Finite Element Analysis of Steel Structure Elements with Ansys Workbench

*1 Koken, A.
[1]Konya Teknik Üniversitesi,
Mühendislik ve Doğa Bilimleri Fakültesi,
Konya, Turkey
E-mail:akoken@ktun.edu.tr

[2]Kurt, E.[2]
[2]Konya Teknik Üniversitesi,
Mühendislik ve Doğa Bilimleri Fakültesi,
Konya, Turkey
E-mail: ertugrul4170@gmail.com

***Abstract*— There are many analytical solution methods used in the analysis of structural systems. With the developed analytical solution methods, it is possible to make faster and real solutions. The ANYSS Workbench program is one of the most important programs based on the finite element method and recently used in the analysis of structural systems. With this program, internal forces, stress-displacements and moment-rotation relationship in elements and joints results can be obtained.**

In this study, analytical solution of steel tension and compression elements has been realized by using ANYSS Workbench program. The stresses and displacements in steel elements subjected to axial tension and compression were obtained both theoretically and using the ANYSS Workbench program. Obtained the results were compared and interpreted.

Keywords—ANSYS Workbench, Steel Structure Elements, Stress analysis, Finite Element Analysis (FEA).

I. Introduction

Steel is a material that can be used in a wide range of areas. The most used area is the area of steel structures. Steel structures are used in industrial buildings, commercial buildings, bridges, residences and many other types of structures today. Structural engineers should select the most appropriate section by analyzing the structural elements before choosing the structural element section to make a safe and economical design.

Various analytical methods have been developed in the analysis of structural systems. Finite element method is one of the most widely used of these methods in recent years. The behavior of structural elements under various loads, internal force distribution and displacements can be obtained with the finite element method. ANYSS Workbench application is a computer application based on finite element method [1].

During the last decade, sophisticated numerical analysis for the design of members has become more and more popular. In addition, in order to limit time consuming and expensive laboratory tests researchers are often tented to base developed design models on exhaustive numerical parametric studies. However, in order to simulate reliably the behavior of structural elements, it is crucial to include realistic assumptions for geometric and material imperfections. If these imperfections are not taken into account, the ultimate resistances may be heavily overestimated. Obviously, the physical imperfections are of random nature and cannot directly be represented in practice [2]

The influence of geometric imperfections on the member resistance has been investigated first by Koiter in reference [3] from a theoretical point of view in the '40s of the 20th century. Beginning from that point, many other studies were conducted on the imperfection sensitivity of shell structures (see for example references [4–6]) and steel members (see for example references [7–8]).

The compressive performance of welded steel sections, including the local buckling and overall buckling of both normal [9–12] and high strength [13-14] steel compression members has been studied, and relevant design methods have been proposed.

The finite element code, ANSYS, is developed to perform elastic buckling analysis and predict critical loads for all tested specimens. Three-dimensional nonlinear finite element analysis results of these steel cellular beams are then compared with the experimental results. The tested cellular beam specimens are designed by using one of the stochastic search techniques called harmony search optimization method [15,16].

II. MATERIAL METHOD

ANSYS program using finite element method; It is a computer-aided engineering program that can be used for modeling and analysis in engineering studies. With the finite element method, problems in complex geometry, which are very difficult to analyze in one piece, can be analyzed separately by dividing them into small and many pieces. A single and consistent analysis result can be obtained by combining the results obtained from the analysis of a finite number of elements.

Linear and nonlinear buckling analyzes can be performed as well as static analysis on elements and connections with the ANSYS program. The shape changes and stress distributions resulting from the force applied to the model can be instantly displayed at the end of the analysis.

The basic process steps in analysis in ANYS Program are as follows;

1. Selecting the analysis system in the ANSYS workbench popup window
2. Introducing the material and properties to be used in the ANSYS workbench.
3. Designing and modeling geometry in ANSYS workbench
4. Meshing process of elements in ANSYS workbench
5. Determination of boundary conditions and loads in ANSYS workbench
6. Analyzing and viewing the results in ANSYS workbench.

