JOURNAL OF MECHANICAL ENGINEERING AND TRANSPORT

Journal homepage: https://met-journal.com.ua/en

Vol. 10, No. 2, 2024 99-108

UDC 621.177; 621.314

DOI: 10.63341/vjmet/2.2024.99

Article's History:

Received: 02.09.2024; Revised: 18.11.2024; Accepted: 26.12.2024.

Maksym Omelian*

Postgraduate Student

Ternopil Ivan Puluj National Technical University

46001, 56 Ruska Str., Ternopil, Ukraine

https://orcid.org/0009-0008-7501-5318

Method of finite element modelling of the stress-strain state parameters of a flat truss with parallel belts

Abstract. The purpose of the study was to create a method for modelling the stress-strain state of a flat truss with parallel belts using multi-package software based on the finite element method. It was established that the software complexes LIRA-CAD 2016 R5 and ANSYS Workbench 14.5 are the most effective for calculating the parameters of the stress-strain state of flat trusses, since they provide high modelling efficiency due to the developed functionality and adaptability to engineering analysis tasks. In the course of the study, the capabilities and interfaces of these software environments were analysed, and simulations were performed for a truss with a triangular grid and dimensions of 18,000×3,600 mm, made of VCt3ps structural steel. Geometric and finite element models of trusses in the media of the identified calculation complexes were developed. For modelling, elements made of rolled corners with a cross-section of 100×100×10 mm were used, and nodal kerchiefs were made of steel sheet with a thickness of 10 mm. For the truss model, ANSYS Workbench 14.5 created a finite element grid with size sampling, in particular for styles, which provides more accurate determination of the parameters of the stress-strain state of SSS in critical zones of the truss. In LIRA-CAD 2016 R5, the truss model was divided into 10x10 mm elements. The proposed method included two main stages: at the first stage, a model was created to determine the stress-strain state of the truss and the environment of the LIRA-CAD 2016 R5 software suite, at the second stage, the results obtained were analysed in the environment of the ANSYS Workbench 14.5 calculation complex to optimise the design parameters. The use of this technique can significantly reduce the time required for design and calculations, which helps to increase the efficiency of designing trusses with parallel belts. The practical value of the development lies in the possibility of optimising the shapes and sizes of cross-sections of elements, which positively affects the economic efficiency of the designed trusses, reducing material costs and ensuring the durability of structures

Keywords: computer simulation, optimisation, rod structures, engineering analysis, variational calculation methods

Introduction

Flat trusses are widely used in various branches of the national economy. Most often, such structures are used in the construction of industrial and public structures, bridge crossings. However, along with this, flat trusses, in particular those with parallel belts, have found application in the transport sector, agricultural engineering, etc. The use of such flat trusses is conditioned by the variety of geometric shapes, a significant degree of automation in manufacturing. In this regard, the issue of reducing the complexity of designing and reducing the metal consumption of trusses with parallel belts is relevant.

S. López *et al.* (2022) presented a methodology for identifying key events leading to progressive destruction on truss-type bridges. The methodology includes a new database that collects detailed information about initial damage and its spread, and the consequences of the collapse. The methodology was implemented by collecting information from 25 case studies presented in the literature. The results

Suggested Citation:

Omelian, M. (2024). Method of finite element modelling of the stress-strain state parameters of a flat truss with parallel belts. *Journal of Mechanical Engineering and Transport,* 10(2), 99-108. doi: 10.63341/vjmet/2.2024.99.

*Corresponding author

revealed the most frequent initial constituted damage state or failure (ICDS) leading to progressive destruction.

Two variants of the algorithm, such as GSSA and GSSB, were proposed to optimise the dimensions of four high-dimensional steel truss samples in the studies by A. Ahrari *et al*. (2020), S.K. Azad & S. Aminbakhsh (2021), H. Cao *et al*. (2023), namely: a single-layer bulb dome of 2,075 elements, a two-layer open dome of 2,688 elements, a two-layer comb dome of 6,000 elements and a two-layer lattice of 15,048 elements in accordance with the AISC-LRFD specification. The obtained numerical results indicate the effectiveness of GSSA and GSSB in working with high-size samples of large-sized steel trusses.

A.A. Mousavi *et al*. (2020; 2022) proposed a method for empirical mode decomposition using the adaptive noise method to determine the presence, location, and severity of damage for a steel truss bridge model. The developed laboratory model was exposed to disturbances, which helped to obtain experimental data. Using the technique, four key signal characteristics were identified: energy, instantaneous amplitude, phase, and instantaneous frequency, which became the basis for qualitative and quantitative damage assessment. To increase the sensitivity of the method to damage detection, improved indices have been developed that combine statistical characteristics of time signals, such as kurtosis and entropy, with energy and instantaneous amplitude parameters. The experimental results demonstrated the high efficiency and sensitivity of the proposed approach, which is superior to existing methods of empirical decomposition of modes in the tasks of detecting, localising, and assessing the severity of damage.

The paper by F. Parisi *et al*. (2022) described a method for detecting and localising damage in steel truss railway bridges using machine learning algorithms. This approach allows automatically analysing raw signals from strain sensors without the need for pretreatment or feature isolation. The results showed that the proposed approach can detect damage with an accuracy of up to 93%, which makes it an effective tool for monitoring the condition of structures.

W. Wu *et al.* (2022) proposed a linear model for predicting the corrosion of truss structures based on the concept of linear cumulative damage to miners to solve the difficulties of conducting accelerated corrosion tests. In addition, an equivalent fatigue test has been developed in combination with a corrosion prediction model. A model for predicting fatigue damage based on corrosion factors was developed and the law of fatigue damage and degradation of structural rigidity in a corrosive environment was determined.

