

<sup>1</sup> Maja HOHNJEC, <sup>2</sup> Mirko KARAKAŠIĆ, <sup>3</sup> Hrvoje GLAVAŠ, <sup>4</sup> Eleonora DESNICA, <sup>4</sup> Ivan PALINKAŠ

## APPLICATION OF THE DIRECT OPTIMIZATION METHOD IN THE DESIGN OF THE STEEL LOAD-BEARING SUBSTRUCTURE OF "SANDWICH" FACADE PANELS

<sup>1</sup> University North, Department of Mechanical Engineering, Varaždin, CROATIA

<sup>2</sup> Mechanical Engineering Faculty, University of Slavonski Brod, Slavonski Brod, CROATIA

<sup>3</sup> Faculty of Electrical Engineering, Computer Science and Information Technology, University of Osijek, Osijek, CROATIA

<sup>4</sup> Technical Faculty "Mihajlo Pupin" Zrenjanin, University of Novi Sad, Zrenjanin, SERBIA

**Abstract:** This paper has optimized the load-bearing steel substructure of facade walls made of sheet metal "sandwich" panels. The substructure consists of steel tubes with a square cross-section of 80×80×3 mm, which are horizontally and vertically placed and interconnected by bolted connections. The substructure is part of the steel hall structure. In order to reduce the consumption of materials required for the construction of the steel hall, the steel substructure was optimized using the direct optimization method. In order to analyze stresses and displacements, a static numerical calculation was performed using the finite element method. The calculation determined that the steel substructure could be optimized. After the optimization, it was determined that the mass of the steel substructure could be reduced by 63,49 %. By reducing the mass, the maximum equivalent stresses according to von Mises increased by 45, 58 % compared to the non-optimized structure. The obtained values of increased stresses were within the permissible stresses, and the optimized structure from the point of view of static calculation is acceptable for production. The obtained optimized steel tubes have a rectangular cross-section of 30×60×2 mm. Modeling of the steel substructure was performed in the SolidWorks software package, and numerical calculation and optimization using the direct optimization method were performed in the Ansys Workbench software package.

**Keywords:** Design; Direct optimization method (DOM); Response surface method (RSM); Finite element method (FEM)

### INTRODUCTION

Design process represents a complex engineering task with the aim of selecting materials and geometry that meet the defined functional and function requirements, while respecting the defined constraints of the design process. For real engineering problems, there are multiple design solutions that meet the set goal, and each of them can represent an appropriate design as a synthesis of functionality and constraints, often relying on empirical methods.

Historically, design relied on the experience of designers before the development of mathematical and computational tools. In the 20th century, scientific advances led to modern and economical designs, both in mechanical engineering and civil engineering, with the application of optimization methods that developed rapidly in the 1960s [1].

An optimal design is defined as a design that achieves lower mass (material savings) and that satisfies the specified design requirements [1]. The application of optimization methods, such as the optimal design method (ODM), explicitly incorporates functional requirements and design constraints, where the optimal design represents the best balance of the most significant functional

requirements while respecting the specified constraints [2].

The application of mathematical programming, as one of the optimization methods, using the objective function, conditional equations (constraints) and variable parameters (geometric quantities), allows achieving optimal design solutions with a clearly defined mathematical algorithm [3]. Of particular importance is the application of the response surface method (RSM), as one of the mathematical programming methods, in the topological optimization (TO) of mechanical structures, combined with the finite element method (FEM) [4, 5].

The aim of this paper is to optimize the steel substructure that supports the facade "sandwich" panels, using the DOM and the RSM. The substructure is part of the steel-concrete hall structure. In order to reduce the mass of the substructure and save material, an initial numerical calculation of the steel substructure was carried out using the FEM, which aimed to determine whether there is a possibility for reliable optimization with respect to the maximum allowable stress of the S235 steel from which the substructure for supporting the panels is made. The

RSM was used to analyze the influence of the input parameters of the steel profiles on the response variables of the maximum equivalent stress according to von Mises and the maximum total displacement.

