Scientific Journal of Impact Factor (SJIF): 5.71
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 6, Issue 02, February -2019

MAPPING OF STRESS DISTRIBUTIONS OF HANGAR ROOF SYSTEMS IN ANSYS SOFTWARE

Selim Taskaya[1], Semih Taskaya[2]*

[1]Department Architecture and Urban Planning, Artvin Coruh University, Artvin, Turkey-08100
[2]Department Metallurgy and Materials Engineering, Firat University, Elazig, Turkey-23119

Abstract — *The hangar roof system is covered with a roof that is carried by overhead steel posts or columns that protect various goods or tools from external influences. Ansys software programs is a simulation software program used in engineering areas in recent years. Studies related to this program are used after the design phase and before the prototype is produced, it allows testing in a virtual environment. With the 3-dimensional 54 edge and 35 corner coordinate points, the hangar roof model formed with 46x50 mm base supporting cylinder pipe diameter was applied 200 Newton force to 5 support beam points. The deformation and stress distributions of the roof top beam and supporting nodes were mapped by the applied force. As a result of the analyzes, it was observed that deformation and stresses in the load-bearing upper beam and base coordinate points increased and the pre-supporting beam points decreased.*

Keywords- *Ansys; Hangar roof system; Mapping; Stress.*

I. INTRODUCTION

Hangar roof systems are designed as a kind of warehouse. In other words, the garage is similar to the closed area where large vehicles such as cars or tractors are stored. Hangars are mostly constructed from closed steel structures for construction sites, factories, airports, large workshops and various workplaces. Because of their large size, especially steel hangars are used as aircraft maintenance area. The use of steel materials in hangar roof systems eliminates the excessive weight conditions and makes the main building structures more resistant. Steel hangars, desired height, desired width and desired product quality, also insulated and is made without insulation. Steel structure hangars can be used with ordinary models and models can be made as desired by users [1]. Ansys is a computer-aided engineering program that can be analyzed and simulated in computer-aided engineering studies. The Ansys program enables effective studies in different disciplines such as mechanical, structural analysis, computational fluid dynamics and heat transfer [2]. Karoki et al. Investigated a reticulated long area steel hangar that did not meet the maximum allowable deviation limits. The aim is to evaluate, test and strengthen the structure to increase its capacity and to prevent excessive deformation. The structure was gradually loaded to the points indicated on the grid to test and evaluate the deformation and loading capacity, and a comprehensive analysis was performed before and after reinforcement. The loading capacity of the hangar element has been significantly improved after strengthening [3]. Yichen et al. In this study, a aircraft hangar maintenance scheduling problem, which is motivated by aircraft heavy maintenance in a hangar operated by an independent maintenance service company, has been investigated. A mixed integer linear programming (MILP) mathematical model was first introduced, combining the relationships between the maintenance schedule and the aircraft layout scheme. In the model, the parking capacity of the maintenance hangar is changed and the aircraft is prevented from coming in and out. Secondly, the model of time-dependent decision variables has been developed by narrowing to the possible entry and exit times of each maintenance request [4]. In Kazakevitch, Riga (Latvia), he examined the results of extensive experimental research in the wind tunnel of wind load on the membrane roof of the hangar at the airport. The cylindrical membrane roof has dimensions of 108 x 60 m in the rectangular plan. As a result of experimental research, he observed the distribution of the wind pressure on the upper and lower surfaces of the roof and the total values formed by the pressure [5]. Karpov et al., Six different shell structures with different radiuses of curvature and thickness and different number of reinforcement ribs and halls for the maintenance and maintenance of hangars of different materials can be used as the roof of the structure of the strength and stability analysis were made. The strength and stability analysis made it possible to select practical shell materials in accordance with the restriction requirements related to the values of the transverse loads under the dimensions of the construction structures which are important for the structural parking and vehicle maintenance of the hangars [6]. Gür et al. designed an isotropic steel material as a 3-dimensional lattice roof and investigated the effects of mechanical stress on beam axes in the Ansys package program according to the different loads according to the finite element method [7]. Taskaya has investigated the deformation, mechanical and elastic stress analysis of beam axes by applying different load and constant pressure to 3D lattice roof model which has an isotropic steel material in Ansys package program [8]. Taskaya et al., in the Ansys software, observed that the rigid axes of the St 70 roof lattice steel according to the St 37 roof truss steel by the finite element method have increased deformation and vector stresses according to both the force and the moment effect [9]. In the Ansys program, Taskaya examined the effect of mechanical stresses in the sheets of the pressure-bonded plates according to the supports in St 37 plate steels [10]. Polat et al. Investigated the comparative analysis of the problem of continuous contact in a homogeneous layer with elastic semi-infinite plane and loaded with rigid two blocks [11]. Kaymaz et al., in the study, consisting of 3 intermediate layers, 7 degree orientation angle with 2 different radial geometry geometry model, x, y, z coordinate dimensions according to the Ansys software in 3-dimensional FEM method. In two different tests, the flat and radial geometry sandwich plates were fixed by linear and linear fixation from the right and left supports, and they performed mechanical stress analysis at 4 MPa under axial pressure. Geometric shapes of the same-bearing different and geometric shapes of different-bearing comparing the same structures have examined [12]. Taskaya and Taskaya in this study, 40 mm outer diameter, AISI 310 stainless steel cylinder containing mechanical properties are modeled in Ansys Workbench 12.0 module. As a result of the analyzes, it was observed that the charge distributions effected by the static structure of the cylindrical steel were concentrated in the upper and lower regions [13]. Taskaya and Taskaya in this study has transformed the St 52 steel with a thickness of 1000x2000 to a 3-dimensional volume in Anys software. In the analysis of the stress and vector simulation of the st 52 steel model volume in Ansys software, they determined that the voltage and pressure change in the axial coordinates of the force increased in direct proportion [14]. Taskaya and Taskaya in this study examined the temperature, pressure, velocity and mass flow behaviors of the hot and cold water fluid mixture passing through a volume of three dimensions. This fluid mixture was determined according to T_1-V_1, T_2-V_2 temperature and speed parameters. The parameters obtained from the simulation analysis data results, according to the fluid of the global axes in the model volume, the fluid in the pressure outlet mixture; The change in mass and knot flow velocity was fixed, temperature and pressure were increased [15].

