Journal of Materials and Manufacturing 3(2): 18-31 (2024)

Journal of Materials and Manufacturing

Design and FEM analysis of hatch ramps for sea freight transport

Omer Sengul[a*], Ferhat Caliskan[a]

Hatice Bayraktar Vocational and Technical Anatolian High School, Kocaeli,4100,Türkiye
(ORCID: 0000-0002-2253-5064), omersengullll@hotmail.com
(ORCID: 0009-0002-5517-0034), ferhatcaliskan_38@hotmail.com

Abstract

The capping ramp is a movable bridge between the ferry and the port during the disembarkation of land, sea and rail vehicles over the ferry to unload their cargo. It is a bridge that can be raised and lowered by means of hydraulic cylinders and placed on the ship to reduce the static and dynamic level differences that may occur in this way in unit distance and to minimize the ramp climbing angle of these level differences, especially in railway vehicles. Ferries are equipped with rail channels and rails to carry railroad vehicles. In this study, the static analysis of the hatch ramp is carried out by finite element method. In the study, the ramp, which consists of four slices, is designed so that a row of wagons can pass over each slice and one loaded truck can pass over each slice. The total wheel mass of each wagon is 100000 kg, and the total wheel mass of the truck is 33500 kg. Since the ramp slices were designed to be equivalent, the rails on the ramp had to coincide with those on the ship, which caused small differences between them. Since this does not affect the results of the finite element analysis, there is no need to calculate each slice separately. The model was built to cover two slices and half of the fixed part. A half model was prepared according to the symmetrical axis parallel to the YZ plane and finite element analysis was performed. The entire ramp beam was modeled, and the semi-model was not used because it is simpler in structure than the ramp. Since it carries only the weight of the ramp, it was modeled and calculated separately. The deflection caused by the wagon is 29 mm and the deflection ratio is 1/700. According to the yield strength, the safety coefficient is 1.87. The weight of the beam itself and the weight of the ramps cause a deflection of 41.5 mm at the midpoint and the deflection value according to the beam length is 1/500. The safety coefficient of the beam is 1.64. It is seen that the stress and deflection values of the beam meet the expectations. As a result, it was found that the compound stress values were satisfactory, and the stress and deflection values met the expectations.

Keywords: Clamshell Ramp, Finite Element Analysis, Static Analysis, Modelling

1. Introduction

Land, sea and rail transportation are among the most widely used modes of transportation in national and international transportation. Today, 90% of transportation in the world is carried out by land, sea and rail [1]. In this way, it facilitates the transportation of large quantities of loads that constitute industrial raw materials from one place to another at one time. This type of transportation is the most widely used mode of transportation in the world because it is reliable, minimizes the loss of goods and is more economical. During the disembarkation of land, sea and railway vehicles over the ferry to unload their cargo, a capping ramp is used to create a movable bridge between the ferry and the port [2]. A capping ramp is a bridge that can be raised and lowered by hydraulic cylinders to reduce the static and dynamic level differences that may occur in the unit distance and to minimize the ramp climbing angle of these level differences, especially in railway vehicles. Ferries are equipped with rail channels and rails to carry railroad vehicles. In this context, academic studies on the hatching ramp are of great importance.

In this context, it would be advantageous to make a computer model of the assembly before starting experimental studies and production [3-4]. In an experimental study, features such as determining the

* Corresponding author
E-mail addresses: *omersengullll@hotmail.com*
DOI: 10.5281/zenodo.14274757
Received: 25 July 2024, Revised: 28 September 2024, Accepted: 16 October 2024
ISSN: 2822-6054

dimensions and parameters of the experimental setup and samples, determining the critical points where data will be obtained during the experiment can be realized by finite element modeling and analysis of these models (Ozer, 2006). In this context, finite element method applications are used in existing package programs. Package programs such as ANSYS, SAP (Structural Analysis Program), ABAQUS are used for finite element modeling [2-5,9]. Such package programs have been made because of the joint work of many experts for a long time. In addition, considering the economical dimension in the preparation of individually prepared programs, it is more advantageous to use existing package programs [6-7].

In literature, many experimental studies have been conducted [8]. Before experimental studies are carried out and production starts, it is thought that making a computer model of the setup and performing finite element analysis will provide great advantages. For this reason, in this study, a finite element analysis of the marine and railway hatch ramp was performed with ANSYS 14.0 package program.

2. Material and Method

In this study, an experimental study was carried out to determine the data to be used in the static analysis of the marine, land and railway hatch ramp and ANSYS 14.0 package program was used for finite element method analysis. AH36 standard steel material was used in the study, the mechanical properties of the steel material are given in Table 1. The material in Table 1 is used as ship and ramp construction material. This material is also commonly used to utilize surplus materials in shipbuilding. The ramp model for the capping ramp used in the study is shown in Figure 1, the ramp beam model is shown in Figure 2 and the overall model for the capping ramp is shown in Figure 3.