In the study by G. Weng *et al.* (2020) to detect stresses in steel structures and elements using existing magnetism, a magnetic stress sensing system was developed that includes a magnetic flux induction coil, a magnetic flux measurement device, a load device, and data acquisition software. In addition, a practical formula for determining stresses was obtained through experiments. On this basis, a typical model of the steel truss structure of the Bailey beam was designed and manufactured under various operating conditions.

M. Hohol *et al.* (2020) performed structural and parametric synthesis of steel combined trusses with a span of 30 m to minimise mass and potential energy. It was shown that the maximum load-bearing capacity of the structure was achieved by applying the criterion of rational design for strength. It was established that the use of combined trusses instead of conventional ones provides significant metal savings, reduces the complexity of manufacturing, and reduces installation time.

The main stages of strengthening steel beams of a sloping roof for a typical metal frame with initial geometric imperfections according to the deformed scheme were considered by S. Hudz *et al.* (2019) based on a new approach to the analysis of internal force factors, the features of the working conditions of a beam under the joint action of transverse bending in two planes and bending torsion resulting from the eccentricity of load stacking were found and described.

Based on this, it is relevant to search for a reliable method for modelling the parameters of the stress-strain state (SSS) of a flat truss with parallel belts, which will give full information about the distribution and magnitude of stresses, the value and nature of deformations depending on the acting external forces. The purpose of the study was to develop a methodology for modelling the SSS of a flat truss with parallel belts using multi-package software tools based on the finite element method (FEM). Objectives of the study: to determine multi-package software tools suitable for calculating the SSS parameters of flat trusses with parallel belts and to analyse the environments of detected software tools suitable for calculating the SSS parameters of flat trusses with parallel belts; to develop a full-size finite element model of a flat truss with parallel belts in the environment of detected software tools. The scientific originality lies in the fact that a two-stage method for modelling the SSS parameters of a flat truss with parallel belts using multi-package software tools based on the finite element method is proposed.

Materials and Methods

The research methodology was based on calculation complexes that work using the finite element method. This universal variational approach is used to solve the most complex problems, in particular, in the mechanics of deformed solids, fracture mechanics, and applied mechanics. The main idea of this method (Hrebeniuk & Homeniuk, 2022) was to divide the continuous volume of a real construct into subunits consisting of finite but small elements called finite elements (FE). Within each such element, the desired function can be replaced with an approximate one. This allows representing any quantity that continuously changes in the volume of a body in the form of a discrete model consisting of a set of piecewise continuous functions. Each of these functions was defined within the corresponding sub-domain, providing an accurate approximation for analysing the actual construct. The finite element method in the current problem was implemented based on the displacement method – functions approximate movements within finite elements.

To calculate the parameters of the stress-strain state (SSS) of flat trusses with parallel belts, software tools were defined: LIRA-CAD 2016 R5 at the first stage and ANSYS Workbench 14.5 at the second stage, respectively. The finite element truss model corresponded to the parameters of a real truss with geometric dimensions of 18,000×3,600 mm. Truss material – VSt3ps steel, the physical and mechanical properties of which included the elastic modulus: $E = 2.06 \times 10^5$ MPa and density $\rho = 7{,}800$ kg/m^3. The truss elements had the following characteristics: the profile of braces and belts – rolled corners 100×100×10 mm, kerchiefs are made of steel sheet with a thickness of 10 mm.

At the first stage of modelling in the LIRA-CAD 2016 R5 environment, a three-dimensional rod geometric model of the truss was created, the physical and mechanical characteristics of materials were set, a finite element grid with an element size of 10×10 mm was formed, and loads were applied. The truss was loaded with a concentrated force of $P = 10$ kN, evenly distributed between two upper belt nodes $P/2 = 5$ kN per node. Calculations were performed using the finite element method, but the kerchiefs in the nodes were not modelled, which was considered in further analysis. At this stage, data on stresses, deformations, and internal forces in the truss elements were obtained.

The second stage involved modelling in the ANSYS Workbench 14.5 environment, where the geometric truss model was built in the Design Modeler module, taking into consideration styles. Similarly to the first stage, the characteristics of materials and loads were set, and a more detailed finite element grid was formed. This helped to increase the accuracy of calculations and consider the nodal features of the design.

Comparison of the results of the two modelling stages allowed evaluating the accuracy and efficiency of the applied software packages. The results included stress distribution, strain shapes, and identification of critical structural zones. The methodology proved that the combination of engineering solutions with modern software tools ensures the efficiency of design and calculations, optimisation of geometric parameters of truss elements, and high accuracy of stress-strain state analysis.

Description of the LIRA-CAD 2016 R5 settlement complex

The LIRA-CAD software package is a modern representative of the Ukrainian software of the LIRA family. LIRA-CAD 2016 R5 (Ukraine) is a multifunctional software package for designing and calculating machine-building and building structures for various purposes (Strelets-Streletsky *et al.*, 2019). The programme allows performing calculations for static and dynamic loads using the finite element method as a basis. Various connected modules (processors) provide the ability to select and check cross-sections of steel and reinforced concrete structures, model the ground, calculate bridges and analyse the behaviour of buildings during installation, etc. The LIRA-CAD PC consists of several interconnected information systems. The organisation of relationships between these systems ensures the manufacturability of working with the complex so that the complex itself guides the user – from creating a calculation model to designing elements. The main graphics system is the VISOR-CAD system (Fig. 1.), a single graphical environment that has a large set of features and functions for forming adequate finite element and super-element models of calculated objects. VISOR-CAD allows conducting a detailed visual survey of the created models and their correction, describing the physical and mechanical properties of materials.

