Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin

e-ISSN	: 2549-1999	No.	: 4	Month	: August
p-ISSN	: 1693-5462	Volume	: 21	Year	: 2023

Article Processing Dates: Received on 2023-05-15, Reviewed on 2023-05-19, Revised on 2023-07-30, Accepted on 2023-08-09 and Available online on 2023-08-25

Optimization process of the Truss Structure using Finite Element Analysis: Step by step from 2D to 3D space

Arhami*, Iskandar Hasanuddin, Masri
Department of Mechanical Engineering and Industry, Syiah Kuala University, Banda Aceh 23111, Indonesia
*Corresponding Author: arhami@usk.ac.id

Abstract

This paper discusses the process of optimizing the truss structure step by step from 2D to 3D space using finite element analysis. This step-by-step optimization process is carried out to simplify the analysis of truss structures from simple to more complex cases. Optimization aims to obtain the minimum cross-sectional area and weight for each truss member. The stages of the optimization process carried out in this study are starting from a 2-dimensional (2D) truss structure with several two and five members to a 3-dimensional (3D) one-level tower with a total of 18 members. The optimum criterion as the constraint used is the full stress design method and the value of the cross-sectional area and weight of the structure as a result of optimization, leading to convergence during the iteration process. The tool used to run the iteration process is performed using Fortran software. The results of this optimization process are the total cross-sectional area (A) and a minimum of weight (W), that is, for a two-member truss A = 1 in^2 and W = 4 lb, for a five-member truss A = 3.48 in^2 and W = 14 lb. Furthermore, for a one-level of tower-space truss with a total of 18 elements, A = 57.91 in^2 is obtained and the optimum weight of the truss structure is W = 134.02 lb. From these results, it can be seen that the optimization process that starts from simple to complex cases can be carried out easily and still takes into account the existing constraints.

Keywords: 2D-truss, 3D-space truss. Optimization, finite element analysis, minimum weight

1 Introduction

The analysis of complex truss structures using traditional methods will be complicated and time-consuming. So, recently various optimization methods have been developed using finite element analysis. Application of the optimization process can be used in decision-making, design, construction, and maintenance of various engineering systems. In this case, the optimization method is used as a tool to optimize the objectives in the process of making a decision.

The main contribution of the optimization methods are often applied to solve engineering problems which can be briefly described as follows: design of aircraft to obtain minimum structural weight; obtaining the optimal trajectory of the spacecraft; water resource system design to maximize benefits; minimizing the weight of the structure against the wind, earthquake, and random loading; various design of civil engineering structures such as frames, foundations, steel space truss towers [1], bridges [2], chimneys, and dams, to obtain minimum cost criteria; optimal plastic structure design; optimal design of joints, gears, cams, machine tools, and other mechanical components; factory or industrial layout; control system optimization [3], [4], [5].

The basic types of trusses shown in this article have simple designs that could be easily analyzed by 19th and early 20th century engineers. A truss is relatively economical to construct because it uses materials efficiently. A truss is a simple structure whose members are subject to single internal force and uniform deformation [6], axial tension and compression only but not bending moment, shear or torsion [7].

There are two types of trusses, namely plane truss and space truss [8]. When all members and applied forces are oriented in the same plane, the structure is the plane truss or 2D truss [9]. While in a space truss, the members and the forces are oriented in three dimensions.

The object of this research is the truss and analysis are carried out in stages and modeled in 2D to 3D planes. Here, the primary purpose was to minimize the cross-sectional area and weight of the structure with the optimization process but must consider certain conditions, namely being able to withstand loads such as the weight of the fluid tank, parabola, wind effect or drag, and the weight of the structure itself. So, to get the relatively minimum cross-sectional area and weight of the truss, optimization is needed without violating the specified constraints.

There are two conditions for the truss optimization process carried out in this study. First, starting from a simple 2D truss that has a total of two and five bars. The next stage is followed by a more complex truss in the form of 3D space with a total of eighteen bars. With so many members, it requires analysis with finite element analysis. Finite element analysis is a way of solving the continuum problem, where the domain is discretized into several small elements. These small elements are called finite elements which are connected at the nodes or joints, forming a series, which together and as a whole approaches the original continuum shape [10].

