Journal of Research and Applications in Mechanical Engineering
ISSN: 2229-2152 (Print); 2697-424x (Online)
(2024) Vol. 12, No. 1, Paper No. JRAME-24-12-013
[DOI: 10.14456/jrame.2024.13]

Research Article

Redesign of the Mini Hydro Turbine Structure Using Finite Element Analysis (FEA) to Solve Resonance Problem

P. Sreejai[1,*]
A. Promwungkwa[1]
K. Ngamsanroaj[2]
[1]Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University, Chiang Mai, 50200, Thailand
[2]Electricity Generating Authority of Thailand, Bang Kruai, Nonthaburi, 11130, Thailand

Received 31 August 2023
Revised 14 November 2023
Accepted 25 November 2023

Abstract:
This study investigates the structural redesign of a mini hydro turbine system in an effort to solve the turbine shaft resonance problem. The original design of the turbine shaft support is examined to identify the root causes of the problem, which result in the failure of the turbine bearings. In order to approximate the natural frequencies, finite element analysis (FEA) models are developed and verified through comparison with experimental data, which may consist of modal analysis, coast-down testing, and deflection tests. Experimental results deviate from the FEA analysis of mode 1 frequency by 6.81%. Analysis of the linear dynamic response reveals that the amplitude of the shaft coupling's vibrations during system start-up and shutdown exceeds the allowable design parameters; this is the primary cause of bearing damage. FEA is used to redesign the shaft structural support in order to eliminate the natural frequencies. Modes 1 and 2 exhibit frequency shifts of 14.57 to 38.16 Hz and 11.36 to 37.57 Hz, respectively. These values are double the frequency at which a typical generator operates. The maximum shaft support deflection is reduced from 0.25–0.39 mm to 0.01 mm, according to the FEA results. Utilizing the redesign parameters, a new shaft support will be constructed. To confirm the FEA results, it is possible to monitor and recheck the data.

Keywords: *Vibration, Resonance, Mini hydro turbine, Finite element Analysis, Redesign*

1. Introduction

The predominant use of irrigation dams in the northern region of Thailand is for agricultural objectives. Nonetheless, electricity generation constitutes an additional benefit. A mini hydro turbine power plant with a capacity of 450 kW was constructed on a dam in Chiang Mai. A vertical propeller turbine equipped with an induction generator constitutes the hydro turbine. However, vibration concerns emerged in both the turbine shaft and the overall structure during the testing phase [1]. The turbine bearing experienced failure after 5,000 hours of operation subsequent to the commissioning of the system. The annual electricity production was reduced from 800,000 kWh to 200,000 kWh. As a result of increased machine downtime caused by this repeating problem. In order to identify the root causes and implement effective solutions to these problems, a thorough investigation was necessary.

Modern vibration problem-solving techniques employ a variety of techniques involving vibration measurement instruments and specialized software for comprehensive analysis. [2, 3] These techniques include finite element analysis, experimental modal analysis, and comparing results to confirm the accuracy of mathematical models [4, 5].

* Corresponding author: P. Sreejai
E-mail address: Prudya.sre@egat.co.th

This technique results in model improvements before the actual modifications, thereby reducing the costs related to the improvements [6, 7]. It not only addresses problems and complications but also improves its applicability to a variety of applications, such as large hydroelectric power plants [8-11].

The objective of the study is to simulate the condition of the mini hydro turbine structure using a mathematical model based on Finite Element Analysis. The accuracy of this model is validated by comparing it to practical testing, including experimental modal analysis, coast-down tests, and deflection tests. Subsequently, finite element analysis models are used to determine the root cause of the turbine bearing's failure. As a result, the structure of the mini hydro turbine is redesigned to address the resonance problem and serve as a prototype for further mini hydro turbines.

2. Verification of the Original Design

To address the problem of turbine bearing damage, which a preliminary engineering analysis relates to the vibration problem, it is necessary to investigate the original design and execute a comprehensive root cause analysis. For this study, a mathematical model based on finite element analysis is applied.

2.1 FEA Modelling and Analysis

A computational-aided design (CAD) application was employed for creating a model representing the microhydro turbine's structure. An overview of the FEA is presented below, as depicted in Fig. 1. Applying the ASTM A36 steel sheet, the material parameters were determined. The model assigns fixed joints to the connections existing between the support surfaces and the concrete structure. The generator, with a mass of 3.2 tons, was subjected to an equivalent force. An axial force of 62 kN was applied relative to the turbine. Setting the pressure within the pipe to 1.2 kg/cm². Furthermore, a damping ratio of 0.01 was taken into consideration. The mesh generation process generated 55,465 elements in total, composed of various element types such as plate elements, line elements, and 3D elements, and employed finite element analysis for computation. Results include static analysis, natural frequency analysis, and linear dynamic response analysis [12].

