Volume 2, Issue 2

March/2021

SCIENNOVATION

A Journal on Structural Science and Innovation

www.sciennovation.net

Volume 2, Issue 2 March/2021

SCIENNOVATION

A Journal on Structural Science and Innovation

SCIENNOVATION is a peer-reviewed international open access journal. It publishes , "original research articles", "review articles" and "editorials". The journal aims to report experimental and theoretical studies on structural engineering.

SCIENNOVATION aims to be a well-known journal incexed by reputable indexes in the field of structural engineering.

SCIENNOVATION includes international scientific developments in the field of structural engineering. It annually publishes two numbers a year and has open access. The September issue which includes Turkish studies. The March issue which includes English studies.

There are 30 days pre-evaluation and 30 days scientific evaluation period for the studies sent to Sciennovation to be published. At the end of each period, the author will be informed about the grounds of acceptance or rejection.

SCIENNOVATION

A Journal on Structural Science and Innovation

CONTENTS

Yayımcı :www.sciennovation.net
Sistem Sağlayıcı :Dergipark, Tübitak Ulakbim – www.dergipark.org.tr
Editör :Ali Gürbüz
Kapak Tasarım :Ömer Gürbüz, TRT World, İstanbul
Kapak Fotoğrafları :Gean Sebastian Rodrigues - pexels.com

	• **Habib UYSAL** huysal@atauni.edu.tr **ATATURK UNIVERSITY**
	• **Hakan YALCINER** hakan.yalciner@emu.edu.tr **ERZINCAN BINALI YILDIRIM UNIVERSITY**
	• **Ivica ZAVRSKI** zavrski@grad.hr **UNIVESITY OF ZAGREB**
	• **Ihsan E. BAL** i.e.bal@pl.hanze.nl **HANZE UNIVERSITY OF APPLIED SCIENCE**
	• **Kurtulus SOYLUK** k.soyluk@gazi.edu.tr **GAZI UNIVERSİTY**
	• **Mehmet H. OMURTAG** omurtagm@itu.edu.tr **ISTANBUL TECHNICAL UNIVERSITY**
	• **Murat SAATCIOGLU** murat.saatcioglu@uottawa.ca **UNIVERSITY OF OTTAWA**

	• **Mustafa S. ALTINAKAR** altinakar@ncche.olemiss.edu **THE UNIVERSITY OF MISSISSIPPI**
	• **Serkan BEKIROGLU** serkanb@yildiz.edu.tr **YILDIZ TECHNICAL UNIVERSITY**
	• **Sakir ERDOGDU** shake@ktu.edu.tr **KARADENIZ TECHNICAL UNIVERSITY**
	• **Sevket ATES** sates@ktu.edu.tr, **KARADENIZ TECHNICAL UNIVERSITY**
	• **Ugur ERSOY** ugur.ersoy@boun.edu.tr **BOGAZICI UNIVERSITY**
	• **Yucel SAYGIN** ysaygin@sabanciuni.edu **SABANCI UNIVERSITY**

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).

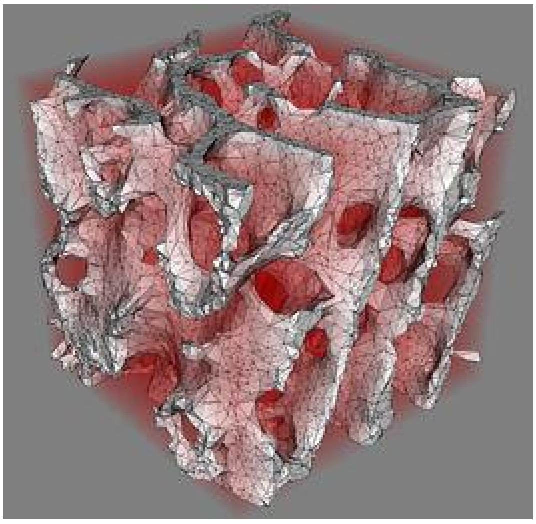


Figure 1. Finite element method sample model (URL-1, 2020).

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:

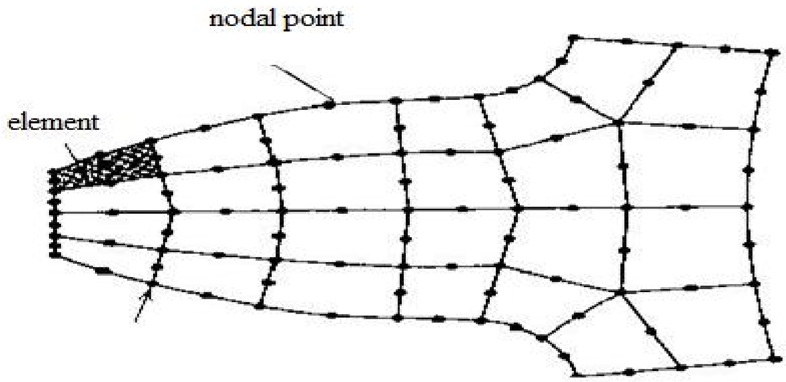


Figure 2. Finite element notation point representation (Url-1, 2020).

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).

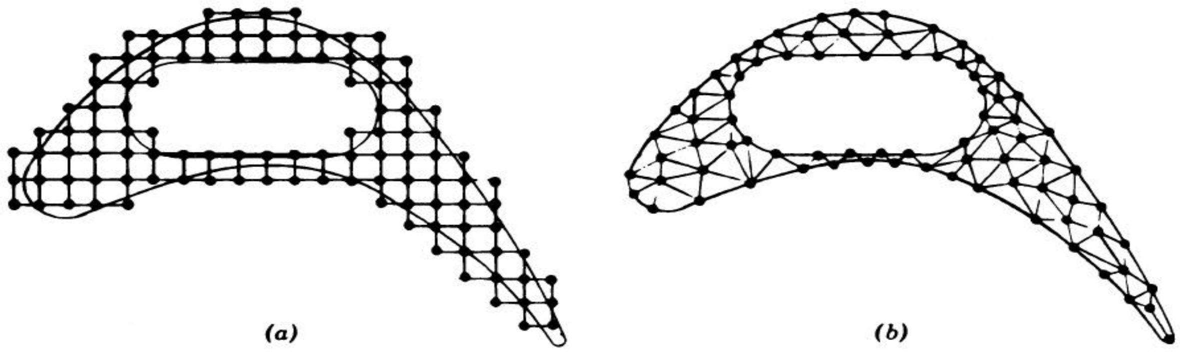


Figure 3. (a) Finite difference and (b) finite element discretizations of a turbine blade profile

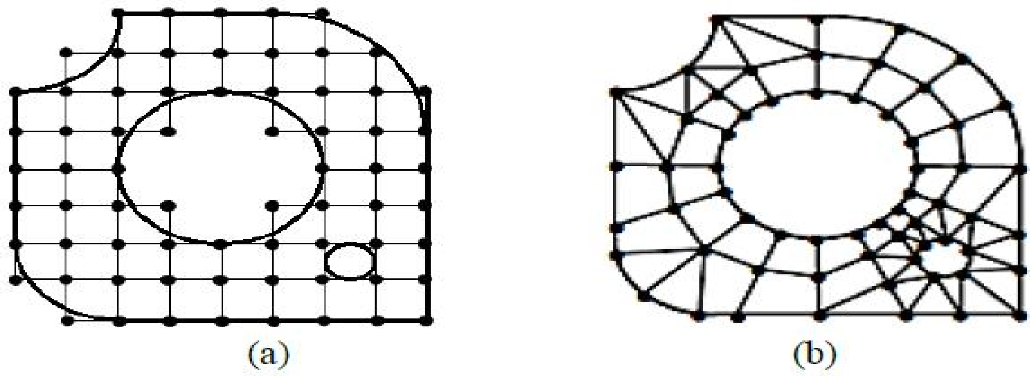


Figure 4. (a) Plate geometry finite difference model and (b) Finite element model.

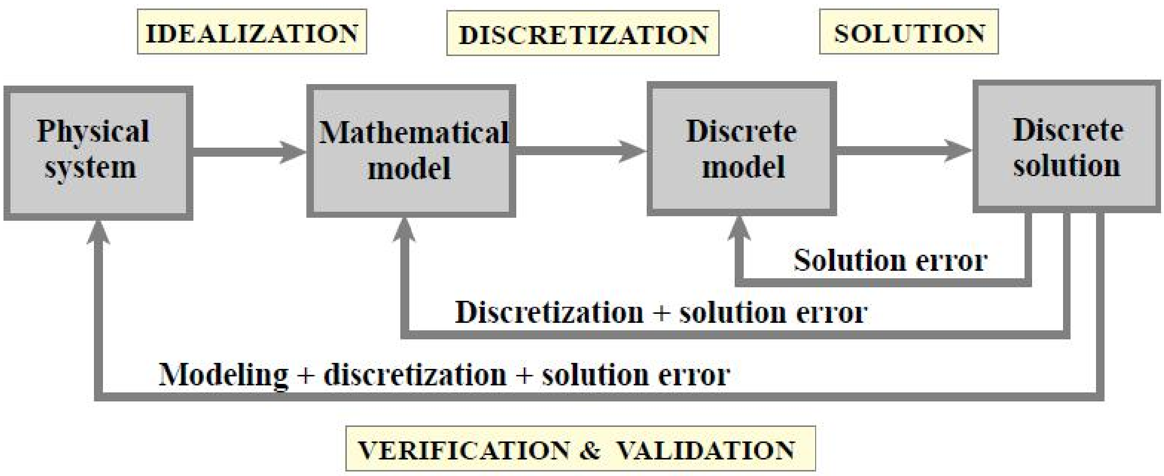


Figure 5. Flowchart of model-based simulation (MBS) by computer.

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 α = 3.523 / 7.86 tan α = 0.448

α = 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

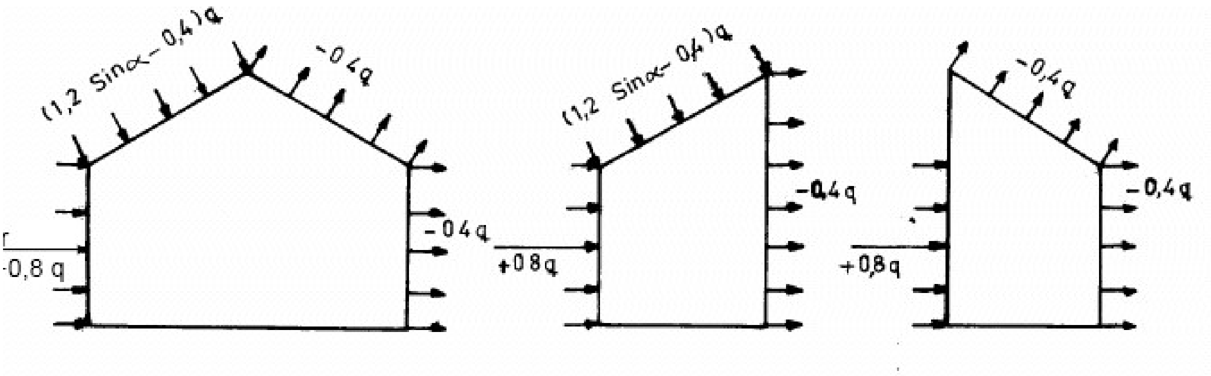


Figure 6. Wind loads (TSE, 498).

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
<u>Stress control:</u>
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

<u>Purlin Calculation on Vertical Surfaces</u>
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

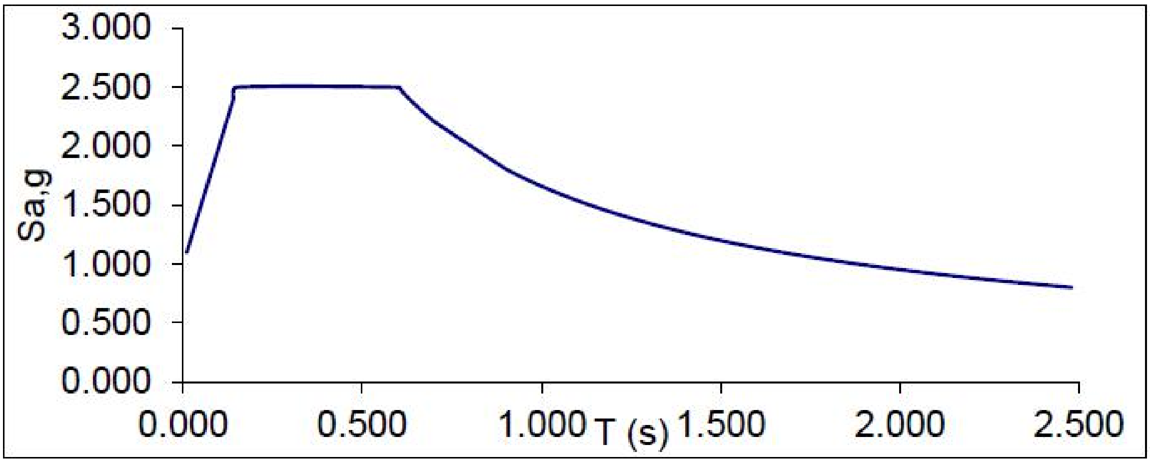


Figure 7. Roof spectrum acceleration chart according to TBDY-2018

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.

