Fatigue Analysis in Ship Engine Bed Construction due to Engine Vibration Using The Finite Element Method (FEM), Study Case of Labritha Karina Ship

Fera Indriani[1], Muhammad Sawal Baital[2]

[1] Students of Marine Construction Engineering Technology, Vocational Collage, Diponegoro University, Semarang, Indonesia
feraindriani@students.undip.ac.id

[2] Department of Industrial Technology, Vocational Collage, Universitas Diponegoro, Semarang, Indonesia
sawalbaital@lecturer.undip.ac.id

Accepted on May 13, 2025
Approved on June 12, 2025

***Abstract*—The engine bed structure on a ship plays a crucial role in supporting the main engine and damping vibrations generated during operation. The Labritha Karina vessel underwent a propulsion system modification, requiring a redesign of the engine bed. This study aims to analyze the structural strength and fatigue life of the engine bed under dynamic vibrations using the Finite Element Method (FEM). The geometry of the engine bed was modeled using SolidWorks software and analyzed with Ansys. The analysis results show that the maximum stress is 35.69 MPa, which is below the allowable stress limit of 235 MPa. The maximum deformation is recorded at 1.353 mm, with a safety factor of 6.58, indicating that the engine bed structure is structurally reliable. The fatigue simulation revealed that the fatigue life of the engine bed is approximately 25,96 years, excluding damping factors and the use of mounting pads.**

***Index terms* —Fatigue; Engine Bed; Vibration; Finite Element Method (FEM);Ansys.**

I. Introduction

The Labritha Karina is a passenger ship that operates between Merak and Bakauheni. The ship had speed problems that affected its performance. Previously, the ship used an SRP propulsion system which was later converted to a *controllable pitch propeller* (CPP) propulsion system with *a Twin Screw Propeller*. The SRP has a vertical shaft that allows the boat to rotate up to 360° without the need for a rudder. For this, the foundation of a new machine was built by moving the engine from *frames* 23 to 28 to *frames* 38 to 42.

Fig. 1. Kapal Labritha Karaina

The ship's Engine Bed is an important component that supports the engine and distributes vibration well. Engine vibration when operating at high speeds can cause repeated stress on the machine's foundation structure, potentially causing material fatigue to structural cracks. Fatigue analysis is important to guarantee the safety of the ship's structure. One good analysis method to address this problem is the Finite Element Method (FEM), which can help estimate the lifespan of a machine's foundation and improve its design to be more resistant to dynamic loads. This study aims to determine the construction strength of the Engine Bed of the Labritha Karina due to engine vibration using the FEM method.

II. Literature Review

A. *Engine Bed*

The ship's Engine Bed is one of the crucial parts in a ship's propulsion system, which serves as the main buffer of the ship's engine and plays an important role in damping the vibrations generated by the engine during the ship's operation. In addition to receiving static loads from the mass of the machine, the foundation of the machine also receives dynamic loads from the vibration and thrust of the trust block[1]. An Engine Bed must be specifically designed to withstand dynamic loads due to the forces that occur during the ship's operation and the static loads of the engine's weight. The design of the Engine Bed is greatly influenced by the main size of the vessel, the type of engine and the conditions when operating. The installation of the Engine Bed must be made in such a way that the straightness of the engine shaft with the propeller shaft remains guaranteed. [2]

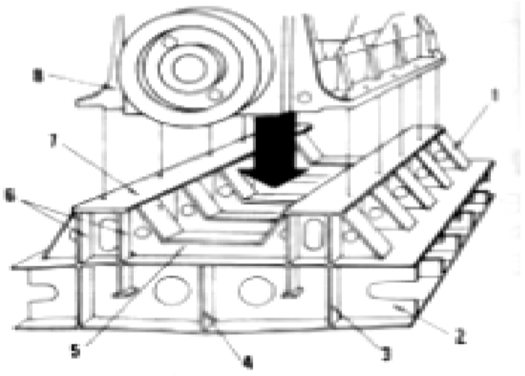

Fig. 2. Machine Foundation Structure
(Source: https://znanio.ru/media/sistemy-nabora-korpusa-sudna-2841669)

The rigidity on the foundation of the machine with a double base construction underneath must meet the Requirements. This is to ensure that construction deformation remains within the permitted limits. From the planning stage to the construction of the Engine Bed, attention must also be paid to the distribution of forces, both transverse and longitudinal of the ship.

