Zeszyty Naukowe Politechniki Częstochowskiej
Budownictwo
ISSN 0526-5916

Zeszyty Naukowe Politechniki Częstochowskiej nr 27 (2021), 69-75
DOI: 10.17512/znb.2021.1.11

Steel cantilever beam optimization with ANSYS software

Anna Jaskot[1]

ABSTRACT:
The problem of analysing complex steel structures in relation to analytical and numerical tests of a single structure element has been presented. The aim was to demonstrate the compliance of the chosen methods for determining the strength parameters of a cantilever beam and to select the best cross-section for the optimization of the structure. The beam deflection and the value of the maximum reduced stresses (according to Huber von Mises' hypothesis) have been determined numerically and analytically. Numerical analysis, on the basis of the finite element method, as well as the optimization, have been performed in the ANSYS environment. Based on the static analysis of the structure, optimization of the beam mass has been made by changing its cross-section dimensions, taking into account the specific optimization criteria, pointed in the work. As a result of optimization, solutions satisfying the required criteria have been selected. The weight of the structure has been reduced by half compared to the initial values of the designed structure. The permissible stresses have not been exceeded.

KEYWORDS:
cantilever beam; numerical analysis; ANSYS; FEM; cross-section optimization

1. Introduction

One of the most common materials used in construction engineering is steel. It is applied in elements to strengthen their bending and tensile strength and can be an independent building material for steel structures or used as a reinforcement (i.e. in concrete structures). Due to its resistance to high loads, it can be used to build structures such as bridges, skyscrapers, halls, landings, cantilevered viewing platforms and other structures with a shell, frame or structural system. The self-load is one of the main loads of all structures, hence it influences the application of the design solution. Steel is a material of large mass, therefore it is replaced with lighter materials while maintaining the highest possible safety factors for the use of the structure. In this way, objects are designed with the use of light steel structures, which are characterized by the use of cold-formed sheet elements in their technology. The design of such structures is extensive in time due to the complex calculation procedures of elements in the IV cross-section class.

Reducing the weight of the structure also takes place through the use of optimization algorithms, the purpose of which is to find the best solution within the given constraints. Therefore, solutions are being sought that will enable the maintenance of strength and functional properties while reducing the weight of the structure. In modern steel construction projects, very large spans are achieved, reaching several hundred meters, which are used in the construction of covers of stands in stadiums, exhibition and sports halls, etc.
Self weight is a very important matter in the design of large span structures. This affects not only the variable load, but also its dynamic properties, low vibration damping and dynamic fatigue.

[1] Czestochowa University of Technology, Faculty of Civil Engineering, ul. Akademicka 3, 42-201 Czestochowa, e-mail: anna.jaskot@pcz.pl, orcid id: 0000-0002-5478-0685

It also comes down to the use of steels with increased durability (e.g. S355 or S460, etc.). In the design and exploitation of the structure, variable loads, such as wind or snow loads, play an important role. In the work [1] detailed visual, numerical and analytical studies of the total collapse of the cantilevered roof structure of a sports tribune has been presented due to lateral buckling and plastic deformation as a result of insufficient consideration of variable loads on the total loads. The influence of variable loads due to wind and snow loads has been considered and described in [2] on the example of a high building with a shaft-frame structure.

The main characteristic of frame structures is the full mutual cooperation of all elements of the structure. This is useful in taking over forces in case of failure of the structure. The systems of the shaft structures have also been extensively described in [2]. Innovative ways of using cantilevered elements in modern building structures for various functional purposes have been presented in [3]. Particular attention has been paid to the use of technologies that reduce weight without losing strength. The results of static and modal analysis have been presented on the example of the viewing platform described in [4]. The viewing platform was made of steel pipes with variable cross-sections depending on the cross-sectional forces.