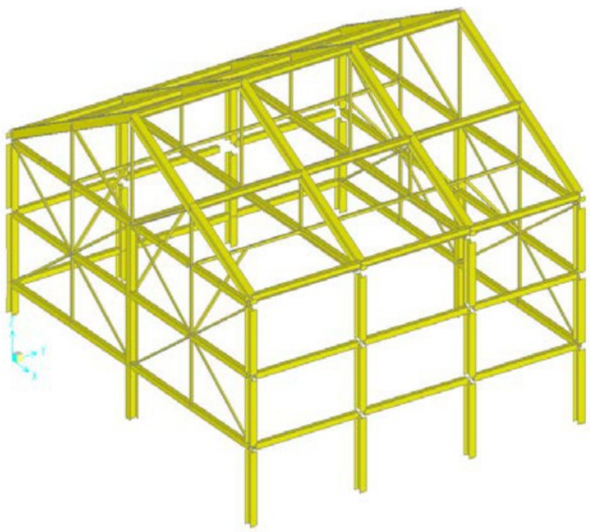


Figure 7. Finite element model of the steel carrier system

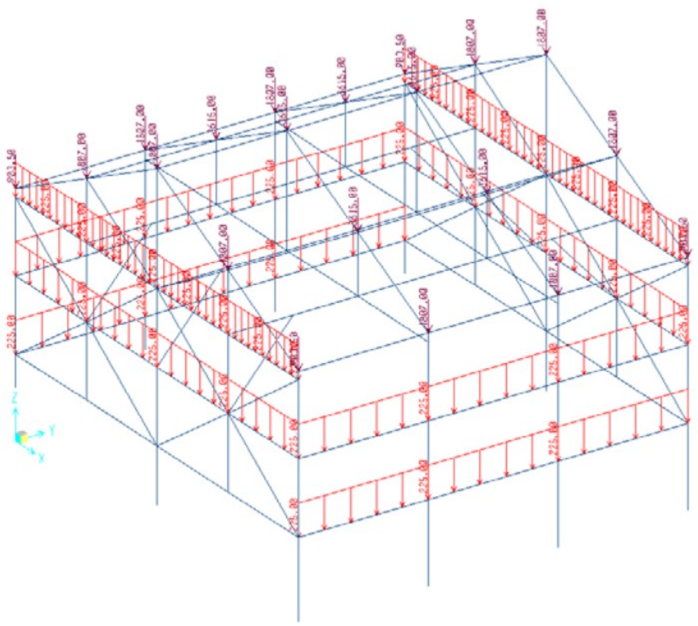


Figure 8. Dead load condition

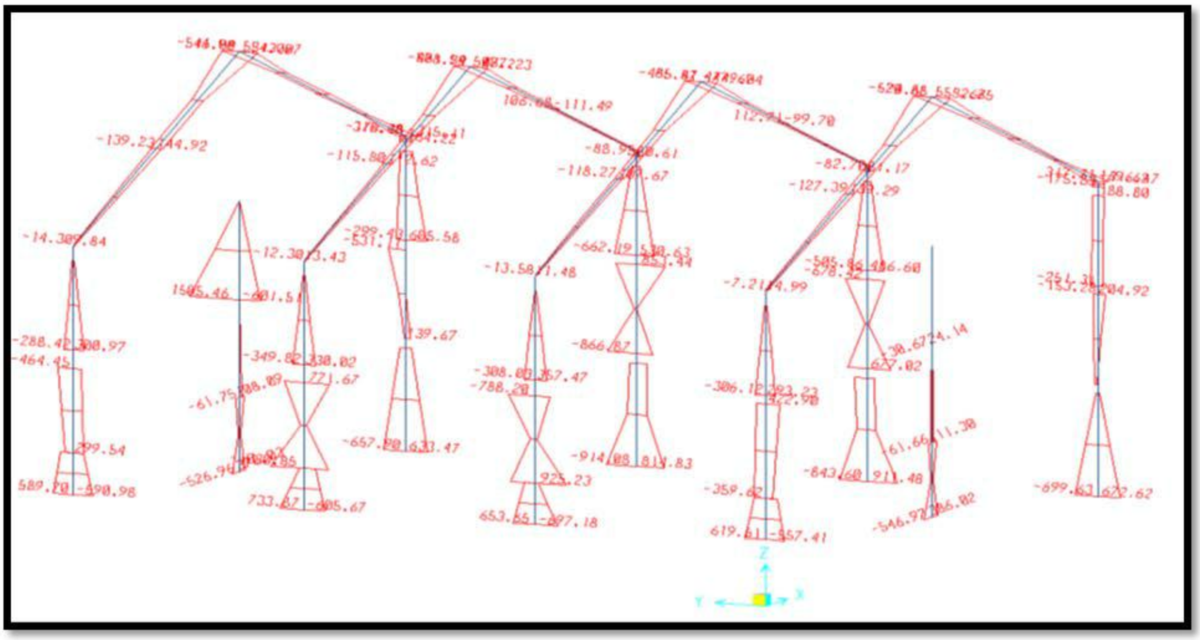


Figure 9. Tension, moment and buckling representation of the profile.

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.

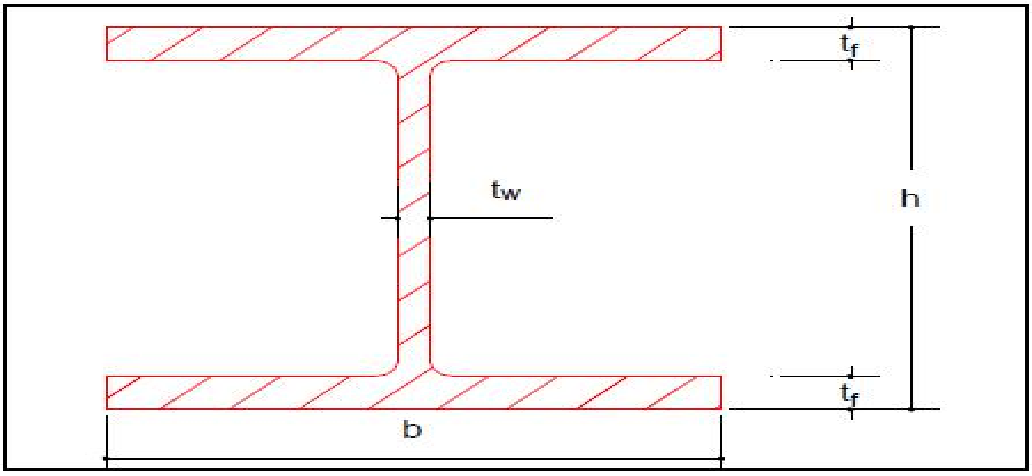


Figure 10. Profile connection points

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

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

SCIENNOVATION
A Journal of Structural Science and Innovation

RESEARCH ARTICLE

Date Received: 17.02.2021
Date Accepted: 12.03.2021

Finite Element Analysis of a Cement Silo in Bar City of Montenegro

Bahadır SADIKLAR[1], Ali GÜRBÜZ[2]

[1]*Çelik Konstrüksiyon Yapi İnşaat San. Ve Tic. A.Ş, Kocaeli, Turkey*
[2]*Recep Tayyip Erdoğan University, Department of Civil Engineering, Rize, Turkey*
Corresponding Author: ali.gurbuz@erdogan.edu.tr

Keywords:
steel roof
wind load
nonlinear analysis

Abstract

The aim of this study is to propose a roof design for cement silos, which is one of the commonly constructed building types. The project discussed within the scope of the study was implemented in the port in Bar, Montenegro. Within the scope of the study, the roof loaded with wind, snow and earthquake has been analyzed considering local regulations. Earthquake effects on the roof were taken into account with spectrum analysis. Radial ribbed steel roof system was preferred in the design of the cement silo. Design and analysis were done with TESLA Structures and CSI SAP2000 programs. For Montenegro, the values in the Eurocode 8 regulation were used. In the results of working; The data obtained from the analyzes were interpreted.

Karadağ'ın Bar Şehrinde Bir Çimento Silosunun Sonlu Elemanlar Analizi

Anahtar Kelimeler;
Çelik çatı
Rüzgar yükü
Doğrusal olmayan analiz

Özet

Bu çalışmanın amacı yaygın olarak inşa edilen yapı türlerinden biri olan çimento siloları için bir çatı tasarımı önermektir. Çalışma kapsamında ele alınan proje Karadağ'ın Bar şehrindeki limanda uygulanmıştır. Çalışma kapsamında yerel yönetmelikler göz önüne alınarak rüzgar, kar ve deprem yüklemesi yapılan çatı analiz edilmiştir. Çatıdaki deprem etkileri spektrum analizi ile dikkate alınmıştır. Çimento silosunun tasarımında radyal nervürlü çelik çatı sistemi tercih edilmiştir. Tasarım ve analizler TESLA Structures ve CSI SAP2000 programlarıyla yapılmıştır. Karadağ için Eurocode 8 yönetmeliğindeki değerler kullanıldı. Çalışma sonucunda; analizlerden elde edilen veriler yorumlanmıştır.

1 INTRODUCTION

Steel construction is generally used as a carrier system in sports facilities, factory buildings, steel industrial structures, silos, aircraft hangars, workshops, hangars and warehouses (Terzi et al., 2015). Steel construction structures have superior mechanical properties, ease of application and static capabilities compared to other carrier systems. In addition to being safe and economical, steel construction structures also provide great convenience in terms of quick installation and removal and transportation. At the same time, using steel structure systems on roofs allows to pass large openings and obtain transparent spaces (Ustabas et al., 2018). It is preferred in terms of ease of change and renewal situations, being a recyclable material, light but ductile building design, shorter construction time compared to conventional methods and indirect cost economy.

Roof element; consists of carrier elements such as roof cover, purlin, truss (Ozturk, 2009). The loads acting on the system from above are transferred first to the roof cover, from there to purlins and truss, then to the columns and from there to the ground. Horizontal forces are transferred to the ground by the columns by means of crosses and belts (Sahin, 2013). Structural analysis is performed according to the prescribed loading combinations in order to determine the required strengths of the components (column, beam, cross, bolt, welding, etc.) in the dimensioning of steel structures.

The entrance of steel to the dome construction coincides with the years of l8ll. After the first years of using cast iron or coarse iron elements, we see that in the 19th century, when there were rapid developments in steel production technology, many different types of steel dome systems were used to cover large openings. Steel domes are often named after the designers. Föble, Zimmerman, Fuller, Mohr Domes can be given as examples (Ay and Durmus, 2002).

The steel dome assembled in our project is a radial ribbed dome type. Radial ribs are formed by connecting to the tensile ring at the base and to the pressure ring or to each other at the top. The systems look like a spider web when viewed from the plan. There is a pressure ring in the middle and a tensile ring at the bottom to ensure the stability of the system. All ribs are supported on this pressure ring. While ribs can be made with a single profile in small openings, curvilinear plane truss systems are used as the span gets larger (Mohammed, 2015).

In this study, the assembled silo roof was modeled in the SAP2000 program. Necessary loads were made according to the Eurocode regulation and spectrum analysis was applied. For this purpose, firstly, the dimensions, sections, material properties of the static project were transferred to the program and the values obtained as a result of the load calculations were activated on the profiles. Then, spectrum analysis was performed and the results were evaluated.

2 METHOD

2.1 Structural Information

The diameter of the radial ribbed steel dome of the cement silo whose implementation project has been completed in Montenegro is 28.5 m. The photos of the steel elements on the roof is as in figure 1

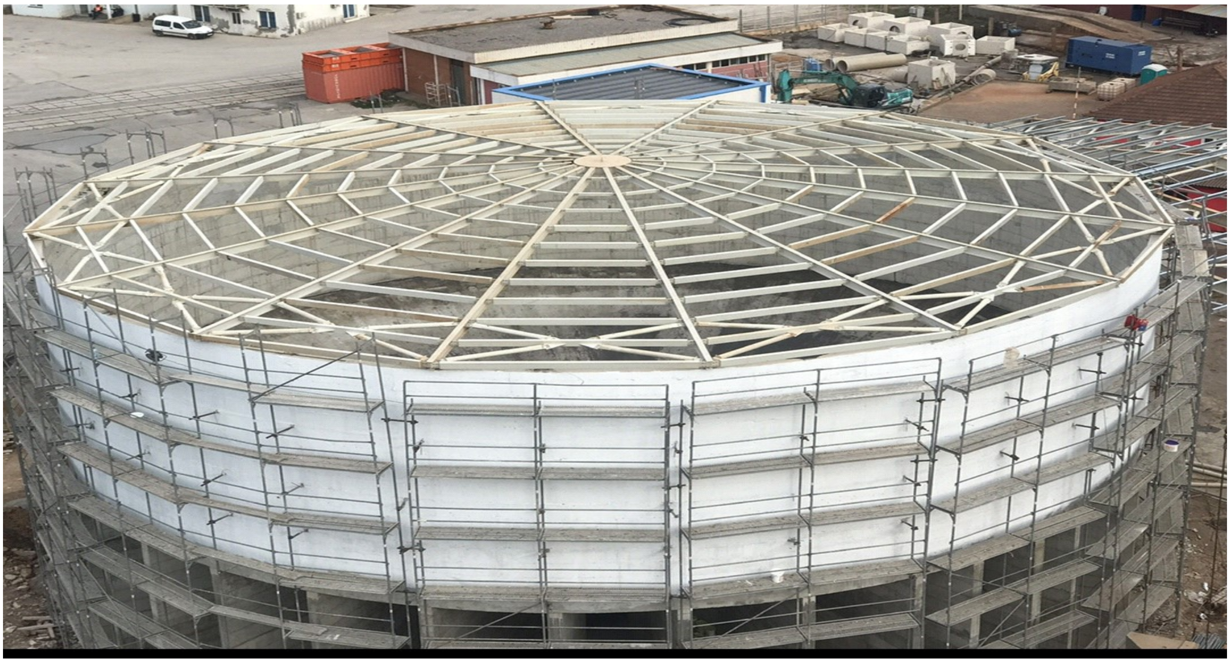


Figure 1. General view of the structure

The dome build from IPE270 Profiles, 120x60x60 box profiles, 200x100x4 wind connections and pipe profiles. The circle with a diameter of 135 cm where the beams meet consists of CHS1250 * 10. Each of the IPE270 beams of the dome is supported by 20mm anchors to the reinforced concrete curtain. The elevation difference between the highest and lowest grades of the dome is 2m.

The steel design of the project was made with the TEKLA Structures program (Trimble, 2021). During the design, attention was paid to the definition of joints as points and frame elements as frames. Then the drawing was transferred to the CSI Sap2000-Version 2020 structural analysis program (CSI SAP2000, 2020).

3 NUMERICAL EXAMPLE

The material sections and type of steel elements are defined in CSI Sap2000 (CSI, 2020). Since the box profiles are not required to take moment where they are supported to the beams, freedom was given to those points. Since no rotation or translation movement is required where the beams are supported on the reinforced concrete curtain, a built-in support was defined in the program.

3.1. Snow Load

Article 2136 of Snow Load Part No 213 in Temporary Technical Code for Building Loads used for load calculations in the country (Jugoslovenski, 1988). According to the Technical Code, a minimum load value of 0.35 kN/m2 should be used in snowy areas. Since the roof slope is 7o <20o, this value is used exactly without any reduction due to the roof slope.

While calculating the snow load to be impacted on the roof elements, first the load falling on the area between 2 consecutive box profiles was calculated. Then, it was applied as a distributed load to the beams to which it was supported. 3D view of the steel roof is shown in figure 2.