The aim of this study is to determine the stresses that will be generated at the support beam points as a result of the application of the 3-dimensional designed hangar roof model to the 5 support beam points.

II. MATERIALS AND METHODS

2.1. Construction of a Hangar Roof Model

In the Ansys Workbench software, the hangar roof model is designed in 3D with 54 edge, 35 corner coordinate points and 46x50 floor supporting cylinder tubes (Figure 1). The beams with a specific general steel material are selected from the data library of the Ansys software as general structural steel. Steel beam layers are systematically covered in a network form.

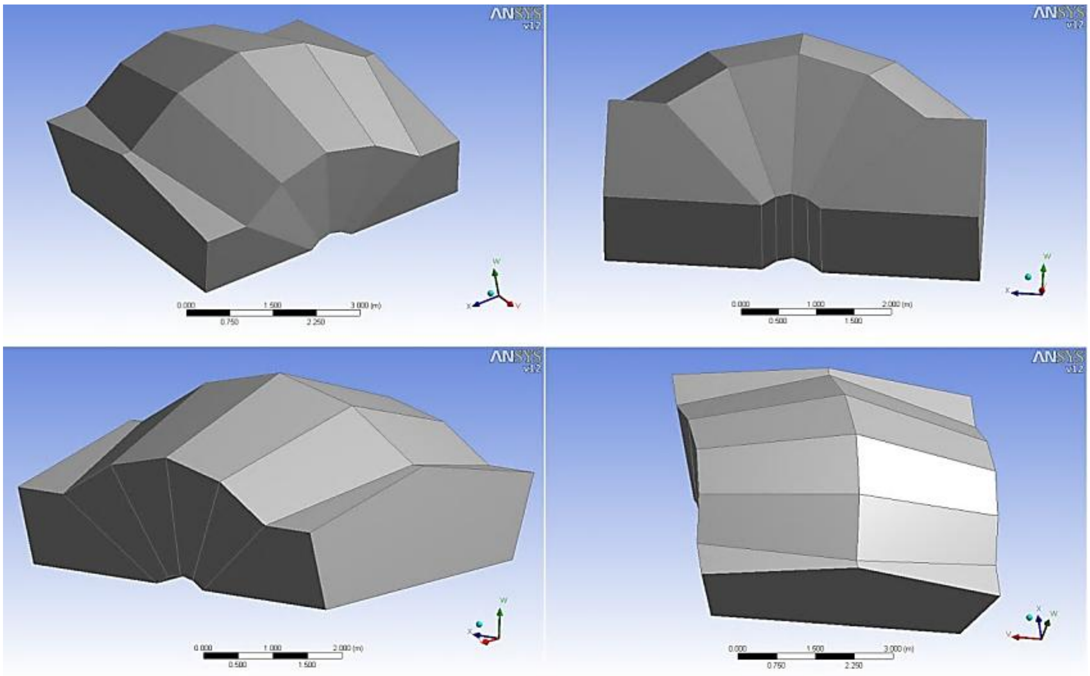

Figure 1: Hangar roof model

The hangar roof model is equipped with reinforced beam structures. In this model, it is essential to map the stress distributions between these beam structures. Figure 2 shows the beam structure of the whole model. Here, the base region points of the model are removed from the model to avoid contact with the cylindrical tube and not to affect the supporting points. Only the foot support points of the hangar roof model and the base remain cylindrical supporting tubes. The pressure of the front beam to the base areas of the upper roof area determines its strength. The connection boundary conditions of these support points are determined during the installation phase.

