Numerical analysis and optimization of large dimensioned structures considering stress concentrations in welded joint

Jaroslav Majko[0000-0001-7136-2656], *Milan* Sága [0000-0002-0938-8772], *Zuzana* Ságová[0000-0002-1550-3212], *Marián* Handrik[0000-0002-8622-3103], *Peter* Kopas[0000-0002-1309-7644], and *Lenka* Jakubovičová[0000-0002-9381-1595]

Faculty of Mechanical Engineering, University of Žilina, Univerzitná 8215/1, 010 08 Žilina, Slovakia

Abstract. The paperpresents an optimal design and computational solution of truss structures consideringstress concentrations in welded joints. The calculation approach for precise analysis of welded joints using finite element method is introduced. The standard finite element analysis of the truss structure is followed by a special calculation procedure. The procedure is based on the known method of sections. Each member of the truss structure joined at the point under investigation is sectioned off the entire structure. In the next step, a new detailed finite element model of the welded joint is created. These analysis results allow define the stress concentration factors for technical problems of this type. Finally, a specific optimization procedure for design of structures based on the principles of discrete optimization is shown.

1 Introduction

Long-term experience in accident assessment of welded structures lead to conclusion, that in practice the usage of modern facilities for simulation of operating states is in limited extension. The applied computational models often contain simplifications, which disqualify the significance and justness of computer simulation. The justness is also limited by application of new technologies (such as IR cameras) [1,2], which can be used to monitor the behavior of system under loading, but appropriate measurement methodologies and assessment methods must be considered [3].
The article focuses on optimal design and computational solution of welded structures of large dimension machines and equipments, which members are characterized as welded truss elements with stress concentrations in joints. The primary aim was to put forward methodology and calculation algorithm based on finite element analysis, which would take into consideration stress concentrations. The principles of virtual modeling were applied using supporting software products such as Pro/Engineer, Cosmos/M [4] and programming languages Cosmos-Language and Matlab [5].
The primary aim of the study is to perform an analysis of theoretical and practical scientific tools for obtaining the relevant results. The attention is focused on tools with linkage to simulation calculations using appropriate mathematical models related to the issue of

effective identification of stress concentrations in welded structures. The considered computational models are based on finite element method principles. A computational algorithm for solving a problem using the method of sections in combination with FEM is proposed, taking into account the synergy of the truss model with detailed model created from 3D finite elements. Subsequently, the application of the discrete optimization algorithm for optimal design of selected welded truss construction (for example parts of rotating tower crane) is expected in terms of load carrying capacity.

The intention of authors is addition of missing computational tools, chain procedures and proposal of simulation calculations system primarily based on the finite element method [6-9]. The methodology is a complex encompassing design, computational and technology areas into a whole unit.

The significant task in described process is mathematical formulation of objective function and constraints for solution of technical problem. In the solution process of specified task were considered criterions [10, 11] related to truss constructions. This means:

- Strength criterion covering stress concentration analysis in places adjacent to welds.
- Stability criterion related to possible loss of structural stability, particularly in cases of local collapse of truss member subjected to high compressive forces.

The solutions of described problems were performed using several computing tools and models [12, 13] in order to realize detection of stress concentrations in welds. The appropriate connections provide a number of possibilities in the process of analysis and optimization of the most complex construction assemblies.

2 Computational identification of stress concentrations in welding joints of truss construction members

This section of paper proposes a methodology of computer prediction of stress values in critical places of welded truss structures. Numerical analysis of local effects in the case of welded structures is presented, which proves to be a necessity corresponding to reality.

The considered truss construction represents the supporting part of the EC-HM tower crane. We assume a construction, whose virtual model was firstly created in Pro/Engineers program (Fig.1) and subsequently transformed into COSMOS/M software into computational model based on finite element method. Selected quantities describing geometric properties of the computational model were parameterized. Fig. 1 shows parameterized cross-sectional quantities of structural bearing frame parts. The FEM model consists of 4320 truss elements and 4024 nodes. Other specified computational parameters are Young modulus ($E=2.10^{11}$Pa), Poisson ratio ($\mu=0.3$) and density of material ($\rho=7800 kg/m^3$). Based on the fundamental principles of mechanics, loading forces on construction were derived from wind load and weight load, which in the worst scenario could act simultaneously.