Table 1. Mechanical Properties of AH36 Steel [7].

Material	Yield Strength (N/mm^2)	Tensile Strength (N/mm^2)	Elongation at Break (%)	Density (kg/m^3)	Modulus of Elasticity (MPa)	Poisson's Ratio
AH36	360	490-630	20	7850	2.1E+5	0.3

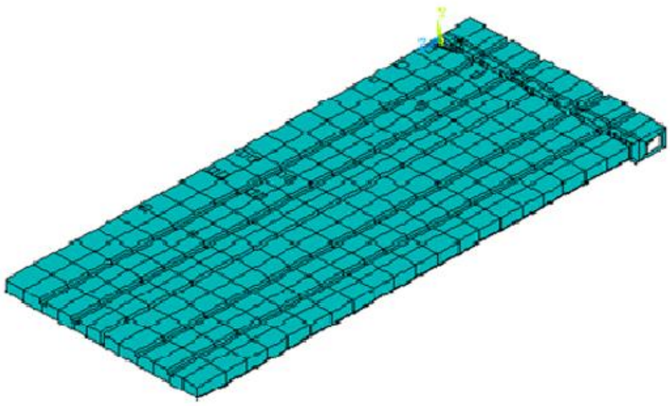

Fig. 1. Ramp model

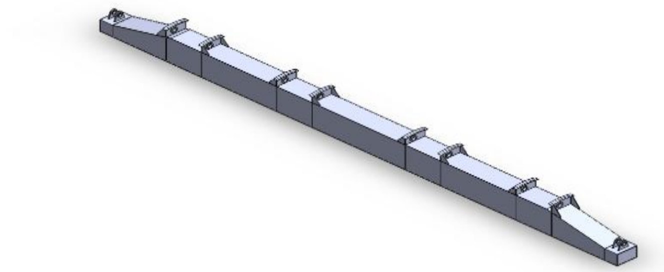

Fig. 2. Ramp beam model

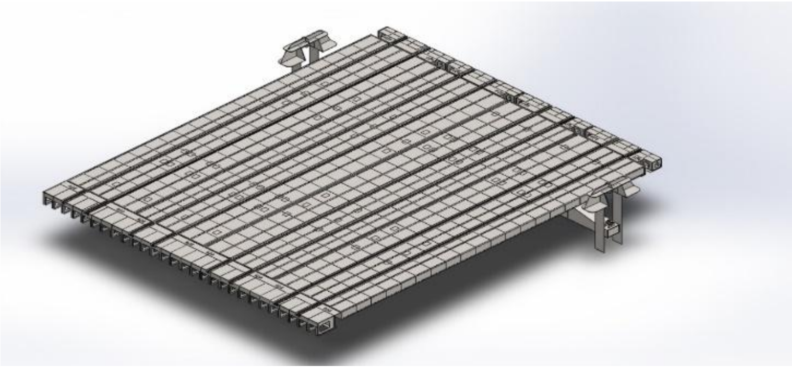

Fig. 3. Full model of the close dump ramp

2.1. Finite Element Method

The structural elements that make up the model are defined in the model by considering the geometric dimensions of the structure, the loads on the structure, the movement capabilities and degrees of freedom of the bearings and joints of the elements. The objective of mathematical modelling is to enable the observation of the actual behaviour of the structural elements of the whole or a certain part of the structure under different loads or different physical effects. The actual behaviour of the structure is quite complex. Therefore, many simplification methods are necessary to design the structure. To make a simple model, it is very important to properly determine the physical properties of the material constituting the structural elements [2-8,10]. In this study, ANSYS 14.0 program was used for the finite element method. Although the ramp slices were designed to be equivalent, the necessity of the rails on the ramp to coincide with those on the ship caused small differences between them.

Since these differences did not affect the results of the finite element analysis, it was not necessary to calculate each slice separately. The model was built to cover two slices and half of the fixed part, i.e. parallel to the YZ plane and half of the model according to the axis of symmetry. Although the solutions were made in the same model, the hinges of the fixed frame and both slices were kept independent of each other, and no contact was made between the hinges. The hinges of the slices were fixed, and truck loads were applied to one slice and wagon loads to the other. The solution within the same model was done for ease of modeling and analysis. When assigning the truck load, the wheel tracks were considered in the downhill direction where the truck would create the heaviest conditions and the center wheel group was in the middle of the ramp. It is assumed that the truck will travel on the edge of the ramp and will also encounter the maximum torsional load of the ramp slice and the wagon pair is assumed to have their wheel groups in the middle.

2.2. Determination of Loads

Truck load values are given in Table 2 and technical specifications are given in Table 3. Truck and wagon loads, wheel tracks were modeled as separate surfaces on the model and the compressive loads on the areas according to the wheel group load values are shown in Figure 4. Ramp and fixed frame hinges were modeled in accordance with their actual dimensions and thicknesses, but in such a way that forces and bearings can be easily given, and the ramp hinges were given bearing properties and the fixed frame hinges were given force values that can be transmitted from the ramp.