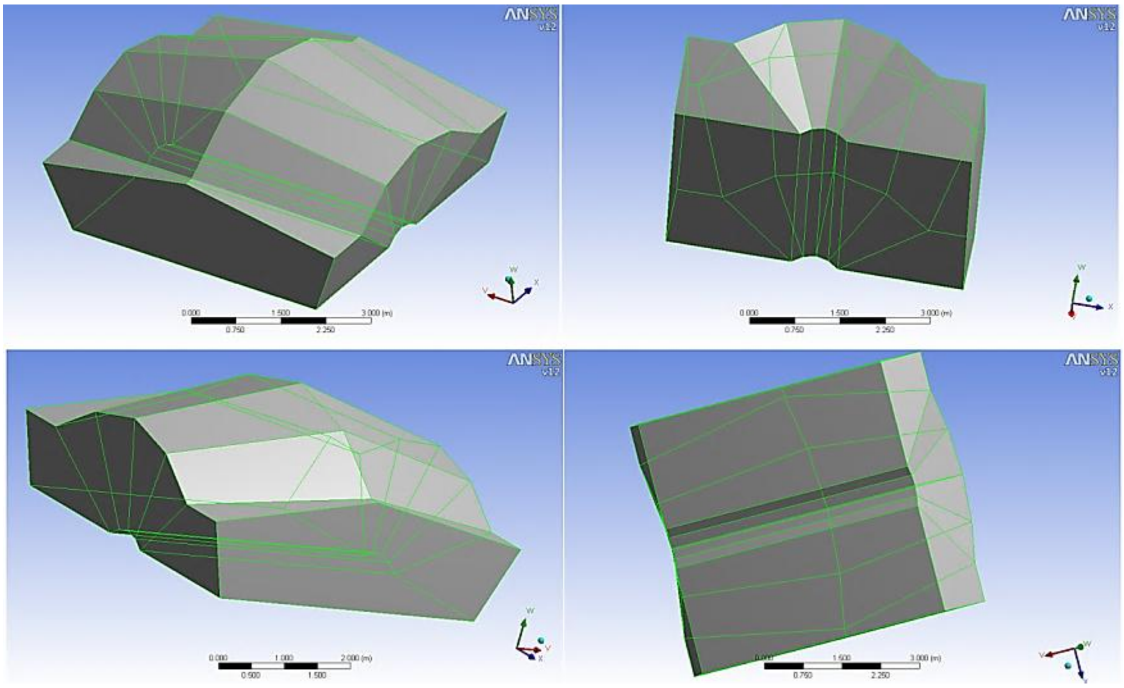

Figure 2: Beam brackets of the hangar roof model

After the beam support points of the hangar roof system are determined, the coordinates are determined according to the axes (Figure 3). After this stage, the beam columns are shown in linear form and all points are determined. The end portions of the supporting point 5 of the base region are in the form of a pointed profile. Cylindrical tubes are installed in these areas to prevent any rupture in the force to be applied in the analysis. In these regions, cylindrical ends having a diameter of 46x50 mm are placed and the strength of the hangar roof system is strengthened (Figure 3).

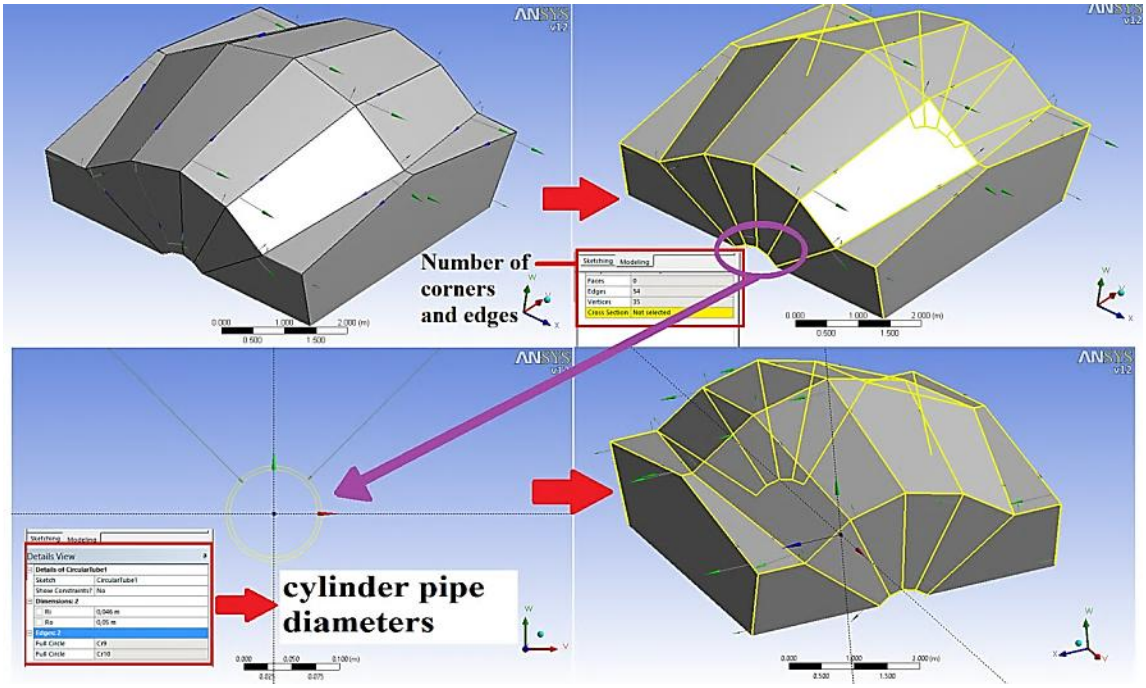

Figure 3: Coordinate points of hangar roof model and placement of cylindrical pipes

Prototype hangar roof model design is completed. Thus, solid model formation is provided. Since the analysis of the roof system will be made to the beams, a solid model is defined from the "cross section solids" module and only the beam support points appear. As seen in Figure 4, these parts are examined.