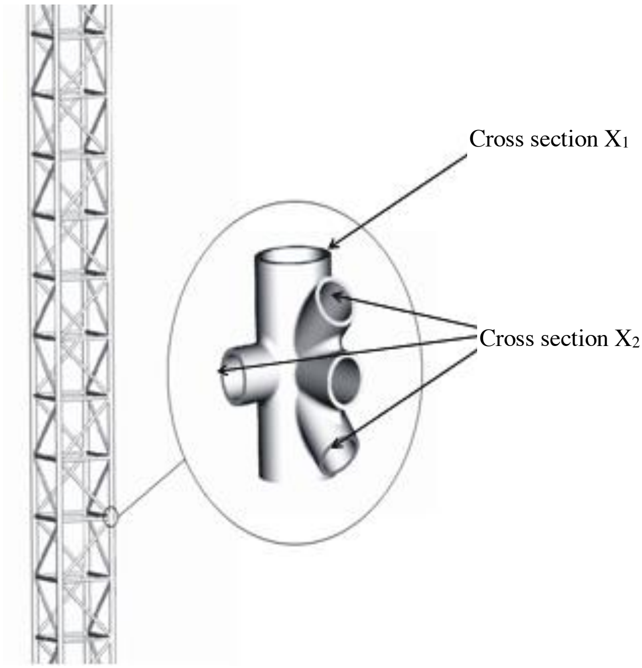

Fig. 1. Part of the truss construction virtual model designed in Pro/Engineer program depicting cross-sectional variables X1 and X2.

The tower crane manufacturer specifies the magnitude of vertical forces to 4x40 000 Newtons. The magnitude of the horizontal forces due to wind was determined according to, where the load up to 40 m height was considered with a maximum pressure of 1300 N / m^2. The recalculation defines modeled forces in nodes specified at various heights in the following order:

- The highest node located at 32 meters ~ 2 x 4550 N
- The middle node located at 15 meters ~ 2 x 2125 N
- The lowest node located at 5 meters ~ 2 x 900 N.

In the Fig. 2, the deformation shapes of the structure and the locations of maximum stress values are shown. The calculations identify the node number 8 as the critical point. The node is the intersection of the following truss members:

- Truss number 7 subjected to axial stress ->σ_x = -98,5 MPa
- Truss number 29 subjected to axial stress ->σ_x = -93,6 MPa
- Truss number 66 subjected to axial stress ->σ_x = 1,6 MPa
- Truss number 67 subjected to axial stress ->σ_x = 5,4 MPa
- Truss number 146 subjected to axial stress ->σ_x = 6,7 MPa
- Truss number 197 subjected to axial stress ->σ_x = -11,1 MPa.

The Fig. 3 illustrates detailed connection of these truss members.

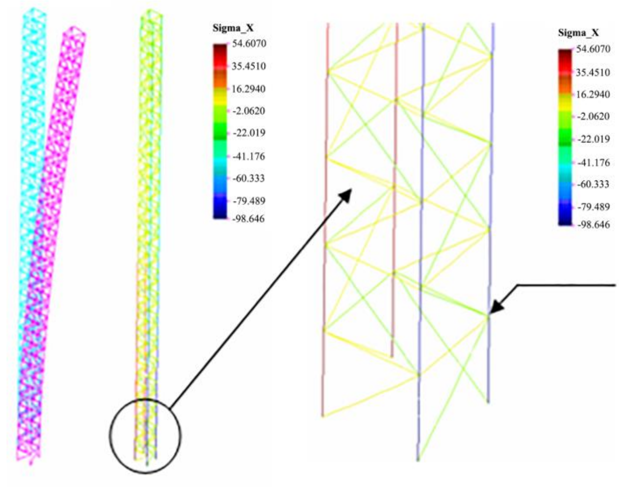

Fig. 2. Graphical presentation of results. Structure shape deformation, axial stress distribution in individual truss member, identification of node number 8 and illustration of trusses connected at the node.

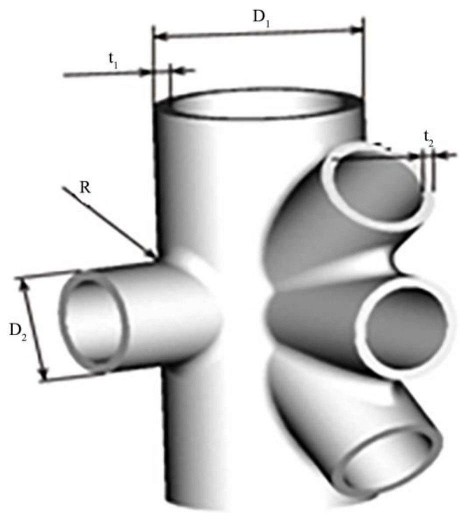

Fig. 3. Virtual model of node detail designed in Pro/Engineer.

