International Journal of Materials Science and Applications
2019; 8(6): 103-108
http://www.sciencepublishinggroup.com/j/ijmsa
doi: 10.11648/j.ijmsa.20190806.12
ISSN: 2327-2635 (Print); ISSN: 2327-2643 (Online)

Elastic Stress Analysis of St 37 and St 70 Steels with Finite Element Method

Semih Taskaya[1, *], Bilgin Zengin[2], Kursat Kaymaz[3], Muzaffer Askin[2]

[1]Department of Metallurgy and Materials Engineering, Faculty of Technology, Firat University, Elazig, Turkey

[2]Department of Electrical and Electronics Engineering, Faculty of Engineering, Munzur University, Tunceli, Turkey

[3]Department of Civil Engineering, Faculty of Engineering, Munzur University, Tunceli, Turkey

Email address:

muh.semihtaskaya@gmail.com (S. Taskaya)

[*]Corresponding author

To cite this article:

Semih Taskaya, Bilgin Zengin, Kursat Kaymaz, Muzaffer Askin. Elastic Stress Analysis of St 37 and St 70 Steels with Finite Element Method. *International Journal of Materials Science and Applications*. Special Issue: *Materials Science and Engineering Model Designs*. Vol. 8, No. 6, 2019, pp. 103-108. doi: 10.11648/j.ijmsa.20190806.12

Received: October 11, 2019; **Accepted**: October 22, 2019; **Published**: November 4, 2019

Abstract: 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 a plurality of simple, small, connected, sub-regions called finite elements. In other words, the problem which is divided into parts connected by a plurality of joint points can be easily solved. St 37 and St 70 steels are materials used in the manufacturing of general building materials, produced by processing the hot-formed steel further through a cold drawing process. Ansys; is a computer aided engineering program where analysis and simulations can be performed in computer aided engineering studies. It enables effective studies in different disciplines such as mechanics, structural analysis, computational fluid dynamics and heat transfer. The finite element method, which enables the solution of complex engineering problems with controllable parts by simplifying, is a common and useful solution method used in many engineering applications. St and St 70 steels are modeled as three-dimensional I-beams 3 mm in thickness in Ansys program package in accordance with the finite element method. Finite element method helps simplifying complex engineering problems and solving them with controllable parts. Elastic stress analyses were performed in X, Y, Z axes by stabilizing the right and left supports of steel beams and applying a pressure of 100 MPa on the top flanges. It was observed in Ansys simulation analyses that elastic stress effect was higher in St 37 steel compared to St 70 steel.