2 Research Methods

The optimization method in this study applies numerical analysis with a computer-assisted program using the Fortran language. The basis in optimization is using the finite element analysis. The first optimization is carried out on a 2D truss with fixed support which has a total of two and five members. The optimization of the next step was carried out on a 3D space truss in the form of a one-level tower with four fixed supports and eighteen members.

2.1 Two-bar truss 2D model

This structure is modeled by three nodes and two bar elements. Fixed support as boundary conditions is given at nodes 1 and 2 with a horizontal force F acting on node 3, as shown in Fig. 1.

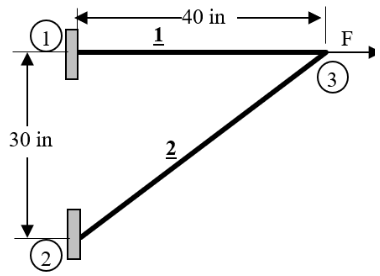

Fig. 1. Fixed support truss model with two members.

2.2 Five-bar truss 2D model

This 2D truss structure model has four nodes and five bars as shown in Fig. 2. This structure was fixed support at nodes 1 and 2, with a horizontal force *F* acting on node 4.

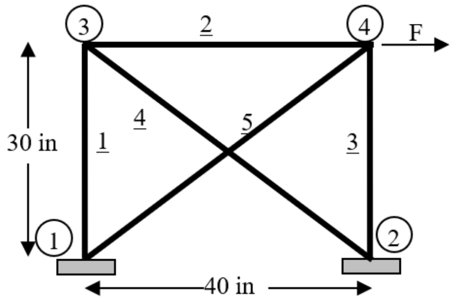

Fig. 2. Fixed support truss model with five members.

2.3 The 3D space truss model with eighteen members

The 3D space truss structural model has eight nodes and eighteen members. This structure is fixed-support at nodes 1, 2, 3, and 4 and external load F_1 and F_2 that work on nodes 5 and 6. The whole dimension of the truss is shown in Fig. 3.

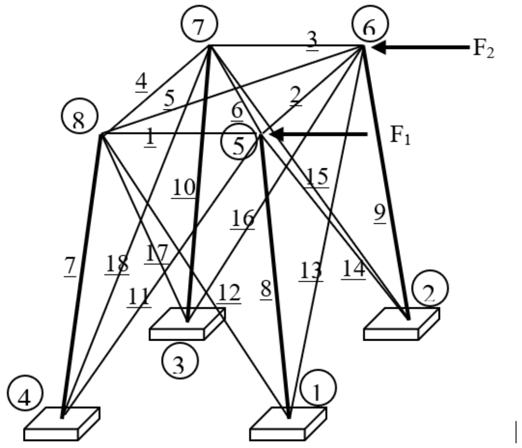

Fig. 3. The 3D space truss model, the form of a level tower with eighteen members.

2.4 Design variables of the truss member

The profile of each truss member as a design variable is L-shape section written as a, b, and t or can be expressed as x_1, x_2, and x_3. Problem design can be defined as a vector that will be optimized in the form of Fig. 4.

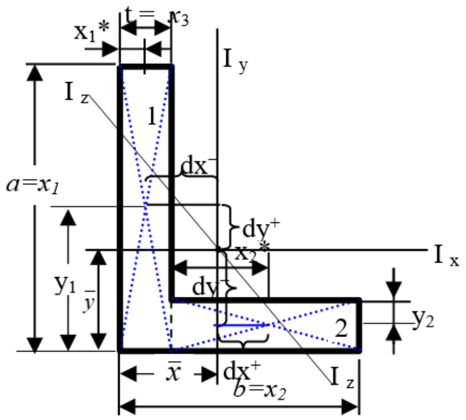

Fig. 4. Truss element profile as a design variable

2.5 Finite Element Analysis

According to [11], the equation for the relationship between forces, truss member stiffness, and nodal displacements is written as Eq.1.