The ramp slices are supported from the hinges connected to the fixed section and the ramp ends resting on the ship. The loads on the beam were assigned as point loads to the keypoints at the top surface center of the slice carriers. The cylinder lugs were modeled as triangular structures in accordance with the actual lug thickness and dimensions and their movements in the X, Y and Z axes were limited at the end points. Hinge assignment loads are shown in Figure 5 and ramp beam loads are shown in Figure 6.

Table 2. Truck load values

Truck front wheel increased pressure (MPa)	Truck rear wheel increased pressure (MPa)	Truck rear wheel increased pressure (MPa)	Wagon increased wheel load (MPa)
0.475	0.253	0.211	11.147

Table 3. Technical specifications

Slice bearing load (Newton)	Ramp slice center of gravity (mm)	Ramp slice mass (kg)	Beam center (mm)	Slice carrier quantity
121 182,352	10 500	40 000	17 000	2

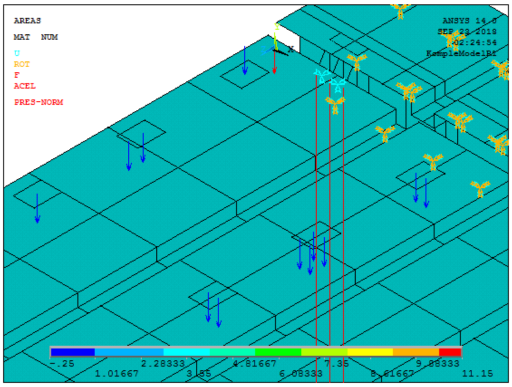

Fig. 4. Pressure loads

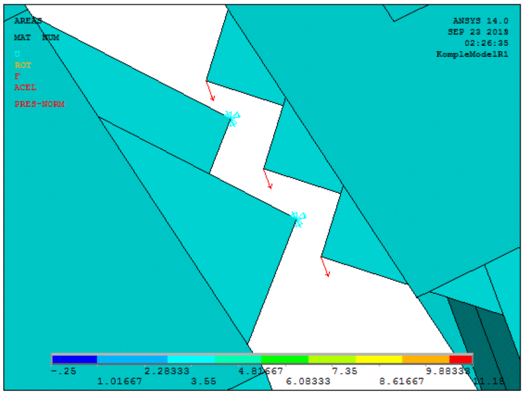

Fig. 5. Hinge assignment loads

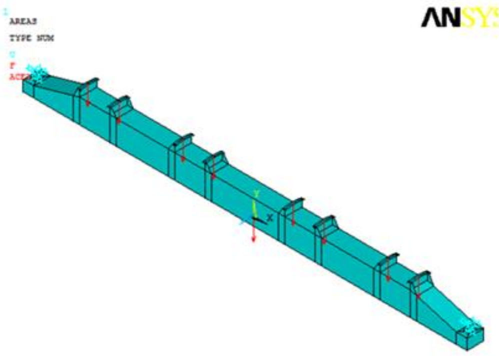

Fig. 6. View of beam compressive loads

2.3. Model

The model was modelled to cover half of the real ramp and is shown in Figure 7. In the ramp consisting of a welded sheet metal structure, the joining of the sheets was modelled with the principle that the point and edge lines are common on the neigh boring surfaces, and it was ensured that this structure did not change during modelling by performing a glue operation. Especially in the lower flag structures and in the areas adjacent to the areas where the forces are given, multi-edged surfaces were created, and the element type SHELL181 was selected to make these surfaces free mesh. The entire beam was modelled. Since it is simpler than the ramp in terms of structure, the half model was not used, since it carries only the weight of the ramp, it was modelled and calculated separately. The modelling technique is the same as in the fixed part of the ramp slices. Sheet metal structures are joined with glue operation using common points and lines and no contact is used. Again, SHELL181 element was used, and free mesh was used. Since the ramp material is AH36 steel material with a yield strength of 360 MPa, standard steel material properties were entered into the program and material properties were defined. Sheet thicknesses were added with the "Section properties" feature after the modelling work was completed. Figure 7 shows the model view of the cover drop ramp.