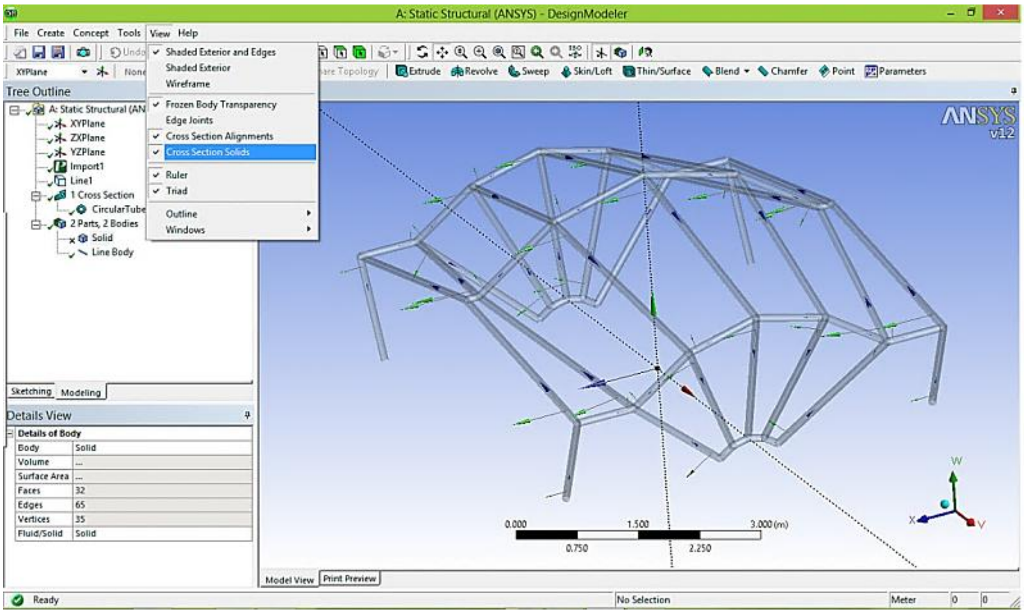

Figure 4: Beam support columns of the hangar roof model

Figure 5 shows the solid model of the hangar roof model and the views of the beam supporting coordinates from different perspectives.

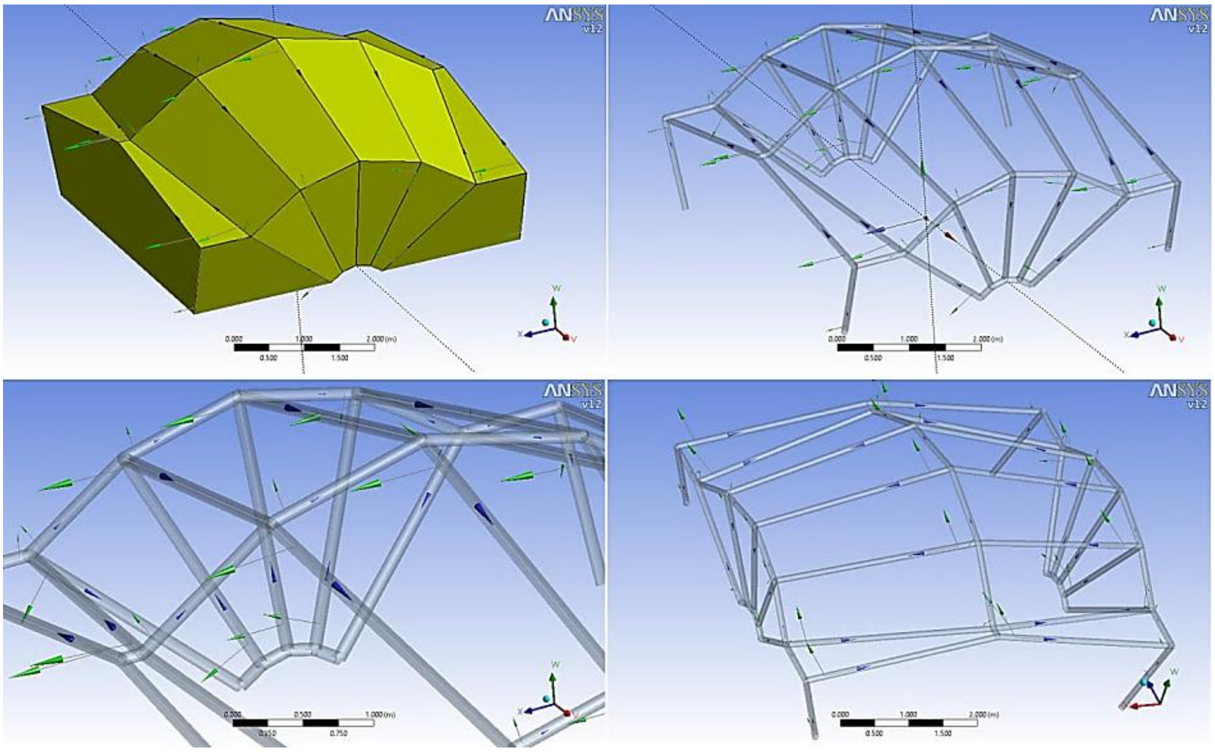

Figure 5: Beam support columns of the hangar roof model

Particularly in these model types, front and rear wing support points are dependent on flexibility coefficients in loading conditions. The deformation and distortions in these regions vary according to the general structure steel properties. As this model uses specific general structural steel, elasticities will not create permanent deformation. In Fig. 6, in the Ansys software, the hangar roof is fixed at the foot support points and the displacement factor is defined in the base cylinder tubes.