$$\{f\} = [k]\,\{d\} \tag{1}$$

By applying the superposition method, the truss element stiffness equation is developed into a global stiffness equation that can be written as Eq.2

$$\{F\} = [K]\,\{D\} \tag{2}$$

Where: $\{f\}$ = element force matrix
$[k]$ = element stiffness matrix
$\{d\}$ = nodes displacement matrix
$\{F\}$ = structure force matrix
$[K]$ = structure stiffness matrix
$\{D\}$ = structure displacement matrix

The 2D truss element stiffness matrix $[k]$ is written as Eq.3:

$$\{k\} = \frac{AE}{L}\begin{bmatrix} C^2 & CS & -C^2 & -CS \\ CS & S^2 & -CS & -S^2 \\ -C^2 & -CS & C^2 & CS \\ -CS & -S^2 & CS & S^2 \end{bmatrix} \tag{3}$$

Therefore, the relationship between the forces, stiffness matrix, and the deformation of two-node elements concerning the x and y coordinates can be written as Eq.4:

$$\begin{Bmatrix} f_{1x} \\ f_{1y} \\ f_{2x} \\ f_{2y} \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} C^2 & CS & -C^2 & -CS \\ CS & S^2 & -CS & -S^2 \\ -C^2 & -CS & C^2 & CS \\ -CS & -S^2 & CS & S^2 \end{bmatrix}\begin{Bmatrix} d_{1x} \\ d_{1y} \\ d_{2x} \\ d_{2y} \end{Bmatrix} \tag{4}$$

Where A, E, and L are known respectively as the area, and modulus of elasticity. While C and S are abbreviations for cos θ and sin θ. Meanwhile, the stiffness matrix for 3D truss elements is expressed as Eq.5:

$$[k] = \frac{AE}{L}\begin{bmatrix} C_x^2 & C_xC_y & C_xC_z & -C_x^2 & -C_xC_y & -C_xC_z \\ & C_y^2 & C_yC_z & -C_xC_y & -C_y^2 & -C_yC_z \\ & & C_z^2 & -C_xC_z & -C_yC_z & -C_z^2 \\ & Symmetry & & C_x^2 & C_yC_y & C_xC_z \\ & & & & C_y^2 & C_yC_z \\ & & & & & C_z^2 \end{bmatrix} \tag{5}$$

Where C_x, C_y and C_z are written as abbreviations for cos θ in the x, y, and z coordinates, respectively.

2.6 Structural Optimization process

The objective function in optimizing the truss structure problem in this study is to minimize the weight of the structure can be formulated by Eq. 6.

Object to: $$f(x) = \sum_{i=1}^{n} \rho\, l_i x_i \tag{6}$$

Constrain to: $\sigma_i \le \sigma_{cr}$ (compression) (5)

$\sigma_i \le \sigma_y$ (tension)

Where ρ is the density of the material, li is the length of each element, x_i is the cross-section of each truss element, σ_{cr} is the critical compressive stress, and the yield strength (σ_y). The value of the objective function confirmed that optimized trusses in equation (6) are within the limits of the design constraints of equation (7).

2.7 Fully Stress Design Method

The full stress design method is used by multiplying the design variables by the stress ratio obtained against the constraint limits that have been determined on the material properties [12]. This concept first analyzes the structure by calculating the stress on each member of the bar under certain loading conditions. The stress for each bar (σ_i) obtained is then compared with the critical stress (σ_{cr}) of the material. The comparison is expected to be as close as possible, namely (Eq.8):

$$\sigma_i \le \sigma_{cr} \tag{8}$$

where: $$\sigma_{i\,cr} = \frac{\pi^2 E_i I_i}{L_i^2}\frac{1}{A_i}$$

$$A_i = \{b_i\, t_i + a_i\, t_i\}$$

and the minimum inertia is oriented in the z direction, then (Eq. 9)

$$I_y = \frac{x_1 x_2^3}{12} + A_1[dy^+]^2 + \frac{x_2}{12}(x_3^3) + A_2[dy^-]^2$$

$$I_{xy} = A_1\big(-(dy^-)\big)(dy^+) + A_2(dx^+)\big(-(dx^-)\big)$$
$$I_z = I_y + I_{xy} \qquad (9)$$

If the comparison conditions in equation (8) are unsatisfied, the design variable needs to be iterated so that an optimum value is obtained.