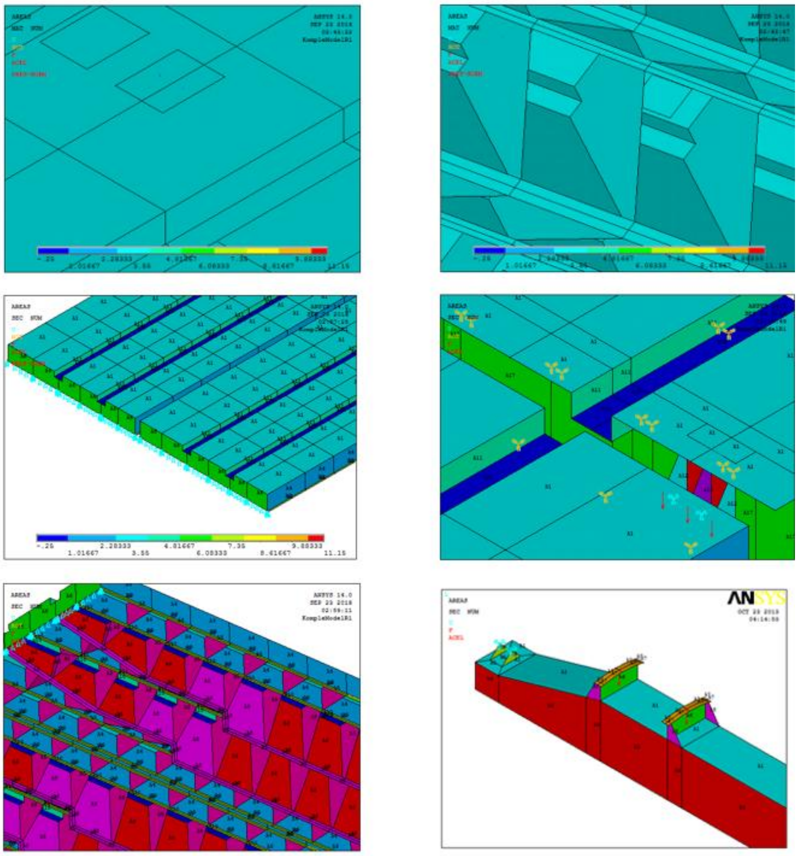

Fig. 7. Lid drop ramp model view

2.4. Ramp fixed section analysis

The fixed section is the fixed steel structure required to connect the ramp sections to the land. Since wagons and trucks also pass over the fixed part connected to the ramp by means of hinges, wheel loads were also considered together with the hinge load. Fixed part hinge model is shown in Figure 8.

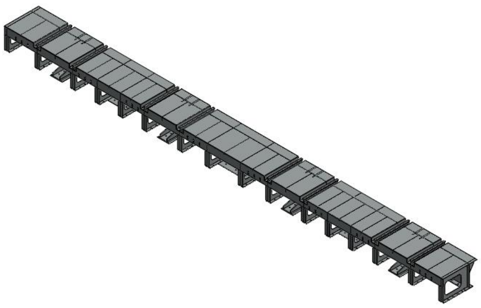

Fig 8. Fixed part hinge model

2.4.1. Determination of loads

Ramp loads are included in the analysis in the form of hinge and wheel loads, which are specified and calculated at the beginning of the document. Half of the fixed part was modelled, and the analysis was performed on this model since it is symmetrical and anchored at multiple points. As in the previous analyses, the hinge eyebolts were modelled as diagonals with the ends where the load would be applied. The fixed section is bounded in three axes by the intermediate bulkhead flanges where the lower anchors are made and the lower flanges of the hinge area. Wheel loads were applied to the divided wheel areas on the respective surfaces. Fixed part hinge loading view is shown in Figure 9.

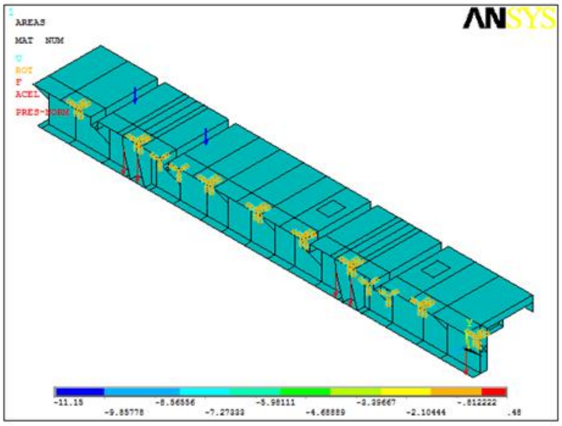

Fig. 9. Fixed part hinge loading view

2.4.2. Model

As mentioned before, the fixed part is modelled in half and no symmetry constraints are required due to the multi-point connection. The modelling technique is the same as in the ramp slices and the fixed part, and the sheet metal structures relate to the use of common points and lines and glue operation, no contact is used. Again, SHELL181 element was used, and free mesh was used.

2.4.3. Material

Since the fixed part material is AH36 with a yield strength of 360MPa, standard steel material properties are entered into the program. Illustration of the sheet thickness of the fixed part hinge is shown in Figure 10.

