SCIENNOVATION
A Journal of Structural Science and Innovation

RESEARCH ARTICLE

Date Received: 07.12.2020
Date Accepted: 25.01.2021

AN INVESTIGATION ON PROFILE TENSION MEASUREMENT AND DYNAMIC LOAD ANALYSIS IN STEEL ROOFS

Selim TAŞKAYA[1*]
[1]Artvin Çoruh Üniversitesi, Meslek Yüksekokulu, Mimarlık ve Şehir Planlama Bölümü, Artvin, Türkiye
[*]Sorumlu yazar: selim_taskaya@artvin.edu.tr

Abstract
Whether new buildings are under construction, or the old building, bathhouse, social complex, etc. Static calculations in building systems and dynamic load analysis are carried out within the margin of error while creating roof systems, whether for the purpose of monitoring the structural behavior of buildings. Roof systems are created using one of many models. One of them is the finite element method. With this method, the distance between nodes from many nodes is measured and modeled. In our study, while creating a steel roof system in a structure of 10 floors 30.50 meters height, 23 * 22 meters width and length, 506 m2, as a result of physical calculations, load analysis, wind load, load combinations, steel frame calculations, equipment, torsion moments according to which parameters analysis was made as needed.

Keywords: Steel Roof, Finite Element Method, Profile and Dynamic Load Analysis

ÇELİK ÇATILARDA PROFİL GERİLMESİ ÖLÇÜMÜ ve DİNAMİK YÜK ANALİZİ ÜZERİNE BİR İNCELEME

Özet
Yeni yapıların yapım aşamasında olsun, isterse de eski bina, hamam, külliye vb. yapıların yapısal davranışlarının izlenmesi amacıyla olsun çatı sistemleri oluşturulurken yapı sistemlerindeki hesaplar statik, dinamik yük analizleri hata payı sınırları içerisinde hareket edilir. Birçok modelden biri kullanılarak çatı sistemleri oluşturulur. Bunlardan birisi de sonlu elemanlar yöntemidir. Bu yöntem ile birçok düğüm noktasından düğümler arası mesafe ölçülmesi yapılarak modellenir. Çalışmamızda 10 katlı 30.50 meter yükseklik, 23*22 metre en ve boy, 506 m^2 lik bir bir yapıda çelik çatı sistemi oluştururken, fiziksel hesaplamalar sonucunda yük analizi, rüzgar yükü, yük kombinasyonları, çelik çerçeve hesabı, teçhizat, burulma momentlerinin hangi parametreler doğrultusunda olması gerektiğinin analizi yapıldı.

Anahtar Kelimeler: Çelik Çatı, Sonlu Elemanlar Yöntemi, Profil ve Dinamik Yük Analizi

1. Introduction

The purpose of engineering structures is to determine the shape and position changes that occur. Temporary or permanent effects occur in engineering structures such as dams, bridges, tunnels, viaducts, towers and their surroundings. Generally, these effects consist of the physical properties of the ground, the existing weight of the building, mobile external loads and similar effects (Yalçınkaya M, Satır B, 2005). Steel is a homogeneous and isotropic material. The quality of the steel produced in accordance with international standards is constantly checked during manufacturing. Thus, the mechanical properties of the material cannot be intervened in the production and assembly stages, which provides the best possible approach to theoretical calculation values. application of steel as a building material in Turkey generally industrial buildings, bridges, warehouses or single - stands out as the roofing system of multi-storey buildings. Steel can be considered as the most suitable building material to construct earthquake resistant structures considering its high strength, lightness and ductility as well as its cost (Karagöz Ö; Özbaşaran H; Doğan M; Gönen H; Ünlüoğlu E, 2015). Physical properties are also taken into account when creating roof systems. It is very important in terms of the subject to collect all the data belonging to the historical buildings, which are the cultural values of societies, before any intervention is made. Using traditional engineering calculation methods to understand the behavior of historical buildings under dynamic effects in the light of the determined information makes the work even more complicated. For this reason, in order to reduce the complexity of the work done and to reduce the processing time, it is a method that has been frequently applied recently to make the structural analysis of historical buildings using the finite element method. Many researchers in the literature have applied the finite element method to determine the behavior of historical buildings under earthquake loads (Demircan R; Kardoğan P; Pınarlık M; Aytekin O, 2017).

Finite Element Method was first developed in 1956 for stress analysis of airframe, and in the following decade it was used in the solution of applied sciences and engineering problems. In the following years, these methods and solution techniques were developed rapidly and became one of the best methods used to solve many engineering problems today (URL-1, 2020).