B. Vibration Theory

According to Keith, vibration is motion that occurs periodically or alternately in a certain time interval[3]. Vibration can also be defined as the movement back and forth (oscillating) through a point of equilibrium in a fixed time interval.[4]

In general, vibration is divided into 2 types, namely forced vibration and free vibration. Free vibration is the vibration that occurs when motion is maintained by gravity or an elastic recovery force in the absence of external forces. Forced vibrations are vibrations caused by oscillating or intermittent external forces on a system[5]. The vibration on the Engine Bed is very closely related to random vibrations because the Engine Bed is part of the dynamic free foundation. Random vibration is a system's response to excitation that is not deterministic, i.e. it cannot be predicted exactly over time. Unlike harmonic or deterministic vibrations that have a defined waveform, random vibrations describe phenomena that are stochastic in nature and can only be analyzed statistically.[6]

The resonance will cause the amplitude of the vibration to rise theoretically with the frequency ideal thus reaching infinity. As a result, if the engine is not supported by a sufficient damping system, the vibrating engine support structure will be damaged. This can be reduced by regulating the impact on frequency and amplitude sighting. The frequency of the excitation of the main engine can be calculated by the following equation.

$$f = \frac{RPM}{60}$$

C. Fatigue

Fatigueis one of the materialdamages caused by cyclic or repetitive load treatment. The part of the structure that must withstand the stresses that fall on it during its operating life.

Fatigue life is the material ability to withstand or receive cyclic loads continuously. Permanent damage can occur When the accumulation of damage reaches a critical level so that the material can no longer support it. Basically, *fatigue* failure begins with the appearance of cracks on the surface of objects that are under tension. This proves that *fatigue* properties are very sensitive to surface conditions which are influenced by many factors including surface roughness, changes in surface properties and residual surface tension.[7]

Generally, in each *cycle* the amplitude *cycle* is not large enough to cause structural failure. However, failure can occur when the accumulation of damage to the structure reaches a critical level. This can be calculated using Palmgren Miner theory to calculate *cumulative fatigue damage*. If the *cumulative fatigue damage ratio* (DM) value is more than 1, then the structure cannot be used. So, based on BKI vol II rules regarding material fatigue for DM values, it can be calculated using equation 1.

$$DM = \frac{NL}{Ni}$$

Where, the way to determine is to use equation 2.N_L

$$NL = \frac{Fo \ x \ U}{4logL}$$

After obtaining the cumulative damage value, the results of the calculation are used to calculate the fatigue life value using equation 3.

$$Fatigue\ life = \frac{Design\ life}{DM}$$

D. Stress and Deflection

The allowable stress is the maximum stress within the permissible safe limits for the material to be loaded. In some structures it is necessary to take care to ensure that the material remains within its elastic range to avoid permanent deformation due to loads. Before that, it is necessary to know the k value (material factor) on the Engine Bed that is in accordance with the BKI 2022 volume II Section 5 rules. Here's how to determine the allowable Stress value using equation 4.

$$\sigma_0 = \frac{235}{k}$$

Deflection is a change in shape that occurs in a material due to the load applied to the material. The deflection that occurs in the material must be at certain limits, must not exceed the permissible deflection limit.[8]Based on BKI Volume II Section 21 regarding the deflection value of the accumulator permit due to load, it can be seen in equation 5.