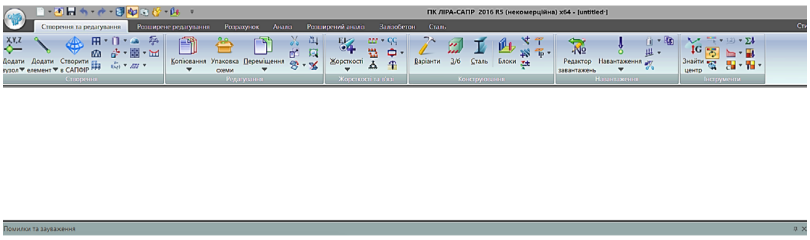

Figure 1. LIRA-CAD 2016 R5 graphics environment window

Source: E. Strelets-Streletsky et al. (2019)

In the same environment, relationships and loads were set, and relationships between different loads were assigned to determine their most dangerous combinations. To calculate the created model, a linear calculation processor was selected, which implements the calculation of linearly deformed structures for static and dynamic impacts. The LIRA-CAD PC includes several calculation processors. All of them were designed to determine the stress-strain state of the structure based on the finite element method in displacements. Calculation processors implement modern advanced methods for solving systems of equations, which have high speed and allow solving systems with a very large number of unknowns. The linear processor was designed to solve problems describing the operation of the material of structures in a linear-elastic formulation. A nonlinear processor allows solving problems related to the physical nonlinearity of a material in the framework of nonlinear elasticity theory and in elastic-plastic formulation (concrete, reinforced concrete, steel concrete, metal, soil). Such problems were solved using the step-by-step and step-by-iterative methods. The nonlinear processor allows solving problems related to geometric nonlinearity (shrouds, large-span coatings, membranes), and structural nonlinearity (contact problems, one-way connections, friction). The library of nonlinear finite elements also includes elements that allow simultaneous accounting of physical and geometric nonlinearities. When calculating nonlinear problems using the step method, the load step was automatically selected based on its history. Calculation processors contained an extensive library of finite elements, which makes it possible to create adequate calculation models with almost no restrictions on describing the real properties of the calculated objects. The software environment of the LIRA-CAD PC provides an opportunity to model complex structures, determine their stress-strain state using linear and nonlinear processors that consider various types of loads, dynamic influences, physical, geometric, and structural nonlinearities.

Description of the methodology of the ANSYS Workbench calculation complex 14.5

The ANSYS Workbench 14.5 PC used the finite element method for mathematical modelling, combining the versatility of algorithms for various tasks with efficient computer implementation of calculations. The problem was solved using the finite element method in the ANSYS software suite in three stages. At the first stage, the foundations of the finite element model of the object under study were formed. This step included the following steps:

- determination of the physical type of the problem (solid state mechanics, heat transfer, hydrodynamics, etc.) and appropriate programme settings;
- selection of the type of finite element depending on the size and characteristics of the object, with the ability to set element properties;
- selection of the object's material and specification of its properties (volume weight, tensile strength, yield strength, Poisson's ratio, etc.), which can be entered manually or imported from the ANSYS material library. Properties determine the material model (linear-elastic, elastic-plastic, bilinear, etc.), which affects the choice of equations for the finite element method;
- creating a geometric solid-state model of an object (3D model) performed in the PREP7 module in classical ANSYS and in the Design Modeler module in Workbench (Fig. 3.). The geometric model can also be exported from any CAD package;
- geometric model was divided into finite elements, with the ability to configure grid parameters.

This step forms a complete mathematical and physical model of the object, which was crucial for obtaining accurate and reliable results in the next stages of modelling.

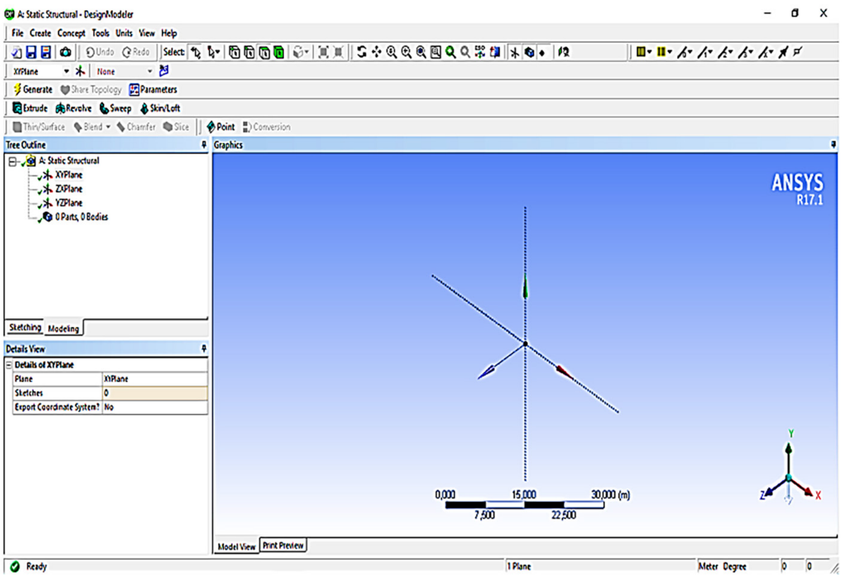

Figure 2. Design Modeler module window in ANSYS Workbench 14.5

Source: L. Danylchenko (2021)

The second step – applying physical conditions and solving the problem – consists of three main steps:

- setting boundary conditions (forces, displacements, connections, etc.);
- selecting the type of analysis (static, dynamic, modal, etc.) and the method for solving equations and parameters of computational procedures (number of load steps, iterations, etc.);
- solving system of equations by the finite element method. As a result, a file was created with a vector of the found degrees of freedom (nodal displacements, temperatures, etc.).