The previous analysis specifies input boundary conditions for generation of detailed computational model of the critical node. The generated computational model includes various tee-joint welds. The difference among these joints was based on different fillet radius (2, 4, 6 and 8 millimeters). These steps allowed identifying critical points around the welds and specifying magnitude of stress concentrators.

The virtual model of critical node detail was generated using Pro/Engineer program (Fig.3). Since the fillet radius R became parameterized dimension, therefore different variations of the computational modelcould be created in COSMOS/M environment. Fig. 4 shows generated finite element mesh with defined boundary conditions. The mesh consists of quadratic tetrahedral elements of size t1/2 and t2/2. The parameters t1 and t2 present the thickness of profiles. It means that each wall of the profile was divided into two elements near the welding domain and its immediate surrounding. The generated mesh satisfy the standards for obtaining a solution with sufficient accuracy.Figure 5 shows two noticeable aspects in the stress concentration domain: the first is a smooth distribution of stress, and the second is a division of the stress concentration area using three or more quadratic elements.The loading is defined as the pressure acting on the cutting surface of the profiles. Previously performed analysis of truss structure specified the magnitude of the loading. Subsequently, the authors selected the node with the highest pressure values. Since the node is part of the truss structure, the force effects induced by the applied pressure are in balance.On the bottom surface of the clipped part of the D2 profile are three points with removed degrees of freedom. The proposed removal of DOF fix the model in space and allow free deformation of the profile surface.Due to the balance of force effects, the reactions at these three points are close to zero. The material added to the model represents the exact shape of the proposed weld (radius of 2, 4, 6 and 8 millimeters).

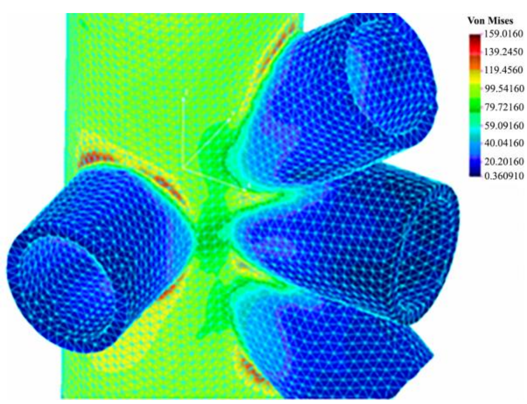

Fig. 4. Equivalent stress (von Mises) distribution at the node number 8 in case of tee joint weld considering modeled fillet radius ρequals to 2 millimeters.

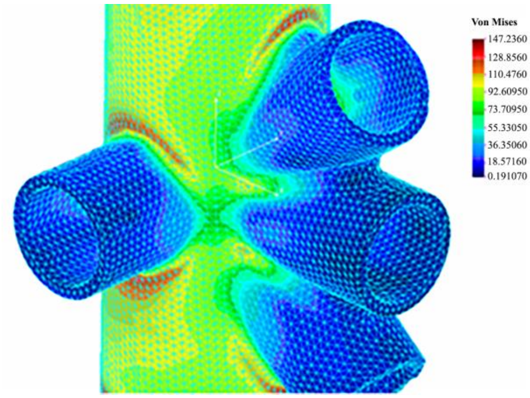

Fig. 5. Equivalent stress (von Mises) distribution at the node number 8 in case of tee joint weld considering modeled fillet radius ρequals to 8 millimeters.

The analyses performed in previous steps confirm that stress concentrators must be taken into consideration in the calculation process. The analysis determines, that axial stress in vertical truss member was σ_x = -98.5 MPa and subsequent detail calculation including weld joints lead to increase of stress range between σ_x = -147 to -159 MPa. This results leads to assumption, that stress construction factor in the case of analyzed truss structure is K= 1.5 or 1.6.
The stress concentrations are located at the beginning of the welds. These observations effect to interesting conclusion, the axial forces and stresses in lateral trusses are less than in vertical member, which is dominantly loaded. However, stress distributions in places near the weld joints are surprising, because the increase of axial stress in vertical truss member by a factor from 1.5 to 1.6 highlights necessary inclusion of local effects especially near the welds in the design process of truss structures.The structure consisted of the same profiles. Therefore in the case of analysis using solid elements, we chose a node subjected to maximum loading. The maximum loading was specified in the previous analysis of the truss structure.That is a reason why the stress construction factor at this point reaches a maximum value. In other parts of the structure, the stress construction factor would have a lower value.The goal of this analysis was to obtain the maximum value of the construction stress factor. Subsequently, this value we used in the optimization of the structure. The calculated value of the stress construction factor is in interval mentioned in the design standards.