There is a need to optimize various types of bar structures in order to minimize the costs (materials, design time), exploitation, while maximizing the functional and strength properties. Hence, through the analysis of simple construction elements, the models built in computer programs are verified in order to demonstrate the correctness of the solution of the computer method with the analytical method. Therefore, in order to analyse complex systems with a large number of elements of bar structures (examples of such systems are presented in [5]), it is possible to examine a single element and verify the correctness of the methods adopted by its analysis. Based on the modelling of a simple structure element, an analysis of its strength properties is carried out in a computer program. The analysis of statics is most often carried out using the finite element method, in which one, two or three-dimensional elements with a varied number of nodes forming the mesh are selected.

This is legitimate from a design point of view. In this way, on the basis of simple elements and fragments of structures, the compliance in the results of the strength properties of the structure obtained in an analytical manner with the results obtained using computer methods can be demonstrated. The methodology can be successfully used to analyse structures with a higher degree of complexity. In this work, the static analysis described in the further part of the work, has been carried out on the example of a cantilever beam modelled in ANSYS. After determining the strength properties, the model was subjected to further tests in order to optimize the structure.

The paper presents an analysis of the optimization of the mass of a cantilever beam subjected to given loads by selecting appropriate dimensions of the beam cross-section. Optimizing the weight of the structure, while limiting the value of reduced stresses (according to Huber von Mises) by selecting the dimensions of the beam cross-section using optimization methods has been made in the ANSYS Workbench program module. The objective function depended on constructive and state variables. The values of the width and height of the rectangular cross-section of the beam were used as the design variables. The state variable has been represented by the value of the maximum reduced stresses (determined using the built-in algorithm according to the Huber von Mises hypothesis). For different types of cross-sections, the dimension of the wall thickness of the cross-section or the height of a section fragment can also be optimized. The results of such analyses, also using the ANSYS software, have been included in [6, 7].

The optimization algorithms are based on some preliminary values, which are then recalculated to find the best possible solution. In the ANSYS environment the module responsible for optimization is the *Response Surface Optimization* module. The main idea of the *RSO* is to use sequences of designed experiments to obtain an optimal response. When it is necessary to formulate a task with many criteria that are usually difficult to compare or are contradictory, multicriteria optimization methods are used. In order to find the best solution, it is necessary to simultaneously assess the impact of all criteria on the final decision whether one is looking for a single solution or a group of solutions that differ from each other, but each of which will be optimal. The usual solution is that the criteria are combined into one objective function that will return one solution. There are also solutions in which tasks with multiple criteria are replaced with a sequence of single-criteria tasks. In most cases, however, when using the multi-criteria optimization method, a set of optimal solutions is obtained. There are many multi-objective optimization techniques. Multi-Objective Genetic Algorithm MOGA is one of the techniques, which is effective for dealing with multi-variable and multi-objective problems. One of the algorithms of the multi-criteria method is the NSGA algorithm. Non-dominated sorting algorithm NSGA is an evolutionary algorithm that selects populations in terms of dominance. The topic has been presented in [8]. The disadvantage of this algorithm is the high computational complexity, and at the same time the lack of sorting the solutions in the order from the best adapted to the weakest. In response to these problems, the NSGA-II algorithm has been developed. With its use, it is possible to use quick sorting of non-dominated solutions and the computational complexity has been reduced. In this work, the possibility of using the NSGA-II algorithm embedded in the ANSYS environment has been used.

Parameters optimized by using MOGA combined with the finite element method FEM have been described in [9], in which the authors have proved that these algorithm parameters can satisfy all requirements with minimum expense and maximum productivity. The advantage of multi-criteria optimization is a set of solutions instead of a single solution. Based on the NSGA-II algorithm proposed in [10], the results of the optimization of mass and deformation of a cantilever beam with a circular cross-section have been presented.

In [11], the subject of cost optimization with explanation of how ANSYS software works has been introduced. As a result, the comparison of an optimized and non-optimized beam under the simulation using the ANSYS tool has been presented. The authors of the work [12] have proposed optimization of the volume of the tapered hollow cantilever beam. In their work the optimization objective has been made to reduce weight under limiting the stress (equal to yield stress). In [13] optimization of a linear elastic isotropic structure subjected to static and self-weight loading conditions has been presented.