2.8 Convergent function

To find out the optimum value, the criteria towards convergent values are used. The optimum criteria are stated as follows (Eq.10):

$$\left|\frac{f(x)_{i+1} - f(x)}{f(x)_{i+1}}\right| \leq \varepsilon, \qquad (10)$$

where:

$f(x)_i$ = objective function at iteration i.
$f(x)_{i+1}$ = objective function at iteration i+1
ε = error limit (ε = 0.0001)
i = iteration number (i = 1, 2, 3,..., n)

If the conditions in this equation are reached or converge, the iteration process stops and the optimum objective value is obtained.

2.9 Finite Element Analysis computational steps

The finite element analysis used to support the optimization process is carried out through the following algorithms:

a) Entering (input) data.
b) Input data as shown in Table 1, includes structural geometry data, number and number of nodes, number of truss elements, loading conditions, degrees of freedom, material type, coordinates of each node, connection between structural members to nodes, boundary conditions, modulus of elasticity of elements, the yield stress of the material used, and the cross-sectional area.
c) Calculating the element stiffness matrix [*k*].
d) Develop the elemental stiffness matrices into a global stiffness matrix [*K*].
e) Modification of the global stiffness matrix due to supports and external forces.
f) Calculation of the displacement of each node.
g) Calculation of the deformation of each bar
h) Calculation of the force F and the stress σ in each member.

The stress results obtained on each bar (σ_i) are then compared with the critical stress (σ_{cr}) of the material.
The general flowchart for the proposed algorithms is given in Fig. 5.

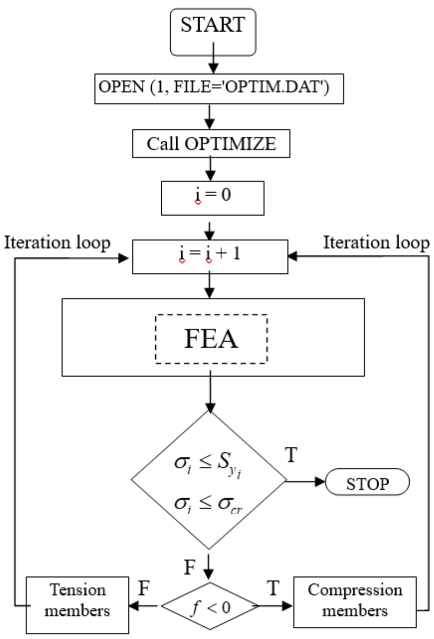

Fig. 5. The flowchart for the optimization algorithms

For example, the input model for design parameter data of the truss assuming material properties is shown in Table 1 and the input data file in Fortran is shown in Table 2.

Table 1. Input data for 2D and 3D space truss

Data design parameters	2D Truss	3D Truss
- Number of nodes	3 dan 4	8
- Number of elements	2 dan 5	18
- Modulus of elasticity	100 x10^6 psi	100 x10^6 psi
- External load	10000 lb	10000 lb
- Yield strength	10000 psi	10000 psi
- Initial value of cross-sectional area	5 and 10 in^2	5 in^2
- Boundary conditions	fixed node 1, 2	fixed node 1,2,3,4
- Load location	node 3 and 4	node 5 and 6
- Dimension of elements	coordinated	coordinate

Table 2. Example of data file input in Fortran for a two-bar truss

Geometries parameters	**Modulus of elasticity**
NP, NE, NLD, NDF, NMAT	NE E
3, 2, 1, 2, 2	1, 10000000
	2, 10000000
Coordinate of bar	**Yield strength**
NP X Y	NE SY
1, 0, 30	1, 10000
2, 0, 0	2, 10000
3, 40, 30	
Element connections	**Setpoint of initial area**
NE NOP1 NOP2 IMAT	NE A
1, 1, 3, 1	1, 5
2, 2, 3, 2	2, 10
Number of boundary cond 2	**Load**
NP NFIX U V	NP F
1 11 0 0	3, 10000
2 11 0 0	

where (NP) is the number of nodes, (NE) is the number of elements, (NLD) is the number of node loads, (NDF, NB) is the number of degrees of freedom, (NFIX, U, V) is the type of support, (NMAT) is the number of material, (IMAT) is the type of material, (E) is modulus of elasticity, (NOP1, NOP2) is connection of node between one element and another, (A) is setpoint as initial cross-sectional area, and (Sy) is yield strength of materials.

