimensions of the structure are presented in Figure 1. The steel profiles utilized in construction are: angle bars 30 x 30 x 2 (mm) – (Figure 1, designation 3); flat bars 20 x 2 (mm) – (Figure 1, designation 2); closed profiles 100 x 50 x 2 (mm) – (Figure 1, designation 3).

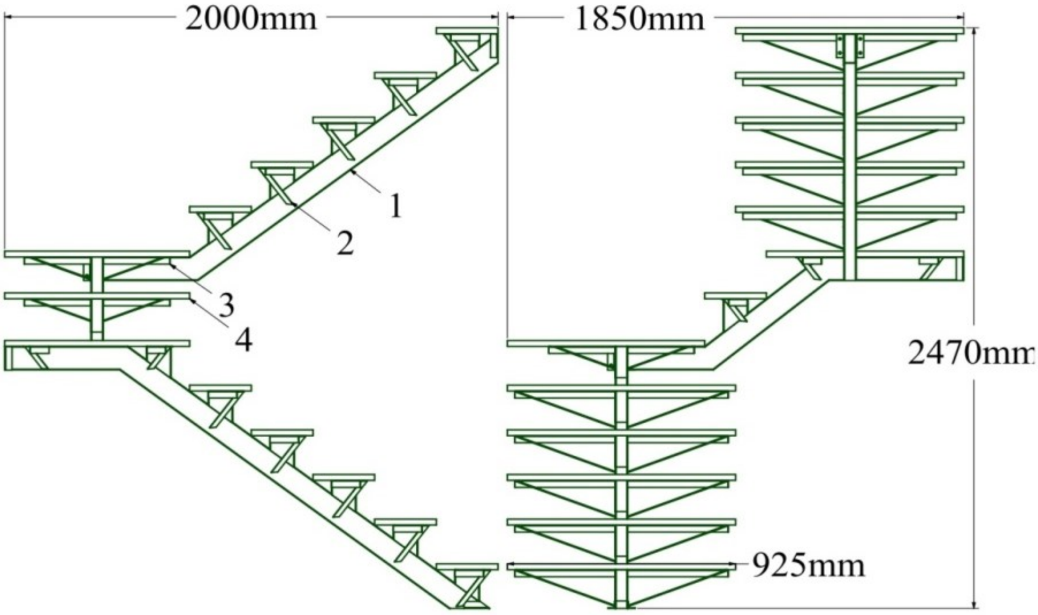

Figure 1. Basic dimensions of the staircase

(source: own study)

Table 1

Parameters of the construction materials utilize in the simulation

Material	Density ($\frac{kg}{m^3}$)	Young's Modulus (MPa)	Yield strength (MPa)	Tensile strength (MPa)
Constructional steel	$7{,}85e^{-7}$	$2e^5$	250	460
Walnut wood	$6{,}12e^{-7}$	12700	50.8	93.1

Source: own study

The optimization and numerical calculations procedure is presented in Figure 2. Firstly the staircase load-bearing structure adapted to the architectural design of a house and accounting for anchorages fitted to construction elements of house's walls has been designed. Subsequently a static strength analysis of a part of the staircase including the main load-bearing profile and step support has been performed using *Static Structural* module. The load applied to the upper surface of a step was 980 (N), a value which corresponds with the load applied by a human.

