International Journal of Offshore and Coastal Engineering
Vol.8 | No. 1 | pp. 1 – 5 | May 2024
e-ISSN: 2580-0914

Submitted: January 6, 2024 | Revised: March 18, 2024 | Accepted: April 14, 2024

Padeye Strength Analysis of Topside Offloading Platform due to Loadout Using Lifting Method

Ilham Kharisma Prayoga[a*], Yoyok Setyo Hadiwidodo [a] and Handayanu [a]

[a)] Department of Ocean Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia
*Corresponding author: ilhamkharisma21@gmail.com

ABSTRACT
Lifting is one method in the loadout process. Lifting is the activity of moving a structure using the help of a crane. The loadout process moves the structure from the yard to the barge. In this study, a lifting analysis was carried out on the topside offloading platform structure. The analysis also considers the center of gravity shift factor, dynamic load factor (DAF), and other safety factors that occur in the structure during the lifting process. Dynamic load factor using Dynamic Amplification Factor. Cog shift during lifting affects the amount of load being lifted. The most significant load received by the lifting point is 872.507 kN at lifting point 3. During the lifting process, no failure was found in the structural members, as evidenced by the maximum UC value of 0.87. The Padeye used is designed according to the DNV OS-H205 criteria. Several checks on the padeye structure were carried out, comparing the stress that occurred with the allowable stress on several stress reviews, such as tensile stress, shear stress, and bending stress. Local analysis was also carried out on the padeye structure to determine the stress on the padeye structure. Local analysis of the padeye structure was conducted using the ANSYS Workbench software. The results of the regional analysis showed the equivalent von-mises of 164.96 MPa in the padeye structure and 140.04 MPa at the joints, with the allowable stress of ASTM A36 steel material of 250 MPa.

Keywords: *Lifting, crane, topside*

1. INTRODUCTION

Indonesia is one of the world's oil and gas-producing countries, and most of the oil and gas are in the ocean. However, oil and natural gas in the earth's bowels are found on land and under the seabed, so a particular structure is required to extract the oil or gas on the seabed.

An offshore Platform is a structure used to explore and exploit oil and gas on the high seas. One of the most critical functions of an offshore platform structure is to support the superstructure and its facilities and ensure safe operations and production activities during operating hours [1].

To develop exploration and exploitation in the oil and gas sector, one of the companies engaged in oil and gas mining plans to build an offloading platform. The offloading platform will be installed near the floating storage regasification unit (FSRU) to assist the unloading process. The construction of offshore structures is carried out separately based on sections or modules. After the fabrication, the structure will be moved from the yard to the barge. This transfer process is called the loadout process. Lifting is one of the loadout methods using a crane.

During lifting activities, structural failures such as fractures are often found. This failure occurs because the calculations are only static and do not consider the actual situation in the field. The lifting method must consider many factors, especially those supporting the lifting process, such as lifting points, hook points, shackles, and padeyes [3]. Padeye is a structure that connects the structure and the shackle. The stress in the padeye must be considered to determine whether or not the padeye fails during the lifting process. Therefore, it is necessary to analyze the padeye structure during the lifting process of the topside offloading platform at the loadout stage.

2. METHODOLOGY

The method used in analyzing the padeye structure in the lifting process of the topside offloading platform is static analysis. The steps taken in this research can be explained as follows:

2.1 Data Collection and Literature Study
A literature study is used to obtain books or journals related to the work of the study, collecting data in the form of structural materials, structural loads, and structural models. The structure used is a topside offloading platform belonging to a company engaged in the O&G sector.

2.2 Structural Modeling
In this study, the structure analyzed is the top side of the offloading platform and the padeye structure. These structures will be modelled using software to perform lifting analysis. The following is a model of the topside and padeye structures:

a. Topside Structure

The topside structure is modelled using SACS 5.7 software. Figure 1 shows the topside offloading platform structure modelling using SACS 5.7.