The described step was necessary for direct numerical analysis of the model using the finite element method. It allows forming a mathematical statement of the problem, considering the physical conditions and load parameters, and getting results that characterise the behaviour of the structure.

The third step was to analyse the results obtained. Physical quantities calculated using the finite element method (displacement, deformation, stress, temperature, etc.) are displayed in the ANSYS Gui as images, tables, graphs, or animations (Danylchenko, 2021). The information provided at this stage provided an understanding of the operation of the structure as a whole, and its individual components. This allows drawing conclusions about the effectiveness of constructive decisions made.

Results

Example of developing a flat truss model with parallel belts

A truss with parallel belts and a triangular lattice was used for finite element modelling (Fig. 3) and geometric dimensions of 18,000x3,600 mm. Truss material – VSt3ps steel, corresponding physical and mechanical properties imported from the VDt3ps steel material model. Elastic modulus of steel $E = 2.06 \cdot 10^5$ MPA, density $\rho = 7{,}800$ kg/m³. Profile of braces and belts – rolled corners with a cross-section of 100×100×10 mm. Kerchiefs in knots are made of 10 mm thick steel sheet.

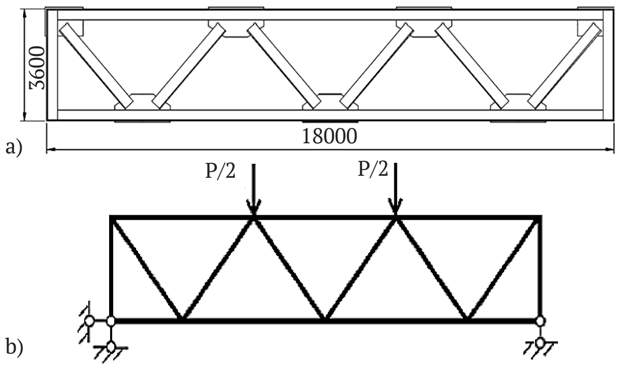

Figure 3. Graphical diagram of the truss

Notes: *a) truss grid diagram; b) truss load diagram*

Source: compiled by the author

The truss is loaded with an external total load in the form of a concentrated force $P = 10$ kN. The force P (10 kN) is distributed between two nodes on the upper truss belt. Thus, each of these nodes accounts for half of the total force, i.e., $P/2 = 10$ kN$/2 = 5$ kN.

At the first stage, geometric and finite element models were constructed using these parameters in the LIRA-CAD 2016 R5 PC (Fig. 4). Based on the calculation the internal forces arising in the structural elements due to the exter nal force $P/2$ applied at the corresponding nodes were determined, as well as the cross-sectional dimensions of the truss elements resulting from the action of these forces. This information allows adjusting, if necessary, the previously specified cross-section dimensions in the truss elements. The importance of the obtained data lies in the fact that they allow carrying out a general assessment of the stress-strain state of the structure, determining the forces in its elements, and clarifying the dimensions of cross-sections. These results are crucial for further import of the model into ANSYS Workbench 14.5. Skipping this stage may result in incorrect truss design and errors in subsequent analysis stages.

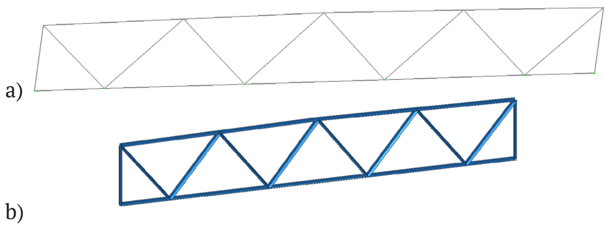

Figure 4. Truss model in the LIRA-CAD environment 2016 R5

Notes: *a) geometric model; b) finite element grid model*

Source: compiled by the author

At the second stage, in the environment of the ANSYS Workbench 14.5 software suite, the calculation data was imported from the LIRA-CAD 2016 R5 PC and the geometric and finite element model of a full-size truss was rebuilt, followed by a division into a finite element grid (Fig. 4). The shapes of cross-sections of braces, belts, and support posts are displayed.

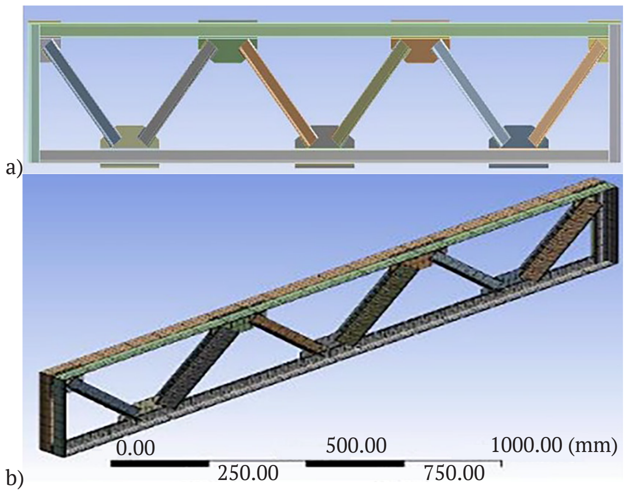

Figure 5. Truss model in the ANSYS Workbench 14.5 environment

Notes: *a) CAD – geometric model; b) FEM –finite element grid model*

Source: developed by author

The three-dimensional geometric structure of the truss, which was developed in the Design Modeler module, is shown. This model is the basis for further analysis and reflects the geometric parameters of the object. The result of splitting the geometric model into finite elements is also shown. This mesh model is used to perform finite element calculations (FEA). The figure illustrates the transition from CAD modelling to the FEM analysis stage, emphasising the importance of the accuracy of constructing a geometric model and the quality of grid creation to ensure the reliability of calculations. The current stage is important for analysing the stress-strain state, since the calculation results in the stress distribution in the cross-sections taken at the first stage (Fig. 5). The ANSYS PC does not programmatically provide cross-section selection functions, unlike the LIRA-CAD PC, but the ANSYS PC allows assessing the stress-strain state more locally, in particular, identifying places of stress concentrations.