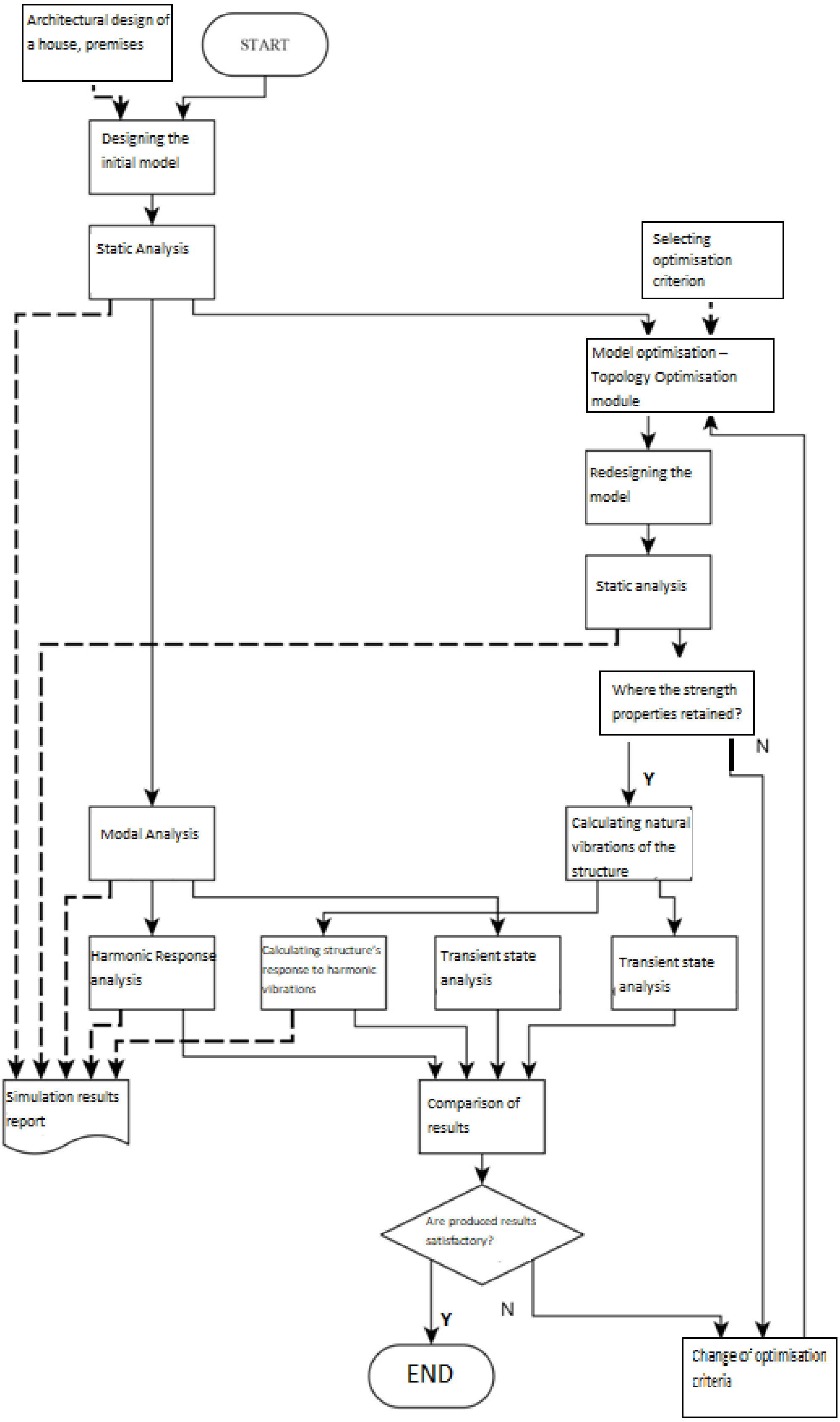

Figure 2. Flowchart of numerical analysis progress

(source: own study)

The data collected during the static analysis were used in the *Topology Optimization* module to determine which areas of the structure were under the smallest load and, in consequence, could be removed from the model. The analysis and removal of appropriate finite elements is being performed iteratively until the optimization criterion is achieved. The detailed analysis of the optimization algorithm utilized in ANSYS software was performed by Dheeraj and Anadi (2012). The *Percent to Retain* coefficient in the *Topology Optimization* module determining retention of mass of the initial model has been set to 60 (%). The repeated static analysis is to check whether the optimized construction will transfer projected loads (the construction stresses calculated in accordance with H-M-H (Huber-Mises-Hencky) stress hypothesis) will not exceed the yield strength).