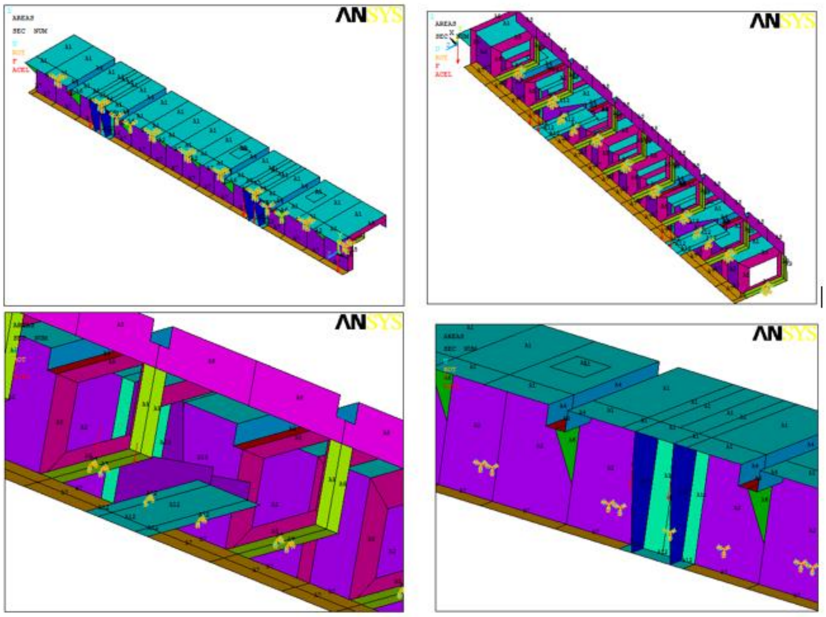

Fig. 10. Illustration of the sheet thickness of the fixed part hinge

2.4.4. Mesh

Free mesh type was used to mesh polygonal surfaces. The total number of elements is 14874. Mesh representation of the fixed part hinge is shown in Figure 10.

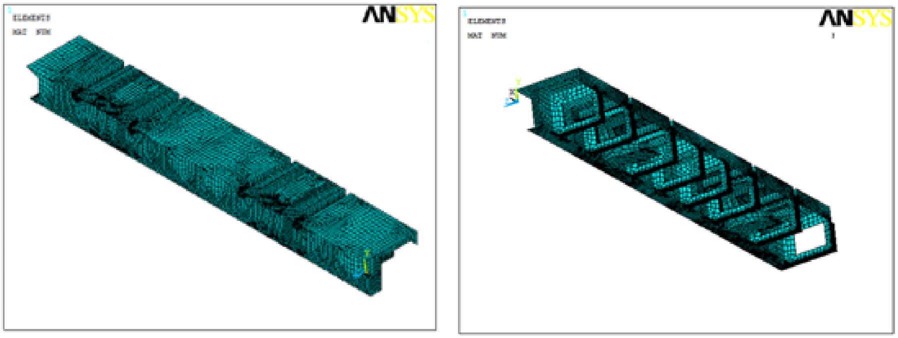

Fig. 11. Mesh representation of the fixed part hinge

2.5. Cylinder tower calculation

The roller tower contains the upper connections of the rollers that allow the ramp to be lifted by means of the lower beam when empty. The towers are welded to plates anchored in the concrete and connected to the concrete in such a way that they fit snugly against these plates. Their load is vertical as they carry the weight of the ramp and beam. Cylinder Tower Model View is shown in Figure 12.

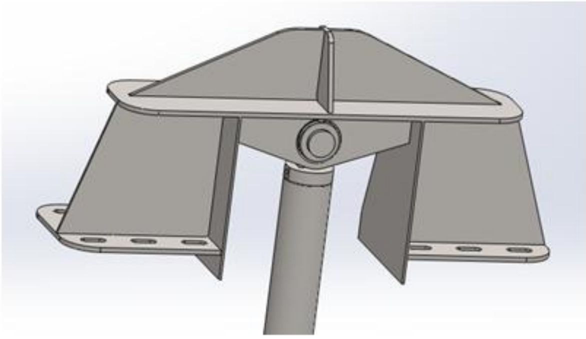

Fig. 12. Cylinder Tower Model View

2.5.1. Determination of loads

The tower loads are assumed to be 40000 kg per tower in case of lifting the ramps of 40000 kg each with a canter of mass of 7 meters from the joint by means of a beam at a distance of 170 meters from the joint. Cylinder Tower Model Loading View is shown in Figure 13.

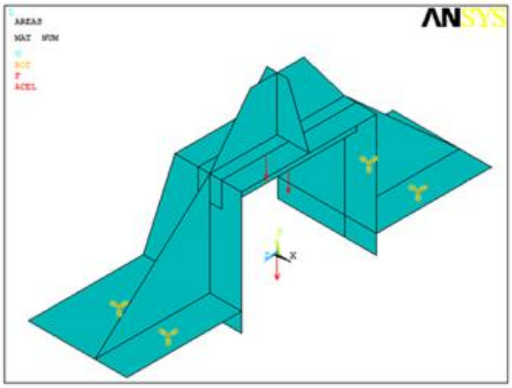

Fig. 13. Cylinder Tower Model Loading View

2.5.2. Model

In the tower model, the cylinder lugs are modelled in a polygonal shape to give realistic reaction loads to the sheet to which they are connected. The cylinder tower, which consists of a sheet metal structure, was prepared as a surface model, SHELL181 element was used, and free mesh was made.