$$\delta_{izin} = \frac{l}{200}$$

E. Safety factor

Every operating structure certainly has a security factor as a guarantee for the safety and usability of a structure. Therefore, it is important to check for these safety factors. Based on BKI Volume II Section 2B regarding the calculation of safety factors is as follows.[1]

$$SF = \frac{Yield\ stress}{Calculated\ stress}$$

F. Finite Element Method

The finite element method is a technique used to solve structural problems using a numerical approach applied to certain problems in the fields of engineering and science.[9] According to isworo and Ansyah, the finite element method is a numerical method to solve differential equation problems, both ordinary differential equations and partial differential equations. According to Zienkiewicz, FEM is a numerical method for reconciling partial and integral differential equations.[10]

The method of this element is very flexible and can be adapted to solve a variety of difficult problems in various fields of engineering science. In addition, it also provides numerical solutions to different types of structural problems. One of the problems that can be solved with the finite element method is the *eigenvalue* problem, where the natural frequency and mode of vibration of a structure need to be calculated. In designing a structure or component where dynamic loads, frequency, vibration are parameters that must be considered when designing. By using the element method up to the Ansys program, the natural frequencies with the vibration mode are obtained.[11] The initial step on the finite element method is to model the structure as *an assemblage* into smaller or simpler parts. This small part is called the until element. The point that connects one element to another is called a nodal point (*node*). The discretization process is the process of dividing a continuous structure model into smaller parts (*meshing*). [12]

III. Research Methodology

A. Research Methods

The method used in this study is a quantitative method using Solidworks software and Ansys software.

B. Design Plan

The research was carried out using the object of the Labritha Karina ship. This ship has an engine power of 2 x 2308 HP with a Merak - Bakauheni shipping area. The main dimensions and main sizes of the engines used on the Labritha Karinawith full scale are as follows.

The main size data of the Labritha Karina ship:

TABLE I. DIMENSIONS

Description	***Full Scale Size***
LOUDSPEAKER	95.795 m
LPP	84.996 m
Breadth (B)	15.00 m
Height (H)	9.85 m
Draft (T)	3.6 m
Speed (Vs)	14 Knots
Sailing Route	Merak-Bakauheni

The main engine data used by the Labritha Karinais the Mitsubishi S16R-MPTK main engine as shown in table 2.

TABLE II. ENGINE DATA

Main Engine Type	Mitsubishi S16R-MPTK
Speed	1800 RPM
Power	2308 HP
Bore	170 mm
Stoke	180 mm
Dimensions(LxWx H)	3066 x 1622 x 1960 mm
Mass	7000 kg
Cylinders	16
Frequency	50 Hz

The process of modelling the construction of the ship's Engine Bed in this study starts from collecting data and then making a 2D model using *AutoCAD software*, which functions to accurately determine the dimensions and basic shape of the construction of the Engine Bed based on the data of the plan drawing. Next, the 2D model that has been created is exported into a 3-dimensional model using Solidworks software, as shown in Figure 3.

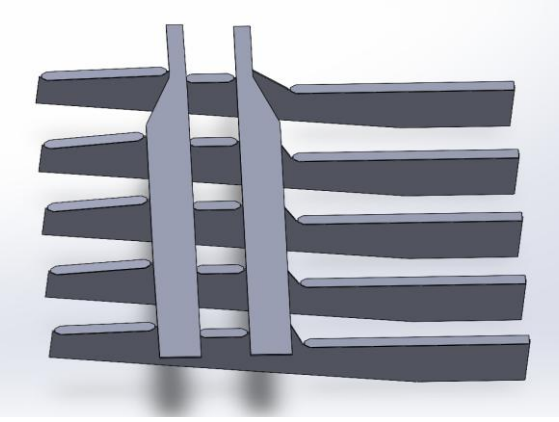

Fig. 3. 3D design of machine foundation

This 3D design is then imported to Ansys for analysis using the finite element method approach.