III. ANALYSIS OF STEEL STRUCTURE ELEMENTS WITH ANSYS

In this study, analysis of two steel elements subjected to axial tensile and compressive forces was carried out with ANSYS program. In this context, two steel elements were modeled in the ANSYS workbench program, the material properties were defined, the appropriate finite element network was created, the boundary conditions were defined and the ANSYS solution was obtained by performing system analysis. In addition, the theoretical solution of both elements has been made and these two results are compared.

Only axial tensile and compressive forces occur in the members of the truss systems where external loads directly affect the nodes. The force that causes elongation in the element is called the axial tensile force (+) and the force that causes the shortening in the element is called the compressive force (-) (Figure 1).

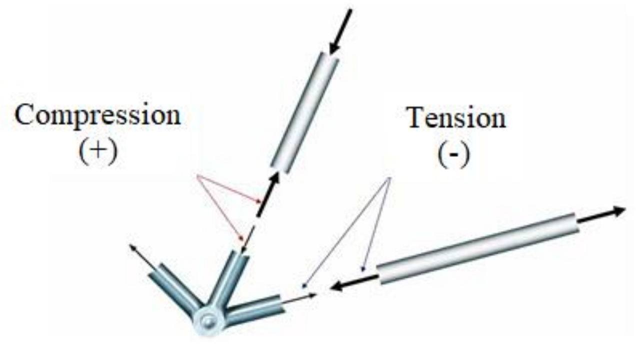

Figure 1. Representation of tensile and compressive forces

Both theoretical and finite element analysis of the IPE270 section steel element loaded with axial tensile force has been performed with ANSYS Workbench program. The two results obtained were then compared.

ELEMENT DIMENSIONS AND LOADING

The 2000 mm long IPE270 section steel element shown in Figure 2 is fixed at one end and loaded with a tensile force of 80 kN in the longitudinal direction from the other end. Steel grade is S235. The tensile stress and elongation amount of the steel element were calculated both theoretically and by using different mesh intervals with the ANSYS Workbench program and the results were compared.

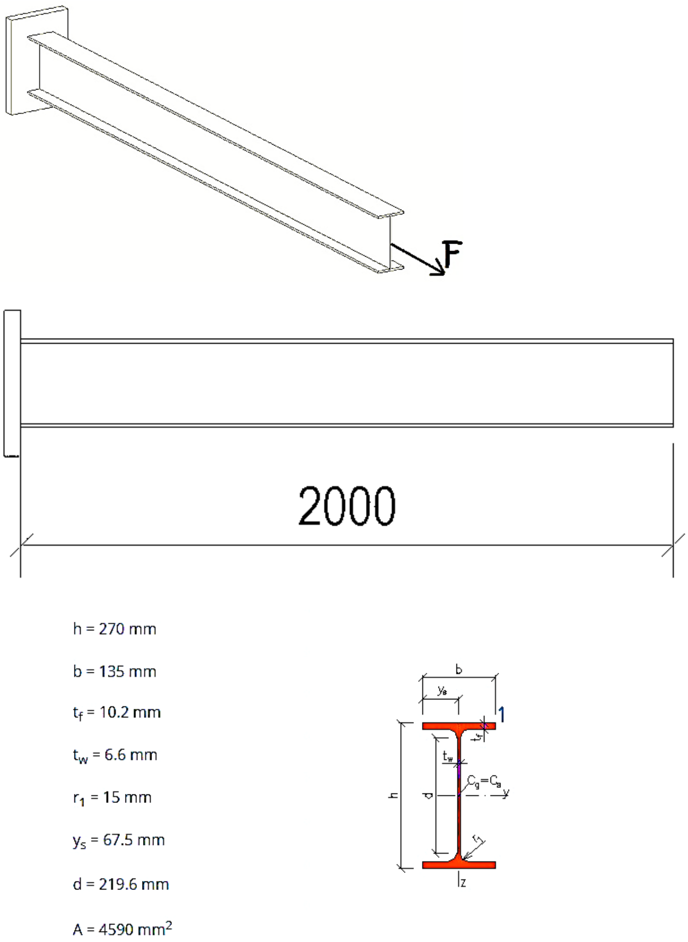

Figure 2 Properties of sectional steel tension elements IPE270

THEORETİCAL SOLUTİON TO THE PROBLEM

The theoretical tensile stress at the IPE270 cross section is calculated below.