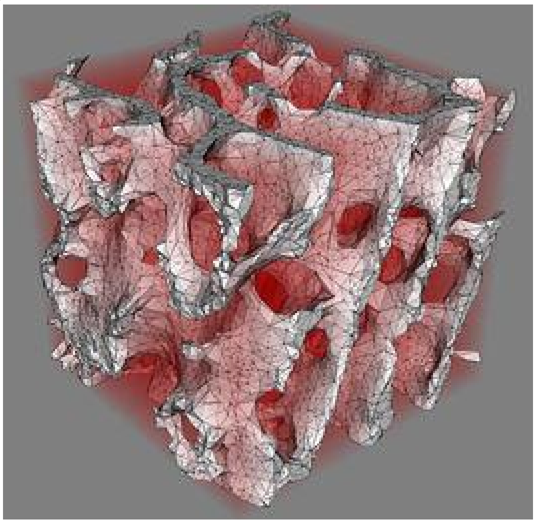

Figure 1. Finite element method sample model (URL-1, 2020).

The basic logic in the finite element method is to simplify and solve a complex problem. In this method, the solution region is divided into multiple, simple, small, interconnected, sub-regions called finite elements. In short, the solution of the problem that is divided into parts connected to each other with many nodes can be done easily. For example, the application of the finite element method in a structural analysis is as follows:

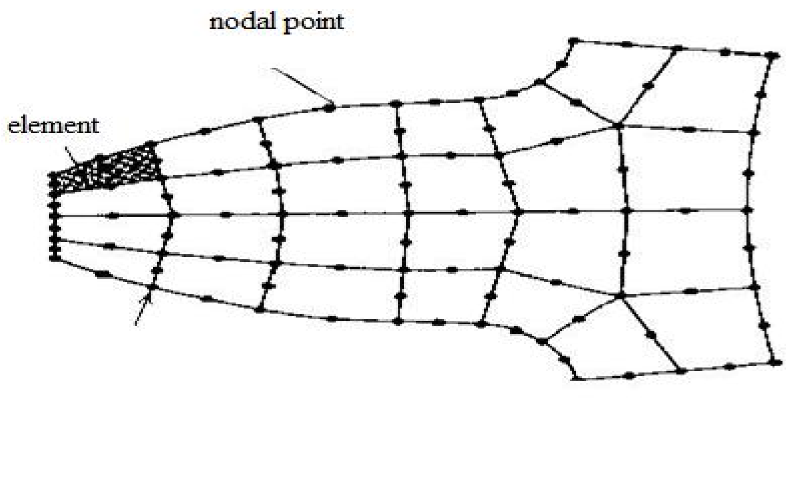

Figure 2. Finite element notation point representation (Url-1, 2020).

The structure is divided into parts with elements containing nodes. The behavior of physical quantities is defined for each element. An approximate system of equations is formed for the whole structure by connecting the elements at the nodes. System equations are solved for unknown values at nodes. (For example displacement) The desired values of the selected elements are calculated (Url-1, 2020).

Modeling and calculation parameters are determined as follows. The building is suitable to be defined with a "general shell element" (SHELL) such as a wall or a roof. For this reason, a centimeter or meter thick wall or roof is modeled with SHELL elements. Columns, main beams and other beams are modeled with FRAME elements (ER AKAN Aslı, 2010).

The mathematical model node prepared for calculations is created using SHELL element and bar element. The thickness of the wall or roof surrounding the building is 0.30 in places. or exceeds 1 meter, the "Thick Shell" option is preferred when modeling the wall or roof in order to be able to calculate the stresses on the inner and outer surfaces in more detail and to take into account the shear stresses in the section plane. The structural function of the spolia column heads located at the top of the columns is defined by releasing the end moments of the main beam elements (moment release). Since it is not possible to take and test material samples, the material properties of the building elements are selected by taking into account the values proposed for masonry or concrete structures or wooden structures in the current earthquake specification, using the correlations produced as a result of previous studies for similar structures and recommended in the international literature. Assuming that the building elements show a single material feature together with the mortar, elasticity module and unit weight assumptions are made. On the calculation model prepared, two different loading cases are applied, considering the forces caused by the constant loads and the ground motion defined by the earthquake spectrum. Spectrum is applied separately in two principal directions, EQx and EQy loading. When calculating the constant loads of the roof section, the weight of the cantilever roof was taken into account in addition to the weight of the main load-bearing wooden elements. In order to evaluate the results easily, two different load combinations are defined as G + EQx (Constant loads + earthquake loading in the x-axis direction) and G + EQy (Constant loads + earthquake loading in the y-axis direction) (ER AKAN Aslı, 2010).

The finite element method (FEM) is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. A finite element model of a problem gives a piecewise approximation to the governing equations. The basic premise of the FEM is that a solution region can be analytically modeled or approximated by replacing it with an assemblage of discrete elements (discretization). Since these elements can be put together in a variety of ways, they can be used to represent exceedingly complex shapes (Yagota V; Sethi A; Kumar K, 2013).