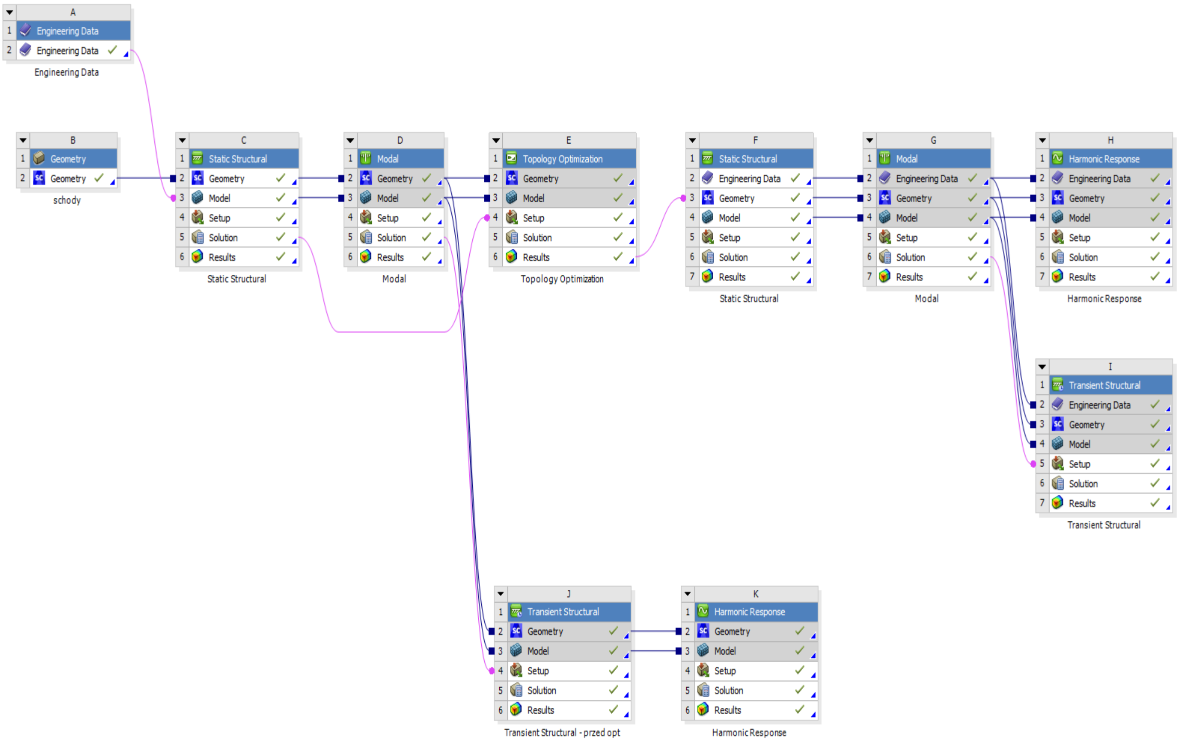

Figure 3. ANSYS software simulation flowchart

(source: own study)

The remaining two calculation models are used with the goal of determining frequency of intrinsic vibrations of the structure (*Modal Analysis*) and structure's response under load and during work in the resonance zone as well as the corresponding mode shape of vibrations (*Harmonic Response*). ANSYS software flowchart for the task has been created following analysis of scientific papers (Mohammad et al., 2015; Obrocki et al., 2014) and supplementing the analysis with own input; the flowchart is displayed in Figure 3 with the individual blocks depicting:

1. A – material data used for numerical calculations,
2. B – geometry of the source model for the designed structure,
3. C, F – static strength analyses module,
4. E – structure geometry optimization module,
5. D, G – modal analyses modules,
6. H, K – modules for calculating forced harmonic oscillations,
7. I, J – modules for analyzing transition state.

3. Results of analyzes

Preparing a calculation environment primarily requires unification of units. In the considered case *Metric* system units (mm, kg, N, s, mV, mA) have been used in both the *Workbench* module as well as the modules utilized in simulation. The spatial model of the structure has been developed in *Space Claim* module and the process of dividing structure's solid figure into finite elements has been performed by using *Mechanical* module for individual settings of meshing – separately for each wooden step as well as the steel structure. In the first case the length of an edge of a finite element had been set to 5 (mm) and to 3 (mm) in the second case. The meshing parameters do not have such an important influence for calculation results in case of the mechanical analysis in comparison to the calculations performed for the issues of flow described by Jurkowski and Janisz (2020) and thus the number of finite elements has been reduced. The character of joints between two solid figures has been set to *Bonded* – a joint which locks all six degrees of freedom between the connected elements. Technical parameters of the model are presented in Table 2 and 3.

Table 2

Compilation of properties characterizing the source model in analyzes

Object	Mass (kg)	Number of nodes	Number of finite elements	Volume (mm^3)
Step	3.538	365975	239986	5.781e^6
Supporting structure	2.972	265463	132525	3.786e^5
Sum	6.51	631438	372511	6.16e^6

Source: own study

Table 3

Compilation of properties characterizing the model post optimization

Object	Mass (kg)	Number of nodes	Number of finite elements	Volume (mm^3)
Step	3.538	365975	239986	5.781e^6
Supporting structure	1.957	177703	88531	2.493e^5
Sum	5.495	543678	328517	6.03e^6

Source: own study

Table 4 presents duration of individual stages of simulation studies as well as the computing power utilized for the presented parameters for models which served as the basis for analyses and calculations. The process of structure optimization has been shortened owing to reducing the number of finite elements and, consequently, reducing the number of nodes for which calculations are performed; however, the size of the file containing results increased due to generating a very complex geometry of the system.

Table 4

Summary of computing time and resources utilized for the source model and optimized model

Stage of analysis	Computation time (min)	Memory use (GB)	Size of output file (MB)
Static Analysis	2.16/1.41	5.89/4.72	428/371
Topology Optimization	28.4/-	5.9/-	10850/-
Modal Analysis	6.58/7.3	5.01/5.66	253/2216
Harmonic Response	39.36/24.6	9.28/10.47	18679/16212
Sum	76.5/33.31	26.08/20.85	30210/18799

Source: own study

a. Static analysis

In terms of strength both structures (the initial structure and the redesigned structure) are fully capable of transferring the projected load. Before improving geometry of the stairs the safety factor, calculated in accordance with formula (1), was 5.95 and it was 4.92 in the modified structure. The distribution of stresses in the structure is presented in Figures 4 and 5.

$$SF = \frac{R_e}{\sigma_{eq}} \tag{1}$$

where: SF – safety factor, R_e – yield strength (MPa), σ_{eq}- maximum equivalent stress (MPa).

In both cases the deflection of structure under load has also been analyzed – it can be observed that the value of maximum deflection increased from 0.068 (mm) to 0.082 (mm) as presented in Figure 5 and 6.