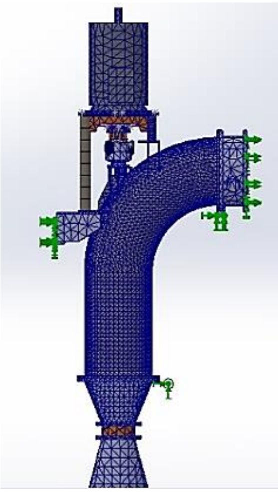

Fig. 1. Original structure overview

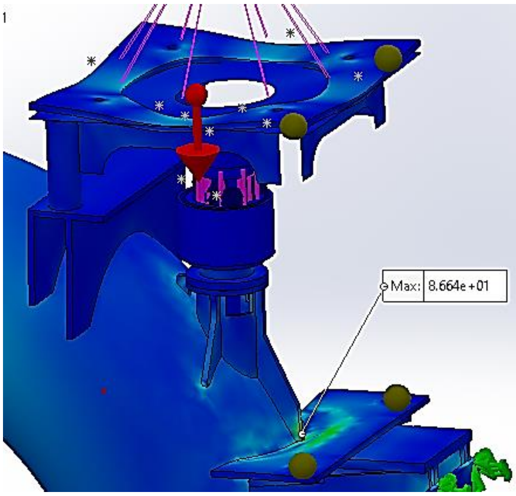

Fig. 2. FEA Von Mises stress

As shown in Fig. 2, the Von Mises stress distribution reveals a maximum value of 86.64 MPa at the bottom plate, with a changing range of 40 to 60 MPa surrounding it. Similarly, the Von Mises stress level varies between 30 and 40 MPa at the generator support plate. When compared to the documented yield strength of ASTM A36 Steel Sheet, which is 250 MPa, it is evident that the turbine's original structure has a safety factor of 2.89.

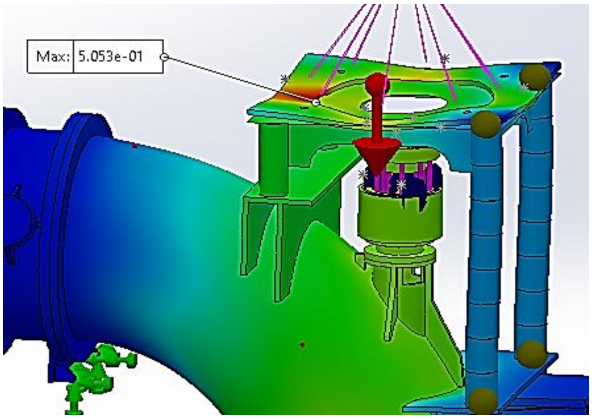

Fig. 3. FEA Structure deflection

Table 1: FEA natural frequency results

Mode No.	Frequency (Hz)
1	11.36
2	14.57
3	36.55
4	40.05
5	48.50

As shown in Fig. 3, the maximum deflection of the generator support plate under the load of the generator is 0.5mm. By comparison, the other side shows deflection between 3 and 5 mm. In addition, Table 1 lists the natural frequency as 13.56 Hz for Mode 1 and 14.58 Hz for Mode 2, which is quite close to the generator's rated speed of 12.5 Hz.

2.2 Finite Element Analysis Verification

For the purpose of validating the FEA model, the model's results are compared to experimental test results. In general, verification and validation techniques frequently suggest a deviation of less than 10%, indicating that the analysis is accurate [5].

2.2.1 Experimental Modal Analysis

The natural frequency of a machine can be determined through modal testing [13] by utilizing experimental data and mathematical equations. In this analysis, the ME-scope VT-540 software [14] is utilized along with Alta Solution's AS-1250FE vibration data recorder. Data is captured in either time or frequency domain format using an IEPE accelerometer from Wilcoxon Sensing Technologies (793 series) roving. Additionally, an impact hammer is fixed for excitation to stimulate the vibrational responses of the structure. The program results for ME's scope include three major parts:

Part 1: Frequency Response Functions (FRFs): Fig. 4 depicts the peaks of FRFs corresponding to vibration modes 1 through 6. Particularly, Mode 1 has a frequency of 11.1 Hz, whereas Mode 6 has the greatest vibrational amplitude at a frequency of 40.5 Hz.

Part 2: Natural Frequency Values and Damping Ratios: Table 2 depicts the natural frequency and damping ratios for modes 1 through 6, which are discussed in the next section. The frequency of Mode 1 is 11.1 Hz, and its damping ratio is 0.874%.

Part 3: Mode Shape Animation: An animation in Part 3 shows the mode shape, which closely resembles the mode shape found using Finite Element Analysis (FEA), as illustrated in Fig. 5.