2.5.3. Material

Since the tower material is AH36 with a yield strength of 360MPa, standard steel material properties were entered into the program.

2.5.4. Sheet metal K2 pediments

Sheet thicknesses were added with the Section properties feature after the modelling work was completed. For the material, the modulus of elasticity and Poisson's ratio, which are common to almost all steels, are entered. Yield and rupture evaluation are interpreted according to the analysis results. Cylinder Tower Model Hair Thickness View is shown in Figure 14.

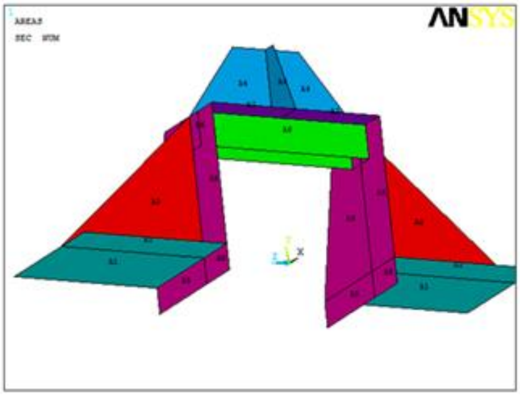

Fig. 14. Cylinder Tower Model Hair Thickness View

2.5.5. Mesh

Free mesh type was used to mesh polygonal surfaces. Since the design model is not complex, a free mesh (general mesh) structure was chosen. The total number of elements is 2129. Cylinder Tower Model Mesh View shown in Figure 15.

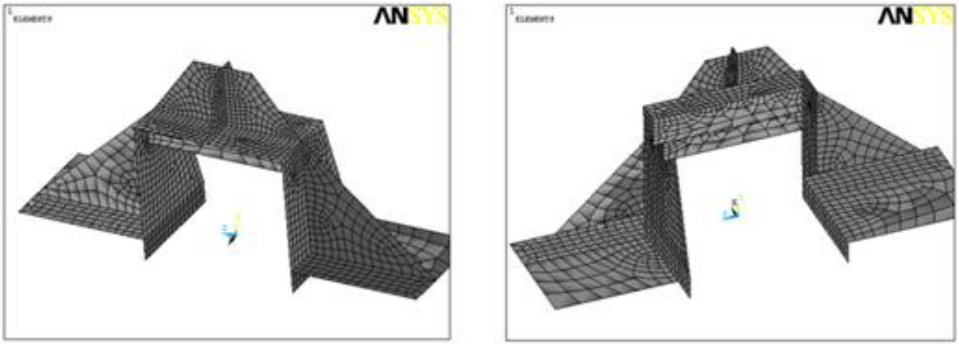

Fig. 15. Cylinder Tower Model Mesh View

3. Results and Discussion

Finite element analysis of the lid dump ramp was performed with ANSYS program. In the analysis, free mesh type was used to mesh polygonal surfaces. The total number of elements is 142 945. The maximum Von

Mises stress value was 192.4 MPa. The yield strength of the material was 360 MPa and the safety coefficient according to the yield strength was 1.87. The maximum value of 192.4 MPa was observed at a point which was too small to be considered at the junction of the ramp slice hinge sheets and longitudinal support elements. The value of 192.3 MPa, which is close to this value, was found in the area with section thinning in front of the ramp slice. If the cross section is thinned in this region by using circular elements, the stress value can be reduced to 140 MPa stress value at the bottom flange of the same longitudinal element. The compound stress value results are satisfactory. Shear stress values are concentrated in the same region and remain around 40 MPa. In the beam, the free mesh type was used to mesh polygonal surfaces [8].

The total number of elements is 123 92. The Von Mises stress value was found to be 540.8 MPa. The part that needs to be examined around the eyebolts is the areas where the cylinder connection eyebolts meet the beam, where the stress value is 220 MPa and the highest stress value is found in this region. For the material with a yield strength of 360 MPa, the safety coefficient was obtained as 1.64. In the middle of the beam, the composite stress value due to shear and bending was found to be approximately 100 MPa. The results obtained were consistent with the literature [5-8]. In the analysis, 33 mm deflection is observed at the ramp. This deflection value is realized under the worst conditions when the truck walks on the edge of the ramp slice. The deflection caused by the wagon is 29 mm and the deflection ratio is 1/700. According to the yield strength, the safety coefficient is 1.87. The weight of the beam itself and the weight of the ramps cause a deflection of 41.5 mm at the midpoint and the deflection value according to the beam length is 1/500. The results obtained were consistent with the literature [7-9]. The safety coefficient of the beam is 1.64. It is seen that the stress and deflection values of the beam meet the expectations. This study will facilitate the detection of engineering errors in ramps to be newly designed and analyzed.