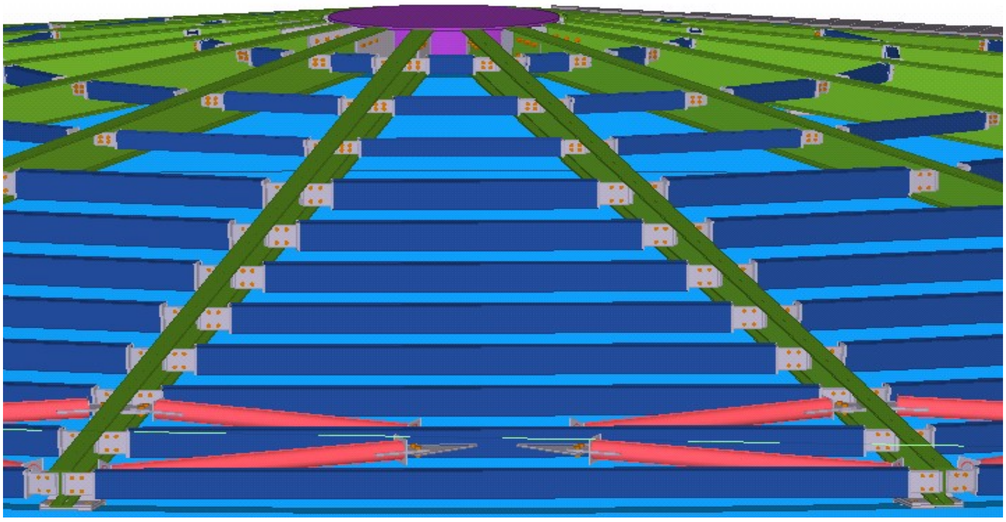


Figure 2. 3D view of the steel roof (TEKLA 2017).

3.2. Wind Load

In the load calculation, the wind should be calculated and activated well. Low-rise and roofed structural systems can be exposed to severe damages by wind effects as in industrial buildings (Huang et al., 2016). In order to prevent this, the effect of wind should be understood very well. Although wind is a load affecting the structure, this load can be impacted as an equivalent static load or as a variable load defined in the time history. However, the random behavior of the wind should not be forgotten during this process. In addition, researches conducted in recent years have shown that positioning the buildings perpendicular to the wind direction and torsional oscillation are as important as being in the same direction with the wind (Kurc et al., 2012) The wind load to be exposed to the roof was calculated according to Eurocode 1: Standard for Actions on Structures (EN 1991) (Handa, 2006). First, the baseline wind speed baseline value (vb,0) 8m/s in the city of Bar was determined (Figure 3). The values of and may differ according to regions, but usually both values can be taken as 1

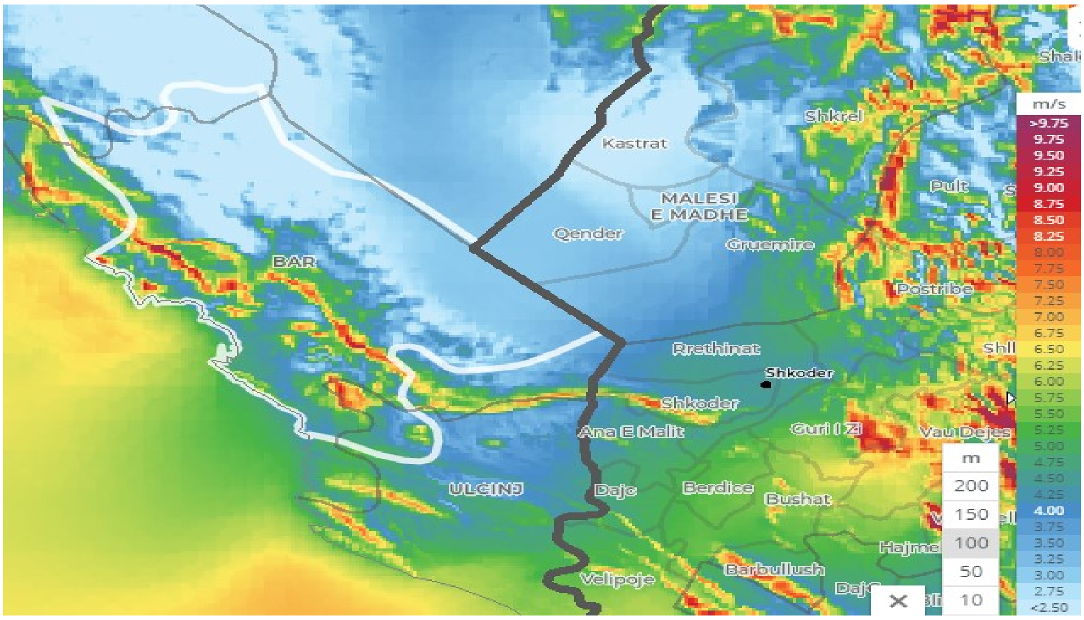


Figure 3. European Wind Map for Basic Wind Speed (WEB-1)

$\nu_b = c_{dir} \times c_{season} \times \nu_{b,0}$ (1)

ν_b : Basic wind speed c_{dir} : Direction coefficent

c_{season} : Seasonal coefficent $\nu_{b,0}$: Basic value of basic wind speed

$\nu_b = 8$ m/s

There is a correlation between the fundamental wind speed and the fundamental wind pressure..

$q_b = \frac{\rho}{2} \times \nu_b^2$ (2)

ρ : Air specific gravity (1.25 kg/m^3)

qb : Basic wind pressure = 40

Since the value at the reference height of the structure is accepted as the basic value of the velocity pressure, it must be changed. The abrupt changes in isometric velocity and wind velocity depend on the roughness of the terrain. The roughness coefficient must be calculated to obtain the main wind velocity at isometric height.

$vm(z)=cr(z)\times co(z)\times vb$ (3)

$vm(z)$: Main wind speed $cr(z)$: Roughness coefficient

$co(z)$: Orography coefficient (Can be taken as 1)

The roughness coefficient depends on the minimum height value.

$cr\ (z) = kr \times \ln(z/zo)\ ,\ z \geq zmin\ = 2.70$ (4)

$kr = 0.19 \times (z/zo,II)\ 0.07$ (5)

kr: Land coefficient zo: Roughness length $zmin$: Minimum height

Land groups required for calculations are as in table 1.

Table 1. Land groups

Land Group	Land characteristic	z_0 (m)	z_{min} (m)
0	Sea or coastal area	0.003	1.0
I	Lakes; unobstructed	0.010	1.0
II	Poor vegetation; Isolated obstacle less than 20 times the height of the obstacle	0.050	20
III	Regular vegetation; Forests, slums, villages	0.300	5.0
IV	Buildings with an average height of at least 15 m and areas covered at least 15%	1.000	10.0

Depending on the group of the land, we can calculate the storm speed from the formula below.

$\nu_p(z) = \nu_m(z) \times G$ (6)

$G = \sqrt{Ce(z)} = \sqrt{1 + 7x\frac{\sigma v(z)}{vm(z)}}$ ve $z \geq z_{min}$ (7)

G: Storm coefficient kI: Turbulence coefficient (usually 1 is taken.)

$vp\ (z)$: Storm Speed

G = 1.34.

qp (z): Storm pressure$q_p(z) = q_b(z) \times [\, c_r(z)]^2 \times [c_o(z)]^2 \times [\, 1 + \frac{7 \times kl}{Co(z) \times \ln(\frac{z}{z0})}\,] = 531.2.$ (8)

$c_o\,(z) = k_I$ = can be taken as 1.0.

Calculation of wind load:

$F_W = c_s \times c_d \times c_f \times q_p(ze) \times A_{ref}$ (9)

F_W : Wind load

cs: Size coefficient (1.0 can be taken.) = 1.0

cd: Dynamic coefficient (1.0 can be taken.) = 1.0

cf: Force coefficient = 0.618

$Aref$: Reference area = 30 m2

$q_p(z) = 531.2$

It was calculated as Fw = 9.848 kN and added to the finite element model.

3.3. Panel + Trapezoidal Load

The weight of the trapezoidal sheet (0.058 kN / m2) + insulation (0.031 kN / m2) + vapor barrier (0.018 kN / m2) + seam (0.043 kN / m2) materials on the roof is calculated as 0.15 kN / m2 and the roof is applied as a constant load. activated.

3.4. Eartquake Load

Spectrum Analysis was used for Earthquake load. There are two different spectrum curves in the Eurocode 8, (2004). One of these curves is valid if the magnitude of the Type 1 earthquake surface wave (Ms> 5.5) is greater than 5.5. If (Ms> 5.5) is less than 5.5, Type 2 is valid. Since the earthquake surface waves in the city of Bar are larger than 5.5, Type 1 was selected in spectrum analysis. The equations for the spectrum to be used during design are given below. Spectrum curve for Type 1 and Type 2 is given in Figure 4.

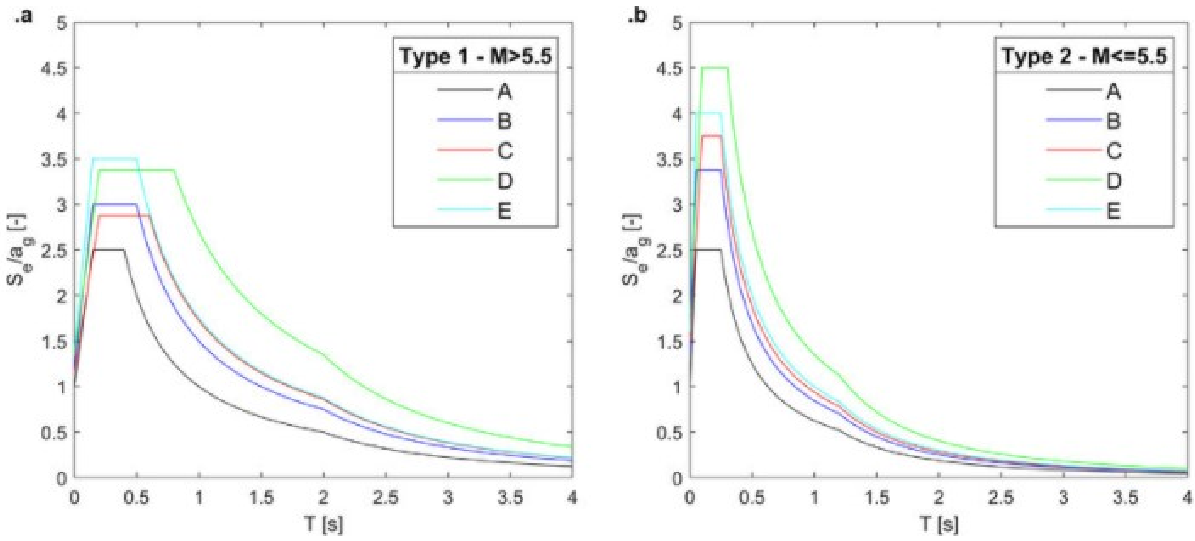


Figure 4. Spectrum Curve Limit Values for Type 1 and Type 2 (Eurocode 8)

5 different soil types are defined in the earthquake code. In addition, two different soil types have been specifically defined. Certain criteria for these floors are given in Table 3. In the city of Bar, the floor type on which the silo will be built was chosen as D as the soil type.

The values used in the drawing of the acceleration-period graph were calculated as in table 2.

Table 2. Spectrum parameteres

Horizontal Ground Acceleration	**0,3**	**Spectrum Period (Tb)**	0,2
Floor Type	D	Spectrum Period (Tc)	0,8
Ground Factor (S)	1,35	Spectrum Period (Td)	2
Behavior Coefficient	2	Spectrum Type	Tip 1

3.5 Load Combinations

The applied load combinations in accordance with Eurocode 3, (2005) are given in table 3.

Table 3. Load Combinations

G+Q+EX+0.3EY	0.9G+0.3EX+EY	1.35G+1.35Q-1.35WY
1.35G+1.5Q	0.9G+0.3EX-EY	1.35G+1.5WX
G+Q+0.3EX+EY	1.35G+1.35Q+1.35WX	1.35G-1.5WX
G+Q-EY+0.3EX	1.35G+1.35Q-1.35WX	1.35G+1.5WY
0.9G+EX+0.3EY	1.35G+1.35Q+1.35WY	1.35G-1.5WY

4. ANALYSIS RESULTS

As a result of the analysis, the displacement of the radial ribbed dome was calculated as 0.106 mm in the combination of 1.35g + 1.35q-1.35wx at joint 120. Table 4 shows the top 5 highest joints.

Table 4. Joint displacements

Combination	Joint	U3 (m)
1,35G+1,35Q-1,35WX	120	-0.000106
1,35G+1,35Q-1,35WX	105	-0.000105
1,35G+1,35Q-1,35WX	121	-0.000105
1,35G+1,35Q-1,35WX	90	-0.000104
1,35G+1,35Q-1,35WX	116	-0.000104

The normal force value has been determined as -135.87 KN at a load combination of 1.35g + 1.35q + 1.35wx at the 1.05m position of the maximum 518 rod element. Table 5 shows the maximum first 5 normal forces.

Table 5. Maximum first 5 normal forces

Combination	Frame Text	Station (m)	P KN	V2 KN	V3 KN
1,35G+1,35Q+1,35WX	518	1,05	-135,87	11,24	-1,677
1,35G+1,35Q+1,35WX	518	0,525	-134,74	10,29	-2,075
1,35G+1,35Q+1,35WX	516	1,27	-133,89	3,91	-2,182
1,35G+1,35Q+1,35WX	517	1,25	-133,75	7,03	5,99
1,35G+1,35Q+1,35WX	518	0	-133,61	9,24	-2,47

The maximum tensile was determined as 93.846.14 KN at the 1.27 m position of the rod element 469 at 1.35g-1.5wx load combination. Figure 5 shows tensile graphs on SAP2000 model of steel roof. Also table 6 shows the maximum first 5 tensile values.