2.2. Fixing of hangar roof model and determination of displacement factors

At this stage, the fixed support process is carried out for the analysis against force from the 4 foot attachment points of the hangar model (Figure 6a). This will cause structural errors if the foot points move in the analysis against force. So these regions are fixed. In figure 6b, the main contact region of the support region of the upper region 5 is made to define the displacement of the cylindrical tubes from the front and rear 4 zones. Thus, the hangar roof system is given its share of stretch. In other words, the model is not intended to be damaged against any strain. Here, the displacement ratio of x, y, z is selected as zero mm. In other words, this displacement provides a swing movement at a constant angle.

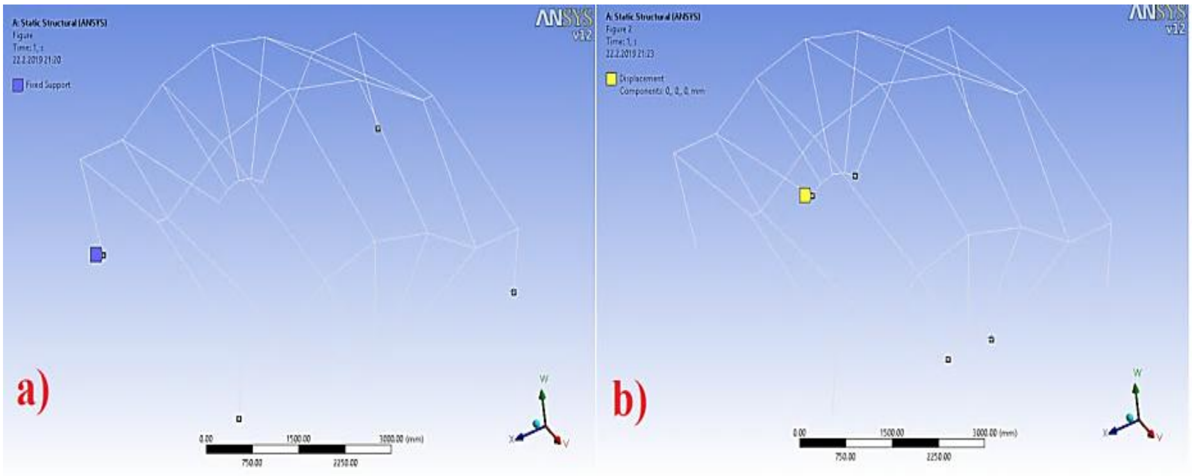

Figure 6: Hangar roof model, **a)** fixing the support points **b)** displacement movement identification

2.3. Meshing of the hangar roof model

In Figure 7, a mesh layer is defined in the hangar roof model. This definition is especially important in order for the stress distributions to be simulated by each other. If this mesh layer is not defined, mapping is not performed. The tensions between the beams cannot be simulated by each other. So the mesh identification process is very important. Makes contact of mechanical data. The mesh layer is defined globally to the entire hangar model. Small mesh structures were formed. The more shallow the mesh layer, the more simulated the distribution of the simulation.

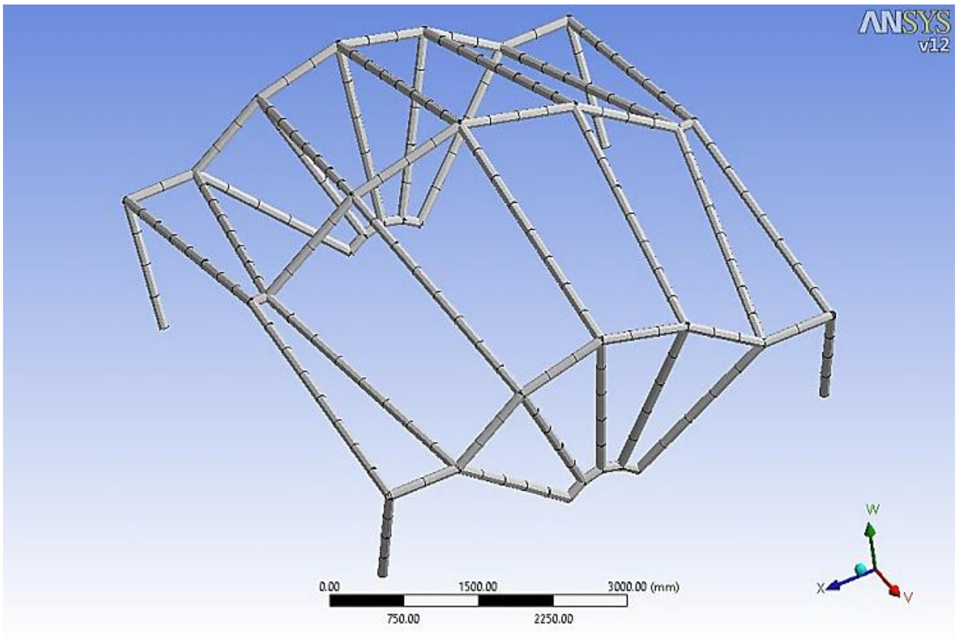

Figure 7: Meshing of hangar roof model (layer of mesh)