Several approximate numerical analysis methods have evolved over the years. As an example of how a finite difference model and a finite element model might be used to represent a complex geometrical shape, consider the turbine blade cross section in Figure 3 and plate geometry in Figure 4.A uniform finite difference mesh would reasonably cover the blade (the solution region), but the boundaries must be approximated by a series of horizontal and vertical lines (or “stair steps”) . On the other hand, the finite element model (using the simplest two-dimensional element-the triangle) gives a better approximation of the region. Also, a better approximation to the boundary shape results because the curved boundary is represented by straight lines of any inclination. This is not intended to suggest that finite element models are decidedly better than finite difference models for all problems. The only purpose of these examples is to demonstrate that the finite element method is particularly well suited for problems with complex geometries and numerical solutions to even very complicated stress problems can now be obtained routinely using finite element analysis (FEA) (Yagota V; Sethi A ; Kumar K, 2013).

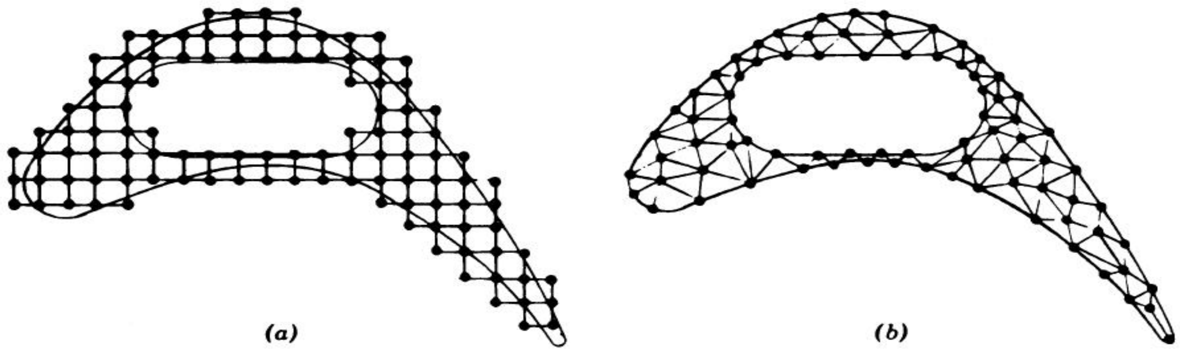

Figure 3. (a) Finite difference and (b) finite element discretizations of a turbine blade profile

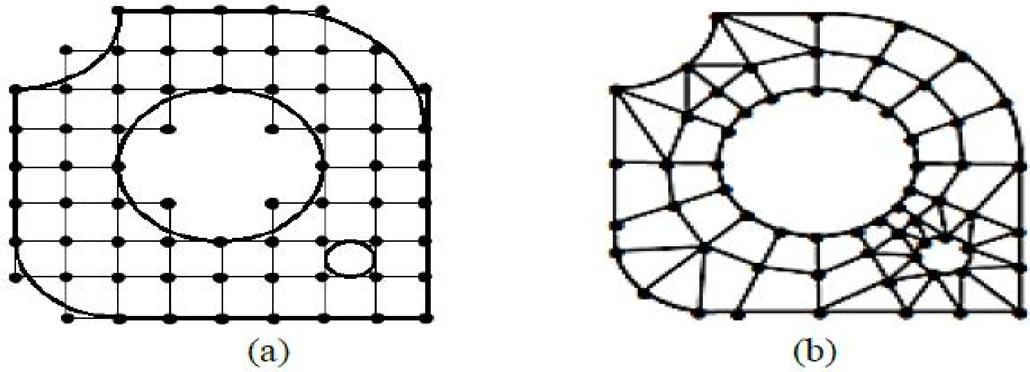

Figure 4. (a) Plate geometry finite difference model and (b) Finite element model.

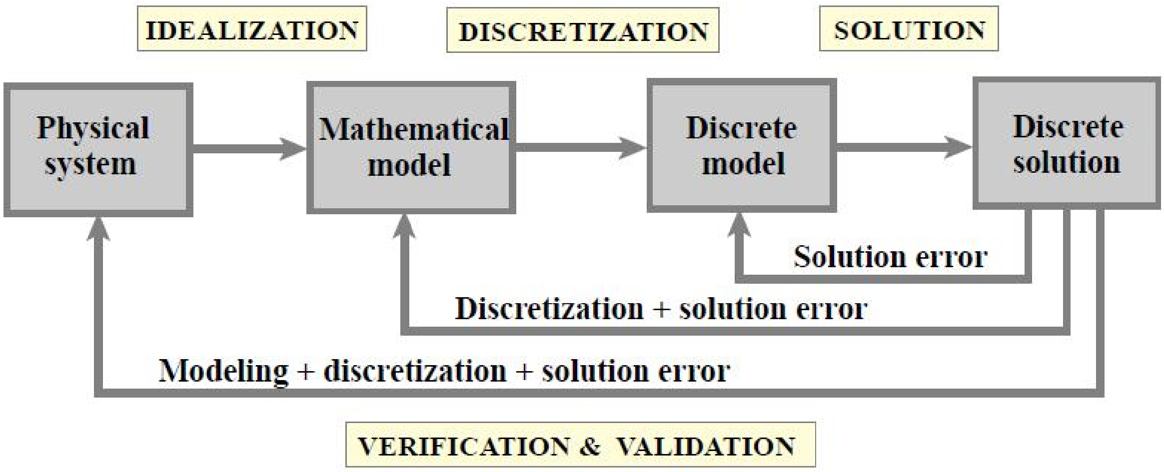

Figure 5. Flowchart of model-based simulation (MBS) by computer.