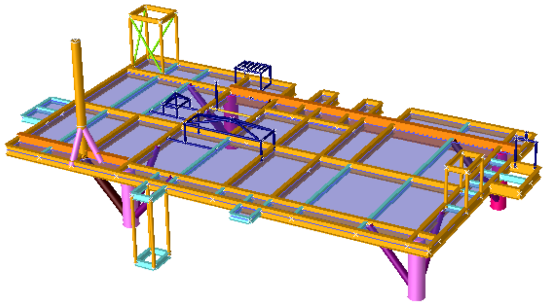

Figure 1. Topside Offloading Platform

b. Padeye Structure

The padeye structure is designed using ASTM A36 steel material with a yield strength of 250 MPa. Padeye structure modelling using ANSYS Workbench software. The padeye structure model and padeye data are shown in Figure 2 and Table 1.

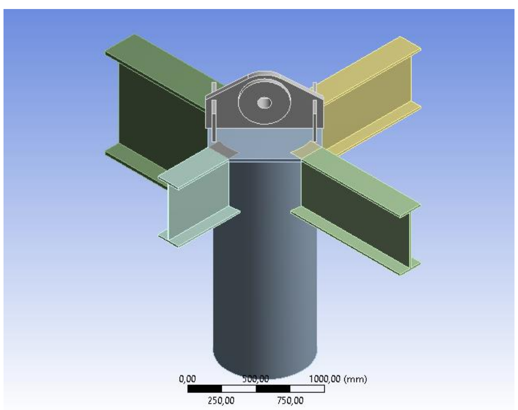

Figure 2. Padeye

Table 1. Dimension of Padeye

Dimension	Symbol	Total	Units
Hole Diameter	Dh	100	mm
Main Plate Radius	Rpl	225	mm
Cheek Plate Radius	Rch	190	mm
Main Plate Thickness	Tpl	50	mm
Cheek Plate Thickness	Tch	36	mm
Padeye Length	L	862	mm
Padeye Height	Ht	475	mm
Hole Height	Hh	250	mm
Hole Distance	Lh	431	mm
Stiffner Height	Hs	250	mm
Stiffner Length	B	400	mm
Thick of Stiffner	s	25	mm

2.3 Calculation of Structure Loads

The weight and COG structure will be obtained after modelling the topside structure using SACS software. The COG location is used to get the hook point. COG Position of the Topside Offloading Structure Platform is X = 12.11 m, Y = 5.84 m, and Z = 0.35 m, with a total structure weight of 1,359.93 kN.

2.4 Static Analysis

The load calculated in lifting the topside offloading plaftorm structure includes the total load of the structure itself and the load above the deck, including equipment, pipes, rigging, and others. The analysis of lifting also takes into account dynamic loads.

The dynamic load in the lifting process is influenced by several parameters, such as rigging configuration, crane condition, the weight of the lift structure, and environmental loads [3]. The dynamic load can be used as a static load factor called a Dynamic Amplification Factor (DAF). Several other factors are used in the analysis of the lifting process, such as the contingency factor, skew load factor, and consequence factor, as safety factors. The following is the load factor used in the lifting topside analysis:

Table 2. Load Factor

Description	Load Factor
Dynamic Amplification Factor (DAF)	1.1
Contingency Factor	1.1
Skew Load Factor	1.25
Consequence Factor	1.15

The static analysis also considers a shift in COG or COG shifting. Based on the lifting document, the cog shift can be calculated as a static load factor. This COG shift is calculated to be able to shift as far as 1-2 meters towards +X+Y, -X+Y, -X-Y, and +X-Y. The output of the static analysis is to get the structure's response in terms of each member's UC value, the stress value that occurs at the lifting point (joint reaction), and the deflection in the structure.

Static analysis was also carried out on the padeye structure by checking the stress on the padeye to determine the strength of the padeye structure. Check stress on padeye structure using AISC 9th Edition codes. Furthermore, regional analysis using the ANSYS Workbench software.

2.5 Local Analysis of Padeye Structure Using ANSYS

To determine the strength of the padeye structure when the lifting process is in progress, a local analysis that reviews the padeye structure as a whole is needed. This analysis uses the finite element method with the help of ANSYS software. It was carried out on the padeye structure, which received the most significant load.

Local analysis of the padeye structure examines two parts: the padeye as a whole and the connection between the deck plate and the padeye.