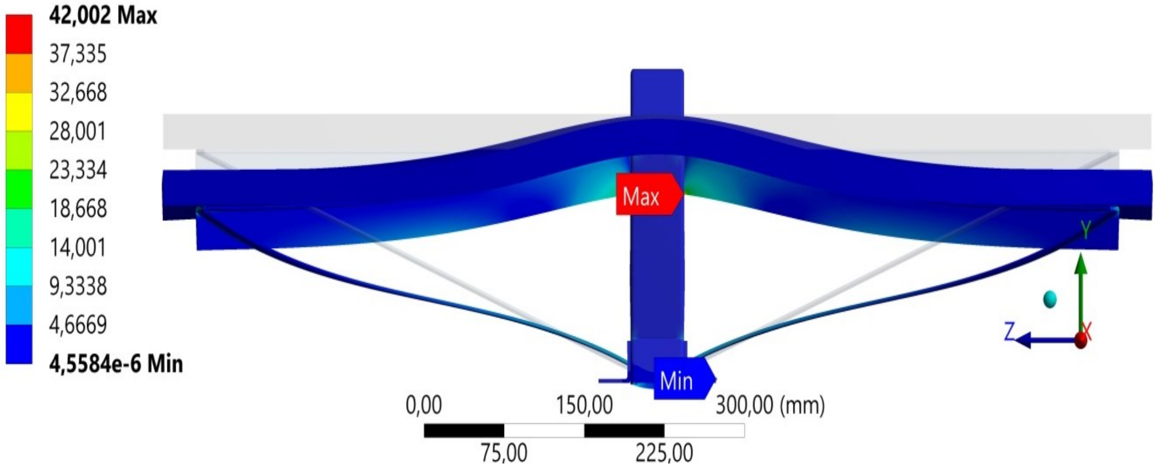

Figure 4. Distribution of stresses in the initial structure

(source: own study)

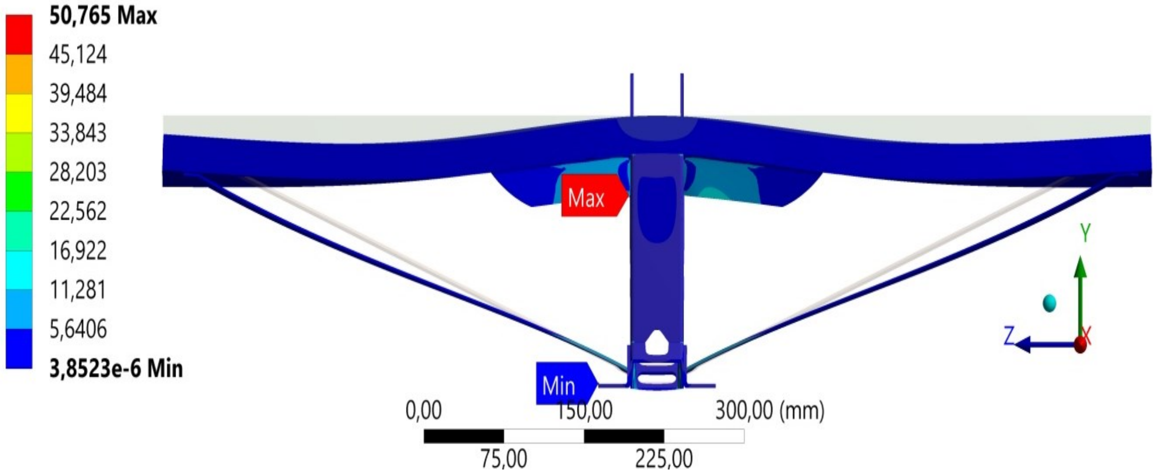

Figure 5. Distribution of stresses in the optimized structure

(source: own study)

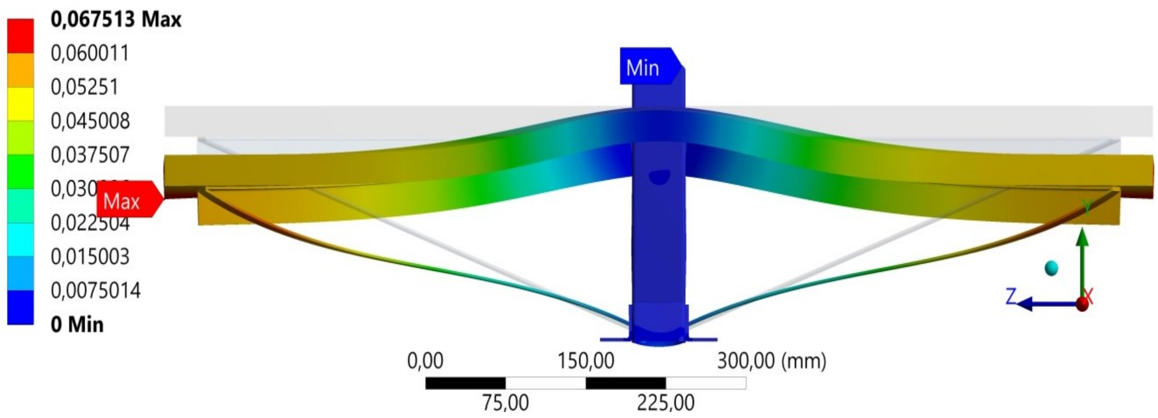

Figure 6. Distribution of deflections in the initial design of the structure

(source: own study)