C. Meshing

This stage can be done after the geometry of the ship's Engine Bed has been completed. Meshing is a process of dividing the entire model into smaller elements to get a detailed analysis of the entire model. The meshing process is very important to pay attention to, because if the meshing is not suitable, the model cannot be analyzed or invalid. In the creation of elements, the determination of the size of the element is very important.

Too coarse an element size can shorten the time, but the simulation results cannot be validated. On the other hand, if the size of the element is too small (smooth) then the simulation time will be longer and require high computer specifications. In this analysis, the size of the mesh element used in the simulation was 12mm, with an average skewness value of 0.3 and an orthogonal quality of 0.65 which stated that the mesh quality was good.

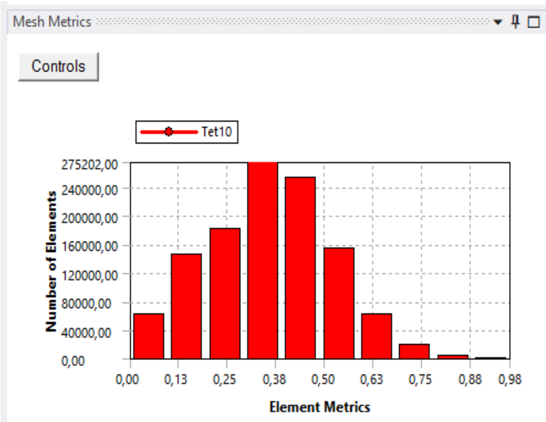

Fig. 4. Diagram Skewness

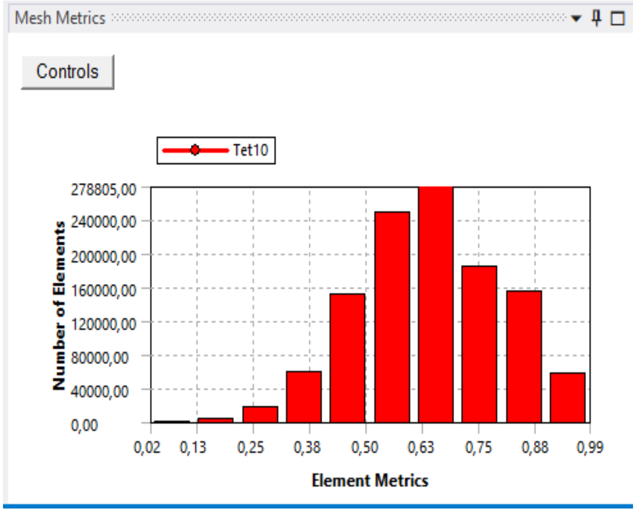

Fig. 5. Diagram Orthogonal Quality

D. Boundry condition

The next stage after meshing is the provision of boundary conditions. In this study, the centre girder of the machine foundation is used as a fixed support, while the top of the foundation is used to receive loads in the form of engine vibrations. The load given at the fulcrum is 68670 N, with the centre girder as the fixed support.

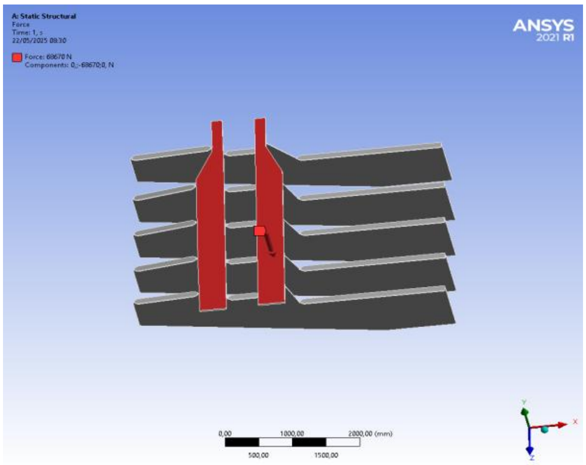

Fig. 6. Load Allocation

IV. RESULT AND DISCUSSION

A. Validasi Meshing

Grid independence is an important step to guarantee the results of the FEM simulation and to ensure that the results of the resulting numerical computation are correct. The simulation was carried out with several different meshing sizes. Then the results are analysed to see if there is a significant difference between meshing one and the other. If there is no significant change from the result, it indicates that the Solution has reached grid independence.