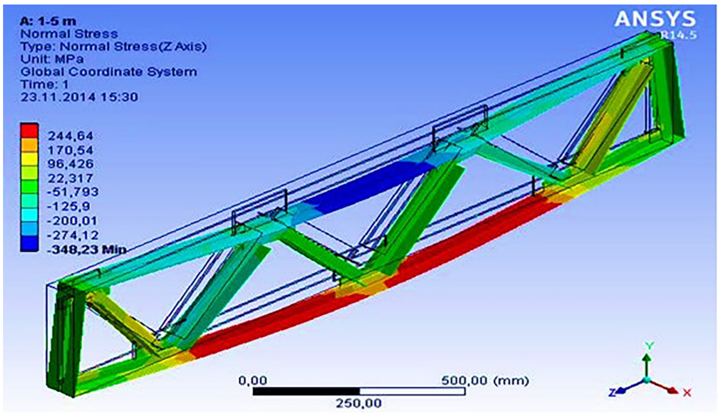

Figure 6. Distribution of normal stresses

Source: developed by author

The finite element grid in ANSYS is created with dimensions of 10x10 mm, and the mesh in the shapes is discrete. In the LIRA-CAD 2016 R5 PC, the finite element grid was created with similar parameters, with the exception of styles. The stages of truss modelling in the LIRA-CAD 2016 R5 software environment, which is widely used for engineering analysis of building structures, are demonstrated. At the first stage, a basic three-dimensional model of the truss is created, which accurately reflects its geometric parameters, in particular, the length, height, and shape of elements. The geometric model is formed in accordance with the specified design characteristics and is the basis for further analysis. A finite element grid model is obtained by dividing the truss geometry into finite elements. The grid model is a key step in applying the finite element method, since it allows translating a physical problem into a discrete form suitable for numerical solution. The model is divided into elements based on the geometric complexity of the object and the required accuracy of calculations.

The truss geometry is divided into finite elements, which provides discretisation of the physical problem for its numerical solution. Loads at the current stage were set similarly to the calculation in the LIRA PC, in the form of a concentrated force $P = 10$ kN distributed between two nodes on the upper truss belt, so each of these nodes accounts for half of the total force, i.e., $P/2 = 10\,\text{kN}/2 = 5$ kN. The grid is constructed considering the geometric complexity of the design and the requirements for the accuracy of calculations, which is key for the effective application of the finite element method. Based on the import of cross-sections and geometry of the CE truss model from the LIRA CAD, the calculation is performed in ANSYS Workbench 14.5, which results in graphic materials with a set of information about the stress distribution in individual truss elements and in characteristic places (nodes, places of load application).

The developed method offers two stages of finite element modelling of the parameters of the stress-strain state of a flat truss with parallel belts. The first stage includes creating a geometric rod model in the environment of the LIRA-CAD 2016 R5 software package, setting cross-sections and physical and mechanical characteristics of materials, applying loads, and further calculation using the finite element method. The external load is a concentrated force $P = 10$ kN. The force from $P/2$ is applied in two truss nodes, which are located on the upper belts. As a result, in the first iteration, a set of data was obtained in tabular form. The Table 1 contains the results of estimating the percentage exhaustion of the stability of elements relative to the OY and OZ axes, the stability of the compressed wall and compressed belt, and the exhaustion of the load-bearing capacity by groups of limit states. The percentage of exhaustion of local stability is indicated separately. These data allow comprehensively assessing the effectiveness of selecting the cross-section of individual structural elements.

Table 1. Result of checking the specified cross-sections of truss elements in the LIRA CAD

Element	OY_1 %	OZ_1 %	SW %	SB %	1LS %	2LS %	L.S.
10	2	2	0	46	2	0	46
10	1	1	0	46	1	0	46
11	2	2	0	46	2	0	46
11	1	1	0	46	1	0	46

Continued Table 1.

Element	OY_1 %	OZ_1 %	SW %	SB %	1LS %	2LS %	L.S.
12	0	0	0	0	2	0	0
12	0	0	0	0	2	0	0
13	1	1	0	46	1	0	46
13	1	1	0	46	1	0	46
14	0	0	0	0	0	0	0
14	0	0	0	0	0	0	0
15	0	0	0	46	0	0	46
15	0	0	0	46	0	0	46
16	0	0	0	46	0	0	46
16	0	0	0	46	0	0	46
17	0	0	0	0	0	0	0
17	0	0	0	0	0	0	0
18	1	1	0	46	1	0	46
18	1	1	0	46	1	0	46
19	0	0	0	0	2	0	0
19	0	0	0	0	2	0	0
1	3	3	0	46	3	0	46
1	3	3	0	46	3	0	46
2	0	0	0	0	1	0	0
2	0	0	0	0	1	0	0
3	0	0	0	0	1	0	0
3	0	0	0	0	1	0	0
4	0	0	0	0	1	0	0
4	0	0	0	0	1	0	0
5	3	3	0	46	3	0	46
5	3	3	0	46	3	0	46
6	2	2	0	46	2	0	46
6	2	2	0	46	2	0	46
7	4	4	0	46	4	0	46
7	4	4	0	46	4	0	46
8	4	4	0	46	4	0	46
8	4	4	0	46	4	0	46
9	2	2	0	46	2	0	46
9	2	24	0	46	2	0	46