2. FINITE ELEMENT METHOD

Although the label finite element method first appeared in 1960, when it was used by Clough (Clough RW, 1960). In a paper on plane elasticity problems, the ideas of finite element analysis date back much further. The first efforts to use piecewise continuous functions defined over triangular domains appear in the applied mathematics literature with the work of Courant in 1943 (Courant R, 1943) . Courant developed the idea of the minimization of a functional using linear approximation over sub-regions, with the values being specified at discrete points which in essence become the node points of a mesh of elements (Yagota V; Sethi A; Kumar K, 2013) .

3. APPLICATION IN ROOF DESIGN

While creating a steel roof system in a 10-storey building of 30.50 meters height, 23 * 22 meters width and length, 506 m2, the physical project calculation is made as follows.

Roof Tilt Angle; $\tan\alpha = 3.523 / 7.86$ $\tan\alpha = 0.448$

$\alpha = 24.1$

Frame Span = L = 22.06 m Purlin Spacing = l1 = 4.255 m

Frame Spacing = L '= 7.13 m l'1 = 4.663 m

Number of Frames = n = 4 Purlins Span = l2 = 7.13 m

Load Analysis

6 + 16 + 4 + 4 Insulating Glass Material g = 90.00 kg / m2 (Roof Plane) g1 = 98.63 kg / m2 (Horizontal Plane)

Purlin Self-weight 6.00 kg / m2 (Horizontal Plane) g2 = 104.63 kg / m2 (Horizontal Plane)

Snow (Region III) Pk1 = 148.5 kg / m2 (Altitude = 1380 m) The value is increased by 10% since the altitude is 1380.

Icing 21 kg / m2 Icing (Ice thickness is accepted as 3 cm thick.) Pk = 169.5 kg / m2

Wind Load

Vwind=	36	m/s
qwind=	83	kg/m^2

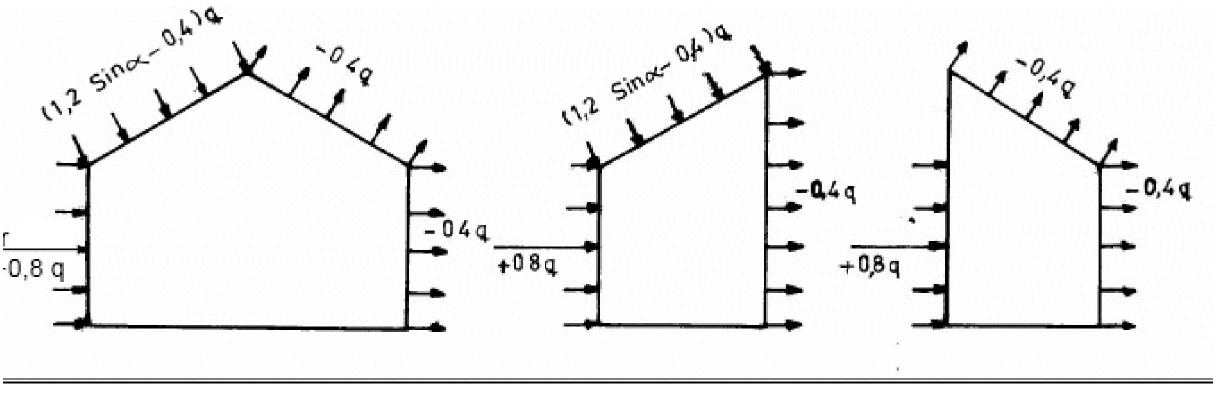

Figure 6. Wind loads (TSE, 498).

Building height wind load gr = 83 kg / m2

Pr1 wind load (1.2 * sina-0.4) * gr == 7.50 kg / m2 (Roof Plane)

Pr2 wind load (-0.4 * gr) -33.03 kg / m2 (Roof Plane)

Pr3 wind load (0.8 * gr) = 66.06 kg / m2 (Vertical Plane)

Vertical Load Affecting the Purlin due to Self Weight and Snow:

q = 1166.41 kg / m (H Loading)

Perpendicular Component of Vertical Load to Roof:

q1 = 1064.4 kg / m (H Loading)

Horizontal Component of Vertical Load to Roof:

q2 = 477.1 kg / m (H Loading)
Purlin Calculation on Inclined Surfaces
Inner Span: Mx = q1 * l2 / 8 Mx = 6763.7 kg.m My = q2 * l2 / 8 My = 757.9 kg.m
Edge Span: Mx = q1 * l2 / 8 Mx = 3381.9 kg.m
My = q2 * l2 / 8 My = 379.0 kg.m Selected Section:
Ix = 8091.0 cm4 Iy = 2843.0 cm4
Wx = 735.50 cm3 Wy = 258.50 cm3
F = 91.04 cm2 G = 71.5 kg / m
Stress control:
1212.8 kg / cm2 <sem = 1440 kg / cm2 Deflection control: 2
0.843 cm fy = 2.48 * qy * e4 / Iy = 0.067 cm
ftotal = 0.910 cm <l / 300 = 2.377 cm