The results of the calculation on FEM can depend on several factors such as the number of or less mesh used during the simulation. Simulation is considered optimal if the simulation results are accurate with efficient time. The data of the number of elements taken is the one that has a difference value of less than 2% with the value that comes out against the smaller number of elements. To facilitate data analysis, the optimal number of elements can be shown as shown in table III.

TABLE III. GRID INDEPENDENCE

Element Size	***Number of Elements***	***Deformation***	***Error%***
25	1153316	1,597	-
20	1152381	1,363	-15%
12	1152916	1,3632	0,01%

Based on the results of the simulation with various numbers of mesh elements as shown in table III, it can be seen that there is a value of the number of elements that is less than 2%. These results show that the deformation does not change significantly, so it can be stated that the simulation results are quite stable and can be used for further analysis of the structural fabric of the machine foundation.

B. Static Structural Analysis Results

This research process was carried out using structural static Ansys. Material selection is carried out with structural steel in the Ansys database.

Allowable Stress

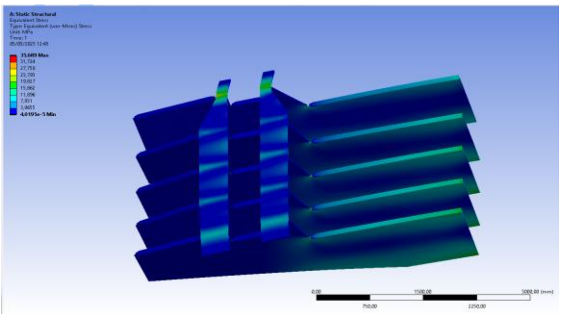

Fig. 7. Maximum Stress

The permissible stress on the analysis of the engine bed structure, with the value of the steel material factor is 1. It can be calculated using equation 4.

$$\sigma_0 = \frac{235}{k}$$

$$\sigma_0 = \frac{235}{1}$$

$$\sigma_0 = 235\ MPa$$

Based on the calculation from equation 4, the value of the permit stress is 235 MPa. Where the value is used as a reference for the criterion that the maximum stress value must not exceed the permissible stress, which is 235 MPa.

The results of the Ansys simulation obtained a maximum stress value of 35.69 MPa and a minimum of 4.019 x 10-5 MPa as seen in figure 4. Thus, it can be concluded that the von mises stress that occurs in the Engine Bed is still below the permit stress and has met the criteria of the BKI rules.

Deflection

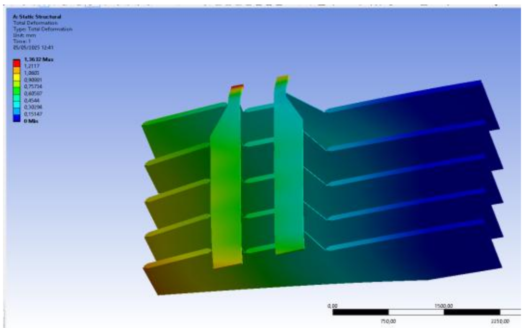

Fig. 8. Maximum deformation

To calculate the value of permit deflection according to BKI Vol II Section 21, it can be seen in equation 5.

$$\delta_{izin} = \frac{l}{200}$$

$$\delta_{izin} = \frac{750}{200}$$

$$\delta_{izin} = 3{,}75\ mm$$

Based on the above value, the permit deflection value was obtained of 3.75 mm. This value is used as a reference for the criteria for the maximum deflection value must not exceed the permit deflection value. From the simulation results, a maximum deflection value of 1,363 mm was obtained. Thus, it can be concluded that the deflection that occurs in the Engine Bed structure meets the criteria in the applicable rules.