2.4. Loading the hangar roof model

As shown in Fig. 8, a 200 Newton load definition is made to the top 5 support beam of the hangar roof model. It is essential to examine the behavior of these 5 support points to the hangar model system by making these process structures individually. Stable equilibrium is achieved by distributing the loads evenly. As a result of this balance, the hangar roof model is analyzed and its structures are examined. The effect of the force varies according to the type of support. In particular, the fixing surface is foot beams which are in charge of carrying the load. Bearing types vary more in sheet steel. The cylindrical portion in the base area changes the deformation behavior of the hangar roof against the load.

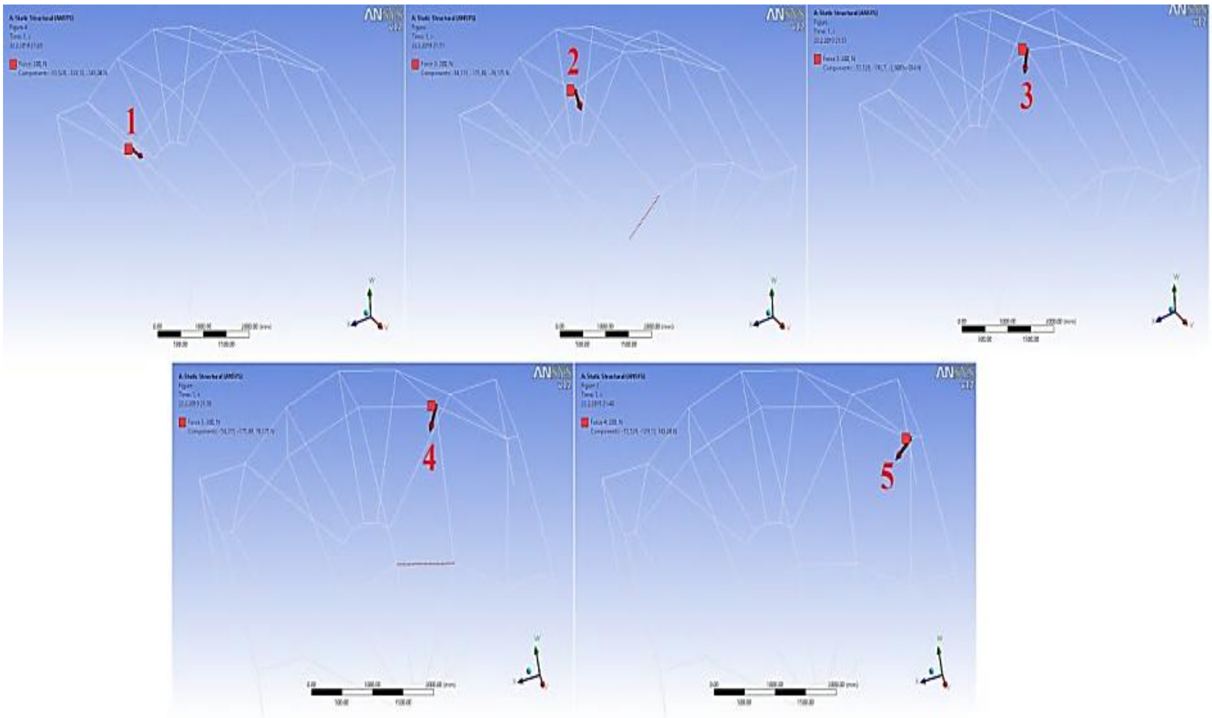

Figure 8: Definition of force to 5 support points of the hangar roof model

III. RESULTS AND DISCUSSION

When the results of the analysis in Figure 9a are examined, direct static stresses can be reached by max. formed on the upper front beam section. Here's the min. stress is the section with 5 supporting points. The mapping of stresses is here along the main side of the multi-side beam. The base cylinder regions are less effective than the upper beam areas. Indirectly, this is caused by the connection points of the main beam points at the 3 connection points. The front, rear and top inlets are connected at one point in the side profiles. Therefore, max. stresses were simulated at this point. The distortions can be observed in the waste stress effects here. When Figure 9b is examined, max. total stresses occur at the tip of the feet carrying the load. Min. Stress effects are formed in the side modules of the 5 support beam regions at the top. The distortion surfaces were mostly examined in the inner middle. Figure 9c de min stresses were detected at the supporting end regions of the hangar roof model. The reason for this is that the amount of load falling to the rear region decreases with the displacement factor in the base cylinder