$$\sigma_t = \frac{F}{A} \qquad (1)$$

$$F = 80000\ N \qquad A = 4590\ mm^2$$

$$\sigma_t = \frac{F}{A} = \frac{80000\ N}{4590\ mm^2} = 17{,}429 \frac{N}{mm^2}\ (MPa)$$

Where:
F: Axial tensile force
A: Cross section area
σ_t: Tensile stress

The amount of elongation in the IPE270 cross section in the longitudinal direction

$$\Delta l = \frac{F.L}{E.A} \qquad (2)$$

$$E = 200000\ MPa \qquad L = 2000\ mm$$

$$\Delta l = \frac{80000\ N\ .2000\ mm}{200000\ N/mm^2.\ 4590\ mm^2} = 0{,}174\ mm$$

Where:
Δl: Lengthening amount
L: Length
E: modulus of elasticity

ANSYS SOLUTİON TO THE PROBLEM:

Entering Material Properties

The properties of S235 steel grade are introduced to the program as in figure 3.

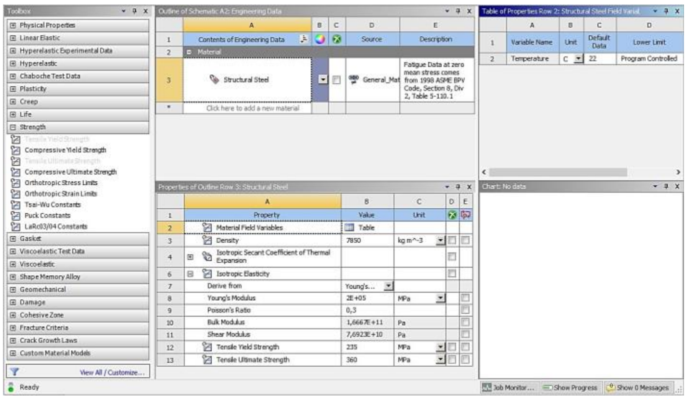

Figure 3. Material properties introduction to the program

Element modeling

IPE270 cross section drawing element was created with the DesignModeler drawing program in the ANSYS Workbench program. The appearance of the drawn element created is shown in Figure 4.

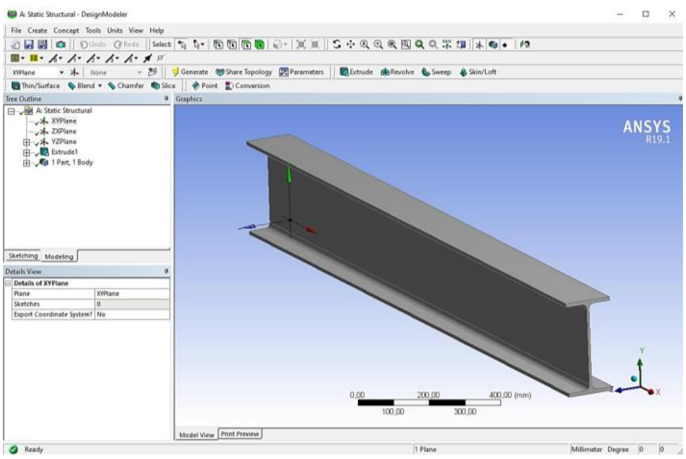

Figure 4. Model view of tension element

Creating a Finite Element Mesh

The mesh type of the building element has been selected in accordance with the geometry by the program. The results were compared by entering the mesh size in different values. Figure 5 shows the division of the element into meshes.