Calculation of Tension Ropes:
tan (b) = l2 / (2 * l1) * cos (a) = 0.765 b = 37.40 cos (b) = 0.794
6 Zmax = 2884 kg
18 Fgç = 2.19 cm2
Stress control:
1318 kg / cm2 <sem = 1400 kg / cm2
Number of tensioner spacing = Total number of purlins (n) = fx = 2.48 * qx * l14 / Ix =
s = Zmax / Fgç =
Chosen Tension (f) =
s = Mx / Wx + My / Wy =
HE 220 B

Purlin Calculation on Vertical Surfaces
Load Calculation in x Direction on Vertical Surfaces
Ld1 = 4.860 m Purlin clearance on vertical surfaces
Ld2 = 2.500 m Purlin spacing on vertical surfaces
gd2 = g * Ld2 = 225.0 kg / m Vertical distributed loads due to glass coating
Load Calculation in y Direction on Vertical Surfaces
Ld1 = 15.000 m Purlin clearance on vertical surfaces
Ld2 = 2.500 m Purlin spacing on vertical surfaces
gd2 = g * Ld2 = 225.0 kg / m Vertical distributed loads due to glass coating
Steel Frame Account
Forces At Node Points Due To Self Weight
Port = g2 * l'1 * l2 = Port = 3478 kg (Dead loads)
Pken = g2 * l'1 * l2 / 2 = Pken = 1739 kg (Dead loads)
Pken = g2 * l'1 * l2 / 4 = Pken = 903.5 kg (Dead loads)
Forces Occurring at Node Points Due to Snow Port = (Pk) * l'1 * l2 = Port = 4937 kg (Snow Load)

Pken = (Pk) * l'1 * l2 / 2 = Pken = 2469 kg (Snow Load)
Pken = (Pk) * l'1 * l2 / 4 = Pken = 1234 kg (Snow Load)
Equipment Loads (gt)
200 kg load is specified for the weight of the VRP system designed for heating and cooling.
Taking this load into consideration, the load distribution was made to the frame system formed at z = 7.72 m elevation.
Forces Occurring at Node Points Due to Wind
Wind Forces on Inclined Surfaces Wind Blows From Left
Pr1ort = 249 Pr1ken = 125 kg (Z-Load)
Pr1ort, X = 102 kg Pr1, x = 51 kg Pr1ort, Z = 227 kg Pr1ken, z = 114 kg Pr2ort = -1098
Pr2ken = -549 kg (Z-Charge) Wind Blows From Right
Pr2ort, X = -449 kg Pr2ken, x = -225 kg
Pr2ort, Z = -1002 kg Pr2ken, z = -501 kg
Pr2ort, X = 102 kg Pr2ken, x = 51 kg
Pr2ort, Z = 227 kg Pr2ken, z = 114 kg

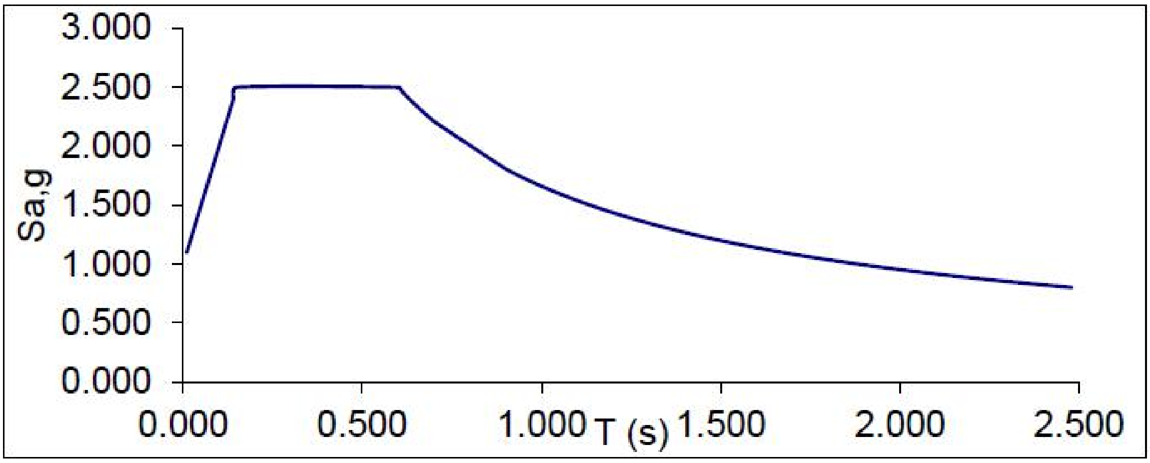

Figure 7. Roof spectrum acceleration chart according to TBDY-2018

4. ANALYSIS RESULTS

4.1 ANALYZES OBTAINED UNDER STATIC AND DYNAMIC LOADS

The steel roof of a 10-storey building with a height of 506 m2 was analyzed using loading combinations. The finite element model of the carrier system is shown in Figure 7, and the loads for the steel construction are given in Figure 8. Modeling was done by combining the nodes using the SAP2000 program.