Keywords: St 37-St70, Ansys, Elastic Stress, Finite Element Method

1. Introduction

St 37 steel is non-alloy steel. Its mechanical properties are above 235 MPa yield strength and 360-510 MPa tensile strength. It is used for riveting, screwing and welding. St 37 steel material is used in many applications by combining good welding properties with high strength structures. There are various grade types and uses including civil, industrial, automotive, machine and aerospace engineering. High strength low alloys have replaced structural steel (for example automotive) where weight reduction is important, but have high strength structures [1]. Mechanical properties of St 70 steel have 365 MPa yield strength and 690-900 MPa tensile strength [2]. The finite element method has a significant effect on the modeling of various mechanical applications related to aerospace and civil engineering. The finite element method, which enables the solution of complex engineering problems with controllable parts by simplifying, is a common and useful solution method used in many engineering applications [3]. ABAQUS, the finite element software package [4-8] commonly used for the analysis of cold-formed steel elements in the past [9] was chosen for quantitative research. Gardner and Nethercot [10] and Jandera et al [11] revealed that numerical models consisting of both membrane and bending components of residual stresses produced similar estimates to models without residual stresses in terms of general material behavior or final load-bearing capacity. Nguyen and Kim [12] used the ABAQUS software to develop a 3D FE model steel-concrete composite connection with headed stud shear connectors. In the developed model, the damage and failure of the concrete (non-linearity of the material), headed stud, steel beam and rebar were taken into consideration. Pavlović et al [13] reported 3D FE steel-concrete composite connection models with embedded, high-strength bolted shear connectors and these models were analyzed using explicit solver and ABAQUS. Strength, failure mode, ductility and load-slip behaviors of steel-concrete composite connectors predicted for 3D FE models exhibited a good correlation with push-out test data. Katwal et al [14] aimed to develop a detailed finite element model for composite beams with steel-profiled slabs by taking into consideration the realistic interaction between different components, failure of shear studs and tension and compression damages of steel-profiled slabs. Taşkaya et al [15] observed that, according to the finite element method in Ansys software, deformation and vector stress increased in St 70 lattice roof steel compared to the St 37 lattice roof steel due to both force and moment effect in beam axes. Polat et al [16] investigated the analysis of continuous contact problem of a functionally graded layer loaded by two rigid blocks and resting on elastic semi-infinite plane by using finite element method. Polat et al [17] investigated the comparative analysis of continuous contact problem of homogeneous layer loaded by two rigid layers and resting on an elastic semi-infinite plane. Gür et al [18] designed a 3-dimensional lattice roof using isotropic steel material and investigated the mechanical stress effects on beam axes under various loads according to the finite element method. Taşkaya [19] applied various loads and constant pressures on the 3D lattice roof model with isotropic steel by finite element method material and investigated the deformation, mechanical and elastic stress in beam axes. Gür et al [20] compared the changes in material by examining the stress analysis in different supports and geometries in the sandwich composite material with the finite element method. Kaymaz et al [21] in Ansys package program 2 different models of 3 intermediate layers, with a straight and 7° orientation angle, are designed as 3 dimensional according to x, y, z coordinate measurements. Sandwich plates with smooth and radial geometries were fixed in two different tests from their right and left supports, linearly and mechanical stresses were analyzed according to axes under 4 MPa pressure. It has been investigated that the compression ratios of straight anchorage support increase linearly as a function of linear meshes in the same geometry but different support stresses. Taskaya and Taskaya [22] work, 40 mm outer diameter, AISI 310 stainless steel cylinder containing mechanical properties are modeled in Ansys Workbench 12.0 module. The design is performed by fixing the cylindrical steel from the substrate support zone. The static structural solution of the steel was analyzed by applying a force of 1000 N in the -y-axis direction from the steel top plate zone. As a result of the analysis, it was observed that the charge distributions effected by the static structure of the cylindrical steel were concentrated in the upper and lower regions. Taşkaya and Taşkaya [23] work, Anys software, St 52 steel with a thickness of 1000x2000 with a thickness of 10 mm was transformed into a 3-dimensional volume. The model of the volume of the left and left supports by fixing the central area 1000 N/mm2 force and 1000 Mpa pressure applied to the axes coordinate points of the tension analysis was performed. In the analysis of voltage and vector simulation of St 52 steel model volume in Ansys software, it was seen that the voltage and pressure change in the axial coordinates of the pressure increased in direct proportion. Taşkaya and Taşkaya [24] work, the temperature, pressure, velocity and mass flow behaviors of the hot and cold water fluid mixture passing through a volume of three dimension designs were investigated. This fluid mixture was determined according to the determined T1-V1, T2-V2 temperature and speed parameters. The parameters obtained from the simulation analysis data results, the fluid of the global axes in the model volume, the change in mass and knot flow velocity according to the fluid in the pressure output mixture were constant, temperature and pressure increased.

2. Method

According to the finite element method in Ansys 12.0 software package, the model structure of St 37 and St 70 steel models is an I-beam with a size of 100x100 mm and a thickness of 3 mm, has an "extrude" 3D design, and is designed in accordance with the mechanical properties given in Table 1 and Table 2.