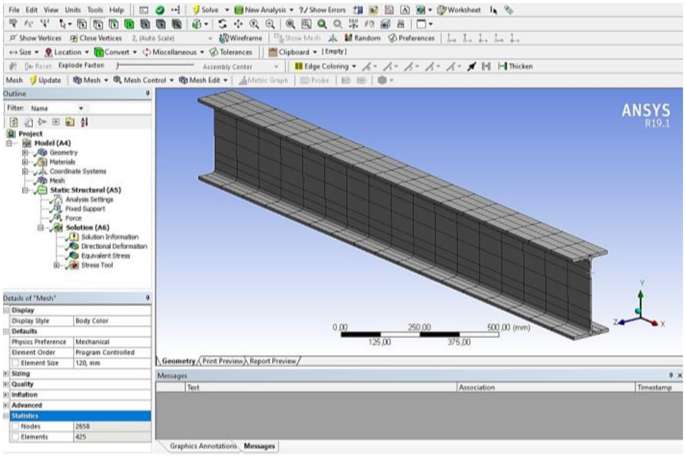

Figure 5. Constructed view of the finite element mesh of the tension member

Creating Boundary Conditions

The form of the boundary conditions and loading of the tensile element is given in Figure 6.

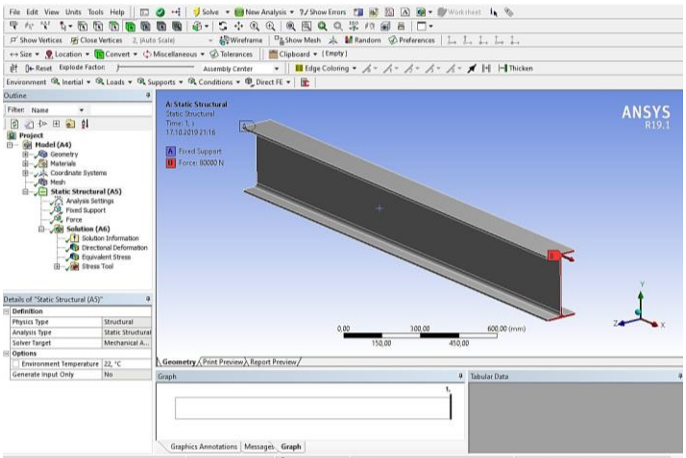

Figure 6. View of the boundary conditions and loads of the tensile member

Viewing and Evaluating Results

The tensile stresses and deformation results in the tensile member are presented in Figure 7.

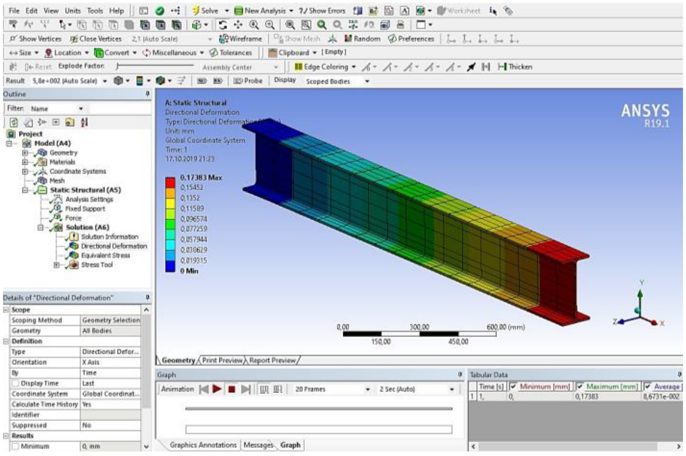

Figure 7. View of the stress and deformation occurring in the tensile member

Comparison of Results

The comparison of the theoretical and ANSYS Workbench results is given in Table 1 below.

Table1. Comparison of Theoretical and ANSYS Workbench Analysis Results

	Tensile stress (MPa)	Elongation (mm)	Max. Tensile stress (Mpa)	Nodes	Elements
Mesh Size (10 mm)	17,412	0,17385	28,695	84903	12800
Mesh Size (40 mm)	17,410	0,17382	23,534	12476	1850
Mesh Size (80 mm)	17,413	0,17380	20,681	3652	575
Mesh Size (120 mm)	17,416	0,17383	19,729	2658	425
Theoretical Result	17,429	0,17429	17,429	–	–

III.B. STEEL ELEMENT LOADED WITH AXIAL COMPRESSIVE FORCE

Both theoretical and finite element analysis of the IPE400 section steel element loaded with axial compressive force has been performed with ANSYS Workbench program. The two results obtained were then compared.