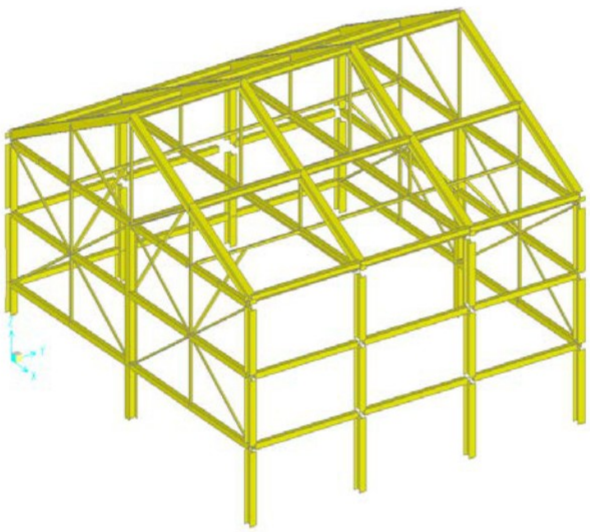

Figure 7. Finite element model of the steel carrier system

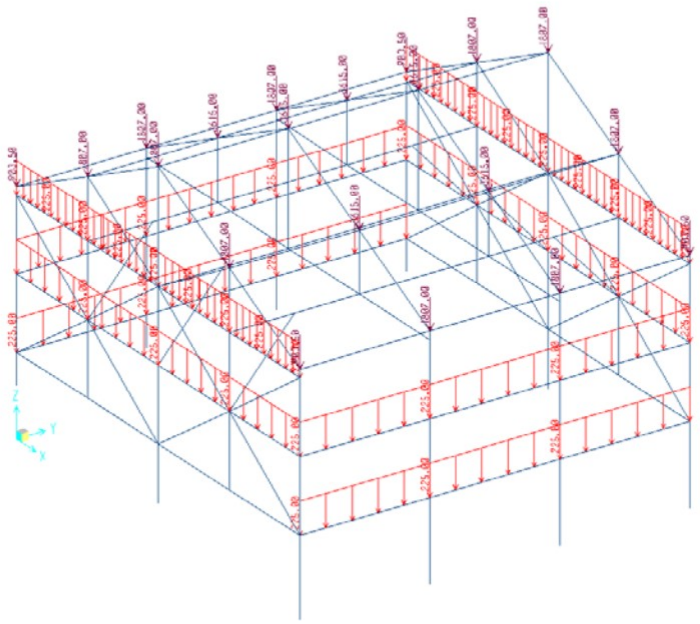

Figure 8. Dead load condition

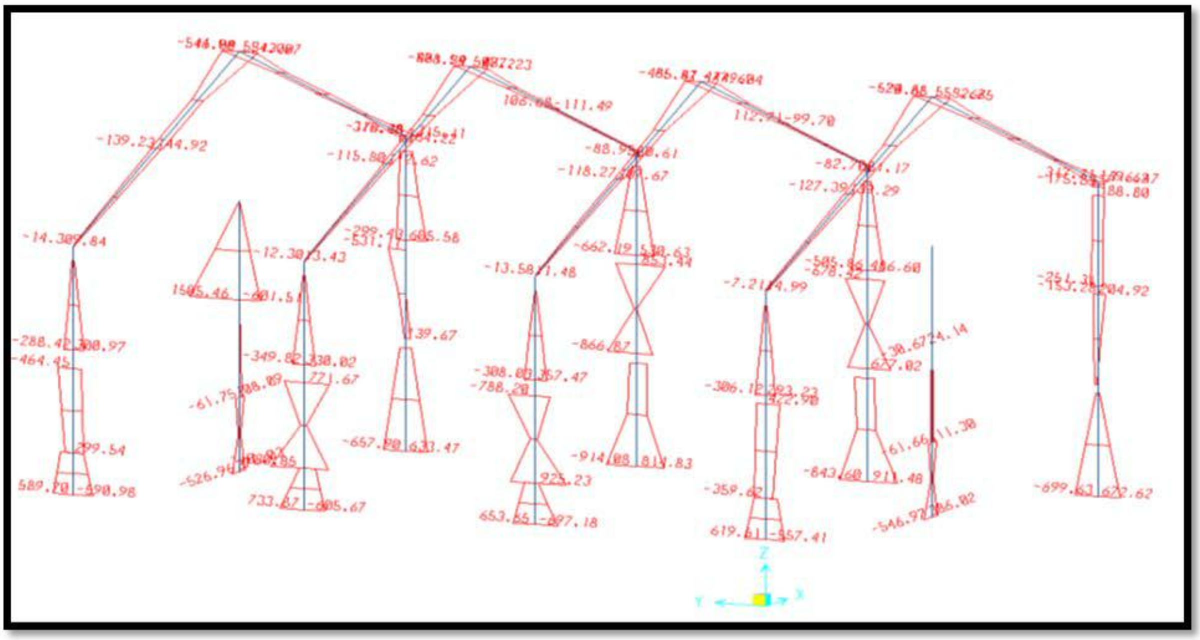

Figure 9. Tension, moment and buckling representation of the profile.