Table 1. *Mechanical properties of St 37 steel [25].*

Material	Density (kg/m^3)	Elasticity Module (MPa)	Elongation (%)	Poisson Ratio	Tensile Strength (MPa)	Yield Strength (MPa)
St 37	8000	210000	15	0.3	360-470	225-235

Table 2. *Mechanical Properties St 70 [26].*

Material	Density (kg/m^3)	Elasticity Module (MPa)	Elongation (%)	Poisson Ratio	Tensile Strength (MPa)	Yield Strength (MPa)
St 70	7700	200000	8-25	0.3	650-880	350-550

2.1. Modeling

For the model design in St 37 and St 70 steel standards, only one I-beam is used for each element type. Following various analyses on the same model, mechanical properties are identified and analysis is performed. In the first stage of modeling, the analysis type was selected statically. The most important parameter during the analysis of the model i.e. the element type is determined. The most commonly used element type in solid modeling "Solid 46" is selected as the element type. The reason for using this element type is to ensure stable transfer of mechanical properties and to reach a conclusion. Mechanical data for the design are defined in the material properties according to Table 1 and Table 2. After the I-beam is designed two-dimensionally as shown in Figure 1 according to the coordinates identified in the first stage, it is converted into 3D with 3 mm thickness by using the "extrude" module (Figure 1 c-d).

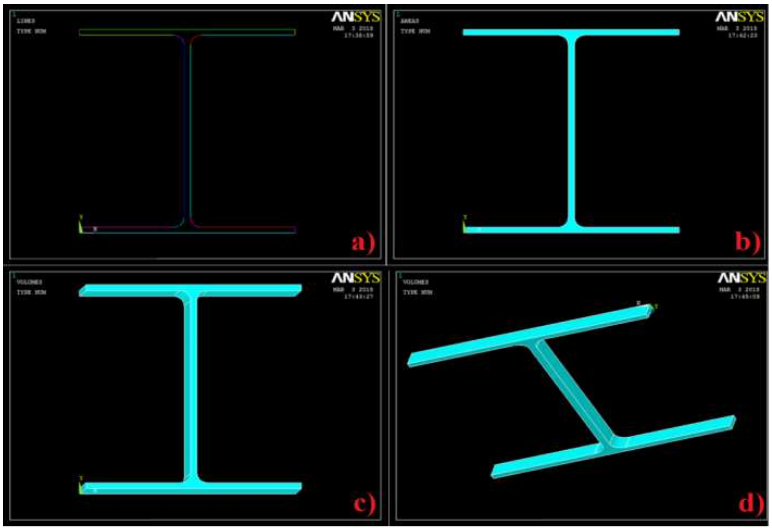

Figure 1. I-beam steel model's a) view of lines, b) creation of an area from lines, c) conversion to 3D, d) a different perspective of the 3D model.

2.2. Mesh Generation for the Model

After the I-beam steel model is designed in 3D, a mesh layer must be created to achieve an analysis in the model and enable simulation distribution (Figure 2). To this end, the mesh degree is determined from the "mesh size" module of Ansys data library. The most suitable mesh degree is selected for the beam, the minimum degree being 1. The neural network mesh is one of the most important phases for mechanical distributions. The purpose here is to facilitate the transfer of material properties.

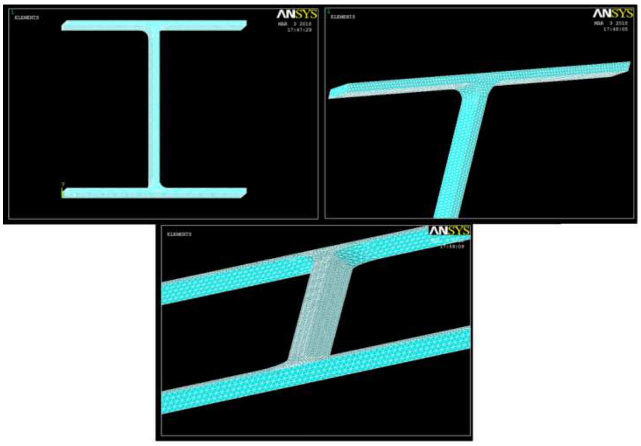

Figure 2. Mesh generation for the I-beam steel.

2.3. Identification of Boundary Conditions of the Model and Application of Pressure

The boundary conditions are identified by stabilizing the I-beam steel model from the bottom right and left supports and applying a pressure of 100 MPa on the top flange (Figure 3).