Safety Factor

The safety factor obtained from the comparison of yield stress with calculated stress can be calculated using the equation of 2.20.

$$SF = \frac{Yield\ stress}{Calculated\ stress}$$

$$SF = \frac{235}{35{,}69}$$

$$SF = 6{,}58$$

Based on BKI rules, the value of the security factor that meets the value must be more than 1. The results of the calculation of the security factor above, a safety factor value of 6.58 was obtained. So it can be concluded that the value of the safety factor of the machine foundation meets the applicable rules criteria.

C. *Capital Analysis*

In this process, the natural frequency and total deformation of the ship's Engine Bed structure are obtained. Using the finite element method, a numerical model of a ship's Engine Bed is simulated to determine how the structure responds to dynamic excitation. The results of the analysis show that there are 50 natural frequency modes with values as shown in table 4.1. Each mode presents a specific form of vibration from the structure when excited at that frequency.

TABLE IV. NATURAL FREQUENCY

Mode	**Natural Frequency**	**Mode**	**Natural Frequency**
1	1,7611	26	30,004
2	5,1394	27	30,863
3	5,3614	28	31,321
4	5,4974	29	31,516
5	5,9929	30	31,663
6	6,0483	31	32,235
7	6,6643	32	32,377
8	6,8279	33	33,437
9	6,8879	34	34,088
10	7,5726	35	40,773
11	8,3936	36	41,183
12	12,434	37	41,421
13	12,448	38	42,009
14	12,492	39	43,386
15	12,571	40	45,239
16	13,347	41	51,03
17	15,252	42	52,217
18	19,894	43	53,306
19	19,942	44	53,966
20	19,996	45	54,058
21	20,386	46	54,311
22	21,152	47	54,535
23	28,668	48	54,789
24	29,351	49	55,547
25	29,512	50	56,286

In the table above, there are 50 vibration modes with a frequency range from 1.76 Hz to 56.29 Hz. The natural frequency vibration modes that are close to the excitation frequency of the main engine are as follows.

TABLE V. EXCITATION FREQUENCY

RPM	***Excitation frequency***	***Deformation***
900	15 Hz	2,070 mm
1440	24 Hz	3,70 mm
1800	30 Hz	2,609 mm
1980	33 Hz	2,825 mm