Notes: OY_1 % – percentage of stability exhaustion along the Y-axis; OZ_1 % – percentage of exhaustion of stability along the Z-axis; SW % – percentage of exhaustion of stability of the compressed wall; SB % – percentage of exhaustion of stability of the compressed belt; 1LS % – percentage of exhaustion of load-bearing capacity for the 1st group of limit states; 2LS % – percentage of exhaustion of load-bearing capacity for the 2nd group of limit states; L.S. – percentage of exhaustion of local stability
Source: developed by the author

The second stage involves importing calculated data from the LIRA CAD and then rebuilding the geometric model in ANSYS Workbench 14.5, followed by setting cross-sections and physical and mechanical characteristics of materials, applying loads and further calculation. At both stages, the physical and mechanical characteristics of the materials, the shape and dimensions of the cross-sections of the truss elements were identical. As a result, the design and calculation of flat trusses with parallel belts becomes more efficient both in terms of time spent and as a result of the efficient arrangement of elements with the required cross-sectional dimensions.

Discussion

The finite element method is a widely used tool in the design and analysis of various structures, in particular trusses. The study by Y. Shved *et al*. (2020) presented a method for determining durability using SOLIDWORKS PC, which allows effectively assessing the reliability of structures. Although this method was based on different software, the results obtained in this paper provide important comparisons regarding the durability of steel trusses. They point out the importance of detecting fatigue cracks, especially in K-shaped nodes on the upper chord of the truss, which is a critical factor in the operation of structures. According to the results of verification, the indicators of fatigue damage were accurate with an error of only 5.7%. This indicates the high reliability of the method used for long-term monitoring of the state of trusses. However, the use of SolidWorks for these calculations has its limitations, especially in modelling complex loads and material properties, where more powerful software tools such as ANSYS can show better results.

The study by M. Basara *et al*. (2021) applied a comprehensive approach that includes both a semi-natural experiment and computer modelling in the ANSYS Workbench environment. Experimental studies of the durability of a rectangular welded truss have confirmed the importance of accurate visualisation of fatigue damage, in particular, in K-shaped nodes, where there is also a decrease in the load-bearing capacity of the structure. Compared to the method proposed by Y. Shved *et al*. (2020), it was found that the use of combined methods allows for more accurate results, although the error in comparison with a semi-natural experiment does not exceed 5.7%. In addition, M. Basara *et al.* (2021) demonstrated that combining real-world tests with numerical methods allows for more accurate predictions, which is important for the practical assessment of structural reliability.

Y. Kovalchuk *et al*. (2023) proposed a new approach to determining the behaviour of welded structures under thermomechanical influences, which achieves high reliability of results due to computer modelling. In particular, it was found that the research results follow a normal distribution with minimal deviations for tensile strength and fluidity (2.1% and 2.6%, respectively). This approach has shown high accuracy and allows more accurately considering the effect of thermomechanical loads on structures, which is important for accurate forecasting of their operational life under conditions of high temperatures and variable loads.

The study by H. Liu *et al*. (2024) presented an innovative approach by applying a deep reinforcement learning method to update steel truss models. This allows integrating the long-term memory network (LSTM) with the deep deterministic policy gradient (DDPG) algorithm, which increases the efficiency of the algorithm in real-world operating conditions. The results obtained using the NF-LSTM-DDPG algorithm were significantly more accurate, with an average relative error of 14.15% and more reliable indicators of changes in the elastic modulus compared to other methods. This study provides important guidance for further improvement of real-world truss modelling and monitoring techniques, particularly for adaptive systems to detect potential damage that may occur over a long period of operation.

The algorithm proposed by N. Khodadadi & S. Mirjalili (2022) generalised normal distribution optimisation (GNDO) used to design truss structures with optimal weight. The ability of the proposed algorithm to deal with such problems is investigated. To evaluate the GNDO algorithm, three reference problems for optimising trusses with frequency constraints were considered. Numerical data show that GNDO is more reliable, stable, and efficient for structural optimisation problems than other metaheuristic algorithms. J. Pierezan *et al.* (2021) presented a modified coyote optimisation algorithm (MCOA), which has the advantage of using chaotic sequences to improve solutions and avoid local minima, and adaptive parameter updates. These improvements allow the algorithm to effectively solve problems of optimising the weight and stability of metal structures. M. Azizi *et al.* (2022) considered the optimisation of lattice trusses using the Chaos Game Optimisation (CGO) algorithm, which showed high efficiency for multi-element structures. A list of other algorithms and parameters for optimising steel structures was given in the studies by A. Kaveh *et al*. (2021), T.N. Huynh *et al*. (2021). G. Caredda *et al*. (2022) considered an adaptive differential evolution (DE) algorithm for truss optimisation by integrating the Q-learning method. This approach is known as q-learning DE (qlDE).

In general, the results of the above-mentioned studies confirm the effectiveness of using ITU to investigate the durability of steel trusses, which allows increasing the accuracy of forecasting and adaptability to real changes in loads and operating conditions. This allows getting more accurate results, which are important for the durability and safety of steel structures in real-world conditions. Moreover, the use of multi-stage methods with multi-package software based on the finite element method has not become widespread. Thus, the method proposed in this study included two main stages of determining the SSS parameters: at the first stage, a model was created to determine the stress-strain state of the truss and the environment of the LIRA-CAD 2016 R5 software suite, and at the second stage, the results obtained to optimise the design parameters were analysed in the environment of the ANSYS Workbench 14.5 calculation complex.