### STRUCTURAL DESIGN OF THE STEEL SUBSTRUCTURE FOR CARRYING "SANDWICH" FACADE PANELS

Halls made of steel profiles for carrying sandwich panels consist of the following basic structural elements: 1-steel cover, 2-portal frame, 3-facade cladding supports, 4-sub-beams, 5-facade cladding and 6-horizontal tie (Figure 1).

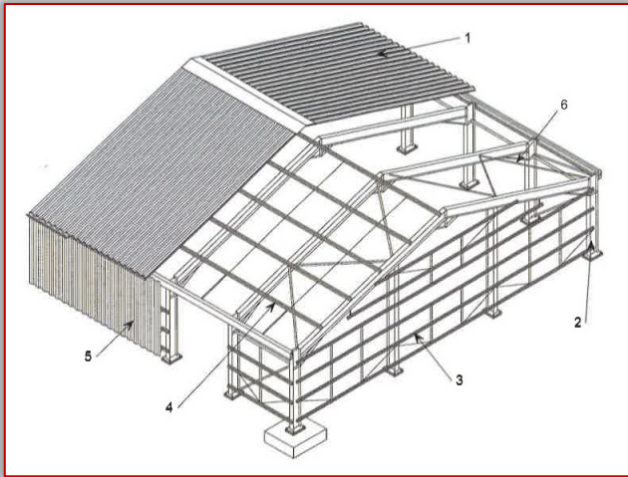


Figure 1. Basic structural elements of the steel hall for carrying "sandwich" facade panels [6]

The roof construction consists of a steel cover, beams and horizontal ties. By combining the horizontal ties and the beams, the stabilization of the roof is achieved, but also the reinforcement of the cover, which is usually made of trapezoidal profiled sheet metal or "sandwich" elements (panels) (Figure 2).



Figure 2. Retractable steel roof with "sandwich" panels [1]

The stability of the steel structure was achieved using the main load-bearing system constructed using a steel frame and beams (Figure 3). Beams are constructed as continuous supports or as supports supported on two supports. Rolled IPE profiles are most often used for beams, but for

smaller loads, it is more economical to use cold-rolled thin-walled profiles.



Figure 3. Steel structure of the hall [1]

The facade cladding is supported by longitudinal beams, or secondary elements. They have no influence on the overall stability of the steel structure. To avoid deformation of the facade beams due to their own weight, systems with tension elements and supporting compression elements are used (Figure 4).



Figure 4. Load-bearing steel structure of the facade and windows [1]

The facade walls are located in the longitudinal direction of the building and on the gable sides. The stabilization of the structure is achieved by using vertical ties (braces) (Figure 5).



Figure 5. Facade wall made of sheet metal "sandwich" panels [1]

## MODELING OF PARAMETRIC CAD MODEL OF LOAD-BEARING SUBSTRUCTURAL ELEMENTS

Modeling of the load-bearing substructure of the facade "sandwich" panels was carried out in SolidWorks 2022 [7] CAD system for parametric modeling. Given that the substructure will be subjected to an initial numerical calculation using the FEM, for the purpose of checking the stress and displacement fields for the needs of the optimization process, certain geometric simplifications were made on the CAD model of the substructure. Such simplifications are common for numerical modeling, to save computer resources and speed up calculation time. The simplifications refer to the screw connection, which is not modeled, and will be defined using boundary conditions when performing the numerical analysis. The actual steel substructure is bolted to the concrete structure that carries the main loads of the hall (Figure 6).



Figure 6. Load-bearing substructure of facade "sandwich" panels attached to a concrete structure [1]

The CAD model of the load-bearing steel substructure of the facade "sandwich" panels was modeled in the SolidWorks 2022 Weldments module. The structural elements were modeled on a spatial skeleton (3D sketch) and the geometric structure consists of square tube profiles with a cross section of 80×80×3 mm (Figure 7).