ELEMENT DIMENSIONS AND LOADING

The 3000 mm long IPE4000 section steel element shown in Figure 8 is fixed at one end and loaded with a compressive force of 100 KN in the longitudinal direction from the other end. Steel grade is S235. The compressive stress, critical compressive stress and shortening amount of the steel element were calculated both theoretically and by using different mesh intervals with the ANSYS Workbench program and the results were compared.

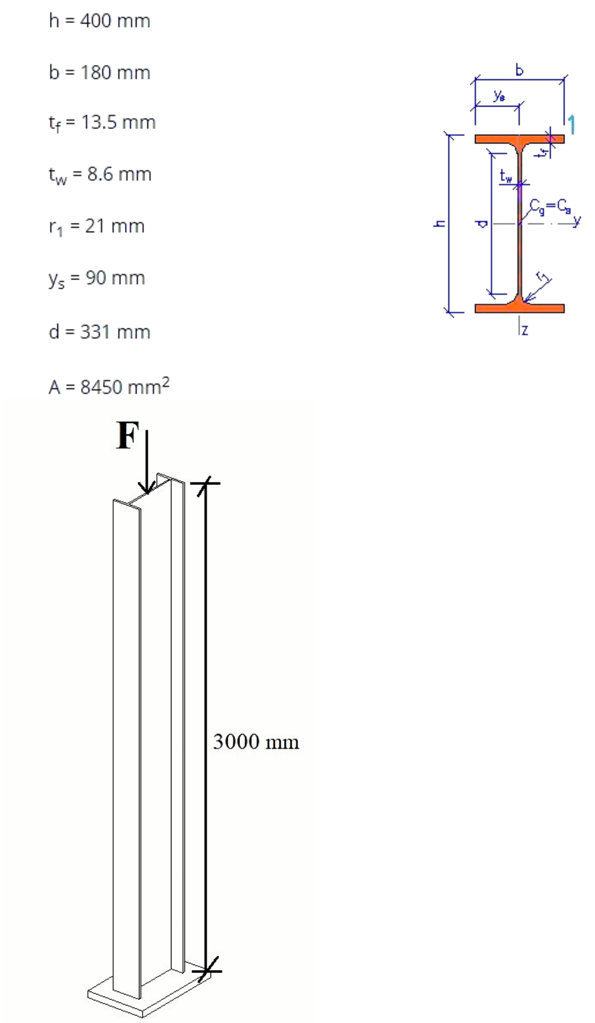

Figure 8. Properties of sectional steel compression elements IPE400

THEORETİCAL SOLUTİON TO THE PROBLEM

The theoretical critical compressive stress and buckling force at the IPE400 cross section is calculated below.

$E = 200000\ MPa \qquad I_x = 13200000\ mm^4$

$$F_{cr} = \frac{\pi^2 . E . I_x}{{L_k}^2} \qquad (3)$$

$$L_k = K . L \qquad K = 2{,}0 \qquad L_k = 2 . L \qquad {L_k}^2 = 4 . L^2$$

$$F_{cr} = \frac{\pi^2 . E . I_x}{{L_k}^2} = \frac{\pi^2 . 200000 . 13200000}{4 . 3000^2} = 723770{,}989\ N$$

Where:
L_k: Buckling length
E: modulus of elasticity
K: Buckling coefficient
I_x: Moment of inertia
F_{cr}: Critical buckling force

The amount of shortening in the IPE400 cross section in the longitudinal direction and critical compressive stress calculated as follows.

$$\Delta L = \frac{P\,.L}{E\,.A} = \frac{100000\,.3000}{200000\,.8450} = 0{,}17751\ mm$$

$$\sigma = \frac{F}{A} = \frac{100000}{8450} = 11{,}834\ MPa$$

$$\sigma_{cr} = \frac{F_{kr}}{A} = \frac{723770{,}989}{8450} = 85{,}653\ MPa$$

Where:

Δl: Shortening amount
L: Length
E: modulus of elasticity
F: Compression stress
A: Cross section area
σ: Compressive stress
σ_{cr}: Critical buckling compressive stress

ANSYS SOLUTİON TO THE PROBLEM:

Entering Material Properties

The properties of S235 steel grade are introduced to the program as in figure 9.