Conclusions

The study examined the method of modelling the parameters of the stress-strain state of a flat truss with parallel belts using multipack software based on the finite element method. The author managed to achieve this goal by developing a two-stage methodology that allows optimising design parameters and increasing the efficiency of designing such structures.

In the course of the study, the functionality of the LIRA-CAD 2016 R5 and ANSYS Workbench 14.5 software complexes were analysed. Geometric and finite element models of a truss with a triangular grid and dimensions of 18,000×3,600 mm, made of structural steel, were developed. At the first stage of the methodology, a basic truss model was created in LIRA-CAD 2016 R5 for preliminary analysis of the stress-strain state. The second stage involved refining the results and optimising the design parameters in the ANSYS Workbench 14.5 environment. The proposed method creates prerequisites for reducing the complexity of design and calculations, ensuring the accuracy of determining the parameters of stresses and deformations of trusses. The practical value of the technique lies in the possibility of optimising the shapes and sizes of cross-sections of truss elements, which creates the basis for reducing material costs and increasing the economic efficiency of structures.

The developed technique is important for studying the stress-strain state of flat trusses, taking into consideration the influence of force factors and ensuring the reliability of structures. It can be used to analyse and optimise other types of structures using multi-package software.

A promising area of further research is the expansion of the methodology to more complex types of structures, the investigation of truss behaviour under nonlinear conditions, and the integration of artificial intelligence algorithms for automating parameter optimisation.

Acknowledgements

None.

Conflict of Interest

None.

References

[1] Ahrari, A., Atai, A.A., & Deb, K. (2020). A customized bilevel optimization approach for solving large-scale truss design problems. *Engineering Optimization*, 52(12), 2062-2079. doi: 10.1080/0305215X.2020.1740690.

[2] Azad, S.K., & Aminbakhsh, S. (2021). High-dimensional optimization of large-scale steel truss structures using guided stochastic search. *Structures*, 33, 1439-1456. doi: 10.1016/j.istruc.2021.05.035 .

[3] Azizi, M., Aickelin, U., Khorshidi, H.A., & Shishehgarkhaneh, M.B. (2022). Shape and size optimization of truss structures by Chaos game optimization considering frequency constraints. *Journal of Advanced Research*, 41, 89-100. doi: 10.1016/j.jare.2022.01.002.

[4] Basara, M., Kovalchuk, Y., & Shynhera, N. (2021). Durability of a welded truss under cyclic loads. *Innovative Solution in Modern Science*, 5(41), 147-158. doi: 10.26886/2414-634X.5(41)2020.11.

[5] Cao, H., Sun, W., Chen, Y., Kong, F., & Feng, L. (2023). Sizing and shape optimization of truss employing a hybrid constraint-handling technique and manta ray foraging optimization. *Expert Systems with Applications*, 213(B), article number 118999. doi: 10.1016/j.eswa.2022.118999.

[6] Caredda, G., Porcu, M. C., Buitrago, M., Bertolesi, E., & Adam, J.M. (2022). Analysing local failure scenarios to assess the robustness of steel truss-type bridges. *Engineering Structures*, 262, article number 114341. doi: 10.1016/j.engstruct.2022.114341.

[7] Danylchenko, L. (2021). *Methodical workshop on the topic: "Engineering analysis in Ansys Workbench" in the discipline: "Computer modelling of materials processing processes" for practical classes and independent work of applicants for the educational level of Doctor of Philosophy in speciality 131 "Applied Mechanics"*. Ternopil: Ternopil Ivan Pului National Technical University.

[8] Parisi, F., Mangini, A.M., Fanti, M.P., & Adam, J.M. (2022). Automated location of steel truss bridge damage using machine learning and raw strain sensor data. *Automation in Construction*, 138, article number 104249. doi: 10.1016/j.autcon.2022.104249.

[9] Hohol, M., Gasii, G., Pents, V., & Sydorak, D. (2020). Structural-parametric synthesis of steel combined trusses. In V. Onyshchenko, G. Mammadova, S. Sivitska & A. Gasimov (Eds.), *Proceedings of the 3rd international conference on building innovations. ICBI 2020. Lecture notes in civil engineering* (Vol. 181, pp. 163-171). Cham: Springer. doi: 10.1007/978-3-030-85043-2_16.

[10] Hrebeniuk, S., & Homeniuk, S. (2022). *Numerical methods for solving mechanical problems*. Zaporizhzhia: ZNU.

[11] Hudz, S., Storozhenko, L., Gasii, G., & Hasii, O. (2019). Features of operation and design of steel sloping roof purlins. In V. Onyshchenko, G. Mammadova, S. Sivitska & A. Gasimov (Eds.) *Proceedings of the 2nd international conference on building innovations. ICBI 2019. Lecture notes in civil engineering* (Vol. 73, pp. 65-73). Cham: Springer. doi: 10.1007/978-3-030-42939-3_8.

[12] Huynh, T.N., Do, D.T., & Lee, J. (2021). Q-Learning-based parameter control in differential evolution for structural optimization. *Applied Soft Computing*, 107, article number 107464. doi: 10.1016/j.asoc.2021.107464.

[13] Kaveh, A., Khodadadi, N., & Talatahari, S. (2021). A comparative study for the optimal design of steel structures using CSS and ACSS algorithms. *International Journal of Optimization in Civil Engineering*, 11(1), 31-54.

[14] Khodadadi, N., & Mirjalili, S. (2022). Truss optimization with natural frequency constraints using generalized normal distribution optimization. *Applied Intelligence*, 52, 10384-10397. doi: 0.1007/s10489-021-03051-5.