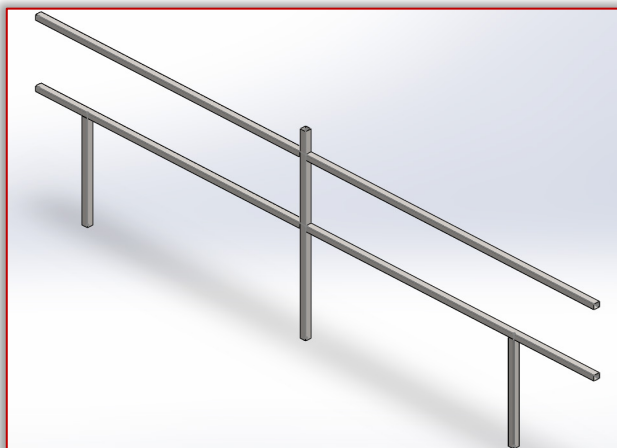


Figure 7. CAD model of 3D load-bearing substructure of facade "sandwich" panels [1]

The connection of the steel profiles was realized using plates with a thickness of 8 mm, while due to the simplification of the CAD model, the modelling of the screw connection was not made. The dimensions of the CAD model of the steel substructure are 7830×2800 mm.

## INITIAL NUMERICAL CALCULATION OF THE LOAD-BEARING SUBSTRUCTURE OF FACADE "SANDWICH" PANELS

The initial numerical calculation was carried out to determine whether the obtained results of the numerical analysis of the load-bearing substructure of the facade panels provide the possibility of additional improvement of the substructure, primarily with the aim of reducing mass and saving materials, i.e. reducing production costs, while at the same time ensuring that the substructure remains reliable, i.e. satisfies the state of maximum stresses and displacements that are conditioned by the mechanical properties of the material from which the structural elements are made. The numerical calculation was carried out using the Ansys Workbench 2022 software [8].

The following boundary conditions were set for the numerical analysis: a bolted connection was created between the steel and concrete structure of the hall using fixed supports (Figure 8).

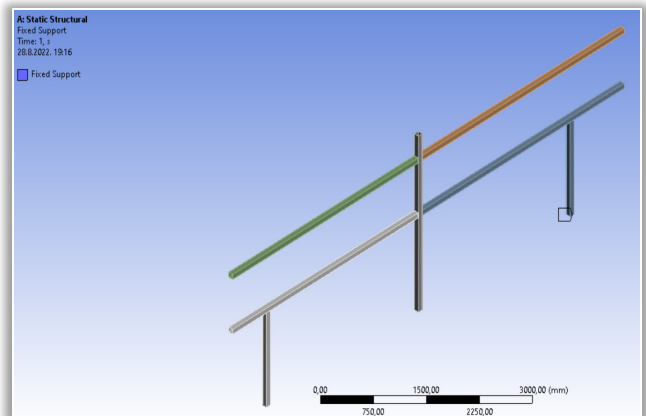


Figure 8. Boundary conditions - fixed supports

The steel substructure is loaded with the weight of the facade "sandwich" panels with a thickness of 80 mm. According to the catalog of the "sandwich" panel manufacturer, the specific mass of the panel per square meter is  $m = 16,4 \text{ kg/m}^2$  (Figure 9). Continuous load of the panels on a steel structure is represented by the product of the acceleration of the Earth's gravity and specific mass, i.e.  $q = 160,884 \text{ Pa}$ .

The load on the structure due to wind acting on the external surfaces, according to [1], is  $0,57 \text{ kN/m}^2$ .

According to the EN 10025 standard, structural steel S235 was selected for the material of the load-bearing steel substructure. The properties of this steel are density  $\rho = 7,850 \text{ kg/m}^3$ , modulus of elasticity  $E = 210 \text{ GPa}$ , yield strength  $R_e = 235 \text{ MPa}$ , Poisson's factor  $\nu = 0,3$  and shear modulus  $G = 81 \text{ GPa}$ .