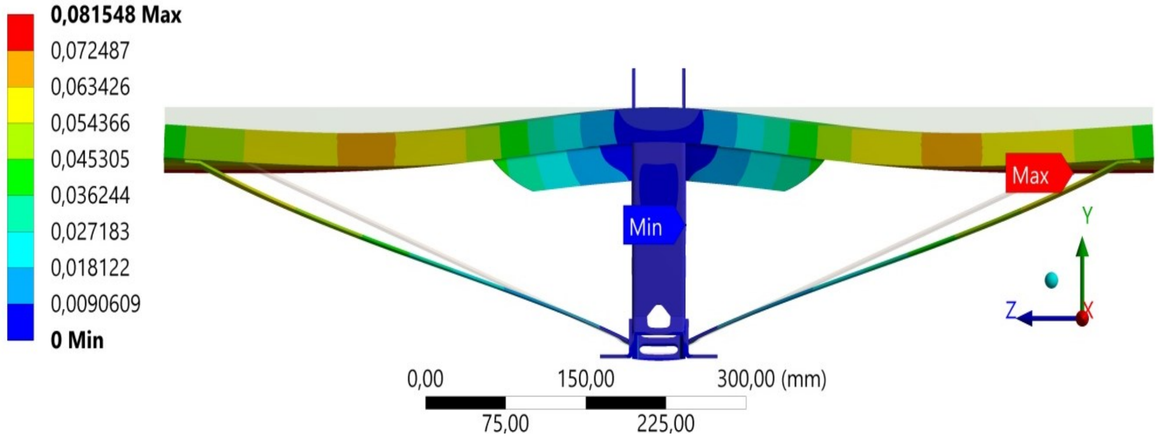

Figure 7. Distribution of deflections in the improved design of the structure

(source: own study)

b. Optimization of the model

The main goal of the computation cycle was reducing mass of the structure. This was reflected in the settings of the *Topology Optimization* module by setting participation of an unaltered part of the model – *the Percentage to Retain* coefficient – to 60 (%). The results of applying the optimization algorithm are depicted in Figure 6 where the parts of the initial structure remaining after applying optimization algorithm are indicated in gray. After generating a raw optimized model two things can be done – the model can be directly converted into an **.stl* file or geometry of the model can be utilized to redesign the structure. In the considered case the second approach has been utilized which enabled obtaining a model with better quality values as well as higher technological aptitude of the structure in comparison to the degenerate "optimal" topology according to Buhl et al. (2000).

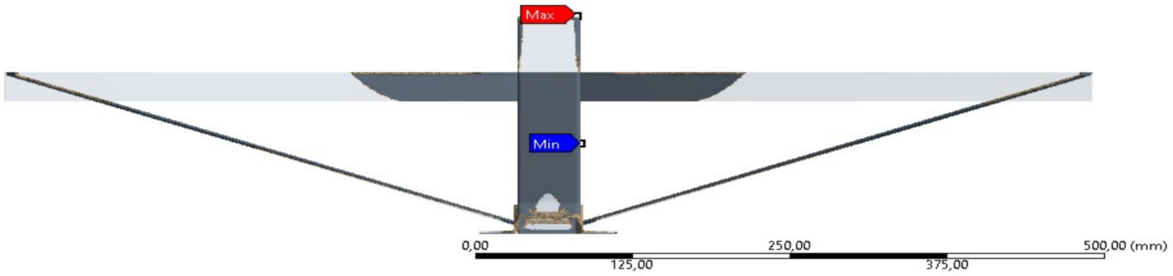

Figure 8. Raw structure model post optimization

(source: own study)

c. Modal analysis

In accordance with the secondary goal of the studies a simulation studies have been performed within the scope of modal analysis in order to determine frequency of intrinsic vibrations as well as their mode shape. The issue of various applications of modal and dynamic analysis of a structure has been presented by e.g. Yongchang et al. (2017). Similarly to ANSYS Advantage Staf (2014) the *Modal Analysis* module has been used to perform analysis of both the initial and optimized structure. The results of analyses of both structures are presented in Table 5. It can be observed that additional mode shapes and corresponding frequencies appear in the process of optimization (Table 5, items 3 and 4) whereas mode shape and frequency for item 12 in Table 5 do not appear. The results of the analysis were compared on the ground of determining the so called "frequency error" determined by the following relation:

$$\varepsilon = \frac{\omega_p - \omega_o}{\omega_o} \cdot 100\% \qquad (2)$$

where: ε – frequency error, ω_p – frequency of oscillations prior to optimization, ω_o– frequency of oscillations post optimization.