Figure 3. a) Stabilization of I-beam steel from the bottom supports b) application of pressure on the top flange.

3. Results and Discussion

The model is analyzed following the application of pressure on St 37 and St 70 I-beam steel models. After the model is analyzed, 2427 elements and 5754 nodes are generated. Constructively, these elements and nodes yield positive mechanical simulation results in the material. In Figure 4, the deformation of beams is investigated.

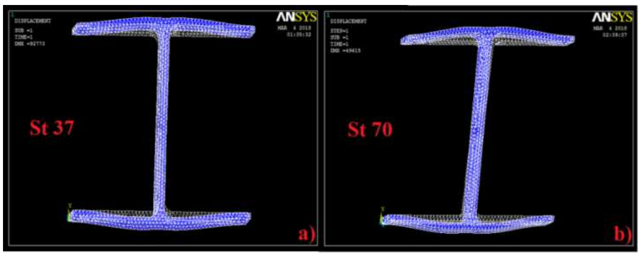

Figure 4. Analysis of I-beam model, a) deformation of St 37 steel, b) deformation of St 70 steel.

In Figure 5, pressure-dependent vector analyses of steel beams are investigated. These analyses are performed based on stress, nodal and resultant vector types.

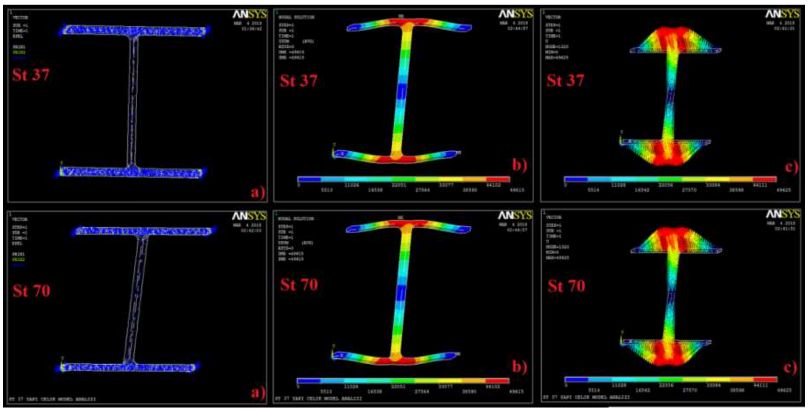

Figure 5. *Vector analyses of I-beam model: a) elastic stress vector distribution of St 37-St 70 steels, b) nodal stress vector distribution of St 37-St 70 steels, c) resultant vector distribution of St 37-St 70 steels.*

In Figures 6 and 7, elastic stress-distance graphical analyses of St 37 and St 70 I-beam steel models in x, y, z coordinates are investigated.

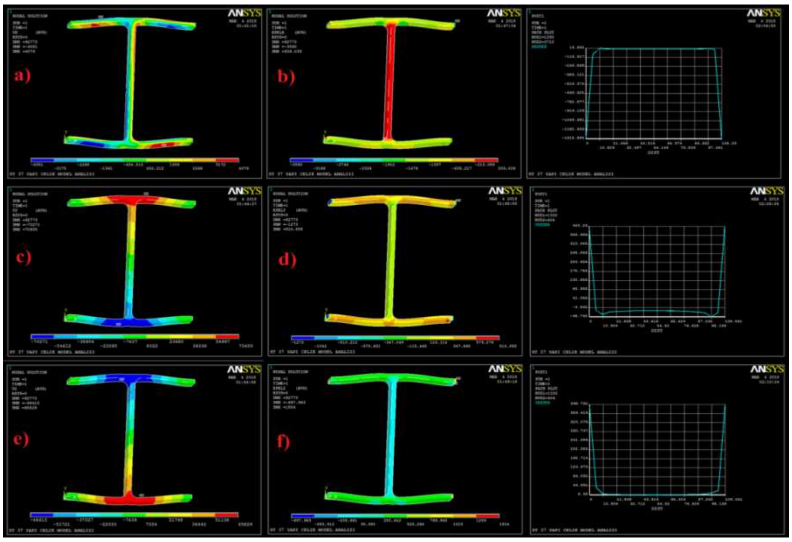

Figure 6. *St 37 steel beam's a, b) graphical analysis of nodal and elastic stress in X axis, c, d) graphical analysis of nodal and elastic stress in Y axis, e, f) graphical analysis of nodal and elastic stress in Z axis.*