BRAND, PRODUCT FAMILY, PRODUCT TYPE	RENDER	JOINT	WEIGHT (Fe 0,85/0,5) [kg/m <sup>2</sup> ]															
			PANEL THICKNESS (mm)															
			50	60	80	100	120	133	150	172	200	220	240	250*				
POWER	T	FTV (facade panel with standard fixing)	n/a	14,4	16,2	18,0	19,8	21,0	22,5	24,5	27,0	n/a	30,6	31,50				
		FTV HL (facade panel with hidden fixing)	n/a	14,6	16,4	18,2	20,0	21,2	22,7	24,7	27,2	n/a	30,8	31,70				
	S	SNV (roof panel)	n/a	15,7	17,5	19,3	21,1	n/a	23,9	25,9	28,4	30,2	32,0	n/a				
		FTV (facade panel with standard fixing)	16,1	17,3	19,7	22,1	24,5	26,1	28,1	30,8	34,2	n/a	39,0	n/a				
		FTV HL (facade panel with hidden fixing)	16,3	17,5	20,0	22,4	24,8	26,3	28,4	31,1	34,4	n/a	39,2	n/a				

Figure 9. Properties of the selected facade "sandwich" panel Tetrahedral finite elements are selected for the finite element mesh. The initial element size of 40 mm was defined, and then it was performed by convergence of the solution, the results of which are shown in Table 1.

Table 1. Convergence of the solution depending on the size of the tetrahedral finite element

Mesh size, mm	Number of nodes	Max. equivalent Von Mises stress, MPa	Max. total displacement, mm
40	72906	37,985	0,37078
30	81471	58,441	0,38637
25	109861	48,665	0,39509
20	119361	54,58	0,41206
19	125231	51,447	0,41553

From Table 1 it is visible that by changing the size of the elements from 20 mm to 19 mm, the difference in deformation is 0,8 %, which is negligible. If the maximum stresses according to von Mises are analyzed, then the difference is 6 %. When comparing other sizes of finite elements (40 mm, 30 mm, 25 mm and 20 mm), the difference is larger. It was not possible to choose smaller finite elements, because the academic version of Ansys Workbench 2022 was used for the calculation. Therefore, the initial numerical calculation was carried out with tetrahedral finite elements of size 19 mm.

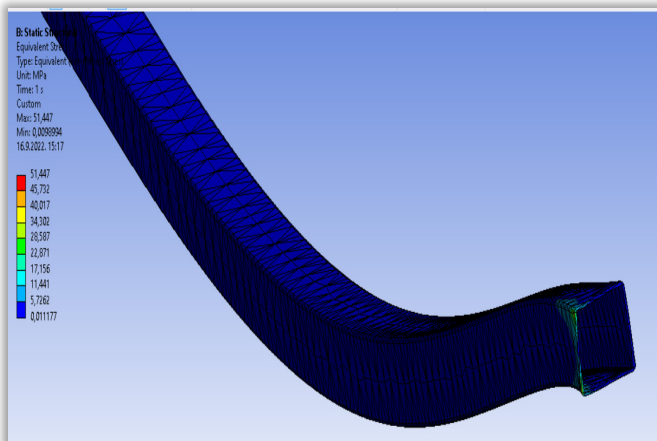


Figure 10. Maximum equivalent stresses according to von Mises

The results of the maximum von Mises stresses are shown in Figure 10. The highest stress points are present where the substructure is attached to the concrete structure of the hall. It can be seen that the maximum equivalent von Mises stress is 51,447 MPa and does not exceed the yield strength of S235 steel.

The displacements are most pronounced in the central part of the steel substructure, and the obtained maximum displacements are 0,41553 mm (Figure 11). The total mass of the substructure in the initial numerical calculation is 150,63 kg.