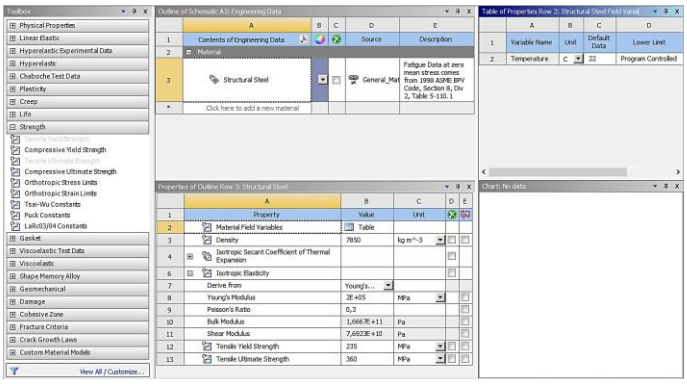

Figure 9 Material properties introduction to the program

Element modeling

IPE400 cross section drawing element was created with the DesignModeler drawing program in the ANSYS Workbench program. The appearance of the drawn element created is shown in Figure 10.

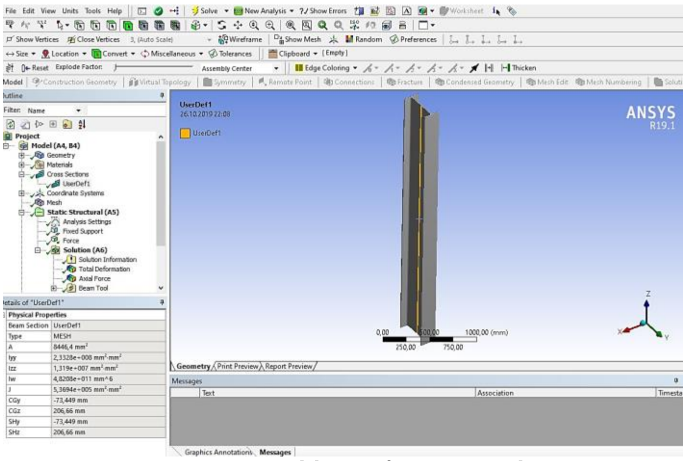

Figure 10. Model view of compression element

Creating a Finite Element Mesh

The mesh type of the building element has been selected in accordance with the geometry by the program. The results were compared by entering the mesh size in different values. Figure 11 shows the division of the element into meshes.

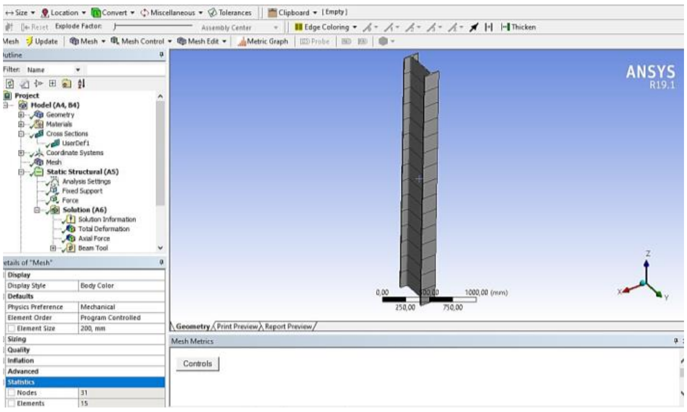

Figure 11. Constructed view of the finite element mesh of the compression member

Creating Boundary Conditions

The form of the boundary conditions and loading of the compression element is given in Figure 12.