3. ANALYSIS AND DISCUSSION

3.1 Topside Static Analysis

The topside structure modelling using SACS 5.7 software is then performed using static analysis to determine the stresses in the topside structure. The analysis results in a unity check on each structure member, the lifting point stresses, and the Center of Gravity value on the topside structure. The following are the results of a static analysis of the topside structure using SACS 5.7.

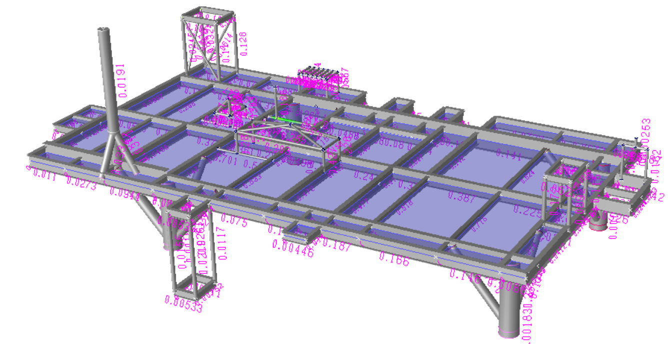

Figure 3. UC Value on Topside Structure

Table 3. Unity Check Maximum

Member	Unity Check Max
0182-0181	0.87
0180-0182	0.87
0111-0126	0.70
0128-0131	0.62

Table 4. Maximum Load at Lifting Point

Lifting Point	Maximum Reaction (kN)
LP1	839,186
LP2	698,999
LP3	872,507
LP4	628,134

The static analysis results show that no member failed during the lifting process. The value of the unity check on the structure is below number one, which means that during the lifting process, no stress is generated that exceeds the structure's allowable stress.

3.2 Padeye Static Analysis

Following the DNV OS code H-205, several things that must be considered in making the padeye design are the shackle has distributed loads, the element's type and size, and the stress or stress that occurs in the pinhole and attachment. Static analysis was carried out on the attachment and pinhole sections of the padeye. Static analysis on padeye aims to determine the stress that occurs on padeye. The stress checks include shear stress, bending stress, tensile stress, bearing stress, and tear-out stress. Allowable stress is used according to AISC 9th Edition codes.

a. Attachment of Padeye

Padeye is designed with attachments. Calculating the stress on the padeye attachment aims to determine the stress in the attachment. Calculations performed are shear stress, tensile stress, and bending stress. Here are the results of the static analysis on the attached padeye:

Table 5. Shear Stress Check on Attachment

Shear Stress in Y Direction					
Horizontal Force (kN)	Shear Area (mm^2)	Shear Stress (MPa)	Allowable Stress (MPa)	UC	Validation
366,786	60,600	6,053	100	0.06	OK
Shear Stress in X Direction					
Lateral Force (kN)	Shear Area (mm^2)	Shear Stress (MPa)	Allowable Stress (MPa)	UC	Validation
26,176	60,600	0.432	100	0.004	OK

Table 6. Tensile Stress Check on Attachment

Tensile Stress					
Vertical Force (kN)	Tensile Area (mm^2)	Tensile Stress (MPa)	Allowable Stress (MPa)	UC	Validation
827,507	60,600	14	14,398	0.096	OK

Table 7. Bending Stress Check on Attachment

In-Plane Bending					
Moment (kN-mm)	Elastic Section Modulus (mm^3)	Bending Stress (MPa)	Allowable Stress (MPa)	UC	Validation
91696	9,794,295.5	9.36	165	0.057	OK
Out-Plane Bending					
Moment (kN-mm)	Elastic Section Modulus (mm^3)	Bending Stress (MPa)	Allowable Stress (MPa)	UC	Validation
6544,1	4,715,156.3	1.39	165	0.008	OK

The stress check calculation in the table above shows that the stress in the padeye attachment is still below the allowable stress set by AISC 9th Edition. So that the stress that occurs in the padeye attachment can be declared safe because the UC value is below 1.

b. Pinhole of Padeye

The calculation of the stress on the pinhole aims to determine the stress that occurs so that the strength of the padeye, especially the pinhole part, can be known. The calculations include tensile stress, shear stress, tear out, and bearing stress.