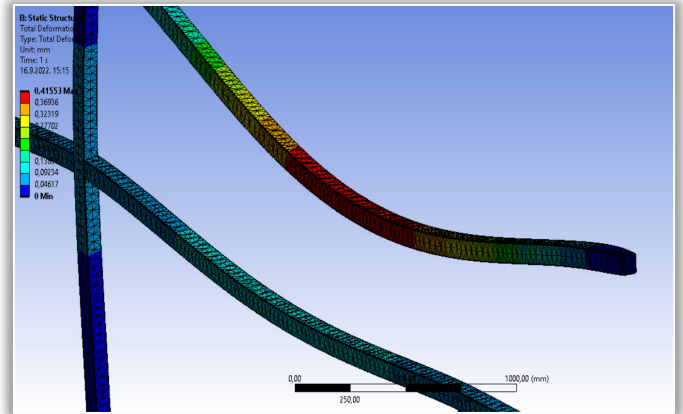


Figure 11. Maximum displacements of the load-bearing steel substructure

### OPTIMIZING THE STRUCTURAL ELEMENTS OF THE LOAD-BEARING SUBSTRUCTURE OF FACADE "SANDWICH" PANELS

According to the results obtained by the initial numerical calculation, with regard to the maximum equivalent stresses according to von Mises, it is a realistic possibility to optimize the substructure in order to reduce its mass and save material for the construction of the supporting steel substructure. Therefore, the objective function of reducing the mass of the supporting substructure was set. In order to carry out the optimization process, the amount of allowable stress was determined, which represents the ratio of the yield strength and the safety factor. For tough materials, the safety factor was determined by the interval  $1,2 \leq s \leq 2$ , and for the purposes of the calculation,  $s = 2$  was selected. Accordingly, the allowable stress is  $\sigma_{dop} = 117,5$  MPa. By determining the amount of allowable stress, the constraint function is defined. The objective function and the constraint function are defined in Ansys Workbench (Figure 12).

Table of Schematic C2: Optimization									
	A	B	C	D	E	F	G	H	I
1	Name	Parameter	Objective			Constraint			
2			Type	Target	Tolerance	Type	Lower Bound	Upper Bound	Tolerance
3	P4 <= 117,5 MPa	P4 - Equivalent Stress Maximum	No Objective			Values <= Upper Bound		117,5	0,001
4	Minimize P6	P6 - Geometry Mass	Minimize	0		No Constraint			
*	Select a Parameter								

Figure 12. Determination of the objective function and the constraint function

In order to reduce the mass, it is necessary to select the input variables (parameters) that will be optimized. The selected parameters are: P1-width of the steel profile, P2-thickness of the steel profile wall and P3-height of the steel profile (Figure 13). Each parameter represents a geometric size and for each parameter it is necessary to set limits within which the optimal values will be sought. The geometric parameters of the height and width of the pipe profile are defined with lower and upper values of 30 to 80 mm with an increment of 10. The increment value of 10 was chosen in order to obtain standardized values of the dimensions of the pipe profiles. For the parameter of the pipe wall thickness, lower and upper limits of 2 to 3 mm are defined (Figure 13).

	A	B	C	D
1	Input Parameters			
2	Name	Lower Bound	Upper Bound	
3	P1 - P3@DS_sirina@Sketch1@Podkonstrukcija.Part	30	80	
4	P2 - P3@DS_stjenka@Sketch1@Podkonstrukcija.Part	2	3	
5	P3 - P3@DS_visina@Sketch1@Podkonstrukcija.Part	30	80	
6	Parameter Relationships			
7	Name	Left Expression	Operator	Right Expression
*	New Parameter Relationship	New Expression	<=	New Expression

Figure 13. Defining the input parameter interval values  
By selecting the DOM and the RSM to display the obtained solutions, the load-bearing steel substructure for carrying the "sandwich" panels was optimized (Figure 14).

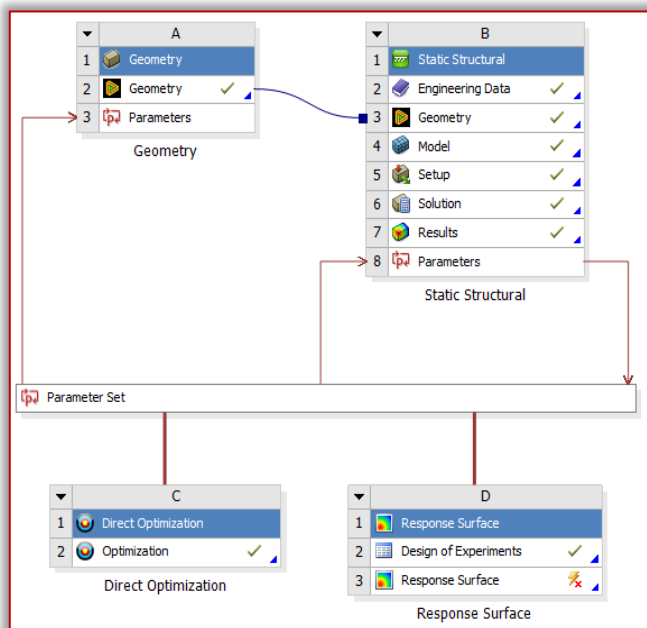


Figure 14. Project scheme for optimizing load-bearing steel substructures

Screening method defined a solution system that included 100 solution combinations, from which 3 solutions were selected that best met the given objective function, taking into account the constraint function (Figure 15).