3 Results and Discussion.

The results of the optimization methodologies detailed gradually from simple 2D truss problems to more complex cases for 3D truss in the previous section are presented in the following section.

3.1 The 2D truss optimization results with two-member elements.

Optimization results for 2D truss with two-bar elements based on finite element analysis obtained optimum cross-sectional area and weight values as shown in Fig. 6 and 7. This value is obtained by using the convergent function of equation 10.

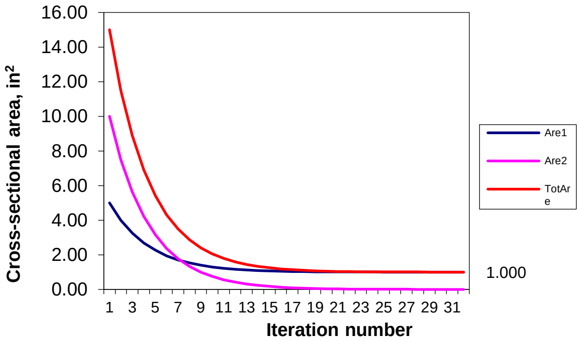

Fig. 6. Iteration wise reduction of the cross-sectional area for 2D trusses with two members

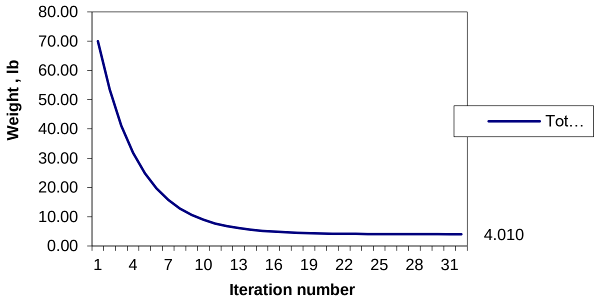

Fig. 7. Iteration wise reduction of weight for 2D trusses with two members

Fig. 6 and 7 show the cross-sectional and weight reduction during iteration using the optimization technique. Given the external load, F = 10000 lb and the yield strength (σ_y) = 10000 psi, the modulus of elasticity, E = 10 x 10^6 psi, the optimum total of the cross-sectional area in this case, is 1.00 in^2, and the total weight of the structure is 4.00 lb.

3.2 The 2D truss optimization results with five-member elements.

The optimum cross-sectional area and weight obtained using the optimization for a 2D truss with five elements are shown in Fig 8 and 9.

At this step, the input data is known by providing an external load, F = 10000 lb and yield strength (σ_y) = 10000 psi, modulus of elasticity, E = 10 x 10^6 psi.

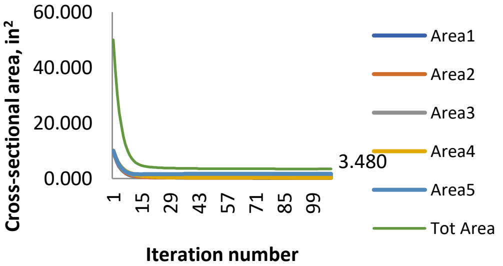

Fig. 8. Iteration wise reduction of the cross-sectional area for 2D trusses with five members

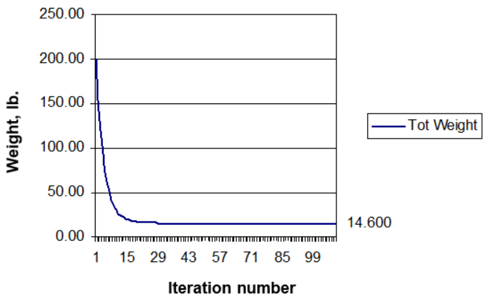

Fig. 9. Iteration wise reduction of weight for 2D trusses with five members

Fig 8 and 9 show the cross-sectional and weight reduction during iteration using the optimization technique. The optimum total of the cross-sectional area in this case, is 3.48 in^2, and the total weight of the five-member structure is 14.60 lb.