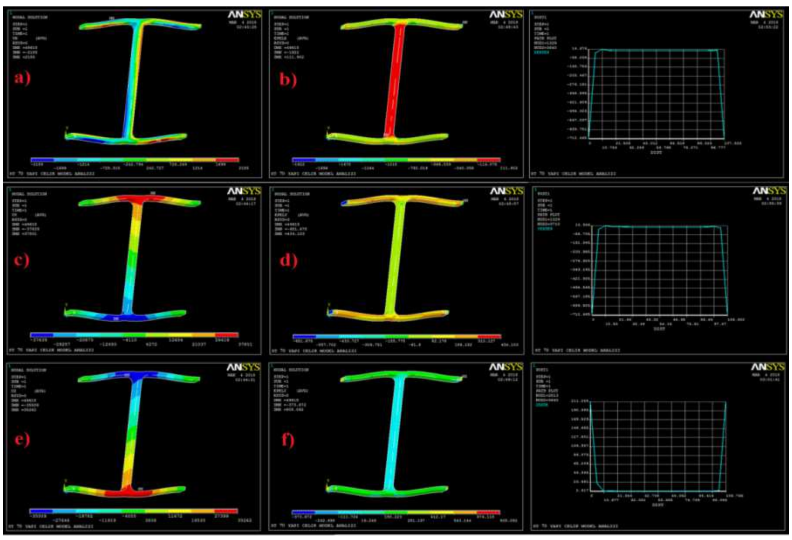

Figure 7. *St 70 steel beam's a, b) graphical analysis of nodal and elastic stress in X axis, c, d) graphical analysis of nodal and elastic stress in Y axis, e, f) graphical analysis of nodal and elastic stress in Z axis.*

In Figure 8, Von Mises elastic stresses of I-beam steel models based on joint shear stresses are investigated.

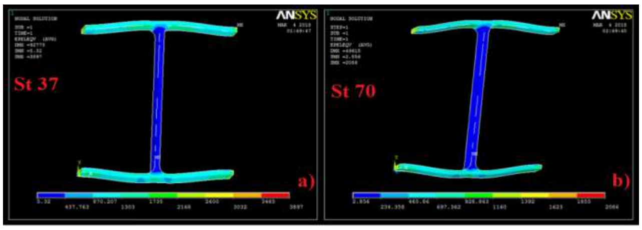

Figure 8. *a) Von Mises elastic stress analysis of St 37 steel beam, b) Von Mises elastic stress analysis of St 70 steel beam.*

In Von Mises test results presented in Figure 8, that is, the average simulation of axial stress and shear stress, it is shown that St 70 steel exhibited higher yield criteria than St 37 steel, because St 70 steel has a higher yield strength than St 37 steel [15-17].

4. Conclusions and Suggestions

In distances from the top node to the bottom node of St 37 I-beam in elastic stress graphs, while a linear increase in stress was observed in x axis, a decrease was observed in y and z axes. In distance-dependent elastic stress analysis of St 70 I-beam steel, on the other hand, while a linear increase in stress was present in x and y axes, a decrease was observed only in z axis. It is thought that the increases and decreases in Y axis result from the tensile strength, yield strength and elasticity properties of the material type. Since the elasticity module of St 70 steel is lower than the elasticity module of St 37 steel [15-17] the tensile and compressive effect of stress between distances increased in Y axis. Since the elasticity module of St 37 steel is higher than the elasticity module of St 70, however, the tensile and compressive effect of stress between distances decreased in Y axis. In X and Z axes, elastic stress distributions demonstrated close results in terms of tensile and compressive effects between distances. Since the pressure is applied toward the Y axis, differences were observed between Y axes of the beams. Distance-dependent stress measurements can also be made from different nodes. During the implementation phase, beam tensile and compressive tests can be conducted and compared with Ansys software simulation results. Beams in various sizes and thicknesses can be modeled, varying support and load quantities can be used, and according to the differences in mechanical results, stress, nodal and vector analyses can be performed and results compared. Deformations in axial regions can also be tested using various finite element programs (such as Apex, Nastran, Patran, ABAOUS) and compared with each other [19].