regions. Therefore, the effect here is min. stress. The stresses in the base cylinder pipes are considerably higher than the stresses in the upper beam regions. Because the main columns carrying the load. The connection points are resistant to high strength. When Figure 9d is examined, directional deformations are seen in the front beam regions. The max. stresses were seen in the base cylinder regions. min. stresses were examined in the upper upper beam regions. Deformation points do not have a permanent effect with oscillation movements. It causes only minor distortions. In Figure 9e, the region with the highest total deformation rate is the top 5 supporting beam points where the force has the main effect. Especially when this section is examined carefully, the stresses are very high in the areas where the connecting elements are connected. Because they are the regions that perform the oscillating motion acting on the load. In other regions, the mapping effect is continued gradually and spreading is taking place. According to the specific elastic coefficient of the general structural steel, these rates vary. In Figure 9f, the intersection of these ports is shown by zooming in the Ansys software. Stresses max. where the junction point is mechanized by simulating other beam elements. The load shows a oscillation from the upper beam region to the base regions. The beam mappings here are highly stable. This confirms the suitability of the analysis results. The data obtained from the analysis results, stress-related displacement factors and stresses and deformations formed a stable balance. In static structural analysis, the distortion and inter-tension stress effects vary according to the amount of load and the fixing angles.

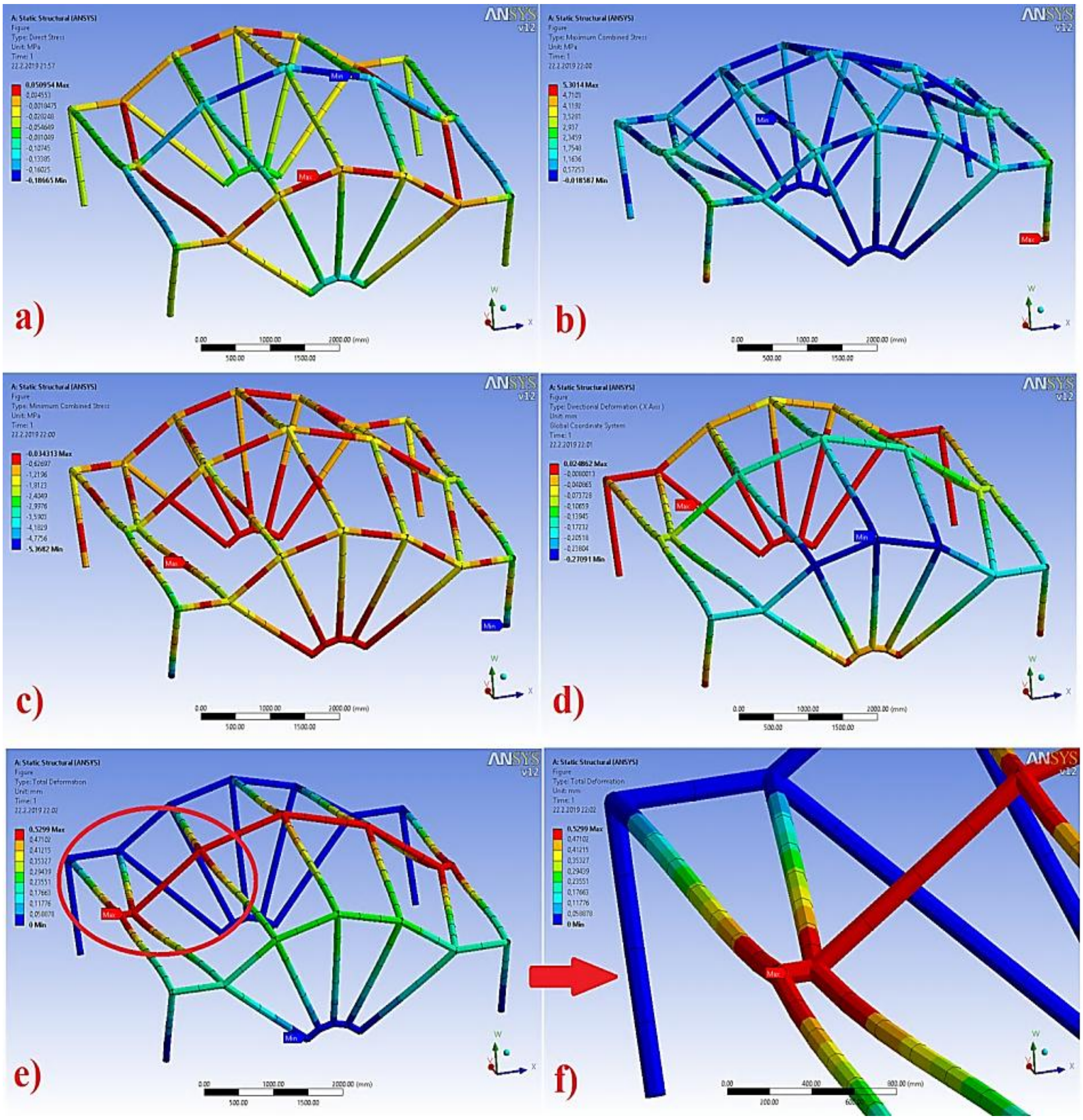

Figure 9: Hangar roof model system **a)** stresses acting directly **b)** max. total stresses **c)** min. total stresses **d)** directional deformation shapes **e)** total acting deformation shapes **f)** total effected deformation on main beam (zoom view)