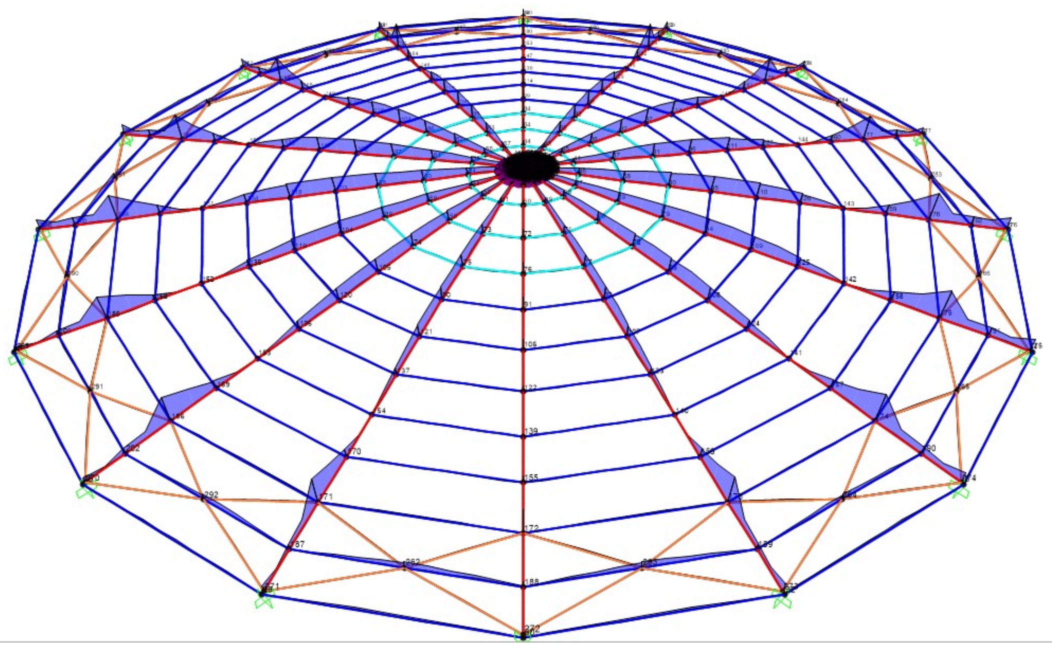


Figure 5. Tensile values on SAP2000 Model

Table 6. Maximum first 5 tensile values

Combination	Frame Text	Station (m)	Smax KN / m^2
1,35G-1,5WX	469	1,27	93846,14
1,35G-1,5WX	481	1,27	93825,02
1,35G-1,5WX	470	0	93433,35
1,35G-1,5WX	482	0	93241,39
1,35G-1,5WX	444	1,27	92473,86

4. CONCLUSION AND SUGGESTIONS

In this study, a cement silo with a diameter of 28.5 meters is considered. It shows that radial ribbed roof systems are a good solution for large span and circular structures such as cement silo under earthquake, snow and wind loads. According to the results obtained from the analysis, the following points can be pointed out.

Smax appeared at the wind junctions of the radial ribbed dome. In the load combinations prepared according to the Eurocode regulation, the maximum tensile value of 93.846,14 kN was determined in the combination of 1.35g-1.5wx.

S_{min} appeared at the wind junctions of the radial ribbed dome. In load combinations, the minimum stress value of -129.087.85 kN was determined in the combination of 1.35g-1.5wx. It can be said that the most critical load in system loading is 1.35g-1.5wx.

Normal force value has been determined as -135.87 KN at maximum 1.35g + 1.35q + 1.35wx load combination. Normal strength values increase on the IPE270 beam from the center of the dome to the outside. As can be understood in the system loading, the loadings have increased as the area from the center to the outside increases.

As a result of the analysis of the dome, the maximum displacement value was measured as 0.106 mm in the combination of 1.35g + 1.35q-1.35wx.

Generally, radial ribbed dome displacements were observed to be at acceptable levels. It has been understood that the tensile and pressure values reflect the properties of the radial ribbed dome and that the type and dimensions of the material used in the assembly can be molded on the safe side with the necessary loads and analysis.

REFERENCES

Autodesk (2019). AutoCAD v19, Commercial Computer-Aided Design and Drafting Software

Ay Z. , Durmuş G. , (2002), " TMH – Türkiye Mühendislik Haberleri " Sayı 418

CSI, SAP2000, (2020) Integrated Finite Elements Analysis and Design of Structures, Computers and Structures, Inc, Berkeley, CA.

Eurocode 3, (2005) Design Of Steel Structures, European Committee For Standartization, Brussels.

Eurocode 8, (2004) "Design of Structures for Earthquake , Doc. CEN/TC250/SC8/N388B. Comité Européen de Normalisation, Bruxelles

Handa, K. (2006). European Standard For Wınd Loads Notes (Eurocode EN 1991-1-4 Wind Actions), 10 p.

Huang, G. , Luo, Y., (2016) Gurley, K. R. ve Ding, J., 2016, Revisting Moment-Based Characterization for Wind Pressures, Journal of Wind Engineering and Industrial Aerodynamics, 151 , 158-168

Jugoslovenski Standard (1988). Riyremeni Tehnijki Propisi Za Opterecenja Zgrada

Kurç, Ö. , Kayışoğlu, B., Shojaee, S. M. N. ve Uzol, O., (2012), Yüksek Binalarda Rüzgar Etkilerinin Rüzgar Tüneli Deneyleriyle Tespiti, İMO Teknik Dergi, 389, 6163-6186.

Mohammed A. , (2015) , " Çelik Kubbe Sistemler " Yüksek Lisans Tezi , Süleyman Demirel Üniversitesi , Isparta

Öztürk, A. Z. , (2009) , " Çelik Yapılar Kısa Bilgi ve Çözülmüş Problemler ", 5. Baskı, Birsen Yayınevi, İstanbul, 232-266.

Terzi, C., Gürbüz, A., Süme, V., & Ustabaş, İ., (2015). Kar Yükünün Tetiklemesi Sonucu Çöken Örnek Bir Çelik Çatının İncelenmesi . 6. Çelik Yapılar Sempozyumu (pp.363-372). Eskişehir, Turkey

Trimble Co (2021). Tekla Structures 2017 Program. www.tekla.com

Ustabas, İ.; Sume, V.; Baki, V. A.; Gurbuz, A., (2018). Evaluation of a Steel Roof of a Football Field That Collapsed due to a Heavy Snowfall . 13th International Congress on Advances in Civil Engineering (pp.1-8). İzmir, Turkey

Web-1, (2021) https://globalwindatlas.info/area/Montenegro/Bar

SCIENNOVATION
A Journal of Structural Science and Innovation

RESEARCH ARTICLE

Date Received: 17.02.2021
Date Accepted: 25.02.2021

Evaluation of Earthquake Behavior of Reinforced Concrete Frame Buildings by Nonlinear Methods

Tuğrul KASAP

Recep Tayyip Erdoğan University, Department of Civil Engineering, Rize, Turkey.
tugrul_kasap20@erdogan.edu.tr

Anahtar Kelimeler;
Dynamic Analysis, Pushover Analysis, Structure Seismic Behavior, Nonlineer Analysis

Abstract

Almost all of our country is within the domains of active faults. Due to urbanization and population growth, the devastating effect of the earthquake is increasing every day. For this reason, it is of great importance to determine the earthquake performances of the structures and to build earthquake resistant structures according to the determined performances. Earthquake-resistant structure the main purpose of the design is to provide sufficient resistance to future loads throughout the life of the structure. In the designs made, the biggest challenges faced are the identification of the effects caused by the earthquake. Linear and nonlinear analysis methods are used in the design of structural systems. Today, they are nonlinear analyses in the field of time-definition, which best reflect the modeling of real behavior and structural seismic behavior. As part of this study, nonlinear pushover and dynamic analysis were performed in terms of studying the earthquake behavior of a concrete frame building. As a result of these analyses, it was seen which analysis method is appropriate in terms of determining the earthquake behavior of the structure more accurately.

Betonarme Bir Binanın Lineer Olmayan Analiz Yöntemleri ile Deprem Davranışının Değerlendirilmesi

Anahtar Kelimeler;
Dinamik Analiz, Statik İtme Analizi, Yapı Sismik Davranışı, Doğrusal Olmayan Analizler

Özet

Ülkemizin neredeyse tümü aktif fayların etki alanları içerisindedir. Kentleşme ve nüfus artışına bağlı olarak depremin yıkıcı etkisi her geçen gün artmaktadır. Bu yüzden yapıların deprem performanslarını belirlemek ve belirlenen performanslara göre depreme dayanıklı yapıların inşa edilmesi çok büyük önem taşımaktadır. Depreme dayanıklı yapı tasarımdaki asıl amaç, yapının ömrü boyunca üzerine gelecek yüklere karşı yeterli dayanımı sağlamasıdır. Yapılan tasarımlarda ise karşılaşan en büyük zorluklar depremden dolayı oluşan etkilerin tanımlanmasıdır. Yapısal sistemlerin tasarımında doğrusal ve doğrusal olmayan analiz yöntemleri kullanılmaktadır. Günümüzde gerçek davranışların modellenmesinde ve yapı sismik davranışını en iyi yansıtan, zaman-tanım alanında doğrusal olmayan analizlerdir. Bu çalışma kapsamında betornarme çerçeve bir binanın deprem davranışının incelenmesi açısından doğrusal olmayan statik itme ve dinamik analizleri yapılmıştır. Yapılan bu analizlerin sonucunda yapının deprem davranışının daha doğru tespit edilebilmesi açısından hangi analiz yönteminin uygun olduğu görülmüştür.

1 INTRODUCTION

Especially in recent years, the building-performance concept has attracted attention in earthquake resistant building design. Various studies are carried out on this concept. Designing and evaluating structural systems according to performance is included in various building codes (Burton et al., 2016). There are various analysis methods to examine the earthquake-affected condition of buildings (Bayat et al., 2017) The main principle in these methods is to obtain the deformations that will occur in the structural system elements under the design earthquake affecting the structure. Performance-based design and evaluation can be made with various nonlinear analysis programs. By analyzing the structure-performance relationship, it can be checked whether the damage occurred in the structure exceeds the strain capacity or not. The limited strain values are compared with the targeted performance values for various earthquakes. As a result of this comparison, it is tried to reach the target values in structural analysis. Thanks to this method, it is possible to make realistic earthquake designs (Işık & Velioğlu, 2017).

Linear and nonlinear analysis methods are used in performance analysis of structural systems. Even within these analysis methods, it is quite difficult to examine the behavior of reinforced concrete structures under the influence of earthquakes (Nobahar et al., 2016). It is impossible to estimate the earthquake behavior of the building realistically with linear analysis methods. Earthquake calculations made with linear analysis methods and earthquake investigation of existing structures are valid until the first plastic hinge is formed in the building (Karabulut 2011). It is extremely difficult to have an idea about how the crashes will occur in the next part after this stage. Nonlinear analysis methods have been developed for more realistic modeling of building behavior (Çavdar, 2019). Among these methods, it is the "nonlinear dynamic time history analysis" method that best reflects the real behavior. However, this method is very complex, time consuming and requires a large number of local earthquake records. For this reason, it is not very suitable for use in the engineering field in practice. As an alternative nonlinear static analysis methods are more commonly used in structural analysis. Studies on the subject show that both methods yield similar results (Yılmaz, 2008).

2. MATERIAL AND METHODS

2.1. Nonlinear Analysis Methods

Structural systems show linear behavior under the loads imposed on them. The displacements, deformations and stresses that occur in the structure are based on linear state assumptions. On the other hand, as the loads to which the structures are exposed approach the bearing strength of the structural elements, the stress and strain exceed the linear elastic limit. Similarly, displacements are increasing unacceptably (Çavdar, 2019). It is not possible to talk about linearity in the situation. In such cases, the nonlinear behavior occurring in the continuation of the linear elastic limit should be evaluated. In the nonlinear theory, the super position principle does not apply. It is taken into account that the charges vary depending on a parameter such that the ratio between them remains constant. Nonlinear theory can be presented in three ways (Tuncer, 2008):

1. Nonlinear theory in terms of materials: Material behavior is not linear elastic.

2. Nonlinear theory in terms of geometry changes: The effects of displacements on equilibrium equations are not small enough to be neglected.

3. Nonlinear theory in terms of material and geometry changes: The material is not linear elastic and the displacements are not very small.

2.1.1 Nonlinear Static Analysis

Nonlinear static analysis is mainly based on the nonlinear theory in terms of material and geometric change. In this method, the horizontal force-horizontal displacement (P-Δ) relationship, which represents the strength of the structures under constant vertical loads and continuously increasing horizontal loads, is obtained (Goel and Chopra, 2004). The analysis is completed by evaluating this relationship for any specified earthquake level. The horizontal force-horizontal displacement relation called the capacity curve is important for nonlinearity. The capacity curve obtained until the limit state of the structure, the locations of weak elements, partial and total collapse mechanisms, horizontal earthquake loads can be obtained (Korkmaz and Uçar, 2005). With the nonlinear static analysis methods, the deformation demands of the structural system elements of existing structures and new structures to be designed can be determined. Thus, it is checked whether the performance level requested from the building is provided for a certain earthquake level. Displacement-based nonlinear static analysis methods used in the performance-based design and evaluation of structures in the literature are generally as follows (Dervişoglu, 2006):

- Capacity Spectrum Method (CSM)
- Displacement Coefficients Method (DCM)
- Secant Method (SM)
- Pour Point Spectrum Method (YPSM)
- Static Pushover Analysis (Modal Pushover Analysis) (MPA)
- Incremental Response Spectrum Analysis (IRSA)
- İnelastic Response Spectrum Method (N2)

2.1.2. Nonlinear Dynamic Analysis

Dynamic analysis is widely used today to examine the nonlinear behavior of an earthquake-affected structure. In determining the earthquake loads affecting the structure, a solution should be made by using sufficient numbers of records depending on the ground conditions (Öncü & Yön, 2016). Dynamic analysis with its general definition; It is the type of analysis in which detailed structural modeling is performed with the help of earthquake ground acceleration records (Zhou et al., 2017). There is relatively less uncertainty in this analysis compared to other analyzes. As a result of dynamic analysis, nonlinear behaviors occurring in the elements for each degree of freedom are determined (Liu et al., 2016). Considering the general results in these elements, results such as relative floor displacement, roof displacement, overturning moment and shear force can be achieved. The most widely used dynamic analysis methods are incremental mode combination method and time history method. Nonlinear dynamic analysis method is shown schematically in Figure 1 (Kayhan & Demir, 2015).