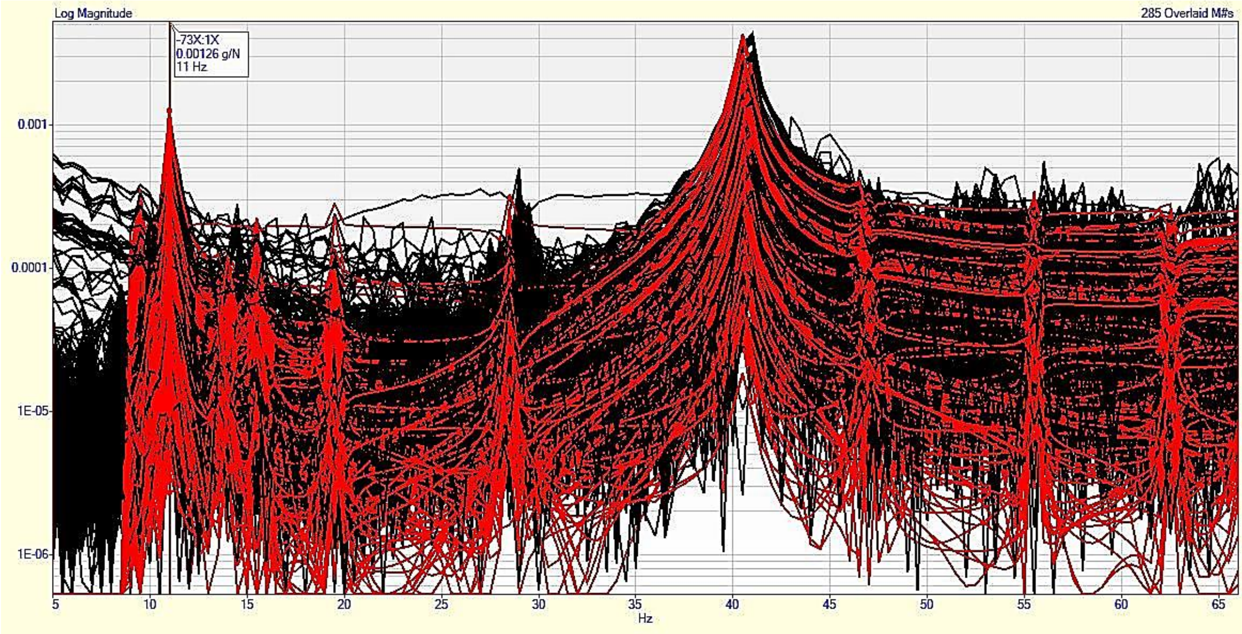

Fig. 4. Frequency response functions (FRFs)

Table 2: Modal testing natural frequency results

Mode No.	Frequency (Hz)	Damping (%)
1	11.1	0.87
2	13.8	0.39
3	15.6	0.74
4	19.4	0.07
5	28.4	0.10
6	40.5	0.78

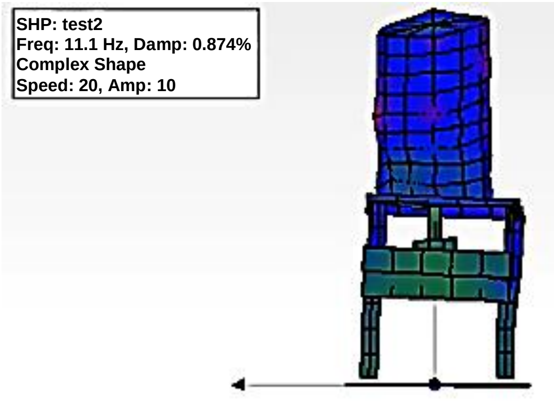

Fig. 5. Mode Shape Animation

2.2.2 Coast Down Test [3, 15, 16]

The Run up/Coast down test mode of the PRUFTECHNIK VIBXPERT® II Series vibration measurement instrument was utilized to collect machine vibration data from starting to shutting down. This included two-axis vibration measurements of the generator's NDE bearing (1) and DE bearing (2). On the Bode plot of Fig. 6, the NDE Bearing (1) indicated a vibration reading of 11 mm/s at a speed of 622 rpm. This measurement was accompanied by a 230-degree phase shift. In addition, the natural frequency of the first mode was determined to be 10.37 Hz.