[15] Kovalchuk, Y., Shynhera, N., & Shved, Y. (2023). Formation of input information arrays for computer simulation of welded trusses behavior under thermal force effects. *Scientific Journal of TNTU*, 110(2), 118-124. doi: 10.33108/visnyk_tntu2023.02.118.

[16] Liu, H., Yang, Z., Zhou, T., Wang, L., & Chen, Z. (2024). Study on updating finite element model of steel truss structure based on knowledge-enhanced deep reinforcement learning. *Engineering Structures*, 316(1), article number 118576. doi: 10.1016/j.engstruct.2024.118576.

[17] López, S., Makoond, N., Sánchez-Rodríguez, A. & Adam, J.M., & Riveiro, B. (2023). Learning from failure propagation in steel truss bridges. *Engineering Failure Analysis*, 152, article number 107488. doi: 10.1016/j.engfailanal.2023.107488.

[18] Mousavi, A.A., Zhang, C., Masri, S.F., & Gholipour, G. (2020). Structural damage localization and quantification based on a CEEMDAN Hilbert transform neural network approach: A model steel truss bridge case study. *Sensors*, 20(5), article number 1271. doi: 10.3390/s20051271.

[19] Mousavi, A.A., Zhang, C., Masri, S.F., & Gholipour, G. (2022). Structural damage detection method based on the complete ensemble empirical mode decomposition with adaptive noise: A model steel truss bridge case study. *Structural Health Monitoring*; 21(3), 887-912. doi: 10.1177/14759217211013535.

[20] Pierezan, J., dos Santos Coelho, L., Mariani, V.C., de Vasconcelos Segundo, E.H., & Prayogo, D. (2021). Chaotic coyote algorithm applied to truss optimization problems. *Computers & Structures*, 242, article number 106353. doi: 10.1016/j.compstruc.2020.106353.

[21] Shved, Y., Kovalchuk, Y., Shynhera, N., & Voronchak, V. (2020). Fatigue damage of the heel joint of welded roof truss. *Scientific Journal of TNTU*, 99(3), 28-33. doi. 10.33108/visnyk_tntu2020.03.028.

[22] Strelets-Streletsky, E., Zhuravlev, A., & Vodopyanov, R. (2019). *LIRA-CAD. Book I. Fundamentals*. Kyiv: LIRALAND.

[23] Weng, G., Wang, J., Liu, Y., Zhu, X., & Dai, J. (2020). Magnetic stress sensing system for nondestructive stress testing of structural steel and steel truss components based on existing magnetism. *Sensors*, 20(14), article number 4043. doi: 10.3390/s20144043.

[24] Wu, W., He, X., He, L., Wu, C., He, J., & Zhu, A. (2022). Joints fatigue damage prediction for a steel truss suspension bridge considering corrosion environment. *Arabian Journal for Science and Engineering*, 47, 4879-4892. doi: 10.1007/s13369-021-06318-8.

Максим Омелян

Аспірант
Тернопільський національний технічний університет імені Івана Пулюя
46001, вул. Руська, 56, м. Тернопіль, Україна
https://orcid.org/0009-0008-7501-5318

Методика скінченно-елементного моделювання параметрів напружено-деформівного стану плоскої ферми з паралельними поясами

Анотація. Мета роботи полягає у створенні методики моделювання напружено-деформованого стану плоскої ферми з паралельними поясами за допомогою мультипакетного програмного забезпечення, заснованого на методі скінченних елементів. Встановлено, що програмні комплекси ЛІРА-САПР 2016 R5 та ANSYS Workbench 14.5 є найбільш ефективними для розрахунків параметрів напружено-деформівного стану плоских ферм, оскільки забезпечують високу ефективність моделювання завдяки розвиненій функціональності та адаптованості до завдань інженерного аналізу. У процесі дослідження було здійснено аналіз можливостей та інтерфейсів зазначених програмних середовищ, а також проведено моделювання для ферми з трикутною решіткою та розмірами 18000×3600 мм, виготовленої з конструкційної сталі ВСт3пс. Розроблено геометричні та скінченно-елементні моделі ферми в середовищах виявлених розрахункових комплексів. Для моделювання застосовувалися елементи з прокатних кутників із поперечним перерізом 100×100×10 мм, а вузлові косинки були виконані зі сталевого листа товщиною 10 мм. Для моделі ферми в ANSYS Workbench 14.5 створено скінченно-елементну сітку з дискретизацією розмірів, зокрема для фасонок, що забезпечує точніше визначення параметрів напружено-деформівного стану НДС у критичних зонах ферми. В ЛІРА-САПР 2016 R5 модель ферми розбита на елементи 10x10 мм. Запропонована методика включає два основні етапи: на першому етапі створюється модель для визначення напружено-деформівного стану ферми і середовищі програмного комплексу ЛІРА-САПР 2016 R5, на другому в середовищі розрахункового комплексу ANSYS Workbench 14.5 аналізуються отримані результати для оптимізації конструктивних параметрів. Застосування цієї методики дозволяє значно скоротити час, необхідний для проектування та розрахунків, що сприяє підвищенню ефективності конструювання ферм з паралельними поясами. Практична цінність розробки полягає в можливості оптимізації форм та розмірів поперечних перерізів елементів, що позитивно впливає на економічну ефективність проектованих ферм, знижуючи витрати на матеріали та забезпечуючи довговічність конструкцій

Ключові слова: комп'ютерна симуляція, скінченні елементи, оптимізація, стрижневі конструкції, інженерний аналіз, варіаційні методи розрахунку