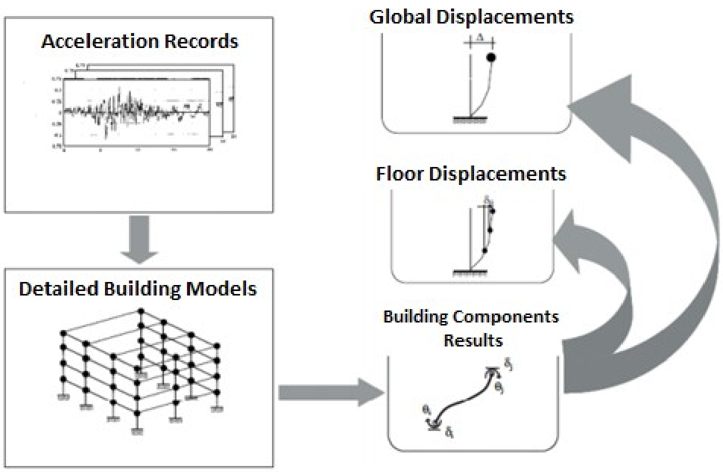


Fig 1. Nonlinear dynamic analysis method (Kayhan and Demir, 2015)

2.2. Materials Used

Within the scope of this study, static repulsion analysis and nonlinear dynamic analysis in the time-history field of a 4-storey reinforced concrete structure were performed with the help of SAP2000 package program. Considering the design conditions given in the 2018 Turkish Earthquake Code (TBEC-2018), C30/37 concrete and S420 steel were used in the analysis. In addition, the building has 4 openings in the X direction and 3 openings in the Y direction, and a seating area of 144 m2 (16m in X direction, 9m in Y direction). The floor plan and 3-dimensional model of the building can be seen in Figure 2. All floor heights of the building are 3 meters. Column dimensions are 40cmx40cm, beam dimensions are 30cmx50cm and slab thickness is 12 cm. In addition, 9Ø16 longitudinal reinforcement and Ø8 / 150 stirrups were used in columns, and 9Ø12 longitudinal reinforcement and Ø8 / 150 stirrups were used in beams. The cover is 4 cm in columns and beams. The building foundation is considered to be rigid. Soil damping is not taken into account. Figure 3 shows the column and beam sections of the building.

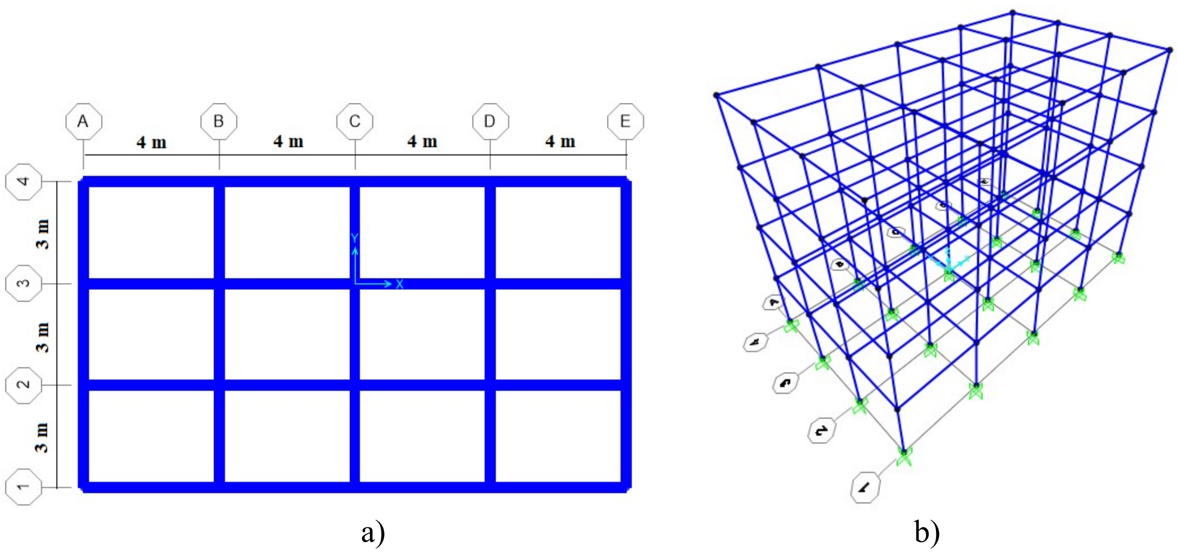


Fig 2. a) Floor Plan b) 3D Model

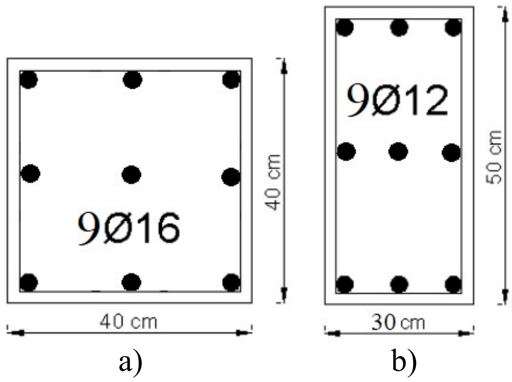


Fig 3. a) Column section b) Beam section

Static pushover analysis was effected according to the dominant mode (first mode) of the carrier system. The structure was first analyzed under vertical loads. Bending stiffnesses and plastic hinges are assigned to columns and beams. Gradually increasing loads were applied to the structure. The static pushover (capacity) curve has been drawn under the gradual increasing load effect of the structure. Capacity diagrams were obtained with the help of the curve drawn. Modal displacements were determined by transitioning from the design earthquake to the acceleration spectrum. In each step, the pushover analysis was performed again until the structural system reached the specified modal displacements.

In nonlinear dynamic analysis in time history, earthquake loads must be applied directly to the structure to determine the behavior of structures under earthquake effect. Therefore, with the help of earthquake database records from AFAD, strong seismic loads were applied to the existing structure in 2 directions. 3 different earthquake records are used to simulate the effects of earthquakes. These;

- Kocaeli Earthquake Record (İzmit, 1999, 6.1 Magnitude),
- Denizli Earthquake Record (Bozkurt, 2019, 6 Magnitude)
- Van Earthquake Record (City Center, 2011 6.7 Magnitude)

Earthquake records used are generally selected from large earthquakes occurring in Turkey. The destructive effect of earthquakes can be associated with the acceleration values they create on the ground. Table 1 shows the maximum acceleration values of the selected earthquakes. The magnitude of these values can be directly related to the magnitude of the stresses that will occur in the building.

Table 1. Maximum accelerations of the earthquake records

City	Station	Station No	Maximum Acceleration (g)
Kocaeli	İzmit	4101	0,312
Denizli	Çardak	2005	0,274
Van	Muradiye	6503	0,181

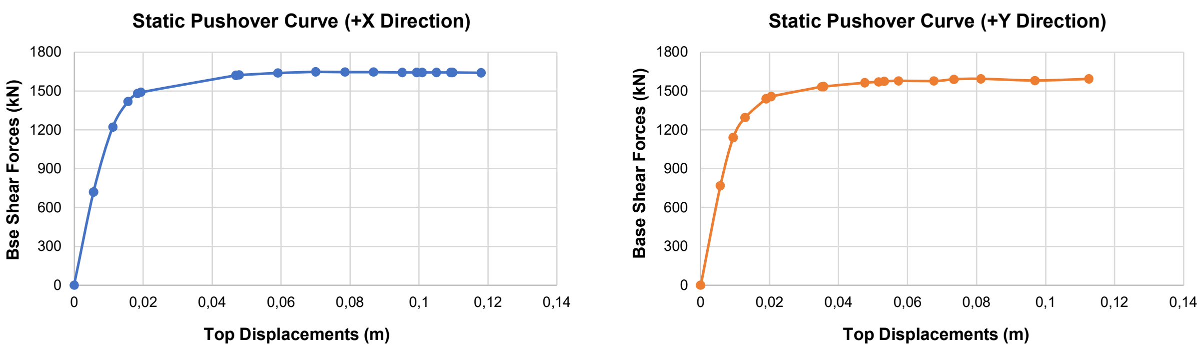
3. RESULTS

Base shear forces and top displacements found with the help of earthquake records and static repulsion analysis are given in Table 2. As a result of the nonlinear static repulsion analyzes performed in the X and Y directions, the peak displacement values were higher than the nonlinear dynamic analyzes performed in the time-history area. It is assumed that this is due to the repetitive loading used in static pushover analysis. Comparing the analyzes made with the help of earthquake records, the base shear and peak displacement values of the Kocaeli earthquake with the highest acceleration value are higher than the other earthquakes. As a result, it is seen from the values that the effect of large acceleration values on the structure will be large.

Table 2. Base shear forces and top displacements

Earthquake Records	Base Shear Forces (kN) Max.	Base Shear Forces (kN) Min.	Top Displacements (m)
Kocaeli-İzmit X	1604	-1607	0,0359
Kocaeli-İzmit Y	1574	-1610	0,0336
Denizli-Çardak X	1391	-1421	0,0270
Denizli Çardak Y	1412	-1332	0,0276
Van-Muradiye X	1467	-1269	0,0190
Van-Muradiye Y	1411	-1228	0,0183
Static Pushover X	1614	-	0,0480
Static Pushover Y	1558	-	0,0446

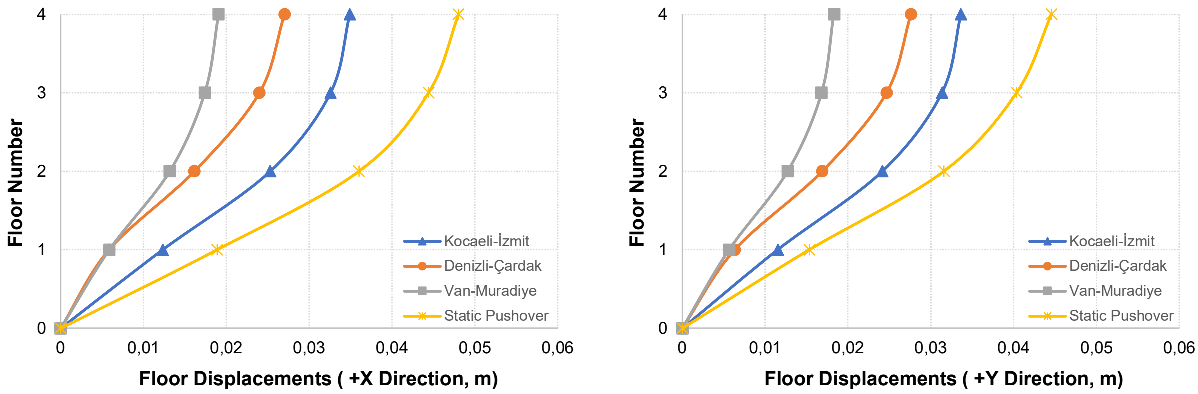
As a result of the static pushover analysis, the base shear forces and the top displacements are similar in X and Y directions, since the structure is symmetrical in both axes (X and Y). In Figure 4, the curves of the base shear forces corresponding to the peak displacements in the X and Y directions of the static repulsion analysis are given.



a) b)

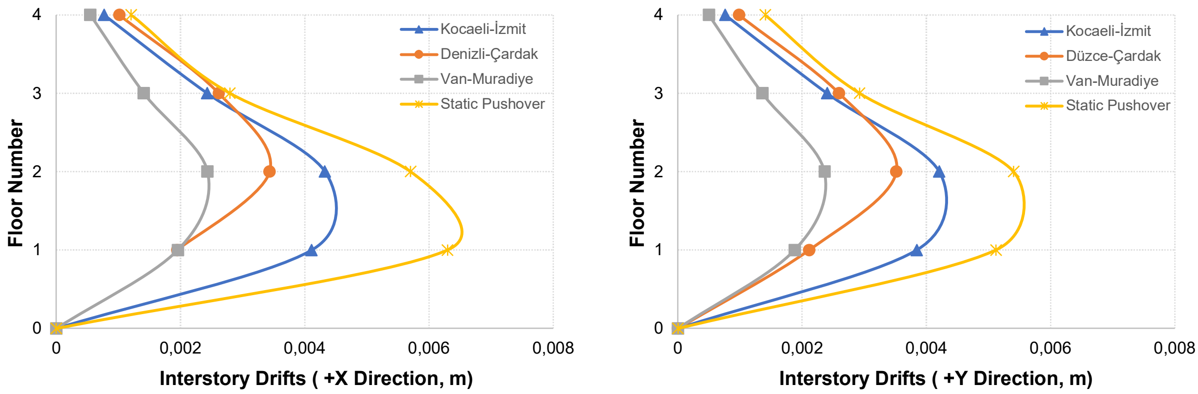
Fig 4. Static Pushover Curves a-) X direction, b) Y direction

Floor displacements found as a result of static pushover analysis and dynamic analysis are as in Figure 5. Interstory drifts are given in Figure 6. When these data were examined, the storey displacements and the interstory drift values obtained at the end of the static pushver analysis were more than the values obtained from the dynamic analysis. In addition, Kocaeli earthquake caused more storey drifts compared to other earthquakes due to its high acceleration value. Based on the analysis made in both directions (X and Y) for structural design, it is clear that if earthquake records are used, the Kocaeli earthquake will cause more damage to the building. This situation shows that the building structural system should be designed on the Kocaeli earthquake scenario.



a) b)

Fig 5. Floor displacements a) X direction b) Y direction



a) b)

Fig 6. Interstory drifts a) X direction b) Y direction

REFERENCES

Bayat, M., Daneshjoo, F., Nisticò, N. 2017. The effect of different intensity measures and earthquake directions on the seismic assessment of skewed highway bridges. Earthquake Engineering and Engineering Vibration, 16(1), 165-179.

Burton, H.V.; Deierlein, G.G.; Mar, D.; Mosalam, K.M.; Rodgers, J.; Gnay, S.: Rocking spine for enhanced seismic performance of reinforced concrete frames with infills. J. Struct. Eng. 142(11), 04016096 (2016).

Çavdar Ö., 2019. Investigation of the earthquake performance of a reinforced concrete shear wall hotel using nonlinear methods, International Journal of Science and Engineering Applications, 8-12, 509-516

Dervişoğlu, Z. (2006). Theory comparison without method of performance evaluation of reinforced concrete buildings affected by earthquakes. Master Thesis. Balıkesir University, Institute of Science and Technology, Balıkesir, Turkey, 235 s.

Goel R. K., Chopra A. K. Evaluation of modal and FEMA pushover analyses: SAC buildings. Earthquake Spectra, Vol. 20, Issue 1, 2004, p. 225-254.