Table 5

Comparison of resonance frequency for source model and optimized model

No. of mode shape	Frequency – source model ωp [Hz]	Frequency – optimized model ωo [Hz]	Frequency error ε [%]
1	51.258	48.786	4.8%
2	51.41	48.864	5.0%
3	x	87.579	x
4	133.46	131.36	1.6%
5	135.16	138.79	-2.7%
6	140.9	145.79	-3.5%
7	150.53	x	x
8	208.49	201.98	3.1%
9	276.95	273.82	1.1%
10	277.74	275.72	0.7%
11	x	297.7	x
12	330.3	x	x

Source: own study

The produced results only allow us to determine the value of frequency and response of the structure to being subjected to resonance frequencies. It can be observed that despite reduction of mass by 40 (%) the frequency of intrinsic vibrations has not changed significantly. However, as suggested by Żółtkowski and Jędrzejak (2015), the calculated deflection resulting from structure's response can be treated only illustratively (e.g. for the purpose of determining the frequencies resulting in the strongest response).

In his book Fiebig (2019) suggests that the FEM analysis should be validated and verified through experimental modal analysis – an action impossible to perform at the design stage. Supplementing modal analysis with analyzing response of the structure enables to confirm correctness of the data collected through modal analysis to a certain degree.

In order to answer the question what deflection and stresses will be recorded within the structure it is necessary to complement the performed analyses with the simulation of forced vibrations under resonance conditions (*Harmonic Response module*).

d. Harmonic Response

The final stage of analyses concerned harmonic response and impulse response simulating jumping on a step by a human. A downward perpendicular force of 980 (N) has been applied to the surface of a step. The vibrations were induced with a frequency ranging from 0 to 300 (Hz).

Comparison of distribution of stresses recorded in two supporting structures – the initial structure (upper graph) and the optimized structure (lower graph) is presented in Figure 9. The maximum stresses of 1.4098 (MPa) were recorded in the initial design at a frequency of 207 (Hz) whereas for the optimized structure the frequency of 201 (Hz) generates stresses with the value of 0.4236 (MPa). Similar results can be observed in comparison to results for natural frequencies, 208.49 (Hz) and 201.98 (Hz) respectively.

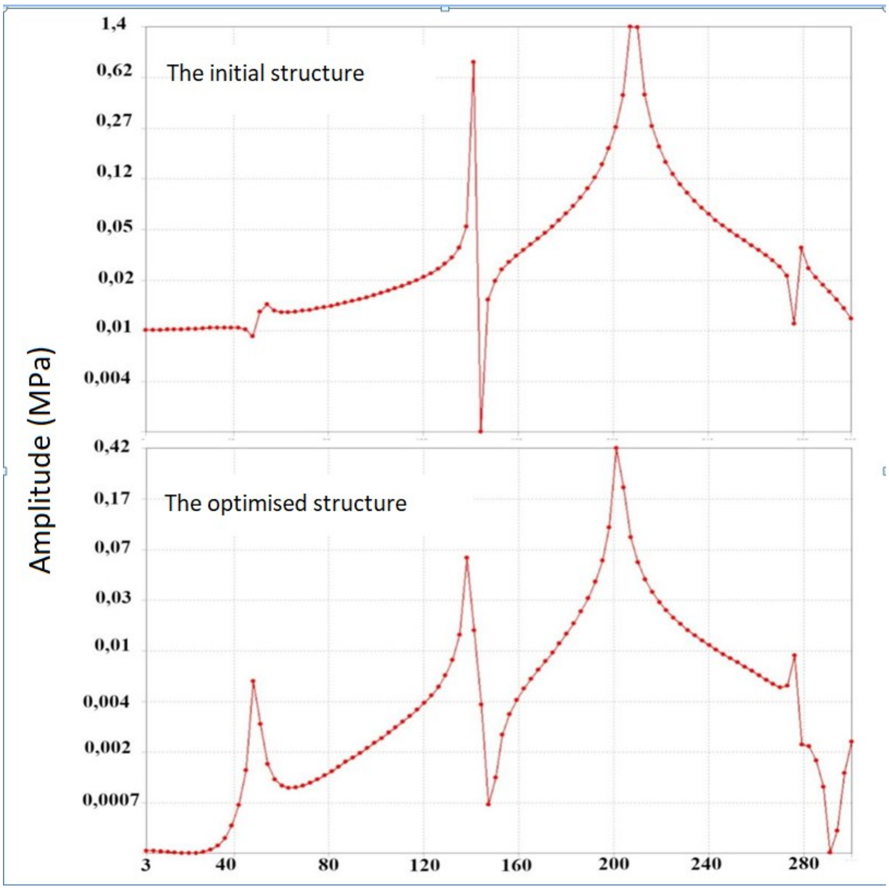

Figure 9. Comparative graph of structure's response to resonance frequencies

(source: own study)