In Figure 9, as a result of the least squares method, the profile, which allows the roof to stand, was mounted to approximately 35 main points calculated in the project in terms of the distance between the start and end points and grad angle evaluation, by reading the angle and distance.

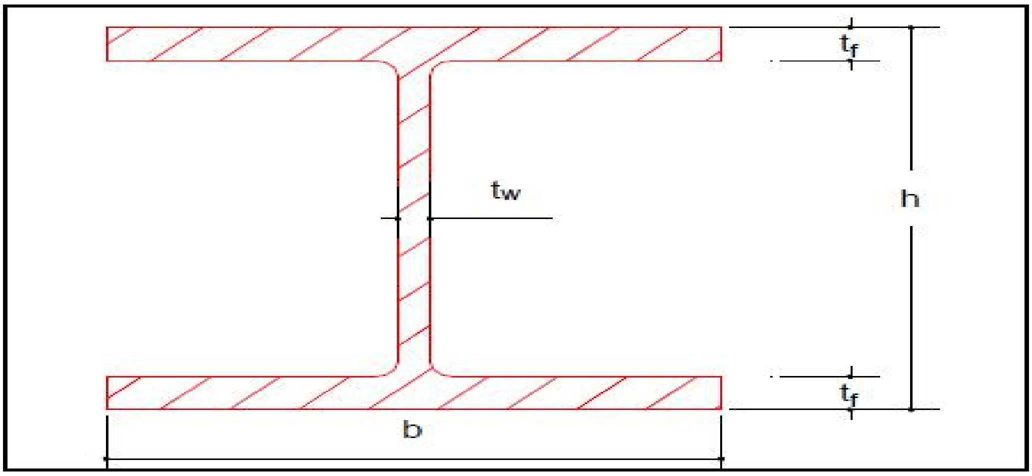

Figure 10. Profile connection points

h = 33 cm b = 30 cm tf = 1.65 cm tw = 0.95 cm Abaş = 99.00 cm2 Agöv = 28.22 cm2

Figure 10 shows the values of 35 main connection points.

KOMB-1	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k$
KOMB-2	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + 0.50\ q_{rüz,x,sag}$
KOMB-3	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + 0.50\ q_{rüz,x,sol}$
KOMB-4	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + 0.50\ q_{rüz,y,sag}$
KOMB-5	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + 0.50\ q_{rüz,y,sol}$
KOMB-6	$1.00\ g_2 + 1.00\ g_t + 0.50\ P_k + 1.00\ q_{rüz,x,sag}$
KOMB-7	$1.00\ g_2 + 1.00\ g_t + 0.50\ P_k + 1.00\ q_{rüz,x,sol}$
KOMB-8	$1.00\ g_2 + 1.00\ g_t + 0.50\ P_k + 1.00\ q_{rüz,y,sag}$
KOMB-9	$1.00\ g_2 + 1.00\ g_t + 0.50\ P_k + 1.00\ q_{rüz,y,sol}$
KOMB-10	$1.00\ g_2 + 1.00\ g_t + F_{dep,X}\ /1.4 + F_{dep,Y}\ /14$
KOMB-11	$1.00\ g_2 + 1.00\ g_t + F_{dep,X}\ /14 + F_{dep,Y}\ /1.4$
KOMB-12	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + F_{dep,X}\ /1.4 + F_{dep,Y}\ /14$
KOMB-13	$1.00\ g_2 + 1.00\ g_t + 1.00\ P_k + F_{dep,X}\ /14 + F_{dep,Y}\ /1.4$

Figure 11. Physical relations of load combinations on connection points

In Figure 11, during the modeling, all the amounts of wind, equipment, buckling, moment and acceleration, and the amount of load to be applied to 35 connection points were calculated by looking at these relations.