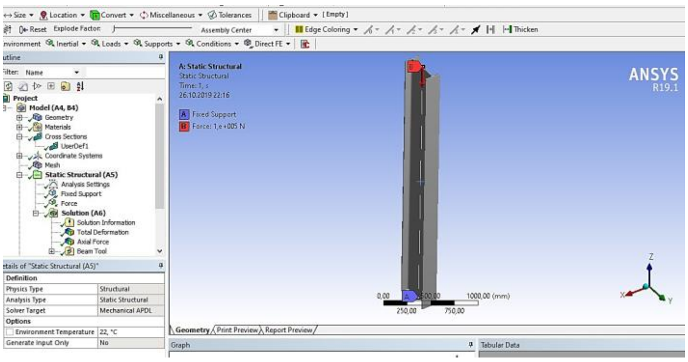

Figure 12. View of the boundary conditions and loads of the compression member

Viewing and Evaluating Results

The compressive stresses and deformation results in the tensile member are presented in Figure 13.

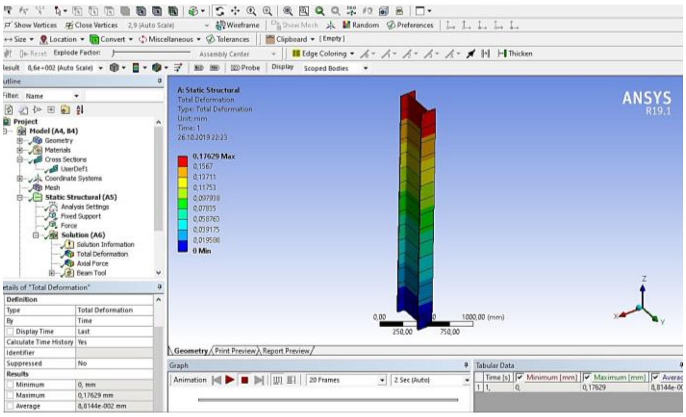

Figure 13. View of the stress and deformation occurring in the compression member

Comparison of Results

The comparison of the theoretical and ANSYS Workbench results is given in Table 2 below.

Table 2. Comparison of Theoretical and ANSYS Workbench Analysis Results

	Compressive stress (MPa)	Critical Force (KN)	Shortening amount (mm)	Nodes	Elements
Mesh Size (200 mm)	11,753	721,730	0,17629	31	15
Mesh Size (500 mm)	11,753	721,740	0,17629	13	6
Theoretical Result	11,834	723,771	0,17751	–	–

Buckling analysis result

As the critical buckling force obtained as a result of the buckling analysis made with ANSYS Workbench is greater than the pressure force acting on the element, the buckling was almost non-existent. The buckling analysis results are presented in Figure 14.

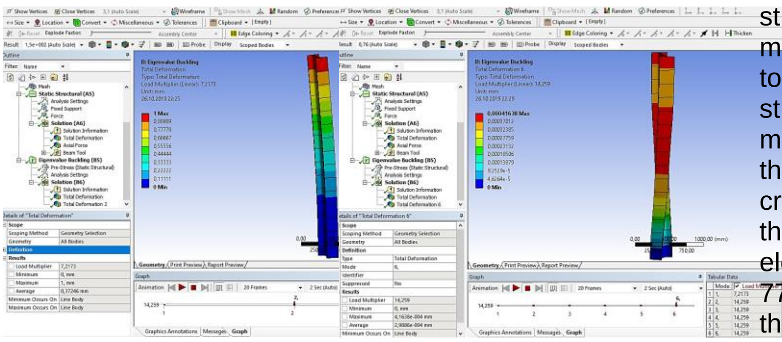

Figure 14. Buckling analysis result

IV. RESULT AND DISCUSSION

In this study, analysis of tension and compression elements with finite element method using ANSYS Workbench program was made. As a result of the analysis, the stress and deformations occurring in the elements were obtained. Also theoretical solutions are made. Theoretical and ANSYS Workbench solutions are presented in Table 3 and the results are interpreted.

Table 3. Theoretical and Ansys results obtained and error rates

	Stress (MPa)		Percentage of error (%)	Displacement (mm)		Percentage of error (%)
	Theoric	ANSYS		Theoric	ANSYS	
Tension Member	17,429	17,416	0,075	0,17429	0,17385	0,252
Compression Member	11,834	11,753	0,684	0,17751	0,17629	0,687

As a result of the studies, the following results have been achieved.