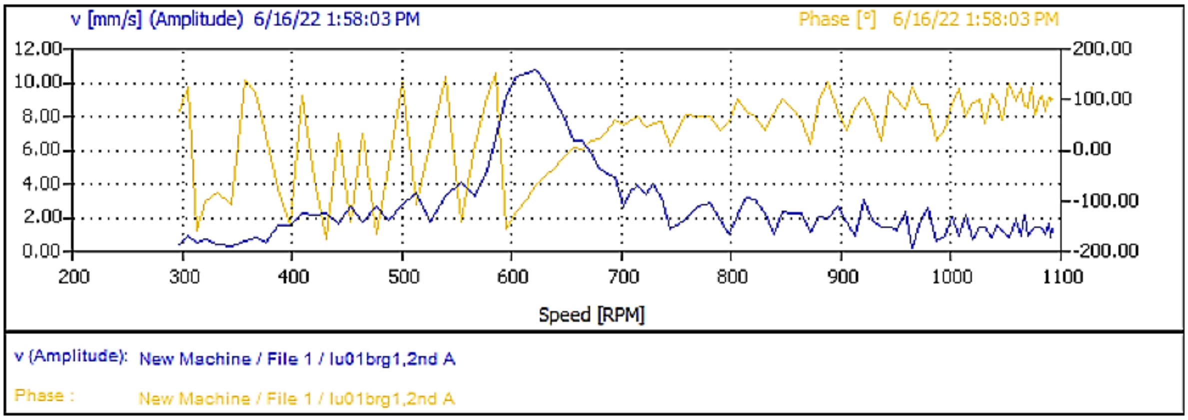

Fig. 6. Bode plot result for NDE Bearing [1]

2.2.3 Deflection Test

Utilize the dial gauge with a precision of 0.002mm, the Mitutoyo 2046S, to measure the generator-supported plate's deflection. Follow these procedures, as demonstrated in Fig. 7: Install the dial gauge at the designated measuring point, use an overhead crane to lift the 3.2-ton generator from the support plate, and then record the support plate's deflection measurement. As shown in Table 3, the results of the deflection inspection test indicate that the values range between 0.25 and 0.39 mm on all four sides.

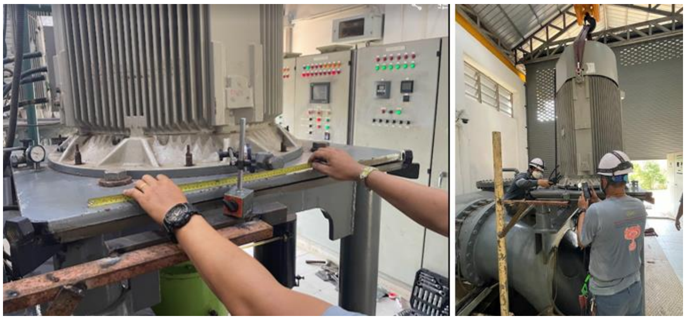

Fig. 7. Support plate deflection inspection

Table 3: Deflection test and FEA comparison

	Deflection test (mm.)
Sensor 1	0.35
Sensor 2	0.39
Sensor 3	0.25
Sensor 4	0.33

2.2.4 Results and Discussions

Comparing the results of the FEA with the results of the deflection test, Table 4 summarizes the findings. It is found that the measurement positions for sensors 1, 3, and 4 were different by 0.01-0.02 mm, which equates to a range of 2.94 to 5.41%. Sensor 2 indicated a difference of 0.07 mm, which is equivalent to 15.22%. Fig. 8 shows the physical measurement location for sensor 2 that deviates from the simplified FEA model because of a steel plate with an adjustable bolt. Consequently, this difference causes an error between the physical measurement result and the FEA result for sensor 2. However, the Mean Absolute Percentage Error (MAPE) of 6.85 percent for the FEA model in the Static analysis is within an acceptable range.

Table 4: Deflection test and FEA comparison

	FEA (mm.)	Deflection test (mm.)	Absolute percent error (%)
Sensor 1	0.37	0.35	5.41
Sensor 2	0.46	0.39	15.22
Sensor 3	0.26	0.25	3.85
Sensor 4	0.34	0.33	2.94
		MAPE	6.85%

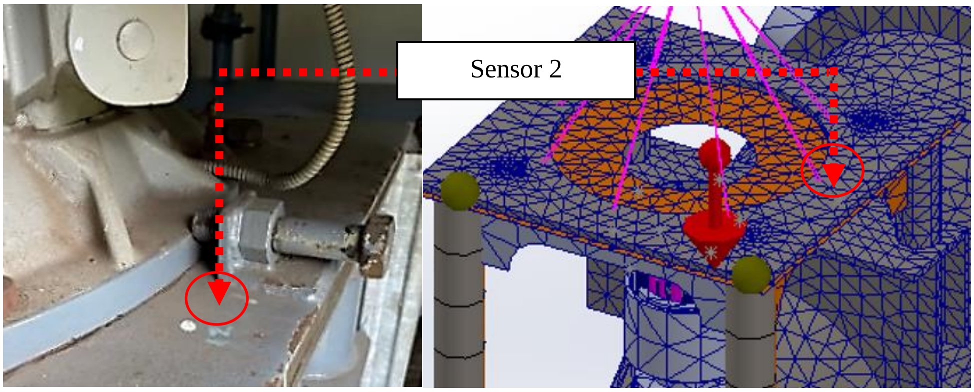

Fig. 8. Comparison between the real position and the model of the measuring point at sensor