4. Conclusion

Von Mises stress value of 96.5 MPa was observed at the fixed part of the ramp. This stress value is observed in point areas that are not considered at the hinge ends and in the rectangular area where the truck wheel load is given. Values of 50-60 MPa are commonly observed in hinge sheet metal. Again, the deflection is around 1.6 mm in the unsupported area where the truck wheel is pressed. These values are shown in Figure 16 and 17.

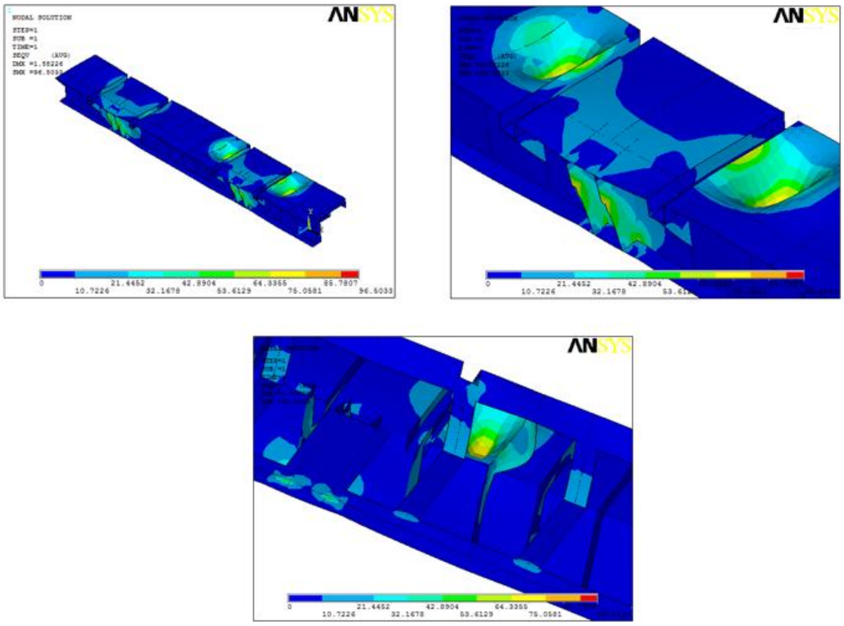

Fig. 16. Van Mises Compound Stress Results of the fixed part hinge

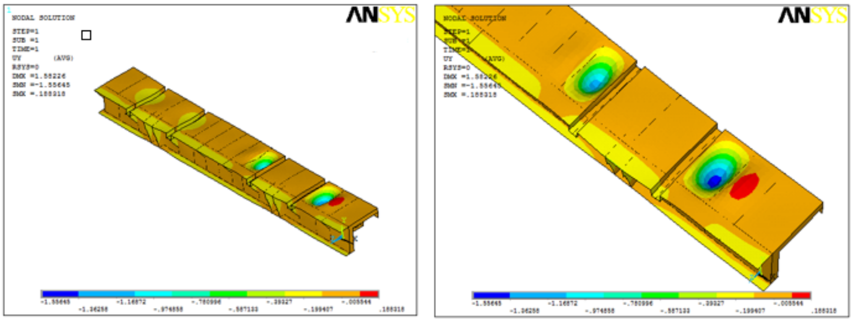

Fig. 17. Shear Stress Results of the fixed part hinge

Shear stresses are around 20 Mpa under the hinge plate and 25-32MPa in the wheel compression areas. Stress and deflection values in the fixed section meet the expectations. These values are shown in Figure 18.

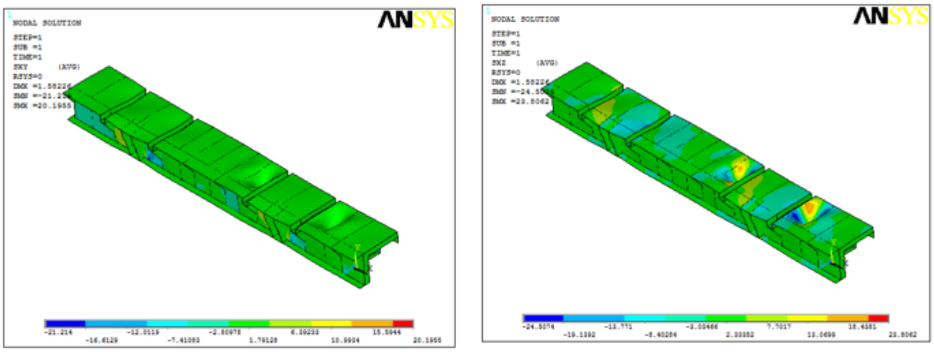

Fig. 18. Deflection Value Results of the fixed part hinge

In the analysis of the Cylinder Tower, the Von Mises stress value was 94 MPa. These values occurred at the point where the force was applied to the polygon modeled instead of the eyebolt and at the connection to the body. The maximum shear stress value was 41 MPa at the connection point of the eyebolt sheet to the top flange. The deflection was 0.36mm at the end of the eyebolt. These values are shown in Figure 19.