A comparison of analysis results shows that deformations, nodal, stress and resultant vector analyses as well as nodal and elastic stresses in x, y, z coordinate axes were observed to increase in St 37 steel compared to St 70 steel, because the elasticity module of St 37 steel is higher than that of St 70 steel by 10000 MPa. Therefore, the elastic character of St 37 steel increases compared to St 70 steel. Steels with high tensile strength show less elastic behaviors. The tensile strength of St 70 is thus higher than that of St 37 steel [15-17] Therefore, the elastic behavior of St 70 steel is reduced compared to the St 37 steel. When we look at the effects of right and left supports of steel beams, it is observed that tensile and compressive effect resulting from the pressure applied on the top flange increased in St 37 steel compared to St 70 steel as shown by counter simulation results in Figures 6 and 7. Pressure-induced differences were not close in axial mechanical effects and resultant vector analyses. There was a significant difference between results. This significant difference is associated with elasticity, stress and yield strength differences of materials.

Acknowledgements

A simplified summary statement of this work was presented at the "3rd International Conference on Computational Mathematics and Engineering Sciences (CMES'2018)-(Turkish Republic of Northern Cyprus), Girne, CYPRUS" conference.

References

[1] Bebon International co., ltd., http://www.steel-plate-sheet.com/Steel-plate/DIN/St372.html. (30 May 2019).

[2] Join- Win Steel, http://www.steel-jw.com/DINEN/ST702-structure-steel-with-competitive-price.html. (30 May 2019).

[3] Simscale, https://www.simscale.com/blog/2016/10/what-is-finite-element-method/. (29 May 2019).

[4] ABAQUS Inc., Abaqus. (6. 13)., (2013). Finite Element Analysis Pawtucket. Computer-aided engineering, USA.

[5] A. D. Martins, PB. Dinis, D. Camotim and P. Providencia, "On the relevance of local-distortional interaction effects in the behavior and design of cold-formed steel columns", *In: Structural stability research council annual stability conference, Structural Stability Research Council (SSRC);* Toronto, On, Canada, 2014, pp. 402-45.

[6] P. B. Dinis, B. Young and D. Camotim, "Local-distortional interaction in cold-formed steel rack-section columns," *Thin-Wall Struct,* 2014, 81, pp. 185-194.

[7] CH. Pham and GJ. Hancock, "Numerical simulation of high strength cold-formed purlins in combined bending and shear," *Research report*, University of Sydney, Department of Civil Engineering, 2009, 904, pp. 1-53.

[8] DCY. Yap and GJ. Hancock, "Post-buckling in the distortional mode and buckling mode interaction of cold-formed thin-walled sections with edge stiffeners," *In: 18th International specialty conference on cold-formed steel structures, recent research and developments in cold-formed steel design and construction, United states: University of Missouri-Rolla;* Orlando, FL, 2006, pp. 71-88.

[9] M. Abambres, D. Camotim, N. Silvestre and K. J. R. Rasmussen, "GBT-based structural analysis of elastic-plastic thin-walled members," *Comput Struct,* 2014, 136, pp. 1-23.

[10] L. Gardner and D. Nethercot, "Numerical modeling of stainless steel structural components-a consistent approach," *J. Struct. Eng.,* 2004, 130, pp. 1586-1601.

[11] M. Jandera, L. Gardner and J. Machacek, "Residual stresses in cold-rolled stainless steel hollow sections," *J. Constr. Steel Res.,* 2008, 64, pp. 1255-1263.

[12] H. T. Nguyen and S. E. Kim, "Finite element modeling of push-out tests for large stud shear connectors," *J Constr Steel Res.,* 2009, 65, (10-11) pp. 1909-1920.