Table 5: Comparison of natural frequencies with different methods

Natural Frequency	FEA (Hz)	Modal test (Hz)	Coast down test		MAPE
			NDE Bearing (Hz)	DE Bearing (Hz)	
Mode 1	11.36	11.1	10.37	10.23	6.98%
Mode 2	14.57	13.8	13.30	13.71	6.81%
Mode 3	36.55	15.6			
Mode 4	40.05	19.4			
Mode 5	48.50	28.4			
Mode 6		40.5			

According to the data in Table 5, the natural frequencies in Mode 1 and Mode 2 remain comparable across all testing methods. In particular, Mode 1's natural frequency varies between 10.23 Hz and 11.36 Hz, with a mean absolute percentage error (MAPE) of 6.98%. Mode 2 contains a natural frequency ranging from 13.30 Hz to 14.57 Hz with a MAPE of 6.63 %. The total MAPE values at 6.81% overall. Knowing the water turbine's rated speed of 750 rpm (12.5 Hz) and its maximum rotational speed during an emergency stop of 1600 rpm (26.7 Hz), it is significant to note that the natural frequencies of the turbine structure in modes 1 and 2 are close to the generator's operational range. This proximity increases the possibility of resonance excitation. In other respects, it is evident that the coast-down test method was unable to identify a higher mode's natural frequency. Mode 4 of the FEA indicates a frequency of 40.05 Hz, which is similar to Mode 6 of the Modal Test, which measures a frequency of 40.5 Hz.

2.3 Failure Mode Analysis

According to the technical specifications of the FLENDER COUPLINGS N-EUPEX 350mm shaft coupling, a radial shaft misalignment of up to 0.5mm is permissible. Fig. 9 displays the Finite Element Analysis (FEA) result for the linear dynamic response in a harmonic study. This analysis reveals that the measured displacement of the turbine coupling is 1.28 mm in mode 1 (at 11.35 Hz) and 0.95 mm in mode 2 (at 14.57 Hz). Based on these results, it can be determined that when the generator operates at a rotation speed equal to the natural frequencies of the first and second modes, it generates a significant amount of vibrational resonance. Consequently, this resonance causes a radial misalignment of the turbine shaft coupling that exceeds permissible limits and causes the bearing to fail quickly. A comprehensive investigation of bearing damage is outside the scope of this study.

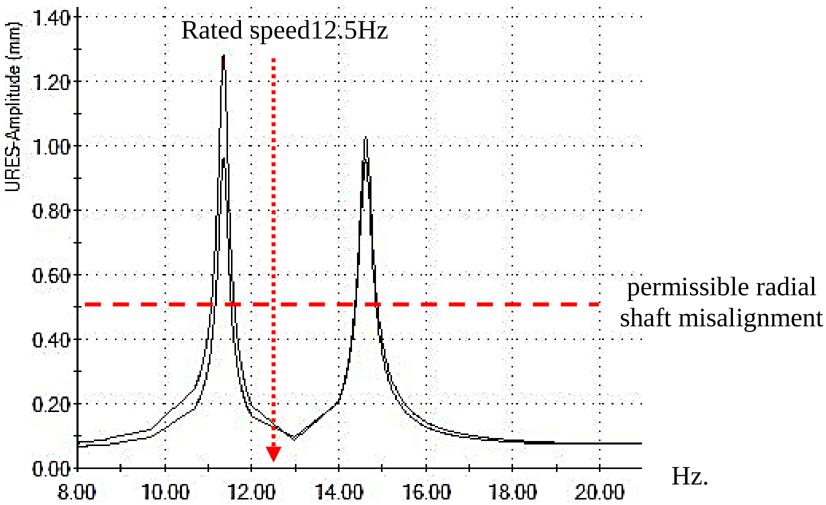

Fig. 9. Turbine coupling displacement

3. Redesign of the Generator support

3.1 Vertical Turbine and Pump Design Related Work

Summary of the studies conducted by El-Shaikh [7], Loeser and Neto [17], and El-Gazzar [18]. Collectively, these studies indicate the resonance problems that vertical pumps with top-mounted motors frequently face, especially in mode 1. Commonly, simulations of finite element analysis are used to modify the structure in order to increase mode 1's natural frequency, ensuring a minimum 20% deviation from the operational speed. To validate the finite element analysis (FEA), the studies referenced in sections 1–2 use comparable methodologies to evaluate the agreement between FEA results and experimental data, with the differences falling within the recommended range of 10 percent. In this study, the Coast down test method was utilized for comparison, ensuring the validity of the combined results.

3.2 Redesign of the Structure

The simulation intended to resolve the resonance problem by modifying the original structure of the mini hydro turbine was unsuccessful. It was unable to change the natural frequencies of modes 1 and 2 beyond the normal rotation speeds of the machine. Therefore, a study of vertical turbine and pump designs was conducted to guide the development of a new concept. The new design adopted a cylindrical shape with reinforcement fins to support the generator and rotating components, diverging from the original design, which featured four long poles and a flat plate for support.