Table 8. Tensile Stress Check on Attachment

Tensile Stress in Vertical Direction					
Vertical Force (kN)	Tensile Area Vertical (mm^2)	Tensile Stress (MPa)	Allowable Stress (MPa)	UC	Validation
946,47	27,580	31,637	112.5	0.28	OK

Tensile Stress in Horizontal Direction					
Horizontal Force (kN)	Tensile Area Horizontal (mm^2)	Tensile Stress (MPa)	Allowable Stress (MPa)	UC	Validation
366,786	38,910	24.32	112.5	0.22	OK

Table 9. Shear Stress Check on Pinhole

Shear Stress in Vertical Direction					
Horizontal Force (kN)	Shear Area (mm^2)	Shear Stress (MPa)	Allowable Stress (MPa)	UC	Validation
366,786	37,660	9,739	100	0.1	OK
Tensile Stress in Horizontal Direction					
Lateral Force (kN)	Shear Area (mm^2)	Shear Stress (MPa)	Allowable Stress (MPa)	UC	Validation
26,176	37,660	0.7	100	0.007	OK

Table 10. Tear-Out Stress Check on Pinhole

Tear Out					
Sling Force (kN)	Tear Out Area (mm^2)	Tear Out Stress (MPa)	Allowable Stress (MPa)	UC	Validation
946,471	37,660	25.13	75	0.335	OK

Table 11. Bearing Stress Check on Pinhole

Bearing Stress					
Sling Force (kN)	Bearing Area (mm^2)	Bearing Stress (MPa)	Allowable Stress (MPa)	UC	Validation
946,471	11,590	81.66	225	0.363	OK

The stress check calculation in the table above shows that the pinhole padeye's stress is still below the allowable stress set by AISC 9th Edition. So, stress in the padeye attachment can be declared safe because the UC value is below 1.

3. 3 Padeye Local Analysis using ANSYS

Padeye modelling will be analyzed using ANSYS Workbench software. After the modelling, entering the load, determining the support, and analyzing the meshing sensitivity is the analysis stage. The analysis carried out on the ANSYS Workbench is static structural with Von-Mises stress output on the padeye and the connection between the padeye and the plate on the deck leg. The stress on the connection needs to be reviewed because the load obtained from the padeye is distributed to the structure through the connection between the padeye and the plate on the deckleg. By reviewing the stress at the connection, it can be seen whether or not the connection fails.

The force included in the padeye model is the force that occurs in the sling. The magnitude of the force entered in the ANSYS Workbench software is the most significant force in the sling. The biggest load occurs on the sling at lifting point 3, and the following is the force input to the ANSYS Workbench in Table 12. After running the ANSYS workbench software, the stress value in the padeye structure is obtained. The following are the results of the stresses in the padeye structure and the connection between the padeye and the deck leg plate with a meshing size of 23 mm, as shown in Figure 4, Figure 5, and Table 13.