[13] M. Pavlović, Z. Marković, M. Veljković and D. Buđevac, "Bolted shear connectors vs. headed studs behaviour in push-out tests," *J. Constr Steel Res.,* 2013, 88, pp. 134-149.

[14] U. Katwal, Z. Tao, and M. K. Hassan, "Finite element modelling of steel-concrete composite beams with profiled steel sheetingi," *Journal of Constructional Steel Research,* 2018, 146, pp. 1-15.

[15] S. Taşkaya, B., Zengin and K. Kaymaz, "Investigation of force and moment effect of St 37 and St 70 roof lattice steels in Ansys program," *Middle East Journal Of Science*, 2018, 4 (1), pp. 23-35.

[16] A. Polat, Y., Kaya and T. Ş. Özşahin, "Fonksiyonel derecelendirilmiş tabakada sürekli temas probleminin sonlu elemanlar yöntemi ile analizi," *20. Ulusal Mekanik Kongresi. Uludağ Üniversitesi*, Bursa, 2017, pp. 332-341.

[17] A. Polat, Y. Kaya and T. Ş. Özşahin, "Elastik yarı sonsuz düzlem üzerine oturan ağırlıklı tabakanın sonlu elemanlar yöntemi kullanılarak sürtünmesiz temas problemi analizi," *Düzce Üniversitesi Bilim ve Teknoloji Dergisi,* 2018, 6, pp. 357-368.

[18] A. K. Gür, S. Taşkaya, N. Katı, and T. Yıldız, "3D Kafes çatı modelinin Ansys yöntemiyle mekanik gerilmelerinin incelenmesi," *2nd International Conference on Material Science and Technology in Cappadocia (IMSTEC'17),* Nevşehir, 2017, pp. 11-15.

[19] S. Taşkaya, "Investigation of mechanical and elastic stresses in Ansys program by finite elements method of 3D lattice roof model," *Mugla Journal of Science and Technology*, 2018, 4 (1), pp. 27-36.

[20] A. K. Gür, S. Taşkaya, N. Katı, and T. Yıldız, "Investigation of stress analysis in sandwich composite plates by Ansys method," *8th International Advanced Technologies Symposium (IATS'17)*, Elazig, 2017, pp. 495-509.

[21] K. Kaymaz, B. Zengin, M. Aşkın and S. Taşkaya, "Sandviç Kompozit Tabakalarında Mekanik Gerilmelerin Basınca Bağlı Olarak Ansys Yazılımı İle İncelenmesi," *Gümüşhane Üniversitesi Fen Bilimleri Enstitüsü Dergisi,* CMES sempozyum ek sayısı, 2018, pp. 79-93.

[22] S. Taskaya and S. Taskaya, "Investigation of Static Structure Effect According to Axial Coordinates by Using Finite Element Method in Ansys Workbench Software of AISI 310 Austenitic Stainless Cylindrical Model Steel," *International Journal of Scientific Engineering and Science*, 2018, 1 (11), pp. 65-70.

[23] S. Taşkaya and S. Taşkaya, "St 52 Çelik Model Hacminin Ansys Yazılımında Eksenel Koordinatlara Göre Kuvvet Ve Basınç İlişkisinin İncelenmesi," *I. Uluslararası Battalgazi Multidisipliner Çalışmalar Kongresi*-Malatya/Türkiye, 2018, Cilt- I, pp. 81-93.

[24] S. Taşkaya and S. Taşkaya, "Musluk Model Hacminin İçinden Geçen Akışkanın Ansys Yazılımında Global Koordinatlara Göre Dağılımınının Simülasyonu," *I. Uluslararası Battalgazi Multidisipliner Çalışmalar Kongresi*-Malatya/Türkiye, 2018, Cilt-I, pp. 114-125.

[25] S. Taşkaya, "Investigation of mechanical stresses dependent on press in St 37 steel Ansys program," *The Journal of International Manufacturing and Production Technologies*, 2017, 1 (1), pp. 39-46.

[26] China steel, http://www.steelgr.com/Steel-Grades/Carbon-Steel/st70-2.html. (30 May 2019).