By applying the database obtained from the original structure's FEA model, including external force, pressure, and damping ratio data. Mixed mesh element approach finite element analysis (FEA) was performed using computer-aided design (CAD) software. As illustrated in Fig. 10, the design allocated the line element for beam components, the plate element for steel plate sections, and the complex component structural elements as 3D elements. This design composition encompassed a total of 97,508 elements, characterized by a standard element size of 163.75 millimeters.

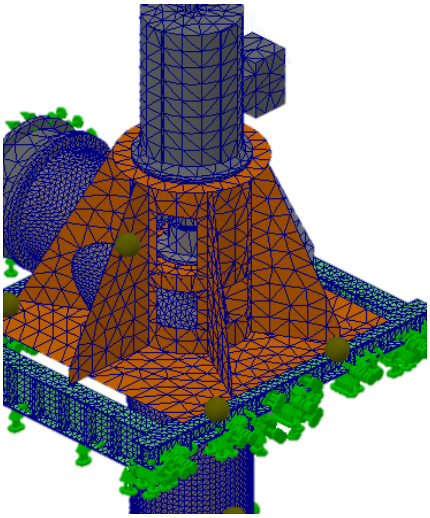

Fig. 10. Finite element model of new design structure

3.3 New Design Finite Element Analysis

Natural Frequency: The redesigned structure shows a mode 1 natural frequency of 37.57 Hz and a mode 2 natural frequency of 38.16 Hz, as shown in Table 6's natural frequency analysis. Moreover, modes 3 through 5 have greater frequencies than modes 1 and 2. Fig. 11 depicts Mode Shape 1 with bending along the X-axis, whereas Fig. 12 depicts Mode Shape 2 with bending along the Z-axis.