3 Proposal and implementation of optimization process

Optimal design of the structure requires a complex solution within a cycle which start and end are based on project documentation [14, 15]. This process ensures cooperation of two interconnected programs (Pro/Engineer and COSMOS/M) with program support created by authors in Matlab. Parametric description of system allows rapid adjustment of the frame through knowledge of analysis results. The definition of the following optimization task was proposed to perform the optimization of the truss structure from the previous chapter:

– minimize weight of structure

$$F(\mathbf{x}) = \sum_{i=1}^{3} F_i(\mathbf{x}) \rightarrow \min ., \tag{1}$$

– using following constraints

$$\sigma_i(\mathbf{x}) - \sigma_{DOV} \leq 0, \qquad i = 1 \text{ až } m \qquad (2)$$
$$\lambda_{\min}(\mathbf{x}) - \lambda_{PRED} \leq 0$$

where:

- the first member defines the strength condition of the maximum allowable stress of the structure,
- the second member defines loss of stability condition,
- $\mathbf{x} = [X_1, X_2]^T$ is a vector of optimization variables,
- $\sigma_{DOV} = 120/1.6 = 75$ [MPa] is the prescribed maximum value of the axial stress in the truss structure. The coefficient equal to 1.6 was described in previous chapter,
- λ_{min} is the minimum safety coefficient due to loss of stability in the case of each loadings,
- $\lambda_P = 7$ [-] is prescribed minimum safety factor with respect to loss of stability for a given case of loadings.

Application of the above mentioned optimization formalism was used in cross sectional optimization of the tower crane truss structure. The optimization process was controlled by optdisk_2.geo program. The evaluation of the objective function was performed in subroutine called analyzer_v1.geo. The mentioned subroutine directly solves the task in COSMOS/M program. The described optimization process reduces the weight of the structure from 5293 kg (original design) to 5165 kg.

Table 1. Overview of rolled tubes used in discrete optimization.

Number of cross section	1	2	3	4	5	6	7	8	9	10
ΦD[mm]	51	54	60	89	95	102	102	108	108	108
Thickness [mm]	6	6	6	7	8	6	8	8	10	12

An overview of the optimization process results is in Table 2, Table 3, Figure 6 and Figure 7.

Table 2. Overview of results after performed optimization process.

Optimization variable	Serial number		ΦD1[mm]	Thickness 1[mm]	ΦD2 [mm]	Thickness [mm]
Starting value	7	3	102	8	60	6
Optimal value	9	1	108	10	51	6

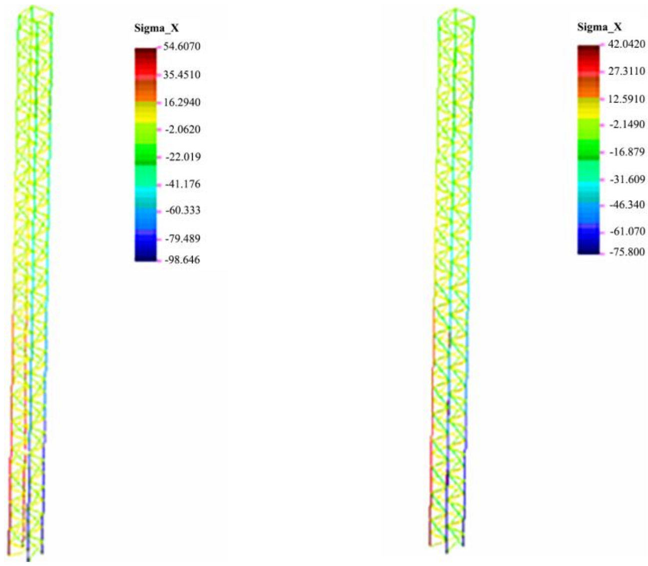

Fig. 6. Axial stress distribution in the truss structure before and after optimization.

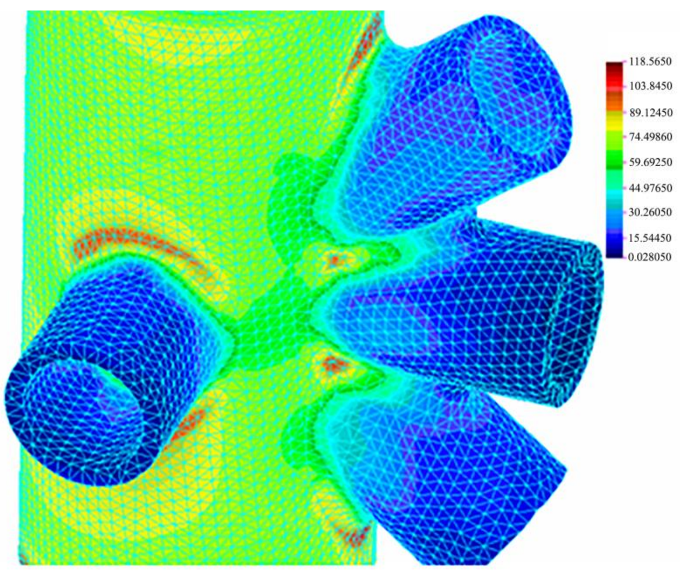

Fig. 7. Stress distribution at the locations of expected stress concentrators after performed optimization of truss structure. Fillet radius is equal to 8 millimeters.