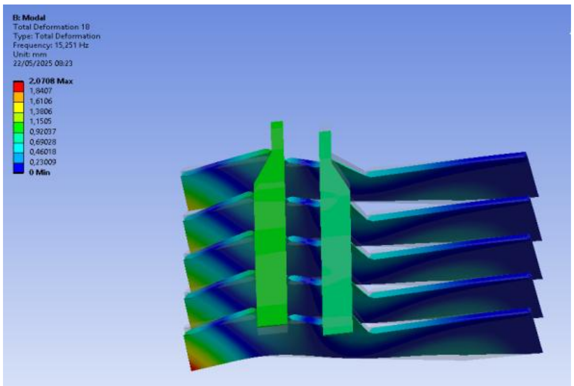

Fig. 9. Deformation with a frequency of 15 Hz

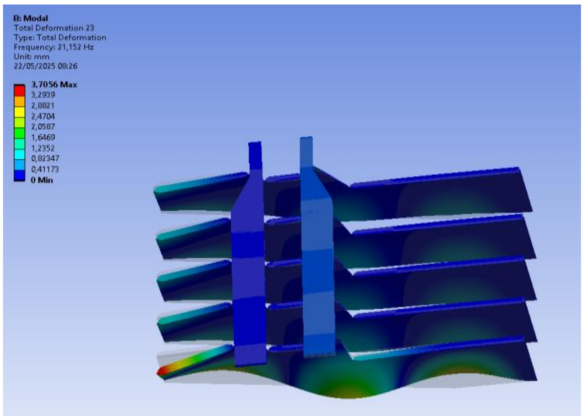

Fig. 10. Deformation with a frequency of 21 Hz

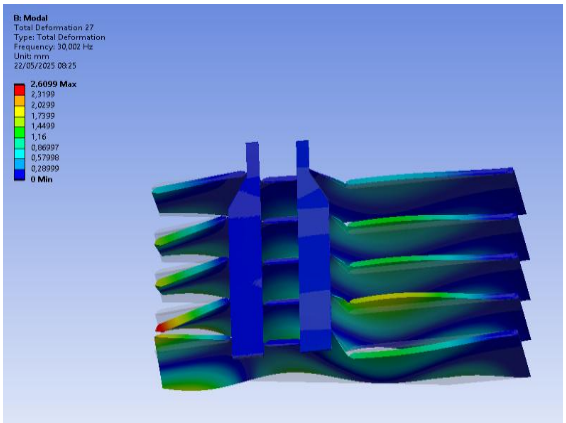

Fig. 11. Deformation with a frequency of 30 Hz

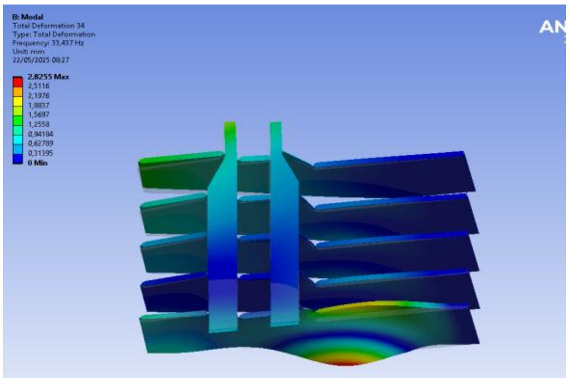

Fig. 12. Deformation with a frequency of 33 Hz

D. Fatigue Life

Fatigue life analysis carried out on the construction of the main Engine Bed using the random vibration method in Ansys software aims to measure the durability of the structure due to random dynamic loads that occur during the machine's operation. The results of the fatigue analysis can be seen as shown in Figure 4.19.

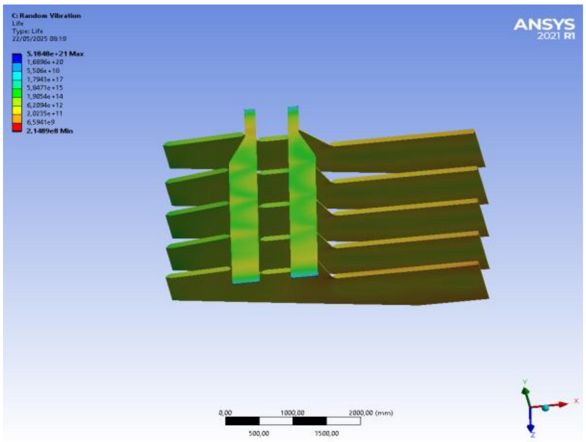

Fig. 13. Cycle values

In figure 13, it can be seen that the results of the analysis of the Engine Bed have a total life cycle of 2.1489e8 cycles. These values indicate the number of load cycles received by each element of the structure before fatigue occurred. The red color in the analysis results indicates the area that has the most critical fatigue value, while the blue color indicates the area that is most resistant to fatigue.

With 50 vibration shape modes and a maximum frequency value of 56.286 Hz as seen in table 4.1. To calculate the cumulative fatigue damage value of the Engine Bed, it can be calculated using equation 1 as follows:

$$NL = \frac{Fo \; x \; U}{4logL}$$

$$NL = \frac{0{,}85 \; x \; 504576000}{4log \; 84{,}996}$$

$$NL = 206874736{,}42$$

Then the cumulative damage value:

$$DM = \frac{NL}{Ni}$$

$$DM = \frac{206874736{,}42}{2{,}1489 \; x \; 10^8}$$

$$DM = 0{,}9627$$

Based on the calculation of equation 2, a DM value of 0.9627 was obtained. To calculate the fatigue life of the main machine foundation construction uses equation 3 assuming the age of 25 years.

$$Fatigue \; life = \frac{Design \; life}{DM}$$

$$Fatigue \; life = \frac{25 \; tahun}{0{,}9627}$$

$$Fatigue \; life = 25{,}96 \; tahun$$

The results of the calculation above are obtained that the life of the ship's Engine Bed without involving the damping factor and the Engine Bed bearing can last for 25.96 years from the time of manufacture.