Table 6: Natural frequency of new design structure

Mode No.	Frequency (Hz)
1	37.57
2	38.16
3	56.11
4	57.88
5	68.21

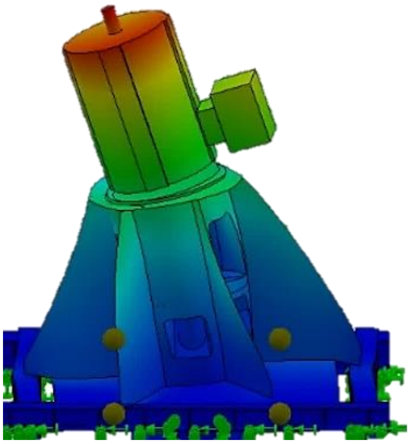

Fig. 11. Mode shape 1

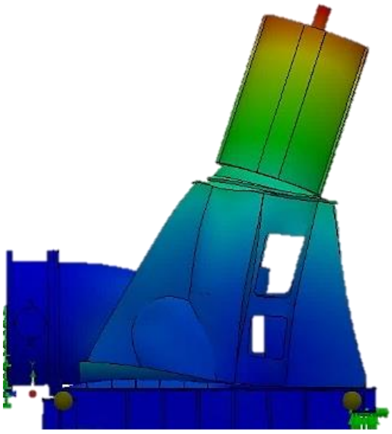

Fig. 12. Mode shape 2

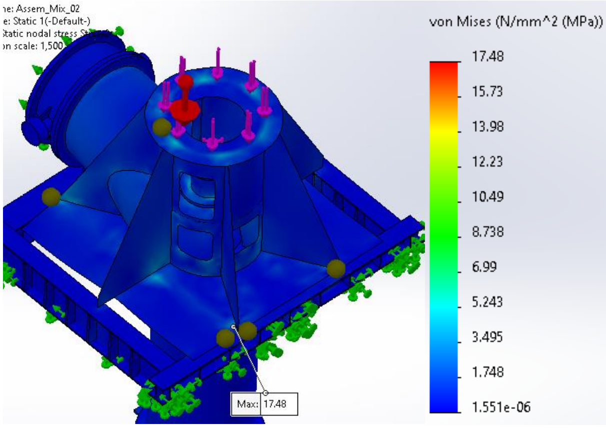

Fig. 13. Von Mises stress of new design structure

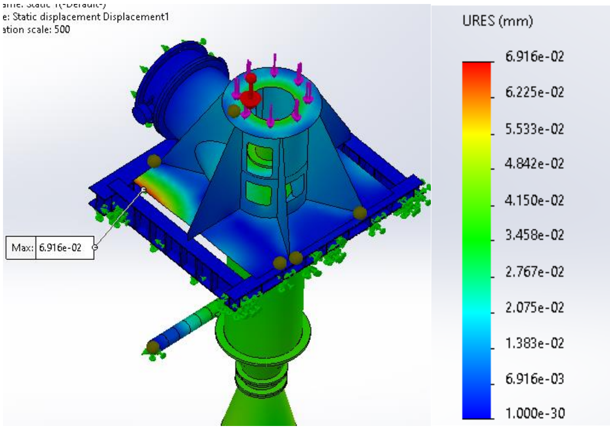

Fig. 14. Deflection of new design structure

Von Mises stresses: As illustrated in Fig. 13, the most significant Von Mises stress in the new structure occurs at the I-beam connection below the floor sheet, with a maximum value of 17.48 MPa. These I-beams are made from SS400 steel with a 215 MPa tensile strength. Considering a safety factor of 12.3, it has been determined that the material is capable of maintaining weight without deforming. Von Mises stresses between 5 and 10 MPa are found to be operationally effective in several of other locations, including the generator base.

Deflection: Fig. 14 shows the deflection of the new structure, which reaches a maximum value of 0.069 mm at the steel sheet base plate. Other locations have deflection values ranging between 0.02 and 0.04 mm. Additionally, the generator support plate's deflection value is 0.01 mm, representing an improvement over the original design.

Strain: Simulation results indicate a total strain value of up to 0.00004 at the reinforced beam's extremity. In addition, there is a notable strain concentration in the surrounding area ranging from 0.00016 to 0.00028.

4. FEA Analysis of Original and New Designs

To establish a relationship between the operating frequency of the machine and the natural frequencies of the Original and New Designs in modes 1 and 2, a comparison of the finite element analysis results can be performed as follows:

Table 7: Comparison of Natural frequency between the original design and new design

Natural frequency	Original (Hz)	New Design (Hz)
Machine rated frequency	**12.50**	**12.50**
Mode 1	11.36	37.57
Mode 2	14.57	38.16
Mode 3	36.54	56.11
Mode 4	40.05	57.88
Mode 5	48.49	68.21

Table 7 presents a comparison of the natural frequencies of the original and new designs. In the new design structure, mode 1's natural frequency is 37.57 Hz, a significant increase from mode 1's 11.36 Hz in the original design. Similarly, the natural frequency of mode 2 has shifted significantly from 14.57 Hz in the original design to 38.16 Hz in the new structure. For modes 3 through 5, the new design's natural frequencies exceed the machine's operational speed.

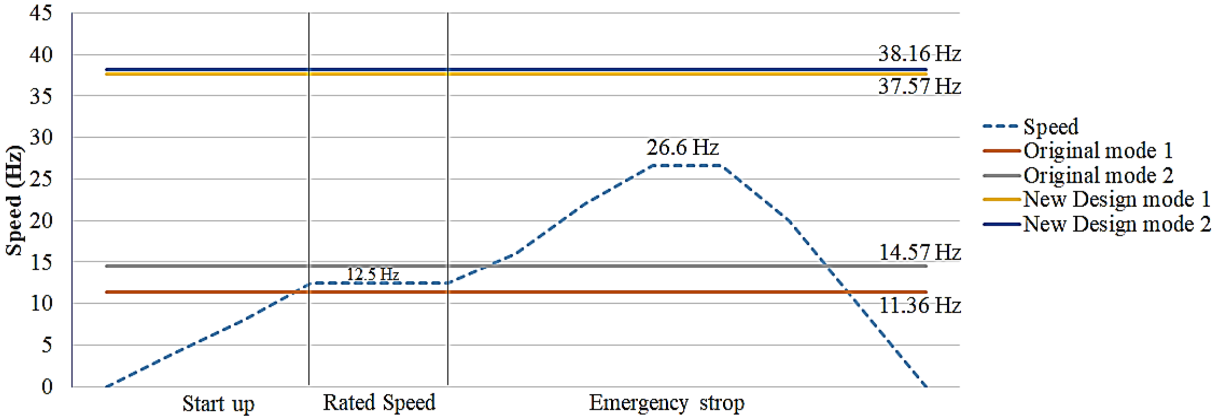

Fig. 15. Relationship between operation frequency and Machine natural frequency

Fig. 15 is a graphical representation of the operational range and natural frequencies of the machine in modes 1 and 2 for both the original design and the new design. In the original design, the natural frequency correlates closely with the operating frequency of the machine, differing by only 9% in mode 1 and 14% in mode 2, causing resonance problems. As a result, the natural frequencies in modes 1 and 2 for the new design structure are significantly higher, exceeding the machine's regular frequency by 200% and its maximum frequency by 41%, as well.

5. Conclusion

The structure of the mini hydro turbine is simulated using finite element analysis (FEA) in order to resolve the resonance problem. The model results are validated via actual testing employing a variety of methodologies. The outcomes of the model and the actual test results prove to be in close correspondence. The mean absolute percentage error (MAPE) for deflection of the support plate is 6.85%, while it is 6.81% for natural frequency modes 1 and 2. Such values fall within the acceptable range for accurate FEA results.