- The theoretically calculated tensile stress value for the element loaded with axial tensile force was found to be 17.429 MPa. As a result of the finite element analysis performed with the ANSYS Workbench program, although the tensile stress value does not differ much according to the mesh spacing, results between 17.410-17.416 MPa were obtained. The amount of elongation formed in the element was obtained as 0.17429 mm in the theoretical solution. As a result of the finite element analysis made with the ANSYS Workbench program, although the length elongation does not differ much according to the mesh spacing, results between 0.17380-0.17385 mm were obtained. The stress and lengthening values found for the theoretical and Ansys Workbench solution are equally acceptable and the error rates are quite small.

- The critical compressive stress calculated for the element loaded with the axial compression force theoretically was found to be 11.834 MPa. As a result of the finite element analysis performed with the ANSYS Workbench program, the compressive stress value was obtained as 11.753 Mpa for both mesh spacing, which did not differ much according to different mesh spacing. The compressive stresses obtained from the finite element analysis made with the ANSYS Workbench program by theoretical calculation are almost the same. The critical buckling force occurring in the element was theoretically obtained as 723,771 KN. In finite element solutions, a critical buckling force of 771,730-771,740 KN was obtained depending on the number of meshes. In the theoretical solution, the length shortening was obtained as 0.17751 mm, while in the Ansys Workbench solution; 0.17629 mm length shortening values were obtained for both mesh spacing. The stress, length shortening and critical buckling strength values found for the theoretical and Ansys Workbench solution are equally acceptable and the error rates are quite small.

REFERENCES
[1] ANSYS version 19.1, (2018). Incorporated programmers manual for ANSYS, ANSYS Inc.
[2] W.E. Ayrton, J. Perry, On struts, The Engineer 62 (1886) 464–515.
[3] W.T. Koiter, On the Stability of Elastic Equilibrium(PhD thesis) Delft Institute of Technology, 1945.
[4] M. Pircher, R. Brigde, The influence of circumferential weld-induced imperfections on the buckling of silos and tanks, J. Constr. Steel Res. 57 (2001) 569–580.
[5] J.G. Teng, X. Lin, J.M. Rotter, X.L. Ding, Analysis of geometric imperfections in fullscale welded steel silos, Eng. Struct. 27 (2005) 938–950.
[6] J.M.F.G. Holst, J.M. Rotter, C.R. Calladine, Imperfections and buckling of cylindrical shells with consistent residual stresses, J. Constr. Steel Res. 54 (2000) 265–282.
[7] ECCS, Ultimate limit state calculation of sway frames with rigid joints, Technical Committee 8, TWG 8.2, Publication n°033, European Convention for Constructional Steelwork, Brussels, Belgium, 1984.
[8] CEN, EN 1993-1-5, Eurocode 3: Design of Steel Structures – Part 1-5: Plated Structural Elements, European Committee for Standardization, Brussels, Mars, 2007.
[9] R. Bjorhovde, Deterministic and Probabilistic Approaches to the Strength of Steel Columns [D], Lehigh University, Bethlehem, 1972.
[10] Y. Fukumoto, Y. Itoh, Evaluation of multiple column curves using the experimental data-base approach [J], J. Constr. Steel Res. 3 (3) (1983) 2–19.
[11] K. Li, Y. Xiao, X. Nao, et al., Column curves for steel compression member [J], Journal of Chongqing Jianzhu University, 1985 (in Chinese).
[12] G. Wang, W. Zhao, S. Wang, Investigation on the bearing capacity of hot-rolled I-section axial compression members [J], 04, J. Build. Struct. 7 (1986) 1–11 (in Chinese).
[13] H. Ban, G. Shi, Overall buckling behaviour and design of high-strength steel welded section columns [J], J. Constr. Steel Res. 143 (2018) 180–195.
[14] G. Shi, K. Xu, H. Ban, et al., Local buckling behavior of welded stub columns with normal and high strength steels [J], J. Constr. Steel Res. 119 (2016) 144–153.
[15] Lee KS, Geem ZW. A new structural optimization method based on the harmony search algorithm. Comput Struct 2004;82:781-98.
[16] Lee KS, Geem ZW. A new meta-heuristic algorithm for continuous engineering optimization: harmony search theory and practice. Comput Meth Appl Mech, 2000.