Reference	Name	P1 - P3@DS_sirina@Part	P2 - P3@DS_stjenka@Part	P3 - P3@DS_visina@Part	P4 - Equivalent Stress Maximum (MPa)		P5 - Total Deformation Maximum (mm)	P6 - Geometry Mass (kg)	
					Parameter Value	Variation from Reference		Parameter Value	Variation from Reference
○	Candidate Point 1	30	2	40	107,47	43,48 %	1,7366	42,221	-23,80 %
○	Candidate Point 2	30	2	50	101,85	35,98 %	1,3539	48,694	-11,95 %
⊗	Candidate Point 3	30	2	60	74,899	0,00 %	1,2442	55,482	0,00 %

Figure 15. Proposed optimal solutions using the DOM

The third proposed solution (Figure 15) best fulfills the mass reduction objective function. The mass of the third solution compared to the reference (initial) mass was reduced by 63,49 %. On the other hand, there was an increase in the maximum equivalent von Mises stress by 45,58 % compared to the stress obtained by the reference (initial calculation) (Table 2).

The difference between the maximum equivalent von Mises stresses of the first and second solution compared to the reference stress obtained by the initial calculation is almost twice as large. On the other hand, the difference in mass between the first and second solution compared to the third solution is not so large, and therefore the proposed third solution was selected as the final solution of the optimization process.

The optimized input parameters of the third solution, obtained by the direct optimization method, have the following values: P1 = 30 mm, P2 = 2 mm and P3 = 60 mm. Therefore, the resulting cross-section of the optimized steel profiles is 30×60×2 mm.

Table 2. Comparison of the results of the output parameters of the initial and optimized models

Output parameters	Initial model	Optimized model	Divergence, %
Mass, kg	150,63	55,482	-63,49
Max. equivalent stress according to von Mises, MPa	51,447	74,899	+45,58
Max. total displacement, mm	0,41553	1,2442	+199,42

The influence of input parameters P1 and P3 on the response variables (maximum equivalent von Mises stress and maximum total displacement) is shown by response surface diagrams (Figure 16 and Figure 17).

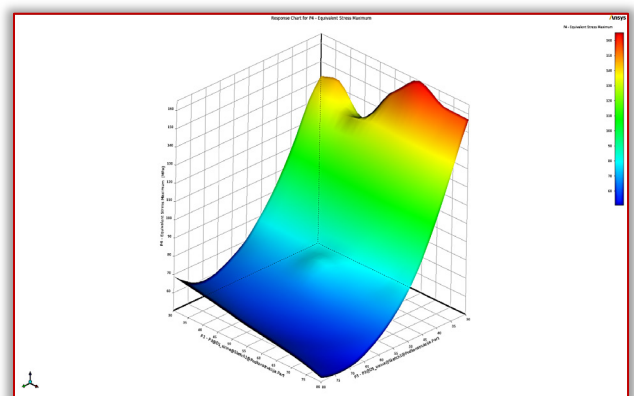


Figure 16. Diagram of the influence of parameters P1 and P3 on the maximum equivalent von Mises stress

From the diagram in Figure 16, it is evident that the highest stresses are present in the pipe size where the profile height is equal to 30 mm, given that in this case the solid support holding the concrete structure is the smallest.

Also, from the diagram in Figure 17 it is visible that the largest displacement is present in pipes with a profile size of 30×30×2 mm. Therefore, it can be concluded that with an increase in profile the total displacement decreases.