The problem of selecting the best dimensions of the beam cross-section has been described in the paper. The beam deflection arrow and the value of the maximum reduced stress (Huber von Mises) have been determined analytically and numerically. On the basis of such a static analysis of the structure, the dimensions of the beam cross-section have been optimized, taking into account specific optimization criteria. The results have been presented in the following chapters of the work.

2. Structural analysis model

The static scheme adapted to the analysis of the internal loads of the system have been presented in Figure 1.

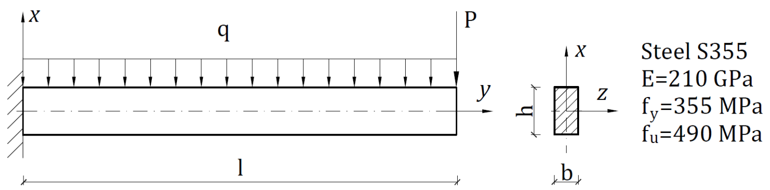

Fig. 1. Model of cantilever beam with its cross-section

The static model of the system has been described in a Cartesian *xyz* coordinate system. The beam, rigidly fixed on one side, has been subjected to a continuous load q and a concentrated force P on its free end. The cross-section of the beam has been defined by the dimensions of width b and height h. The beam has been designed from S355 structural steel and the beam length has been denoted as l. Available computer programs can be used to carry out a numerical analysis of the working parameters of the structure. In [14], the beam deflection calculations have been performed using the *Mathematica* program. In this paper, the *SMath* program has been used to carry out the calculation process.

To perform strength calculations of a structure model in ANSYS, it is necessary to enter first the material characteristics and geometry. Then the boundary conditions and loads can be determined. The static calculations in the *Mechanical* module can then be made.

Due to the complex state of stresses, the value of reduced stresses was determined analytically from the general formula (eq. (1)) according to the Huber von Mises hypothesis:

$$\sigma_{red} = \sqrt{\sigma^2 + 3\tau^2} \tag{1}$$

where: σ – normal stresses in bending, τ – maximal stresses (τ = max(shear stresses; tangential stresses) in bending.
The deflection has been obtained according to the formula (eq. (2))

$$f = \frac{Pl^3}{3EI} + \frac{ql^4}{8EI} \tag{2}$$

where: P – concentrated force at free end, q – continuous load, E – Young's modulus, I – moment of inertia of the cross section ($I = bh^3/12$).
The results of the optimization have been included in the next chapter.

3. Results of the static analysis

The material data and initial geometric parameters presented in Table 1 have been adopted for the purpose of the research. After conducting a static strength analysis using the finite element method in ANSYS, the obtained values have been compared with the values of parameters determined by the analytical method.

Table 1
Material properties and geometrical parameters of the analysed beam made of steel S355

Parameter	Value	Unit
Cross section width (initial) b	0,035	m
Cross section height (initial) h	0,055	m
Beam length l	1,0	m
Young's modulus E	210	GPa
Poisson's ratio ν	0,3	–
Force at free end P	2	kN
Continuous load q	0,45	kN/m

The results have been compared and presented in Table 2.

Table 2
Results of static analyses

	Analytical	ANSYS FEM	Relative error
Deflection [m]	0.0069713	0.0069723	0.01%
Reduced stresses (Huber von Mises) [MPa]	127.50	137.17	7.58%

As a result of the statics of the cantilever beam, the compliance of deflections with an accuracy of 0.01% and of reduced stresses with an accuracy of 7.58% has been demonstrated.
On the basis of a simple element such as a cantilever beam, optimization has been carried out to improve the design process at an early stage.

4. Optimization with ANSYS software

The parameters have been defined as shown in Figure 2.