Işık, E. and Velioğlu, E. (2017). A study on the consistency of different methods used in structural analysis. *Afyon Kocatepe University Journal of Science and Engineering Sciences,* 17(1), 1055-1065. https://www.doi.org/10.5578/fmbd.66120.

Karabulut, A. (2011). Comparison of nonlinear analysis methods defined in TBDY2007 regulation and FEMA 440 report for existing reinforced concrete buildings. Master Thesis. İstanbul Teknik University, Institute of Science and Technology, İstanbul, Turkey, 331 s.

Kayhan, A. H. and Demir, A. (2015). Comparison of maximum displacement demands obtained by nonlinear static and dynamic analysis in single degree of freedom systems. *Turkey Earthquake Engineering and Seismology third Conference*, İzmir, 14-16 October, 1-14.

Korkmaz, A., Uçar, T. (2005). Performance based analysis of reinforced concrete structures. *Earthquake Symposium*, Kocaeli, 23-25 Mart, 430-432.

Liu, S. W., Bai, R., Chan, S. L. 2016. Dynamic time-history elastic analysis of steel frames using one element per member. Structures, 8, 300-309.

Nobahar, E.; Farahi, M.; Mofid, M.: Quantification of seismic performance factors of the buildings consisting of disposable knee bracing frames. J. Constr. Steel Res. 124, 132–141 (2016).

Öncü, M. E. and Yön, S. M. (2016). Evaluation of the earthquake behavior of reinforced concrete buildings by the method of incremental dynamic analysis. *Dicle University Faculty of Engineering Journal of Engineering,* 7(1), 23-32.

SAP2000 V21.1.0, 2019: Structural Analysis Program, Computers and Structures Inc., Berkeley, California

TBEC-2018, (Turkish Building Earthquake Code), Specifications for buildings to be built in seismic areas. Ministry of Public Works and Settlement, Ankara, Turkey

Tuncer, Ö. (2008). Determination of earthquake performance of reinforced concrete structures by linear and nonlinear methods. Master Thesis. İstanbul Technical University, Institute of Science and Technology, İstanbul, Turkey, 300 s.

Yılmaz, C. (2008). Performance evaluation of an existing reinforced concrete structure with static push analysis. Master Thesis. İstanbul Technical University, Institute of Science and Technology, İstanbul, Turkey, 111 s.

Zhou, Y., Ge, P., Han, J., Lu, Z. 2017. Vector-valued intensity measures for incremental dynamic analysis. Soil Dynamics and Earthquake Engineering, 100, 380-388.

SCIENNOVATION
A Journal of Structural Science and Innovation

RESEARCH ARTICLE

Date Received: 25.022021
Date Accepted: 15.03.2021

A Simulation Technique for Controlled Demolition of Buildings

[1]Volkan TAVŞAN, [1]Ali GÜRBÜZ, [2]Temel TÜRKER

1Recep Tayyip Erdogan University, Department of Civil Engineering, Rize, Turkey.
2Karadeniz Technical University, Department of Civil Engineering, Trabzon, Turkey.
Corresponding Author: ali.gurbuz@erdogan.edu.tr

Keywords:
Controlled demolition
Simulation of demolition
Demolition techniques

Abstract

The fact that there are some structures, which is unlicensed or is vulnerable to or expire their lifetime, has led to the search for new demolition techniques in order to be carried out urban transformation studies faster and more effectively. Due to the many advantages of controlled demolition of buildings by explosives, which have been successfully applied for many years. When this method applied properly on structures, such as towers, chimneys, silos, bridges etc., it has wider application are at hant raditional demolition techniques due to lower cost and faster method. The most common problem inbuilding demolition by explosives is that the building does not collapse or collapse in the desired direction. The amount of charge, hole design, delay interval and blasting sequence must be determined correctly for the most accurate planning of controlled demolition of buildings. Simulation of the finite element model in the planning of his process is beneficial for the design of the demolition that is closest to reality. For this purpose, a simulation program has been prepared to analyze the structural behavior of the buildings which planned to be demolished by using explosives and to ensure that the demolition mechanism can be estimated as close to reality as possible. In this simulation program, demolition design of a typical reinforced concrete structure has been prepared.

1 INTRODUCTION

The buildings that have been completed in the past century need to be designed and renovated or demolished in accordance with engineering criteria for the different reasons (Gunes et al., 2019). Considering Turkey as an example, after the Van earthquake in 2011, necessary steps were taken to renew/demolish unauthorized or dangerous structures. Accordingly, 2012, on May 31, a new law (Number: 6306) called "Renewal of Areas at Risk of Disaster" was prepared and thus urban renewal projects were started throughout Turkey (Kentsel Dönüşüm, 2014). Due to the fact that a large part of the country is located within the first and second degree earthquake zone. It is predicted that approximately 6700000 houses will be demolished and rebuilt in the next 20 years, which means that an average of 334000 buildings per year must be demolished and rebuilt (Kentsel Dönüşüm, 2014).

The demolition of buildings by conventional methods is disturbing the environment because it

takes a long time; it also causes high costs and has unsafe working conditions (Özyurt et. al., 2016). The demolition of these structures by traditional methods by inexperienced and uninformed people leads to dangerous consequences. Therefore, a more useful technique was explored and as a result of the studies which started on the basis of the British Standard titled "BS 6187:2011 Code of Practice for Full and Partial Demolition", the Turkish Standard titled "TS 13633" was published in October 2014.

The construction demolition technique by explosives has advantages such as fast application, low cost in high-rise buildings, minimising work accidents, limiting and controlling the impact on the environment over a short period of time (C. Jimeno, E. Jimeno and Carceda, 1995; Koca,2006). However, the construction demolition technique by explosives has disadvantages such as the possibility of damage to nearby buildings, failure to completely demolish the structure, failure to perform the demolition in the desired direction (Tavsan et al., 2020). Demolition planning should be carefully prepared in order to avoid an undesirable situation in building demolition by explosives, blasting demolition work should be carried out by people that have experience and necessary simulations should be carried out in order to predict realistically the demolitionmechanism.

The correct preparation of demolition planning and the investigation of structural behavior is very important for the successful implementation of explosive demolition method and there are various studies in this area (Koca, 2006).

Although there are quite developed professional softwares for controlled demolition of structures by explosives, due to high price, operations are usually performed depending on experience. As a result undesirable situations have been occured.

The method proposed in this study is based on the carrier elements to be detonated on the finite element model are selected and erased. Thus evaluation is possible the deformation shape of the structure. As a result of the evaluation, if the demolition direction is in an inappropriate direction, the demolition process is returned to the beginning and the process is repeated. It is very important that these processes, which take a lot of time, can be performed more effectively and quickly. In this study, in order to prevent time loss and to determine the demolition direction of the structure realistically, in the simulation program created by enabling MATLAB and SAP2000 to work together, the carrier elements to be detonated on the finite element model are selected and erased, then the deformation shape of the structure is examined step by step. In this way, the order of the carrier elements that must be detonated in order to collapse the structure in the intended direction can be determined and as a result the most suitable demolition design can be obtained. If the demolition direction of the simulated system in the program that is created is not as desired , the process steps can be taken back and a new design can be created. As a result of this, the design of the detonation that will allow the structure to fall in the desired direction can be presented.

2. COMPUTER MODEL OF SELECTED BUILDING

In the simulation program, the demolition of a selected typical structure was simulated. The chosen structure is a 10-storey reinforced concrete building that has an axle length of 4.5x5m. The building concrete class was assumed to be C25, the steel class is S420, all beams are 30x60 cm and all columns are 35x35 cm. The formation of plastic hinges in the structure has been neglected during the blasting period as the sudden collapse condition is desired to occur in the demolition design. Horizontal cumulative displacements are taken as a basis in the design of the demolition. The largest horizontal displacement value was determined by considering the relative storey displacement rate. In order to be analyzed controlled demolition by blasting in a computer environment, a structure model must be created by any finite element program (Yılmaz, 2006; Özyurt et. al., 2016; Özmen et. al., 2017). In this study, the structure modeled by SAP2000 program and the floor plan of the structure are given in Figure 1.

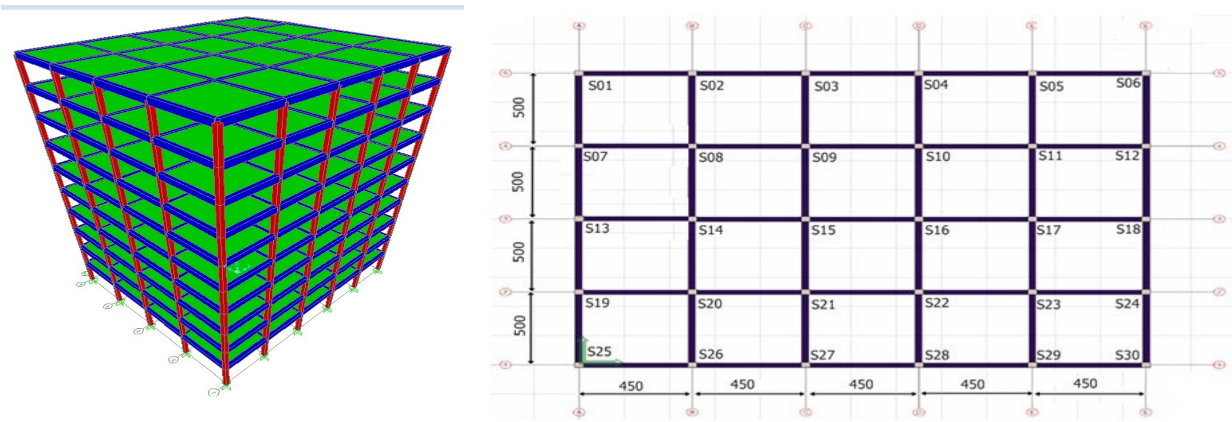


Figure 1. Floor plan and 3D model of the structure to be demolished

3. DEVELOPED SIMULATION PROGRAM

the program purposed in this article is created by enabling MATLAB and SAP2000 to work together. The steps for the demolition simulation program developed are presented in the algorithm of Figure 2.

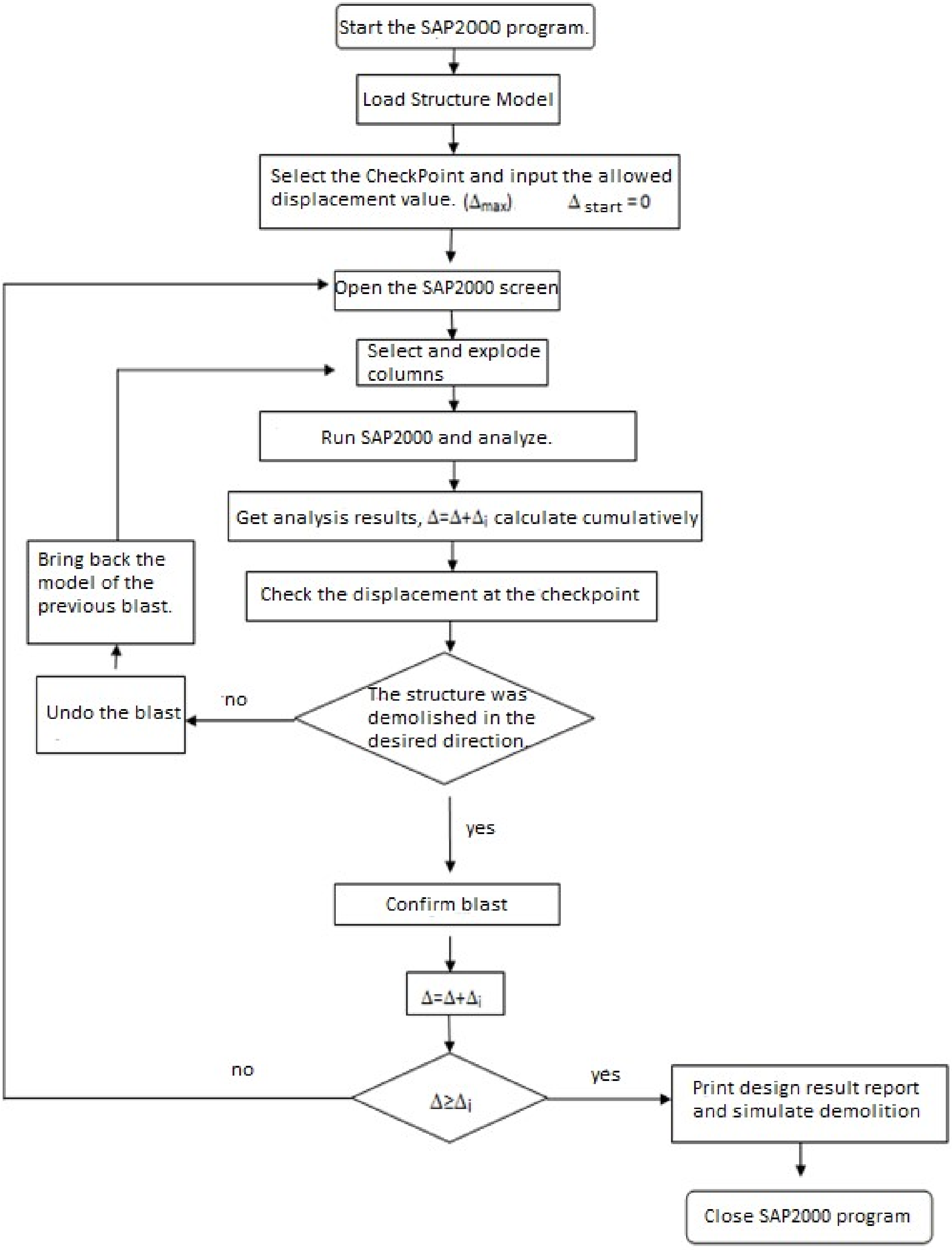


Figure 2. Algorithm of simulation program

In the first step, Sap2000 model of the structure is created ordinary. In controlled demolition of any structure, the carrier elements must be detonated in a certain order to be determined the direction of demolition.Inorder to be determined this order, the analysis must be performed step by step as elements are removed from the structure. This simulation program that works together with MATLAB program was created for the step-by-step analysis of the controlled demolition of a structure (Figure3

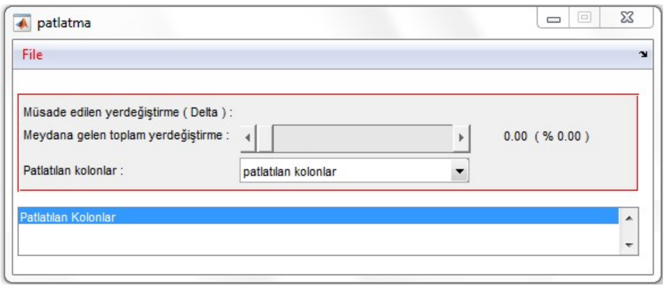


Figure 3. Interface of simulation program

Second step is transfer of 3d model of the example structure to the simulation program. For detailed analysis of the cumulative deformation of the structure model, in the simulation program, by being clicked on the ‘Open_SAP2000_sdb_file’ button in the ‘File’ tab, SAP2000 program is automatically opened in the background. In the ‘Select the sdb file’ section, the structure model was named “model_1” or etc. Transfer of structure model to the simulation program is shown on figure 4.