Table 12. Stress on Padeye

Force	Total Force (kN)
Sling Force (Fsl)	946,471

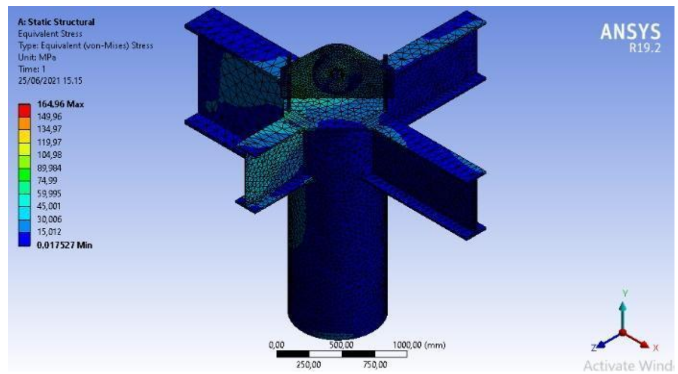

Figure 4. Output Stress Analysis on Padeye

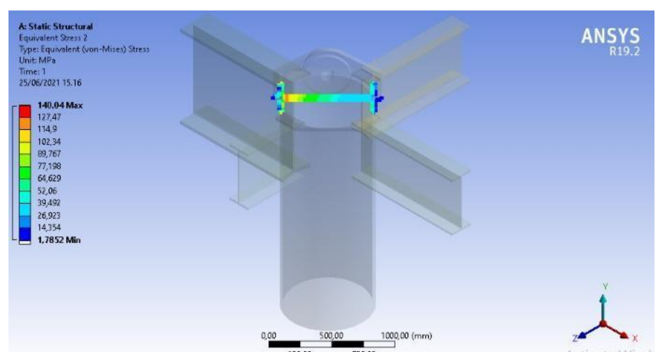

Figure 5. Output Stress Analysis on Padeye Concetion

Table 13. Von-Mises Stress (Maximum – Minimum)

No	Stress on Padeye	Stress on Connection
1	164.96	140.04
2	149.96	127.47
3	134.97	114.9
4	119.97	102.34
5	104.98	89.767
6	89.98	77.198
7	74.99	64.629
8	59.99	52.06
9	45	39.492
10	30	26.923
11	15.012	14.354
12	0.0175	1.785

Figure 4 and Figure 5 shows the results of the von-mises stress and stress modeling in the padeye and the connection between the padeye and the deckleg plate. The analysis results in Table 13 show that the maximum Von-Mises voltage that occurs is 164.96 MPa at the padeye and 140.04 at the junction. With the allowable stress on ASTM A36 steel material being 250 MPa, it can be said that padeye is safe to use in the lifting process, with UC 0.66 for stress on padeye and UC 0.56 for stress on joints.

4. CONCLUSION

The conclusions obtained from the results of this study include:

1. The analysis found that the structure of the topside offloading platform, with a total weight of 1,359.93 kN, did not fail in the structural members. Global stress analysis was carried out using SACS 5.7 software. The highest Unity Check occurred in members 0182-0181 and 0180-0182, 0.87, with a load factor of 1.74.
2. The ANSYS Workbench software performed Local modelling and analysis on the padeye structure. After doing a local analysis on the padeye structure, the results obtained from the equivalent von-mises stress that the largest is 164.96 MPa on the padeye and 140.04 MPa on the padeye connection, with the allowable stress on the ASTM A36 steel material of 250 MPa.

REFERENCES

1. API RP 2A WSD. 2005. Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms – Working Stress Design. Washington DC: American Petroleum Institute.
2. AISC. 2010. ANSI/AISC 360 – 10 Specification for structural Steel Buildings. Chicago. American Institute of Steel Construction.
3. DNV OS H205 Lifting Operation. 2014. Rules of Planning and Execution of Marine Operations. Norway: Det Norske Veritas.
4. GL 0001-ND rev1,1. 2016. General Guidelines for Marine Projects. Recommended Practice. Germanischer Lloyd.
5. GL 0013-ND rev11,2. 2016. General Guidelines for Marine Projects, Recommended Practice, Germanischer Lloyd.
6. Gorat, M. L. P., 2017. Analisis Stabilitas Crane Barge saat Operasi Heavy Lifting. Surabaya: Tugas akhir jurusan Teknik Kelautan ITS.
7. Novanda, A. K., 2012. Analisa Lifting Topside Platform dengan Pendekatan Dinamik Berbasis Resiko. Jurnal Teknik POMITS, I(1), pp. 1-6.
8. Putra, B. K., 2020. Analisis Kekuatan Padeye Pada Saat Lifting Struktur Topside Wellhead Pada Proses Loadout. Surabaya: Tugas Akhir Jurusan Teknik Kelautan ITS.
9. Soelarso, 2015. Analisa Struktur Ula Well Platform Tahap Lifting Dengan Menggunakan Software SACS 5.2 (Studikasus Proyek PT. Bakrie Construction). Jurnal Pondasi, IV(1), pp. 12-19.
10. Yansah, A. R., 2016. Analisis Konfigurasi Rigging dan Padeye Pada Saat Proses Installation Deck Strukture ULA Platform Dengan Cara Lifting. Surabaya: Tugas Akhir Jurusan Teknik Kelautan ITS.