2	⊟ ✓ Design of Experiments	
3	⊟ Input Parameters	
4	⊟ Static Structural (A1)	
5	P12 - wys	☑
6	P14 - szer	☑
7	⊟ Output Parameters	
8	⊟ Static Structural (A1)	
9	P7 - Solid Mass	
10	P13 - Equivalent Stress Maximum	

Fig. 2. Design of experiments for the chosen parameters

The section parameters have been assigned to input parameters (height – P12 and width – P14), and output parameters (mass – P7 and permissible stresses – P13).
The values of the width parameter ranged between 20 and 38 mm, the height ranged between 36 mm and 56 mm, in both cases with a change of 0.5 mm. With such defined dimensions, the selected points have been proposed for the experiment. As a result, a set of 10 points that will be further analysed have been defined. The set of the determined points has been shown in Figure 3.

1	Name		P12 - wys (mm)	P14 - szer (mm)	P7 - Solid Mass (kg)	P13 - Equivalent Stress Maximum (Pa)
2	6	DP	38	20	5,966	4,5616E+08
3	4	DP 1	47	20	7,379	2,9703E+08
4	2	DP 1	38	28	8,3524	3,24E+08
5	7	DP 1	56	20	8,792	2,0886E+08
6	1	DP 1	47	28	10,331	2,1522E+08
7	8	DP 1	38	36	10,739	2,5787E+08
8	3	DP 1	56	28	12,309	1,5721E+08
9	5	DP 1	47	36	13,282	1,7645E+08
10	9	DP 1	56	36	15,826	1,2916E+08

Fig. 3. Design of experiments – determined set of points

The generated results have been shown in Figure 4.

⊟ Optimization Study			
Minimize P7; P7 <= 7 kg	Goal, Minimize P7 (Default importance); Strict Constraint, P7 values less than or equals to 7 kg (Default importance)		
Maximize P13; P13 <= 4,08E+08 Pa	Goal, Maximize P13 (Default importance); Strict Constraint, P13 values less than or equals to 4,08E+08 Pa (Default importance)		
⊟ Candidate Points			
	Candidate Point 1	Candidate Point 2	Candidate Point 3
P12 - wys (mm)	40	40,5	38,5
P14 - szer (mm)	20,5	20	22
P7 - Solid Mass (kg)	★★ 6,437	★★ 6,3585	★ 6,649
P13 - Equivalent Stress Maximum (Pa)	★★★ 4,0634E+08	★★★ 4,0326E+08	★★★ 4,0685E+08

Fig. 4. Response Surface Optimization – candidate points satisfying the requirements

All the points for which the objective function is determined and for which the constraint conditions are met define a set of feasible solutions. In Table 3 the relative difference in the values of individual parameters under the influence of the change in cross-section have been presented.

Table 3
Comparison of results before and after optimization for Candidate Point 1

	Before optimization	After optimization	Relative difference
Mass [kg]	15.111	6.437	–57%
Reduced stresses (Huber von Mises) [MPa]	137.17	406.34	196%

The limitation of the optimized parameter has been made by defining the weight of the structure less than 7 kg and defining values of stresses close to the permissible stresses, but do not exceeded it (σ_{dop} < 408 MPa, for safety coefficient γ_M = 1,2). The generated candidate points represent the best solutions of the problem. The reduced stresses have not exceeded the value of the permissible stress, while the mass of the construction is significantly lower.

5. Conclusions

The optimization of the structure described in the paper has been carried out effectively. The weight of the structure has been reduced by 57%. The aim of the work has been achieved and the results have been presented. On the basis of the analysis of simple construction elements, the results obtained by both methods have been verified in order to demonstrate the correctness of the solution of the computer method with the analytical method. By using this method, optimization can be made by selecting more stringent constraints to achieve even more satisfactory results. Therefore, the results obtained in the analysis and the achieved compliance proved the adopted methodology can be successfully applied to the analysis of structures with a higher degree of complexity.

References

[1] Altunişik A.C., Ateş S., Hüsem M., Lateral buckling failure of steel cantilever roof of a tribune due to snow loads, Engineering Failure Analysis 2017, 72, 67-78.

[2] Rzymski E., Dohojda M., Projekt budynku wysokiego w konstrukcji trzonowo-szkieletowej, Przegląd Budowlany 2019, 3, 38-48 (in polish).