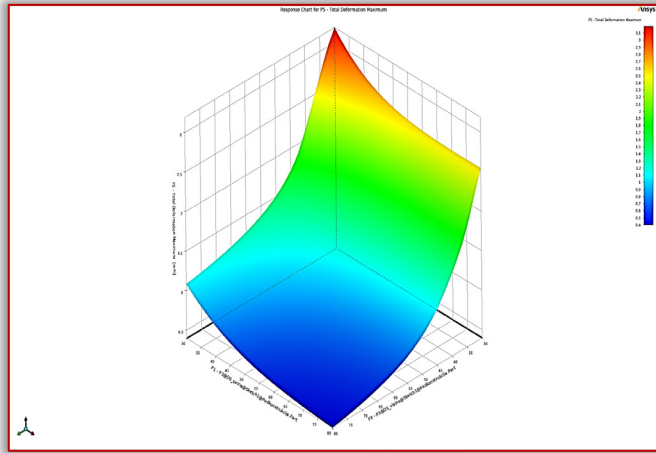


Figure 17. Diagram of the influence of parameters P1 and P3 on the maximum total displacement

## CONCLUSION

Steel structures are a key element of the mechanical engineering and civil engineering industry due to their universal application and flexibility in adapting to different requirements. In order to fully meet these requirements, it is necessary to design and optimize them properly. Most structures can be modified to achieve "optimal" properties. By applying modern mathematical and computational tools, engineers are enabled to carry out an efficient optimization process with a relatively simple procedure and obtain reliable results.

In this paper, a steel substructure for carrying "sandwich" panels made of S235 steel, according to the EN 10025 standard, which was statically determined, manufactured and installed on the hall building, was optimized. Using the direct optimization method, the mass of the steel structure was reduced by 63.49%, which achieved the mass reduction objective function. The mass reduction resulted in an increase in the maximum equivalent von Mises stress by 45.58%. The amount of the maximum stress obtained remained within the permissible limits, i.e. it remained lower than the permissible stress of the steel structure material. By optimizing the steel profiles, the amounts of the input optimized parameters were obtained, the dimensions of which are 30×30×2 mm. The influence of the optimized input parameters on the response parameters of the maximum equivalent von Mises stress and maximum total displacements was shown through

response surface diagrams. By setting the mass as the objective function, the maximum total displacements increased. Therefore, it would be desirable in further analyses to consider all influential factors, including boundary conditions, when making a decision on choosing the optimal solution.

It is important to note that the numerical model of the steel structure is simplified due to the limitations of the academic (student) version of Ansys Workbench 2022, which may cause certain errors in the accuracy of the obtained results.

## References

- [1] Hohnjec, M. Optimiranje nosive čelične konstrukcije sendvič fasadnih panela, master thesis, University North, Varaždin, 2022.
- [2] Johnson, R. C. Optimum Design of Mechanical Elements, Michigan University, John Wiley & Sons, second edition, 1980.
- [3] Ravindran, A., Ragsdell, K.M., Reklaitis, G.V. Engineering Optimization-Methods and Applications, John Wiley & Sons, second edition, 2006.
- [4] Xiangjie, J., Cai, Z., Chong, W. Response Surface Optimization of Machine Tool Column based on Ansys Workbench, Academic Journal of Manufacturing Engineering, vol. 18, no. 2/2020, 2020.
- [5] Shuhui, D., Jindong, B., Jingliang, H., Haixia, W., Fai, M. Mechanical Characteristics Analysis and Structural Optimization of Key Component of Self-Moving Temporary Support, Applied Sciences, 12(21), 2022, 10745.
- [6] Androić, B., Dujmović, D., Džeba, I. Čelične konstrukcije 1, IA Projektiranje, Zagreb, 2009.
- [7] SolidWorks 2022, <https://www.solidworks.com/>
- [8] Ansys Workbench 2022, <https://www.ansys.com/products/ansys-workbench>



ISSN: 2067–3809



copyright © University POLITEHNICA Timisoara,  
Faculty of Engineering Hunedoara,  
5, Revolutiei, 331128, Hunedoara, ROMANIA  
<http://acta.fih.upt.ro>