Results of the simulation confirm the infinitesimal probability of reaching resonance frequency during normal use of the staircase. Quoting Smith et al. (2009) Hypki and Karas emphasize that during pedestrian traffic frequency of forcing varies between 1.5 to 3.5 (Hz) in case of a single person and reaches up to 4.5 (Hz) at maximum in case of multi-person traffic. In turn, the PN-EN 1990 standard accepts resonance frequencies no lower than 5 (Hz). Both the initial and the optimized structure meet these criteria. Only during operations of mechanical devices such as an electrical engine, a power drill or a public address system there is a possibility of the structure resonating but the structure's response can be discounted from the point of view of safety and comfort of use of the staircase. Furthermore, redesigning the structure lowered the value of the highest resonance frequency stress by approximately 1 (MPa).

The improvement of dynamic properties of the structure may be caused by interruption in continuity of an angle bar acting as a support for a step; instead of connecting with the angle bar the flat bar supporting the step is attached directly to the wooden step.

The characteristic of the impulse response is based on the results of the experiments performed by Maji (2012) during which the author measured the forces applied to the base surface after landing of an unsuspended human following a free-fall drop from the height of 430 mm (simulated jumping on a step). Characteristic of the forces applied in a usual direction to a base surface are presented in Figure 10.

For the discussed case a drop in the maximum stress by 54.73 (MPa) has been recorded during the transition state analysis of the improved structure. The simulated conditions reflect the extremely adverse impulse load the construction of a step can be subjected to. As demonstrated by Figure 10 redesign resulted in lowering the maximum stress significantly below the tensile strength of the material.

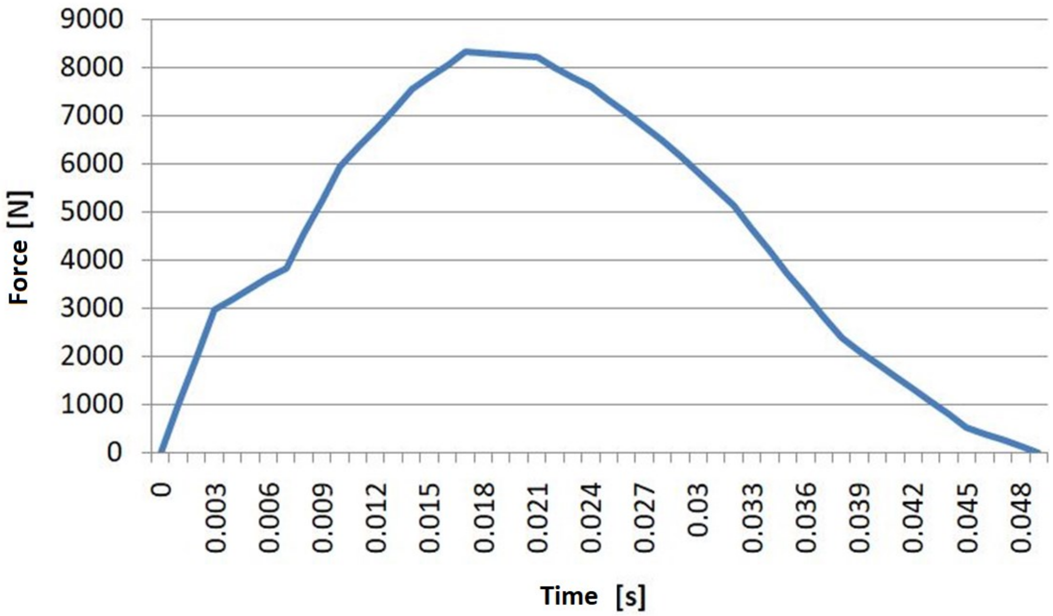

Figure 10. Characteristics of the impulse response

(source: own study)