3.3 The 3D space truss optimization results with eighteen-member elements.

The optimum results of the cross-sectional area and weight obtained from the optimization for a 3D space truss with eighteen bar elements are shown in Fig. 10 and 11. In Fig. 10, only five bars are shown.

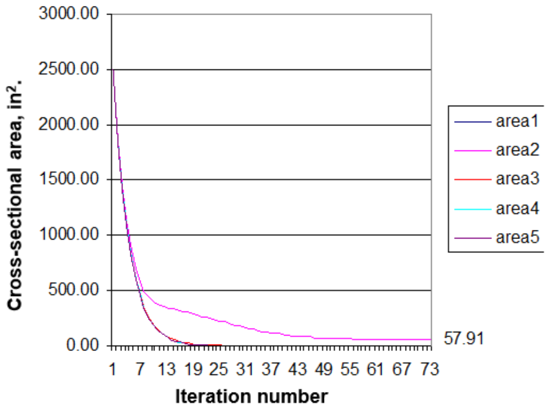

Fig. 10. Iteration wise reduction of the cross-sectional area for 3D trusses with eighteen members

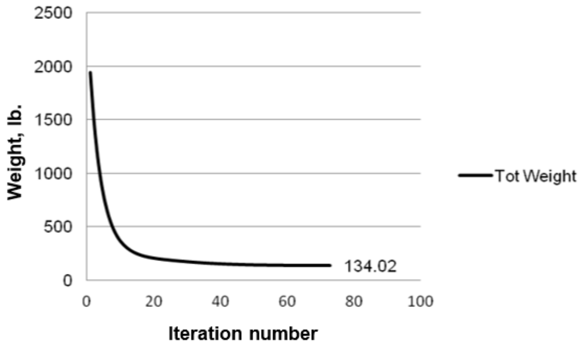

Fig. 11. Iteration wise reduction of weight for 3D space trusses with eighteen members

The results of this step, the input is known by providing two external load locations namely, at nodal 5, F_1 = 10000 lb and at nodal 6, F_2 = 10000 lb with yield strength (σ_y) = 10000 psi, and modulus of elasticity, E = 10 x 10^6 psi. The optimum total cross-sectional area in the 3D case with eighteen members is 57.91 in^2, and the total weight of the structure is 134.02 lb. The output units depend on the input units. From the results (Fig. 10) it is shown that many cross-sectional areas with very small values are even zeroed out because they are deemed unnecessary. However, in practice in the field, cross-sectional areas with a zero value should not exist, but these bars must be available to strengthen the structure with consideration of the varying or random force directions.

3.4 Validation

Evaluation against valid results in this study can be done by simulation the various input initial values with higher and lower numbers. In this regard, for a truss with two members (Fig. 1), the initial value of cross-sectional area is given 5 in^2 and 10 in^2 as the higher initial values. While the lower initial value given is 0.4 in^2 and 0.8 in^2 respectively. As can be seen, after programming was running and towards convergence on the 19th iteration, the results obtained the optimum values.