V. CONCLUSION

A Based on the results of the analysis of fatigue on the ship's Engine Bed caused by engine vibration, it can be concluded that by using the element method up to (FEM), the maximum and minimum stress values were obtained at 35.69 MPa and 4.019 x 10-5 MPa, and the maximum deformation value was 1.353 mm. The result meets the standards of the stress and deflection limits allowed under BKI rules, so that the Engine Bed is declared structurally safe.

The value of the safety factor of the machine foundation was obtained as 6.58. The results of this calculation are well above the minimum limit (>1), which indicates that the Engine Bed has excellent structural resistance to dynamic loads due to engine vibration. From the modal analysis, 50 natural frequency modes with a frequency range ranging from 1.76 Hz to 56.29 Hz were obtained which provided an important picture of the dynamic behaviour of the structure to the excitation of machine vibrations. The results of the fatigue analysis with a load of 68670 N and a cycle of 2.1489 x 108 cycles, the construction can last for 25.96 years. The results were calculated without taking into account the damping factor and the use of engine bearings.

ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to Universitas of Diponegoro for providing the necessary facilities and unwavering support throughout the course of this research. The resources and environment offered by the university have been instrumental in enabling us to conduct our study effectively.

REFERENCES

[1] B. Adityo Nugroho and I. PujoMulyatno, “Analisa Kekuatan Struktur Pondasi Mesin Dengan Interaksi Trust Block Pada Kapal Ropax 5000 Gt Dengan Metode Elemen Hingga”, 2015.

[2] M. Sofi`i, “Teknik Konstruksi Kapal Baja,” 2008.

[3] R. K. Mobley, “ROOT CAUSE FAILURE ANALYSIS,” 1943.

[4] G. A. Sarojo, Mekanika, 5th ed. Jakarta Selatan: SalembaTeknika, 2014.

[5] R. C. Hibbeler, Mekanika Teknik Dinamik, 2nd ed. Jakarta: PT Prenhallindo, 1999.

[6] Daniel J. Inman, Engineering Vibration (Edisi ke-4), 4th ed. Pearson Education, 2013.

[7] K. M. Y. Emily, N. A. N. Priyambodo, and B. Herijono, “Kajian KekuatanStrukturPondasi Mesin IndukHarbour Tugboat,” vol. 2, no. 1, pp. 1–7, 2021, doi: 10.31331/maristec.v2i1

[8] Basori, syafrizal, and Suharwanto, “Analisis Defleksi Batang Lenturmenggunakan Tumpuan Jepit Dan Rolpada Material Aluminium 6063 Profil U Dengan Beban Terdistribusi”, 2015.

[9] Z. Hilmy*et al.*, “Sudut Kemiringan V Bevel Pada Proses Pengelasan Stainless Steel Butt Joint Menggunakan Metode Elemen Hingga”, *JURNAL MARITIM*, vol. 4, no. 1, 2022.

[10] O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu, “The Time Dimension: Semi-Discretization of Field and Dynamic Problems,” *The Finite Element Method: its Basis and Fundamentals*, pp. 379–405, Jan. 2013, doi: 10.1016/B978-1-85617-633-0.00012-5.

[11] H. Isworo and P. R. Ansyah, “Buku Ajar Metode Elemen Hingga HMKB654”, 2018.

[12] W. F. Tjong, “Metode ElemenHingga Untuk AnalisisStruktur,” 2021.