IV. CONCLUSIONS AND RECOMMENDATIONS

The deformation and stress effects of the 3-dimensional hangar roof model system with the effect of 200 Newton load and displacement factor in Ansys Workbench software were investigated. The results obtained are given below:

- Direct static stresses can be reached by max. formed on the upper front beam section. Here's the min. stress is the section with 5 supporting points. The mapping of stresses is here along the main side of the multi-side beam. Indirectly, this is caused by the connection points of the main beam points at the 3 connection points.
- Max. total stresses occur at the tip of the feet carrying the load. Min. Stress effects are formed in the side modules of the 5 support beam regions at the top. The distortion surfaces were mostly examined in the inner middle.
- Min stresses were detected at the supporting end regions of the hangar roof model. The reason for this is that the amount of load falling to the rear region decreases with the displacement factor in the base cylinder regions. Therefore, the effect here is min. stress. The stresses in the base cylinder pipes are considerably higher than the stresses in the upper beam regions.
- Directional deformations are seen in the front beam regions. The max. stresses were seen in the base cylinder regions. min. stresses were examined in the upper upper beam regions. Deformation points do not have a permanent effect with oscillation movements.

- The region with the highest total deformation rate is the top 5 supporting beam points where the force has the main effect. the Stresses are very high in the areas where the connecting elements are connected.

From this model test results, the hangar roof structure from constructive aspect yielded the expected positive results from the different modules in Ansys Workbench software. All results from the model can be tested for all stresses from different regional axes. Model measurements, mesh structure, load amount, fixing angle and direction, complex model structures, changes in the type of analysis can be compared by changing the structure. Different simulation tests can be carried out using different finite element software (such as Apex, Nastran, Patran) [8].

REFERENCES

[1] Roof Celik Holding, https://www.roofcelik.com/blog/hangar-cati (Last access date: 20 February 2019).
[2] CadSay, https://cadsay.com/ansys-nedir (Last access date: 20 February 2019).
[3] K.E. Karoki, L. Shen and J. Gong, “Capacity analysis, investigations and retrofitting of a long span steel grid hangar”, Engineering Failure Analysis, 78, pp. 1-14, 2017.
[4] Y. Qin, Z.X. Wang, F.T. Chan, S.H. Chung and T. Qu, “A mathematical model and algorithms for the aircraft hangar maintenance scheduling problem”, Applied Mathematical Modelling, 67, pp. 491-509, 2019.
[5] M. Kazakevitch, “The aerodynamics of a hangar membrane roof”, Journal of Wind Engineering and Industrial Aerodynamics, 77, pp. 157-169, 1998.
[6] V. Karpov, A. Panin and A. Semenov, “Calculation of Reliability of Hangars for Parking and Maintenance of Vehicles”, Transportation Research Procedia, 20, pp. 261-266, 2017.
[7] A.K.Gür , S. Taşkaya, N. Katı, and T. Yıldız, “Investigation Of Mechanical Stresses of 3D Truss Roof Model By Ansys Method”, 2nd International Conference on Material Science and Technology in Cappadocia (IMSTEC’17), Nevsehir / Turkey, pp. 11-15s, 11-13 October 2017.
[8] 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, 4(1), pp. 27-36, 2018.
[9] 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, 4(1), pp. 23-35, 2018.
[10] S. Taşkaya , “Investigation of mechanical stresses dependent on press in St 37 steel Ansys program”, The Journal of International Manufacturing and Production Technologies, 1(1), pp. 39-46, 2017.
[11] A. Polat, Y. Kaya and T.Ş. Özşahin , “Analysis of Frictionless Contact Problem for A Weighted Layer on An Elastic Half Plane Using FEM”, Düzce Üniversitesi Bilim ve Teknoloji Dergisi, 6(2), pp. 357-368, 2018.
[12] K. Kaymaz, B. Zengin, M. Aşkın and S. Taşkaya, “Investigation of Mechanical Stresses on Sandwich Composite Layers According to The Pressure By Making Use of Ansys Software”, Gümüşhane Üniversitesi Fen Bilimleri Enstitüsü Dergisi, (CMES 2018 Sempozyum Ek sayısı), pp.79-93, 2018.
[13] 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, 2(11), pp. 65-70, 2018.
[14] S. Taskaya and S. Taskaya, “St 52 Çelik Model Hacminin Ansys Yaziliminda Eksenel Koordinatlara Göre Kuvvet Ve Basinç İlişkisinin İncelenmesi”, I.Uluslararası Battalgazi Multidisipliner Çalışmalar Kongresi-Malatya / Turkey, Cilt-1, pp. 81-93, 7-9 December 2018.
[15] S. Taskaya and S. Taskaya, “Musluk Model Hacminin İçinden Geçen Akışkanın Ansys Yazılımında Global Koordinatlara Göre Dağılımınının Simülasyonu”, I.Uluslararası Battalgazi Multidisipliner Çalışmalar Kongresi-Malatya / Turkey, Cilt-1, pp. 114-125, 7-9 December 2018.