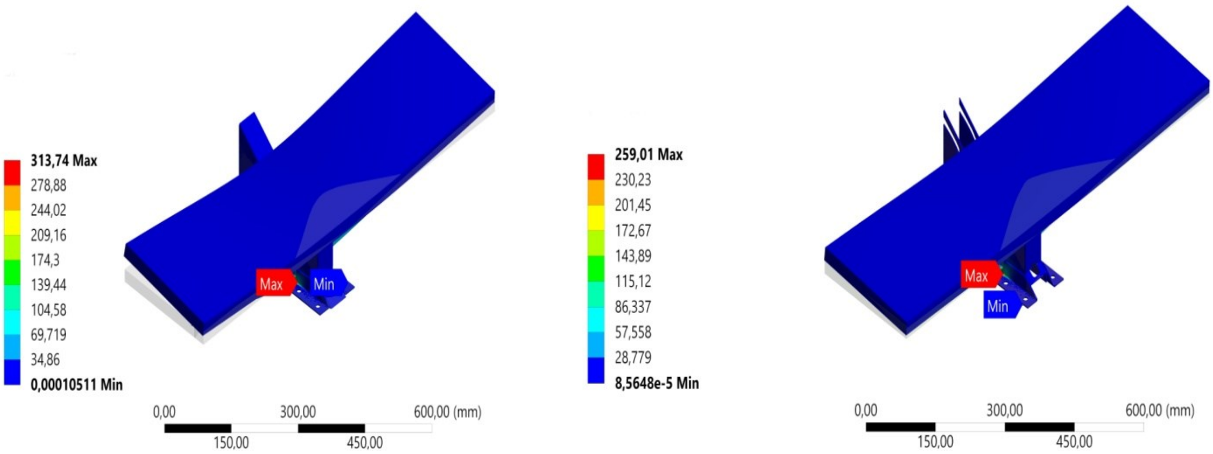

Figure 11. Distribution of stresses within the structure prior to redesign (on the left) and post redesign (on the right) under pulse load

(source: own study)

4. Conclusions

The goal of the study was to determine the influence of the process of optimization on the strength and dynamic properties of a self-supporting staircase. As it could have been projected reduction of mass (in this case by 40%) as a direction for optimization results in lowering mechanical robustness and strength of a structure as evidenced by increase in the maximum stress within the structure from 42 to (Mpa) 50.77 (Mpa). This fact in no way threatens safety of use (safety factor is almost 5) or impairs comfort of use (maximum deflection does not exceed 0.1 (mm).

Although new resonance frequencies have been observed in terms of dynamic properties the harmonic response amplitude of the modified structure subjected to harmonic response test dropped significantly – by 70% – which in case of achieving resonance frequency of 201 (Hz) will result in lower structure displacement amplitude – drop from 0.027 (mm) to 0.023 (mm). Significant improvement in strength properties of the structure was recorded also during the impulse response transition state analysis – the maximum recorded stress dropped by 54.73 (Mpa). Improvements regarding vibrations can be considered in the discussed case as a side-effect as the process of optimization did not directly concern modal analysis. Decrease in mass in combination with redesigning of the connections between metal and wooden components resulted in a structure less susceptible to resonance.

References

ANSYS Advantage Staf. (2014). Violet satellite project. ANSYS Advantage, vol. VII, issue 1. Downloaded from: https://www.ansys.com/-/media/ansys/corporate/resourcelibrary/article/violet-satellite- project -academic-aa-v8i1.pdf?la=en&hash=CE1A9147FCA07A74D30F094097F51C7E160437CB.

Buhl, T., Pedersen, C., Sigmund, O. (2000). Stiffness design of geometrically nonlinear structures using topology optimization. *Struct Multidisc Optim., 19*, 93-104.

Dheeraj, G., Anadi, M. (2012). Topology optimization of continuum structures using optimality criterion approch in ANSYS. *International Journal of Advances in Engineering & Technology, 5*(1), 470-485.

Fai, C.J., LaMont, D., Zhanjing, Y., Murat, K., Deb, H. (2002). *Finite Element Analyses of Seismic Responses of the Berkeley Town House*. Lawrence Berkeley National Laboratory.

Fiebig, W. (2019). *Drgania i hałas w inżynierii maszyn*. Warszawa: Wydawnictwo Naukowe PWN S.A.

Hypki, M., Karas, S. (2015). Badanie komfortu wibracyjnego schodów stalowych w nowej siedzibie NOSPR w Katowicach. *Inżynieria i Budownictwo, 71*(1), 371-376.

Jurkowski, S., Janisz, K. (2020). Analiza wpływu parametrów siatki obliczeniowej na wynik symulacji przepływomierza. *Autobusy, 12*/2019, 129-134.

Maji, K. (2012). *Static and Transient stress analysis of Human Bone*. National Institute of Technology Rourkela.

Mohammad, Al B.M., Mohammad, H.A.H., Abdul, R.N.M. (2015). Determination of Natural Frequencies through Modal and Harmonic Analysis of Space Frame Race Car Chassis Based on ANSYS. *American Journal of Engineering and Applied Sciences, 8*(4), 538-548.

Obrocki, W., Krupa, K., Sieniawski, J. (2014). Metodyka eksperymentalnych badań wytrzymałości zmęczeniowej łopatek sprężarki silnika lotniczego. *Mechanik, 87*(2), 117-120.

PN-EN 1990:2004 Eurocode: Basics of structural design.

Smith, A.L., Hicks, S.J., Devine, P.J. (2009). *Design of Floors for Vibration: A New Approach*. The Steel Construction Institute.

Yongchang, Y., Shuaijun, Z., He, L., Xiaofei, W., Yadong, T. (2017). Modal and Harmonic Response Analysis of Key Components of Ditch device Based on ANSYS. *Procedia Engineering, 174*, 956-964.

Żółtowski, M., Jędrzejak, M. (2015). Analiza modalna w badaniu budowlanych elementów konstrukcyjnych. *Przegląd Budowlany, 3*, 49-53.