Table 3. Overview of optimization process results.

Observed parameter	Start	Optimum
Extreme axial stress $\lvert\sigma max\rvert$ [MPa]	98.6	75.8
Safety factor λ	6.5	8.4

4 Conclusion

The intention of authors was to propose computer based procedure of applying the theoretical apparatus to the design process of welded frame structure of machines and equipments. In particular, the main task was optimization of welded truss structures regarding the expected occurrence of stress concentrators in the welds. The main part of work in not solely presentation and description of theoretical foundation about successful problem solving, but also application of the theoretical information and knowledge into software form. Introduction and interconnection between CAD program and FEM software is an optimal way to solve similar problems in technical practice. In order to satisfy determined objectives, it was necessary to carry out a number of analyzes and scientific calculations. Additionally, special proposed auxiliary programs were created especially to process number of input and output information. The primary example of the study is a proposed numerical approach to identify stress concentrators in the structure and to perform optimization of welded truss structures based on series of FEM simulations.

Acknowledgements

This work was supported byThe Ministry of Education, Science, Research and Sport of the Slovak Republic on the basis of project contract *Výskuminteligentnýchsystémov a procesov s použitímprincípov Industry 4.0 so zameraním na spájanie ťažko spojiteľných materiálov vysoko koncentrovanými zdrojmi energie - laserom a elektrónovýmlúčom*, Contract no. MŠSR 1227/2018.
This paper is supported by project KEGA 037ŽU-4/2018.

References

1. M. Kubiak, T. Domanski, V. Dekys, A. Sapietova Measurement of strain during tension test of welded joint using multi-camera 3D correlation system; XXI Polish-Slovak Scientific Conference Machine Modeling And Simulations MMS 2016. Procedia Engineering 117, 107-113 (2017).
2. M. Sapieta, A. Sapietová, V. Dekýš Comparison of the thermoelastic phenomenon expressions in stainless steels during cyclic loading. Metalurgija 56, 1-2, 203-206 (2017).
3. P. Krawiec, M. Grzelka, J. Kroczak, G. Domek, A. Kołodziej A proposal of measurement methodology and assessment of manufacturing methods of nontypical cog belt pulleys. Measurement, 132, 182–190 (2019).
4. COSMOSM 2.85, Geostar online help, Structural Research & Analysis Corp. (1999).
5. MATLAB, http: //www.mathworks.com - access 15.07.2020.
6. M. Krejsa, J. Brozovsky, D. Mikolasek, P. Parenica, J. Flodr, A. Materna, R. Halama, J. Kozak Numerical modeling of steel fillet welded joint. Advances In Engineering Software 117, 59-69 (2018).
7. K.J. Bathe, E.L. Wilson Numerical Methods in Finite Element Analysis. Prentice-Hall, New Jersey (1976).
8. Y.W. Kwon, H. Bang The Finite Element Method using MATLAB. 1st edn. Boca Raton: CRC Press, Boca Raton (1996).
9. M. Macko, A. Mroziński, A. Prentki Simulations CAE of wood pellet machine. MATEC Web of Conferences, 254, 02028 (2019).

10. J. Murín Metóda konečných prvkov pre prútové a rámové konštrukcie. STU Bratislava, Bratislava (1999).
11. J.L. Buchanan, P.R. Turner Numerical methods and analysis, McGraw-Hill, New York (1992).
12. B. Leitner, M. Vaško Design and Modelling of Tank Wagon Assembly Operations in CAM Environment. Transport Means 2015 I & II, 87-90 (2015).
13. W. Piekarska, M. Kubiak, M. Vaško Numerical estimation of the shape of weld and heat affected zone in laser-arc hybrid welded joints. Procedia Engineering 177, 114-120 (2017).
14. J.S. Arora, M.W. Huang Discrete structural optimisation with commercially available sections: a review, J. Struct. Earthquake Eng. JSCE 13, 93-110 (1996).
15. M.W. Huang, J.S. Arora Optimal design of steel structures using standard sections, Structural optimisation 14, 24-35 (1997).