[3] Generalova E., Generalov V., Kuznetsova A., Cantilever structure in modern construction, MATEC Web of Conferences 2017, 117, 1-7.

[4] Górecka A., Bosak G., Static and dynamic analysis of the steel viewing platform, Technical Transactions Civil Engineering 2016, 1-B, 19-26.

[5] Żółtowski K., Siwowski, T., Recent steel structures in Poland – the selected projects, 13th International Conference on Metal Structures ICMS, 2016, 40-52.

[6] Alteveb M.S., Jolgaf M., Optimization of cantilever beam for minimum weight using finite element analysis, Proceedings of FLCMCPT September 12-13, 2017, Sidi Al.-Sayeh, Tripoli, Libya, 1-8.

[7] Abosbaia A.A., Jolgaf M., Shape optimization of a hollow cantilever beam for weight minimization using finite element method, International Science and Technology Journal, 2019, 17, 1-16.

[8] Gryniewicz-Jaworska M., Przegląd wybranych metod ewolucyjnych w optymalizacji wielokryterialnej, Informatyka, Automatyka, Pomiary w Gospodarce i Ochronie Środowiska 2014, 4, 32-34 (in polish).

[9] Chen W., Zhang Y.-Z., Ma J.-H., Wang B.-X., Chen Y., Wang C., Optimization of processing parameters for beam blank continuous casting using MOGA combined with FEM, Reviews on Advanced Materials Science 2013, 33, 337-341.

[10] Gurugubelli S., Kallepalli D., Weight and deflection optimization of cantilever beam using a modified non-dominated sorting genetic algorithm, IOSR Journal of Engineering 2014, 4(3), 19-23.

[11] Gurung B., Kaur M., Beam cost optimization with differentia approach and simulation on ANSYS tool, International Journal of Advance Research in Science and Engineering 2017, 6(12), 1896-1910.

[12] Indu K. Optimization of tapered cantilever beam using genetic algorithm: interfacing MATLAB and ANSYS, International Journal of Innovative Research in Science, Engineering and Technology 2015, 4(10), 10145-10150.

[13] Naman J., Rakesh S., Effect of self-weight on topological optimization of static loading structures, Alexandria Engineering Journal 2018, 57(2), 527-535.

[14] Mazur-Chrzanowska B., Chrzanowski R., Modelowanie analityczno-numeryczne parametrów pracy belki wspornikowej jednostronnie utwierdzonej z zastosowaniem programu Mathematica, Problemy Nauk Stosowanych 2015, 3, 43-48 (in polish).

Optymalizacja stalowej belki wspornikowej z użyciem oprogramowania ANSYS

STRESZCZENIE:
Przedstawiono problem analizy złożonych konstrukcji stalowych w odniesieniu do badań analitycznych i numerycznych pojedynczego elementu konstrukcji. Celem było wykazanie zgodności obranych metod wyznaczania parametrów wytrzymałościowych belki wspornikowej oraz dobranie najlepszego przekroju poprzecznego w optymalizacji konstrukcji. Analitycznie oraz numerycznie wyznaczono strzałkę ugięcia belki oraz wartość maksymalnych naprężeń zredukowanych (wg Hubera von Misesa). Analizę numeryczną, z użyciem metody elementów skończonych, oraz optymalizację przeprowadzono w środowisku ANSYS. Na podstawie analizy statycznej konstrukcji przeprowadzono optymalizację masy belki poprzez zmianę jej wymiarów przekroju poprzecznego, przy uwzględnieniu wskazanych w pracy kryteriów optymalizacyjnych. W wyniku optymalizacji zostały wybrane rozwiązania spełniające wymagane kryteria. Masa konstrukcji została zmniejszona o połowę w porównaniu do wartości początkowych projektowanej konstrukcji. Dopuszczalne naprężenia nie zostały przekroczone.

SŁOWA KLUCZOWE:
belka wspornikowa; analiza numeryczna; ANSYS; MES; optymalizacja przekroju poprzecznego