The FEA's linear dynamic response analysis revealed a significant insight: when the machine is in operation, it coordinates with modes 1 and 2's natural frequencies, causing resonance problems. These resonance-induced vibrations are mainly responsible for causing damage to the turbine bearings by causing excessive misalignment of the turbine shaft coupling.

Extensive research has been conducted into resonance problems to redesign the structure of the mini hydro turbine. The new design structure's natural frequencies are 37.57 Hz in mode 1 and 38.16 Hz in mode 2. In addition, these values exceed the generator's rated speed of 12.5 Hz and its maximum speed of 26.6 Hz, indicating a 200% increase over the rated speed and a 41% increase over the maximum speed. The original deflection values for the generator support plate and turbine bearing support, which ranged from 0.25 to 0.39 mm, have been successfully reduced to 0.01 mm. The construction project has received approval and is awaiting budget allocation in 2024. The FEA of the new structure should to be verified through experimental modal analysis and the coast down test. The new structure of the mini hydro turbine can be effectively applied to solve the actual structure's resonance problem. This improved design can also serve as a prototype for similar mini hydro turbines.

Acknowledgments

This research was supported by the EGAT-CMU Academic & Research Collaboration Project. Additionally, the authors would like to thank EGAT and CMU for the software ME'scope VT-540 License Number 19280 and Solidworks Serial Number 9710 0260 4659 3852 7D8B 9F32.

References

[1] Hydro power plant part maintenance and manufacturing section EGAT. Installation Maintenance and commissioning of EGAT irrigation Lam-rang Mini hydro power plant. Nonthaburi: EGAT; 2016.

[2] Finley WR, Sauer BJ, Loutfi M. Motor vibration problems: how to diagnose and correct vibration errors. IEEE Ind Appl Mag. 2015;21(6):14-28.

[3] Queen R, Chatlos G. Coast-downs and run-ups: understanding vibration responses. 2018 IEEE Petroleum and Chemical Industry Technical Conference (PCIC); 2018 Sep 24-26; Cincinnati, USA. USA: IEEE; 2018. p. 289-96.

[4] Mohamad Faizul Aslam MI. A study of correlation between finite element analysis and experimental modal analysis in structural dynamic analysis. Pahang: Faculty of Mechanical Engineering, University Malaysia Pahang (UMP); 2018.

[5] Srinivas K. Verifications and validations in finite element analysis (FEA). Ahmedabad: Advanced Scientific and Engineering Services (AdvanSES); 2020.

[6] Gautam A. Structural analysis of Kaplan turbine shaft using ANSYS. Haryana: Department of Mechanical Engineering, National Institute of Technology Kurukshetra; 2013.

[7] El-Shaikh SAA. Experimental investigation and finite element analysis for solving vertical pumps structural weakness. Eur J Mech Eng Res. 2016;3(2):50-63.

[8] Thongsiri G. Mechanical vibration of water turbine in Sirikit hydro power plant unit. Chiang Mai: Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University; 2012.

[9] Gustavsson R, Nässelqvist M, Österud J. Radial dampers impact on shaft vibration at resonance. IOP Conf Ser: Earth Environ Sci. 2019;240:022011.

[10] Valentín D, Presas A, Egusquiza E, Valero C, Bossio M. Dynamic response of the MICA runner. Experiment and simulation. J Phys: Conf Ser. 2017;813:012036.

[11] Xie J, Huang B, Fu L. Reinforcement and resonance control of head cover of Francis turbine by finite element analysis and modal testing. IOP Conf Ser: Earth Environ Sci. 2020;560:012051.

[12] Promwungkwa A. ME422: Introduction to finite element method. Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University; 2021.

[13] Allemang RJ, Brown DL. Chapter 21 Experimental modal analysis. In: Piersol AG, Paez TL, editor. Harris' Shock and Vibration Handbook. 6th ed. NewYork: McGraw-Hill; 2010. p. 21.

[14] Bliss Services Thailand. Modal analysis Me'scope training for EGAT 2022. Nonthaburi: EGAT; 2022.

[15] Eisenmann Sr RC, Eisenmann Jr RC. Machinery malfunction diagnosis and correction. Menlo Park: Hewlett-Packard effective; 2005.

[16] Wilcox Ed. Vibration analysis for turbomachinery. The 45th Turbomachinery and the 32nd Pump symposia; 2016 Sep 12-15; Houston, USA. p. 1-50.

[17] Loeser S, Neto MM. Design methodology for vertical centrifugal pumps. 2011 International Nuclear Atlantic Conference – INAC2011; 2011 Oct 24-28; Belo Horizonte, Brazil. p. 1-10.

[18] El-Gazzar DM. Finite element analysis for structural modification and control resonance of a vertical pump. Alexandria Eng J. 2017;56(4):695-707.