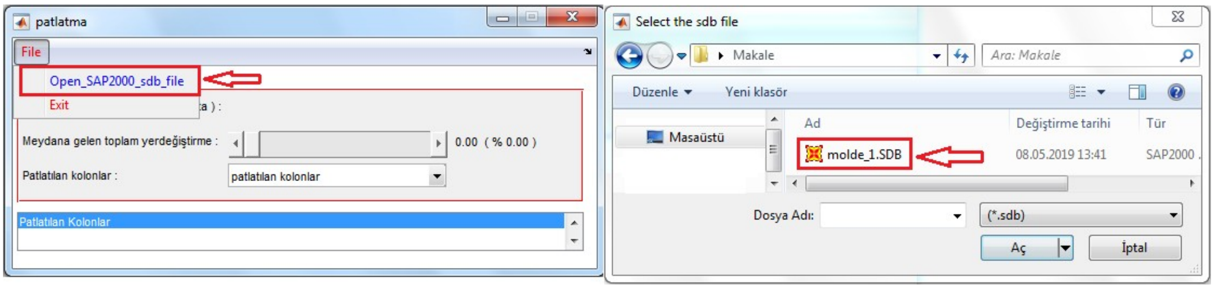


Figure 4. Transfer of structure model to the simulation program

In this transfer process, the data file of the structure is transferred to the MATLAB program, so every change made in this data file is automatically saved to the MATLAB program.

In third step; SAP2000 program calculates the value of the deformation that occurs in the demolition of the structure model, can not give information about whether the structure will collapse or not. Therefore, in order to be determined whether the structure will collapse or not, a different control point is selected for each structure on the program and the displacement value at this checkpoint is compared to the maximum displacement value assigned to the program. In this way it can be decided whether the structure has collapsed or not. For this example structure, the maximum horizontal displacement at point 303 is considered as 120cm.

In the structure, click on the ‘Explode Column’ button and “confirm detonation” button to be determined the order of the bearing elements to be detonated and the SAP2000 program is opened automatically.

According to the intended demolition direction, the user decides which carrier elements to delete in the structure model in the SAP2000 program.

In the next step, if the structure tends to demolish in the desired direction but the deformation value (in this example; 0.0030221 m) does not exceed the maximum deformation value (in this example 1.2 m), a second blasting design is needed.

In the second blasting design, the structure model formed by the first detonation is taken into account (Figure 14). Then, the carrier elements are selected and deleted in the program according to the order determined for the second time (Figure 17&Figure 18).Inthe SAP2000 program, the final structure model is analyzed.

In the next step, if the structure tends to demolish in the desired direction but the deformation value (in this example; 0.003+0.0135=0.0165 m) does not exceed the maximum deformation value (in this example; 1.2 m), a third blasting design is needed. The analysis is continued until sufficient carrier elements are deleted.

If the structure does not collapse in the desired direction in the simulation, the structure model is automatically restored to its previous state by being clicked on the ‘Undo explosion’ button in the interface program. In this step, the model must be restored to the previous one. Because the structure tends towards the Y-axis and this situation is not desired. Then a different blasting design must be created.

For the controlled demolition of the example structure, the carrier elements were selected and deleted in the program according to the order determined for the fifth time. In fifth blasting, the structure collapsed to the desired direction and the deformation value (3.6865 m) exceeds the maximum deformation value (1.2 m). The last step of demolition design is shown on figure5. Finally, this design can be used as a “demolition design” (Figure26).

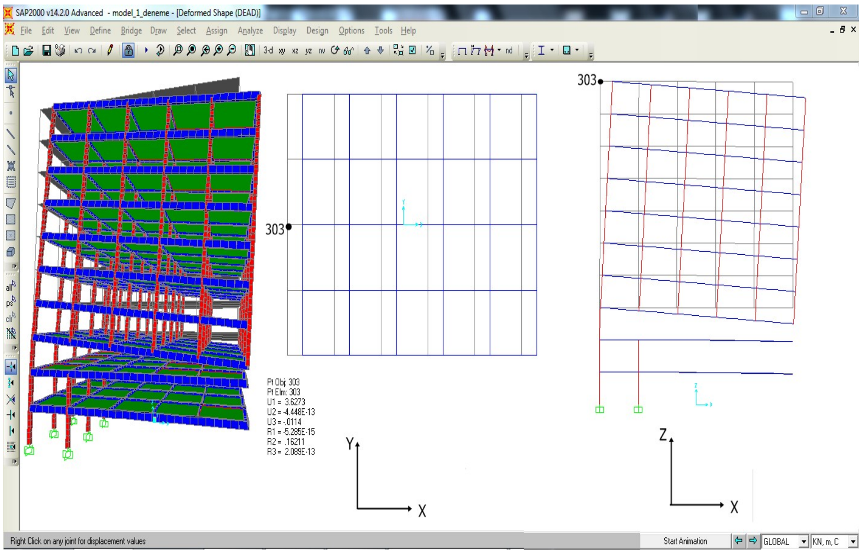


Figure 5. Analysis results of the last blasting design

After the suitable demolition design is obtained, the blasting process is stopped and the design report is prepared automatically by the program (Figure 6).

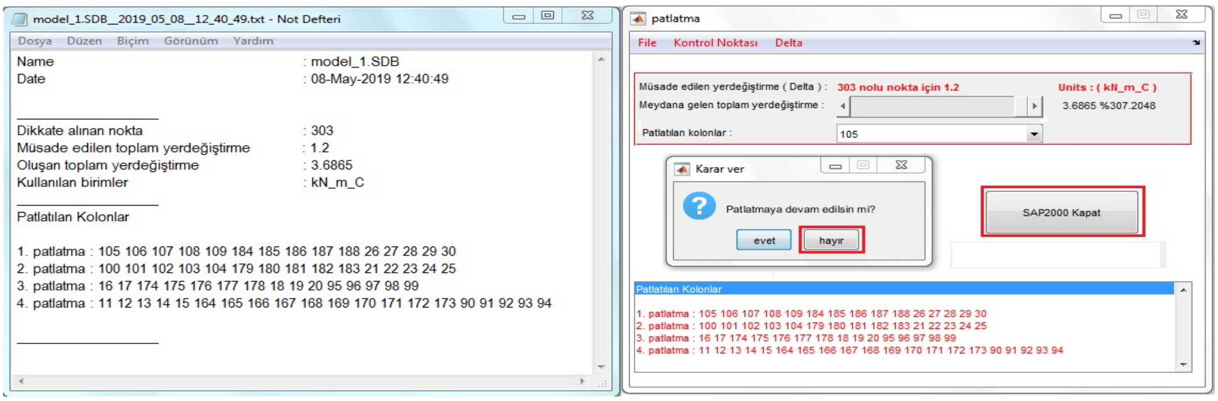


Figure 6. Finalizing the demolition design and design report

By being clicked on the “No” button in the ’decide‘ window, the data of the demolished structure is automatically saved in txt format along with the analysis date and time in the folder named ’Blasting reports'. Finally, the "Design results’ is opened and the number and order of the bearing elements to be detonated is determined as shown in Figure 6 also.

4. CONCLUSION

In this study, a simulation program that based on structural analyses that performed on finite element models was created in order to prevent to demolished in undesired directions, the most important problem faced in the demolition of structures by explosives. The applicability of this program is demonstrated by being designed demolition of a selected typical reinforced concrete structure in the desired direction. The following results were obtained from this study:

- It has been revealed that controlled demolition design of structures by explosives can be made more effectively with structural assessments that based on finite element analyses.
- It has been shown that the demolition of the selected reinforced concrete building can be carried out in the desired direction with this simulation program.
- Thanks to the report generated by the simulation program, the order of the elements to be detonated in demolition was determined.
- Because the demolition is aimed to be carried out in the most economical way, a blasting design which includes blasting a minimum number of structural elements has been revealed.

Demolition design by explosives is a applicable approach for structures such as buildings, towers, bridges. It is thought that blasting demolition can be done more effectively, if this approach and simulation program which is created by running matlab and sap2000 programs together.

REFERENCES

Afet Riski Altındaki Alanların Dönüştürülmesi Hakkında Kanun [Law on Transformation of Areas at Disaster Risk]. (2012, 31 May).*Resmi Gazete*(Sayı:6306). Retrieved from https://www.resmigazete.gov.tr/eskiler/2012/05/20120531-1.htm

Bahadır, F., & Açıkel, H., (2009, May). Controlled demolition of a reinforced concrete industrial structure by explosives in a computer environment. 5. Uluslararası İleri Teknolojiler Sempozyumu (IATS’09), Karabük, Turkey. Retrieved from https://docplayer.biz.tr/17458824-Patlayicilar-le-kontrollu-yikimi.html

Doğan, E., Uzal, B., Pehlivanoğlu, K. and İycil, E. (2009). Patlayıcı kullanılarak betonarme bir su kulesi yıkımı [Demolition of a reinforced concrete water tower using explosives]. Türkiye Mühendislik Haberleri (TMH)–457, pp. 35-44. Retrieved fromhttp://www.imo.org.tr/resimler/dosya_ekler/ec667c928319eed_ek.pdf?dergi=141

Erkoç, O.Y., Sunu, M.Z., Aldaş, G.G.U. and Özkazanç, M.O. (2001). Patlayıcı madde kullanarak su deposu yıkımı [Demolition of water tank using explosives]. Türkiye 17. Uluslararası

Madencilik Kongresi ve Sergisi, Turkey, pp. 23-27. Retrieved fromhttp://www.maden.org.tr/resimler/ekler/31cc28f8747a032_ek.pdf

Gunes, B., Cosgun, T., Sayin, B., Mangir, A. (2019). Seismic performance of an existing low-rise RC building considering the addition of a new storey. Revista de la Construcción. Journal of Construction, Vol 18, No 3, pp. 459-475. DOI: 10.7764/RDLC.18.3.459

Jimeno, C.L., Jimeno, E.L. and Carceda, F.J.A. (1995). *Drilling and blasting of rocks (pp.312)*, Netherlands: A. A. Balkema Publisher.

Kentsel Dönüşüm Türkiye (2014) [Urban Transformation, Turkey], Cushman ve Wakefield , Turkey.

MATLAB: MathworksInc, Computer Program, Natick, MA, 1999.

Koca, O. (2006). Patlayıcı maddelerle kontrollü yapı yıkımı [Controlled structure demolition with explosive materials]. Master Thesis (Thesis number: 179096). Obtained from YÖK National Thesis Center database.

Oloffson, S.O. (1980). Applied Explosives Technology for Construction and Mining, pp. 268- 277.

Özmen, H., Soyluk, K. and Anıl, Ö. (2017). Betonarme binaların patlayıcı kullanılarak yıkımında yapı davranışının analizi [Analysis of structural behavior in demolition of reinforced concrete structures using explosives]. 4.Uluslararası Deprem Mühendisliği ve Sismoloji Konferansı, Eskişehir,T u r k e y .

Özyurt, M.C. (2013). Patlayıcı madde kullanılarak yapıların kontrollü yıkılması ve verimliliğinin incelenmesi [Controlled demolition of structures using explosive materials and research of their efficiency]. Master Thesis (Thesis number:332340). Department Of Mining Engineering, İstanbul University, İstanbul, Turkey..

Özyurt, M.C., Özer, Ü., Karadoğan, A. and Kalaycı, Ü. (2016). Betonarme bir binanın patlayıcı ile yıkılması ve veriminin incelenmesi [Controlled demolition of structures using explosive materials and research of their efficiency]. Uludağ University, Mühendislik Fakültesi Dergisi, c. 21, n. 2, pp. 44.

SAP2000, Computer Program, Computers and StructuresInc, 2008.

Sikiwat, T., Breidt, M. and Hartmann, D. (2009). Computational Steering for Collapse Simulation of Large Scale Complex Structures, 18th International Conference on the Application of Computer Science and Mathematics in Architecture and Civil Engineering, Weimar, Germany, pp. 1-8.

Tavsan, V., Gurbuz, A. (2020). Controlled Demolition Techniques and Demolition Direction, Sciennovation, 1(2), pp. 1-10. ISSN 2687-377X.

Yılmaz, B. (2006). Betonarme yapıların onarımı ve güçlendirilmesi/güçlendirmenin ekonomik olmaması durumunda patlayıcı madde kullanılarak kontrollü yıkımı [Repair and strengthening of reinforced concrete structures/controlled demolition by using explosive materials in case the strengthening is not economical]. Master Thesis (Thesis Number: 183908). Department Of Civ

SCIENNOVATION
A Journal of Structural Science and Innovation

REVIEW ARTICLE

Date Received: 28.01.2021

Date Accepted: 12.03.2021

A Review of Concrete-Filled Steel Tube Elements

Sena KURT[1]

Chamber of civil engineers, freelance civil engineer, Trabzon, Turkey
Corresponding Author email: senavkurt@gmail.com

Keywords:

Concrete-filled steel tube
Special-shaped column
Flexure strength
Finite element modeling

Abstract

Structural elements such as composite columns and beams filled with concrete compared to reinforced concrete elements, maintenance-free concrete inside the pipe, being able to pour concrete without molds, having high rigidity and strength, increasing the strength of the concrete inside the pipe by limiting the lateral deformations and pressure elements part of it has advantages such as increasing the local buckling strength. Steel columns filled with concrete and reinforced concrete inside the USA, Japan, France, England, Azerbaijan and some other countries; widely used in high-rise buildings, bridge and pier piers, oil extraction platforms and various other engineering structures.