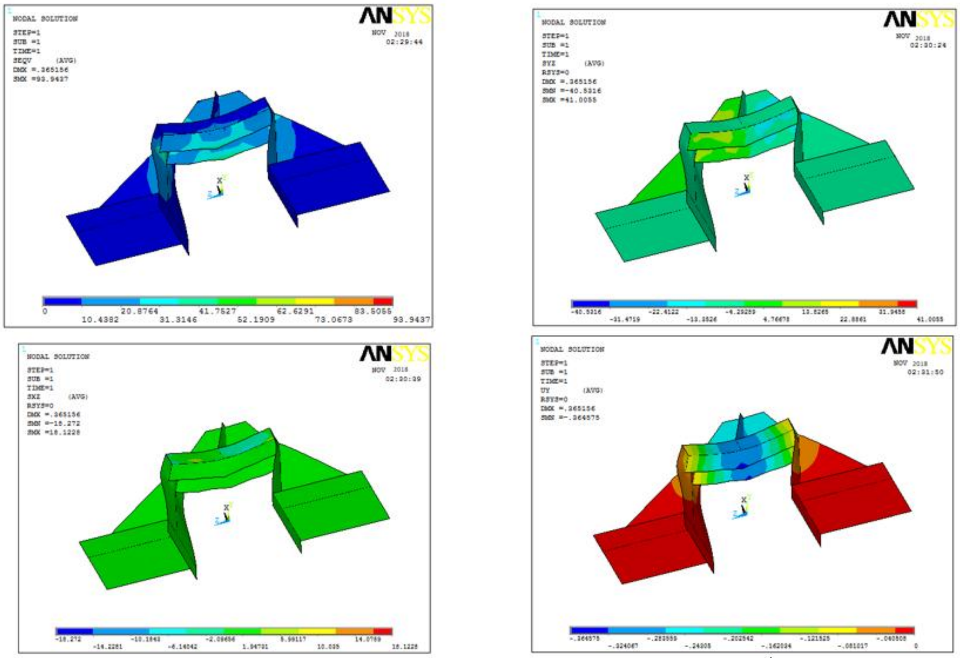

Fig. 19. Cylinder Tower Analysis Result

References

[1] Yıldız, A., "The history of shipyards in Turkey and the development of shipbuilding industry", Journal of Engineer and Machinery, 578 (49): 23-47 (2008).

[2] Şengül, Ö., Kurt, M., Mutlu,B., Atakök, G., (19.10.2017 -22.10.2017) , Publication Place:8th International Advanced Technologies Symposium (IATS'17) , 2017 Motorcycle Tire Design and FEM Analysis

[3] Parliamentary Research Commission, "Tuzla report", Official Gazette, (26868): 8-85 (2008).

[4] Lee, J. M., Seo, H. D., and Chung, H., "Efficient welding distortion analysis method for large, welded structures", Journal of Materials Processing Technology, 256 (November 2017): 36-50 (2018).

[5] Wang, J., Zhao, H., Zou, J., Zhou, H., Wu, Z., and Du, S., "Welding distortion prediction with elastic FE analysis and mitigation practice in fabrication of cantilever beam component of jack-up drilling rig", Ocean Engineering, 130 (March 2016): 25-39 (2017).

[6] Sun, G. F., Wang, Z. D., Lu, Y., Zhou, R., and Ni, Z. H., "Investigation on microstructure and mechanical properties of NV E690 steel joint by laser-MIG hybrid welding", Materials and Design, 127 (January): 297-310 (2017).

[7] Li, L., Cao, J., Gu, H., Yang, P. and Luo, G. (2015). Strength analysis of single module of VLFS based on direct calculating method, Shipping Engineering, 37, 67-71(in Chinese).

[8] DNV (2010). Recommended practice DNV-RP-C205: Environmental conditions and environmental loads.

[9] Özüpak, Y. (2023). Elektrikli araçların kablosuz güç aktarım sistemi ile şarj edilmesi için kullanılan farklı transformatör modellerinin sonlu elemanlar yöntemi ile tasarımı ve analizi. Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi, 39(2), 1113-1122. https://doi.org/10.17341/gazimmfd.1202033

[10] Yildiz, A. (2008), History of Shipyards in Turkey and Development of Shipbuilding Industry, Journal of Engineer and Machinery, 578(49), 23-47.