Frame	Output Case	Comb.	Text	P	V2	V3	T	M2	M3	λ	ω	σ	σ_{em}
El No	Text			kgf	kgf	kgf	kgf.cm	kgf.cm	kgf.cm			kg/cm^2	kg/cm^2
1	0.15	KOMB-14	Max	320	-426	29	15891	0	0	56	1.00	4.41	<Sem
1	0.61	KOMB-14	Max	320	-296	29	15891	1021	23125	56	1.00	56.41	<Sem
1	1.06	KOMB-14	Max	320	-167	29	15891	2041	40331	56	1.00	94.48	<Sem
1	1.52	KOMB-14	Max	320	-37	29	15891	3062	51618	56	1.00	118.63	<Sem
1	1.97	KOMB-14	Max	320	93	29	15891	4083	56986	56	1.00	128.86	<Sem
1	2.43	KOMB-14	Max	320	223	29	15891	5104	56435	56	1.00	125.16	<Sem
1	2.89	KOMB-14	Max	320	353	29	15891	6124	49965	56	1.00	107.54	<Sem
1	3.34	KOMB-14	Max	320	482	29	15891	7145	37577	56	1.00	76.00	<Sem
1	3.80	KOMB-14	Max	320	612	29	15891	8166	19269	56	1.00	30.53	<Sem
1	4.25	KOMB-14	Max	320	742	29	15891	9186	-4958	56	1.00	28.86	<Sem
1	4.71	KOMB-14	Max	320	872	29	15891	10207	-35103	56	1.00	102.17	<Sem
1	0.15	KOMB-14	Min	-218	-572	-22	-11562	0	0	56	1.31	3.94	<Sem
1	0.61	KOMB-14	Min	-218	-442	-22	-11562	-1308	16475	56	1.31	37.89	<Sem
1	1.06	KOMB-14	Min	-218	-312	-22	-11562	-2615	27032	56	1.31	65.80	<Sem
1	1.52	KOMB-14	Min	-218	-183	-22	-11562	-3923	31670	56	1.31	79.79	<Sem
1	1.97	KOMB-14	Min	-218	-53	-22	-11562	-5231	30388	56	1.31	79.85	<Sem
1	2.43	KOMB-14	Min	-218	77	-22	-11562	-6539	23188	56	1.31	65.99	<Sem
1	2.89	KOMB-14	Min	-218	207	-22	-11562	-7846	10069	56	1.31	38.20	<Sem
1	3.34	KOMB-14	Min	-218	337	-22	-11562	-9154	-8970	56	1.31	3.51	<Sem
1	3.80	KOMB-14	Min	-218	466	-22	-11562	-10462	-33927	56	1.31	59.14	<Sem
1	4.25	KOMB-14	Min	-218	596	-22	-11562	-11770	-64803	56	1.31	128.69	<Sem

Table 1. First port load combinations spreadsheet

In Table 1, the weight and load combinations applied to the first connection point from all intersections were found as a result of the physical calculations. The maximum imposed load range was found in the modeling.

5. CONCLUSION AND RECOMMENDATIONS

In the construction of art structures such as new or historical buildings, the design process is carried out before the application in the area where it will be built. In the design phase, the most suitable method is determined and the best solution is reached. One of the most suitable solution methods in this type of design process is the finite element method. Accordingly, the roof model to be placed on the building is determined by creating connection points with the concept of cubage and cross section such as area, height. Steel roof modeling is done by combining these nodes and determining the load calculation. In our application, we create a steel roof system in a structure of 10 floors 30.50 meters height, 23 * 22 meters width and length, 506 m2, as a result of physical calculations, load analysis, wind load, load combinations, steel frame calculations, equipment, torsion moments according to which parameters analyzed that it should be.

References

Chamber of Surveying and Cadastre Engineers, Engineering Measurements STB Commission 2. Engineering Measurements Symposium, 23-25 November 2005, ITU – Istanbul Turkish standards institute, 498

TBDY, (2018). Turkish Building Earthquake Code, Ministry of Environment and Urbanization, Ankara

HE IS. Karagöz, H. Özbaşaran *, M. Doğan, H. Gonen, E. Ünlüoğlu 6.SYMPOSIUM ON STEEL STRUCTURES,

Reinforced Concrete Structure - Steel Cage roof Interaction: Earthquake Response, Eskişehir Osmangazi University, Engineering - Faculty of Architecture, Civil Engineering Department, 26480, Eskisehir / Turkey

Rüya Kılıç Demircan1 *, Pınar Sezin Öztürk Kardoğan2, Murat Pınarlık3, Osman Aytekin4, Analysis of Historical Şavşat Tibet Monastery with Finite Element Method, 2nd International Mediterranean Science and Engineering Congress (IMSEC 2017) Çukurova University, Congress Center, October 25-27, 2017, Adana / TURKEY Pages: 2049-2057, Paper ID: 918

Aslı ER AKAN, Determination of Bearing System Performance by Finite Element Analysis of Historical Wooden Column Mosques, SDU International Journal of Technologic Sciences, Vol. 2, No 1, February 2010, pp. 41-54

Vishal JAGOTA1, Aman Preet Singh SETHI2 and Khushmeet KUMAR1, Finite Element Method: An OverviewWALAILAK JOURNAL, Walailak J Sci & Tech 2013; 10 (1): 1-8.

RW Clough. The finite element method in plane stress analysis. In: Proceedings of the 2nd ASCE Conference on Electronic Computation, Pittsburgh, 1960.

R Courant. Variational methods for the solutions of problems of equilibrium and vibrations. Bull. Am. Math. Soc. 1943; 49, 1-23.

Url-1;https://www.elektrikport.com/teknik-kutuphane/sonlu-elemanlar-metodu-nedir-nerelerde-kullanilir/11661#ad-image-0