Beton Dolgulu Çelik Boru Elemanlarının Gözden Geçirilmesi

Anahtar Kelimeler:

Beton dolgulu çelik boru
Özel şekilli kolon
Eksenel mukavemet
Eğilme mukavemeti

Özet

Yapı elemanlarının betonarme elemanlara kıyasla dayanım-ağırlık oranının yüksek olması, boru içerisindeki betona bakım gerektirmemesi, kalıpsız beton dökülebilmesi, yüksek rijitlik ve dayanıma sahip olması, boru içerisindeki betonun yanal deformasyonlarının sınırlandırılmasıyla dayanımının yükselmesi ve basınç elemanlarının boru kısmının yerel burkulma dayanımının yükselmesi gibi üstünlükleri bulunmaktadır . İçerisi beton ve betonarme ile doldurulmuş çelik kolonlar ABD, Japonya, Fransa, İngiltere, Azerbaycan ve diğer bazı ülkelerde ; yüksek yapılarda, köprü ve iskele ayaklarında, petrol çıkartma platformlarında ve diğer çeşitli mühendislik yapılarında yaygın olarak kullanılmaktadır.

1 INTRODUCTION

Structural engineering is one of the leading sectors of both the ancient times and the times we live in. The field, which has been the subject of many studies since ancient times, has developed over time. This development is seen both in the construction market to meet the needs and in academic studies. Different building examples seen in the world and in our country are the products of this development. Steel and concrete are two indispensable building materials for structural engineering. They are used in building construction in different techniques, sometimes alone or together. One of these techniques is concrete filled steel structures (CFSS). CFSS, which is especially preferred in high buildings, has high strength. Strength / weight ratios of structural elements such as composite columns and beams filled with concrete are higher than reinforced concrete elements. Concrete inside the steel pipe does not require maintenance. Concrete can be poured without mold. It has high rigidity and strength. Lateral deformations of the concrete in the steel pipe are limited. Buckling strength increases. Steel columns filled with concrete and reinforced concrete are widely used in high-rise buildings, bridge and pier piers, oil extraction platforms and various other engineering structures in the USA, Japan, France, England, Azerbaijan and some other countries. It is necessary to know the mechanical properties of the concrete in the pipe in order to calculate the strength, buckling and local buckling control of such building elements (Eyyubov and Adıbelli, 2000).

Concrete filled steel pipes are often coded as CFST in the literature. This construction method is highly preferred in recent years. Many researches, experimental studies and theoretical studies have been done on CFST. Many of the results of these studies have positive impressions (Huang et.al., 2016; Dong et.al., 2017; Hai-Tingand Ben Young, 2018).

Figure 1. Building examples made within the method (AYDIN C., 2008)

The emergence of concrete-filled steel structures begins with the construction of very high-rise steel structures. Especially in the 1950s, methods for the calculation and design of these structures began to be developed (Kloppel et. Al., 1957). Numerous experimental and theoretical studies have been developed to make concrete and steel perform better together. New calculation methods have been introduced with each passing day (GardnerandJacobson, 1967; Knowles, and Park1969). In 1979, the British Standards Institute published a standard named "Concrete and Composite Bridges" (BSI, 1979). Thus, the principles of concrete filled steel structure design have started to be included in official standards. However, the 90s were the golden age of CFST (LukshaandNesterovich, 1991). It has become a regularly applied method. At this time, Europe, America and Japan almost competed to build skyscrapers. This construction technique has gained more rapid acceptance in these countries. With the spread of the system in the market, regulations for Eurocad 4, (1994) in Europe, AIJ, (1997) in Japan, and AISC-LRFD (1999) in America were developed. Extensive research and method development efforts on CFST continued in the 2000s. (LamandWong, 2005) If we look at the world-wide, studies have been made on parameters by forming the basis for this method. This method, which has developed very rapidly in the construction sector, is still widely used today (Krishan et al. 2016). Today, researches on the method continue to gain momentum (Moon et al., 2013).

In recent years, research on the seismic performance of CFST elements has gained momentum (Zhu et. Al., 2018; Buiand Kim, 2021; Liu et al., 2021). The history of concrete-filled steel structures does not date back to ancient times. Large-scale earthquakes in our country triggered this initiative. The increasing need for housing after the 17 August 1999 Gölcük Earthquake played a role in the development of this method.

Figure 2. Examples from around the world (AYDIN C., 2008)

2. ADVANTAGES AND DISADVANTAGES OF CFST

As with every method, there are advantages and disadvantages in this method. Considering all these pros and cons, the optimum path should be followed for construction. If we look at the advantages of concrete filled steel structures, we can explain as follows.

The steel wall surrounds the concrete core in a spiral fashion, allowing triaxial stress to form around the core. The concrete layer formed around the spiral increases the carrying capacity and ductility. In addition, when the steel tube is under extreme loads, it acts as a protector that prevents the concrete core from being destroyed and broken. Steel pipes reduce mold costs as they act like molds during manufacturing. The fact that steel tubes can be made in several layers allows the construction process to continue without waiting for the concreting process. Work time and labor costs are reduced in multi-storey buildings. Because the concrete core prevents local buckling in CFST elements, thinner-walled sections are used compared to steel elements. In the example given in Figure 3, the use of CFST is seen on a bridge pier with a high height and payload.

Figure 3. Pillar column (AYDIN C., 2008)

It is very difficult and costly to construct the basement floor with steel elements in multi-storey high-rise buildings built using only steel. For this reason, basements are generally made of reinforced concrete and are mounted on a steel construction reinforced concrete structure. This situation may cause the formation of very large column sections. This problem can be solved thanks to the high ductility feature of composite structures. Thus, it has become possible to construct high-rise structures with less cost. Famous and special structures such as the Twin Towers of the USA and Petronas Towers of Malaysia, built with this system in the world, have gained the reputation of being the symbolic structures of their countries. Almost all of the tall structures in the world are designed and built as composite. Composite floors are a good alternative as a flooring system for steel structures. Composite structural elements provide better protection of steel by resisting corrosion. Such elements are preferred in structures where high strength, total weight of the building, climatic and geographical environment are compelling factors and chemical effects are important. Safe and economical design of composite elements is not only about sizing but also about economy.

In addition to the relevant standards in the design of composite elements, complying with the recommendations of steel sheet and slip connection manufacturers will also reduce the errors and construction costs that may occur during production (Yorgun, 2005).

If we look at the disadvantages, we can list it as follows. Using two different types of materials together creates calculation difficulties due to the different behavior of these materials. Re-checks in account transactions should be strictly made. Calculating the geometrical moment of inertia in moment calculation presents difficulties. There is limited information about the force transfer and adherence at the contact surface of the steel and concrete being used. Multiple factors need to be analyzed, structurally, behaviorally and environmentally. It is also a disadvantage that the calculation and design part takes a long time. While calculating the modulus of elasticity of the composite joint, we will face some drawbacks and difficulties. In general, successful results are obtained with a good planning and engineering considering these pros and cons.

3. CASE STUDY

High-rise buildings designed with a traditional approach are mass consumers of energy and natural resources. However, during the 120 years that have passed since the first high-rise buildings, models and methods have been developed to reduce these environmental impacts, changes have been made in laws and regulations, and innovations have been made in architectural thinking as well as in materials and technology (Sev at all.2011). It is possible to see many structures related to the subject in the world and in our country. If we first look at examples from around the world, we can list the following: Canton Tower in China Guanzhong, Ganhaizi Bridge in China, Xiangjiaba Bridge and Zidong Bridge are examples to be given. As can be seen from the examples, we can observe these structures widely in China. Wushan, Zhijing River, Maocaolie, Taiping Lake Bridge are among the examples that can be given. The following examples are Petronas Tower, Sail Singapore, World Financial Center and Sky Tree for the system commonly used in high-rise buildings.

Figure 4. Ganhaizi Bridge, China (Web source-1).

Figure 5. Canton Tower, Çin (Web Source-1).

Figure 6. Petronas Tower (Web Kaynağı-2)

One of the best examples in our country of CFST type structures, which are widely preferred in high-rise buildings in the world, is the Nurol Life building. It said structure 2 is shown as Turkey's highest building is built with construction techniques such as CFSTR world examples.

Figure 7. Nurol Life Prejesi’nin tamamlanmış hali (Web Kaynağı-3)

4. CONCLUSION

We had the opportunity to observe many different engineering structures within the scope of this study. However, we could access limited information about the combination elements of these structures. We examined the advantages and disadvantages of the CFST construction style. We concluded that civil engineering is a very successful method in terms of safety, aesthetics and economy, which are the three basic elements of civil engineering, especially for high-rise buildings. Of course, good planning plays a key role in this success.

In today's world where the increase in the number of residences and the demand for tall buildings is increasing day by day, this application is an alternative to meet the needs. CFST elements can serve as a savior for bridges as well as building-type structures in situations where geographical conditions are unfavorable. It is possible to observe many similar examples. Concrete filled steel structures are at the top level in terms of convenience and frequency of use in composite systems. We believe that this demand will continue to increase depending on the population increase.

References

AIJ (1997),"Recommendationsfordesignandconstruction of concretefilledsteeltubularstructures", ArchitecturalInstitute of Japan, Tokyo, Japan.

AISC-LRFD (1999), LoadandResistanceFactor Design SpecificationforStructural Steel Buildings, 2nd ed., AmericanInstitute of Steel Construction (AISC), Chicago, U.S.A.

Aydın, Ç. (2008). Beton Dolgulu Çelik Kutu Kolonlar Ve Dolu Gövdeli Çelik (I) Kirişlerde Oluşan Çerçevelerin Yapı Davranış Katsayısı (R) İrdelenmesi, Yüksek Lisans Tezi.

BSI, (1979). British Standard Institute, BS5400, Part 5, ConcreteandCompositeBridges, London, U.K.

Bui, V.T.; Kim, S.E (2021).Nonlinearinelasticanalysis of semi-rigidsteelframeswithcircularconcrete-filledsteeltubularcolumns,InternationalJournal of MechanicalSciences,

Dong, C.X, Kwan, A.K.H, Ho, J.C.M. (2017). Effects of externalconfinement on structuralperformance of concrete-filledsteeltubes. J Constr Steel Res; 132: 72–82.

Eurocode 4 (1994),"Design of compositesteelandconcretestructures", Part1.1: General rulesandrulesforbuildings (togetherwith United KingdomNational Application Document). DD ENV 1994-1-1:1994, British StandardsInstitution, London W1A2BS.

Eyyubov, C., Adıbelli, H. (2000). İçerisi Beton İle Doldurulmuş Çelik Boru Yapı Elemanlarının Dayanımının Araştırılması, Erciyes Üniv., Müh. Fak. İnşaat Müh.Böl., Kayseri-Türkiye.

Gardner, J. andJacobson, R. (1967),"Structuralbehaviour of concretefilledsteeltubes", ACI J., 64(7), 404-413.

Huang, L, Yin, P, Yan, L. Behavior of hybrid GFRP–perforated-steeltube-encasedconcretecolumnunderuniaxialcompression. ComposStruct 2016; 142: 313–324.

İnce, G , İnce, H , Kaya, F . (2015). Kompozit Yapı Sistemlerinin İncelenmesi . Mehmet Akif Ersoy Üniversitesi Fen Bilimleri Enstitüsü Dergisi , 6 (1) , 43-47 . Retrieved from https://dergipark.org.tr/tr/pub/makufebed/issue/19417/206601

Kloppel, V.K. andGoder, W. (1957),"An investigation of theloadcarryingcapacity of concrete-filledsteeltubesanddevelopment of designformula", Der Stahlbau, 26(1), 1-10.

Krishan, A.L., Chernyshova, E.P., Sabirov, R.R (2016). CalculatingtheStrength of ConcreteFilled Steel TubeColumns of Solid and Ring Cross-section, ProcediaEngineering,Volume 150,Pages 1878-1884,

Knowles, R.B. and Park, R. (1969),"Strength of concretefilledsteeltubularcolumns", J. Struct. Div., ASCE, 95(ST12), 2565-2587.

Kvedaras, AK. (1999). Journal of Constructıonal Steel Research, Volume: 49 Issue: 2 Pages: 197-212.

Lam, D.; Wong, K. (2005). AxialCapacity of ConcreteFilledStainless Steel ColumnsStructuresCongress,pp. 1-11, 10.1061/40753(171)105

Liu, H.; Hao, J.; Xue, Q.; Sun, X (2021). Seismicperformance of a wall-typeconcrete-filledsteeltubularcolumnwith a doubleside-plate I-beamconnection,Thin-WalledStructures,Volume 159,

Luksha, L.K. andNesterovich, A.P. (1991),"Strengthtesting of larger-diameterconcretefilledsteeltubularmembers", Proc. Third Int. Conf. on Steel-ConcreteCompositeStructures, ASCCS, Fukuoka, Japan, 67-70.

Moon, J.; Lehman, D.E.; Roeder, C. W.; Lee, H.E (2013).Strength of circularconcrete-filledtubeswithandwithoutinternalreinforcementundercombinedloading, J. Struct. Eng.Volume 139 Issue 12

Schneider, S.P. (1998),"Axiallyloadedconcrete-filledsteeltubes", J. Struct. Eng., ASCE, 124(10), 1125-1138. https://doi.org/10.1061/(ASCE)0733-9445(1998)124:10(1125)

Sev, A. (2011). Geçmişten Geleceğe Enerji Etkin Yüksek Yapılar ve Uygulama Örnekleri. X. Ulusal Tesisat Mühendisliği Kongresi – 13/16 Nisan 2011/İzmir.

Yorgun, C. (2005). Çelik Sac-Beton Kompozit Döşeme Sistemlerinin Uygulamalarına Yönelik Değerlendirmeler. TMH - Türkiye Mühendislik Haberleri Sayı 435.

Zhu, C. Y.; Zhao, Y. H.; Sun, L. (2018). Seismicperformance of FRP-reinforcedconcrete-filledthin-walledsteeltubeconsideringlocalbuckling vol.37, pp.9,

WEB KAYNAĞI 1 https://www.slideshare.net/HarikrishnaMS1/concrete-filled-steel-tubes.

WEB KAYNAĞI 2https://en.wikipedia.org/wiki/Petronas_Towers.

WEB KAYNAĞI 3 https://www.nurolgyo.com/tr/proje/nurol-life.html.

SCIENNOVATION

A Journal on Structural Science and Innovation

ISSN: 2687-377X