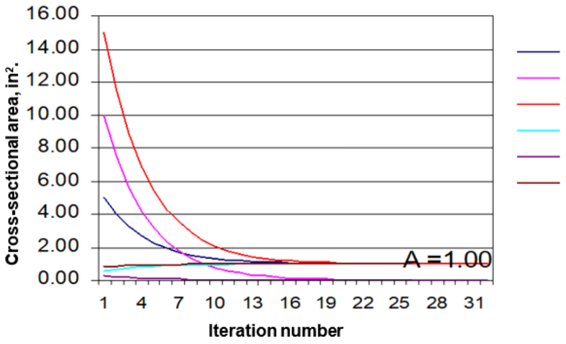

Fig. 12. Validation of optimum cross-sectional area during iteration

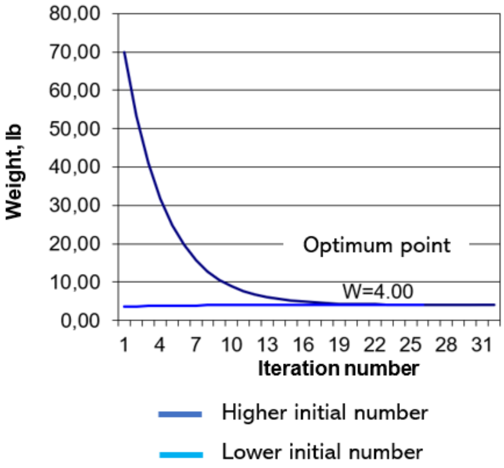

Fig. 13. Validation of optimum weight during iteration

Fig. 12 and Fig. 13 show the optimum total cross-sectional area in this case is 1.00 in^2, and the weight of the structure is 4.00 lb. These results are consistent with results in section 3.1, focus on Fig. 6 and Fig. 7.

4 Conclusions.

The optimization process to obtain the minimum weight of the truss structure can be carried out in stages based on finite element analysis. Optimization techniques have been successfully carried out with the full stress design constraints used by multiplying the design variables with the stress ratio obtained.

The optimum total yield area in the 2D case and the two bars is 1.00 in^2, and the weight of the structure is 4.00 lb. The optimum total area in the 2D case with five members is 3.48 in^2, and the weight of the structure is 14.6 lb. While the optimum total area in the 3D case with eighteen bars is 57.91 in^2, and the weight of the structure is 134.02 lb. All output units resulting from the optimization process depend on input units.

In this study, it was shown that many cross-sectional areas with very small values were even zeroed out because they were deemed unnecessary. But in practice in the field, there should not be a possible cross-sectional area with a zero value, but there must be a bar to strengthen the structure and consideration of varying or random force directions.

References.

[1] M. Artar and A. T. Daloğlu, "Optimum design of steel space truss towers under seismic effect using Jaya algorithm," *Structural Engineering and Mechanics*, vol. 71, no. 1, 2019, doi: 10.12989/sem.2019.71.1.001.

[2] Ankit Sharma and Sumit Pahwa, "Truss bridge structure frame section analysis by using Finite element analysis," *International Research Journal of Engineering and Technology*, 2018.

[3] S. S. Rao, *Engineering optimization: Theory and practice*. 2019. doi: 10.1002/9781119454816.

[4] T. A. J. Nicholson, *Optimization in Industry*. 2017. doi: 10.4324/9781315125831.

[5] R. K. Arora, *Optimization: Algorithms and applications*. 2015. doi: 10.1201/b18469.

[6] D. Wang, M. Shang, and P. Sun, "Deformation Performance Analysis of a Truss Structure Based on the Deformation Decomposition Method," *Buildings*, vol. 12, no. 3, 2022, doi: 10.3390/buildings12030258.

[7] P. Saket and K. Pathak, "Bridge truss structure analysis with various Sections by using Finite element analysis," *International Journal of Research and Analytical Reviews (IJRAR) www.ijrar.org*, vol. 82, no. 2, 2019.

[8] Kunal M. Porwal, Jasmin Gadhiya, and Hardik Patel, "Comparative analysis of 2D roof truss configuration," *Journal of Emerging Technologies and Innovative Research (JETIR)*, vol. 5, no. 4, pp. 494–497, 2018.

[9] C. W. Yee Zaw, K. Thu Zar, and K. Thant, "Roof Truss Design for Industrial Buildings," *International Journal of Advances in Scientific Research and Engineering*, vol. 5, no. 7, 2019, doi: 10.31695/ijasre.2019.33425.

[10] M. Kuryla, "Introduction to Finite Element Analysis (FEA) or finite element method (FEM)," *Freak Weather*, 2018.

[11] Saeed Moaveni, *Finite Element Analysis: Theory and Application with ANSYS*, Fourth Edition. Pearson Education, 2015.

[12] M. L. Malito, "Foundations of structural optimization: A unified Approach," *Journal of Mechanical Working Technology*, vol. 10, no. 3, 1984, doi: 10.1016/0378-3804(84)90054-8.