Recent Trends in Civil Engineering and Built Environment Vol. 5 No. 1 (2024) 93-103

RTCEBE

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rtcebe
e-ISSN :2773-5184

Comparison Between Cracked and Normal Residential House at Peat Soil Area by Using Finite Element Modelling

Mohd Amirul Haqimi Mohd Hussin[1], Tuan Norhayati Tuan Chik[2]*

[1]Faculty of Civil Engineering and Built Environment,
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

*Senior Lecturer, Faculty of Civil Engineering and Built Environment, Universiti Tun Hussein Onn Malaysia

DOI: https://doi.org/10.30880/rtcebe.2024.05.01.010
Received 9 January 2022; Accepted 1 January 2024; Available online 30 June 2024

Abstract: Peat soils have a very high moisture content, low bearing capacity, and high compression. Due to its characteristics, peat soil is less preferred for the construction of civil buildings. However, every problem must have a solution, for example needing improvements on peat soil or mitigation technique on building to control the damage on buildings. If the issue persists, contamination from peat must be taken into consideration. It can be unsafe and fail to control. The structural condition of the peat soil will affect humans and severe damage to the building, such as cracks in plaster, cracks in beams, columns, and floors. In addition, it can also cause a building to collapse. Therefore, mitigation techniques are essential to control or reduce the impact of damage to a building erected on the surface of peat soil. Thus, the purpose of this study is to investigate a dynamic response on cracked and normal residential houses at peat soil area and to compare the performance of both houses by using Finite Element Modelling (FEM). The study will focus on two locations where existing residential buildings are damaged and residential buildings used mitigation techniques. The location of these two buildings is in Ayer Hitam, Muar, Johor. Both of these buildings were modelled with Finite Element Methods (FEM). The Finite Element Analysis, also known as Modal Analysis, was carried out using the ANSYS software. After evaluation, the natural frequency mode shape 1 from building 1 was 5.15Hz, while the natural frequency mode shape 2 from building 2 was 9.42Hz. According to the natural frequency data, Building 1 has the lowest value compared to Building 2. The possibility of lower natural frequency causes building 1 to suffer severe damage, including cracks on non-structural and structural cracks. While building 2 has been strengthened structurally in theory, mitigation has been added to the building to enhance the structure further. However, the value of the natural frequencies of building 2 is more acceptable than the value of the natural frequencies of building 1.
Keywords: Peat soils, cracks, residential houses, Finite Element Modelling, ANSYS

*Corresponding author: thayati@uthm.edu.my

1. Introduction

Malaysia is one of the developed countries. In 1991, during the presentation of the Sixth Malaysia Plan, the fourth and seventh Prime Ministers of Malaysia, Mahathir Mohammad, introduced the vision of 2020 to make Malaysia a developed country on par with foreign countries. However, there is a land that is rarely chosen for the building, namely peat soil. Peat soils have a very high moisture content, low bearing capacity, and high compression. Due to its characteristics, peat soil is less preferred for the construction of civil buildings. However, every problem must have a solution, for example needing improvements on peat soil or mitigation technique on building to control the damage on buildings. Before any development, land is essential as a building support. In addition, the Balance of the building also plays an important role to erect on peat soil to prevent the building from tilting. Damage and failure to buildings will have a significant impact on the community. Many cases of soil problems that occurred in our country are the failure of the soil structure. In addition, the ANSYS software tool is chosen because it helps obtain results by using the data or information of the building with mitigation and without mitigation that analyses the strength of the building. The mitigation techniques that have been performed will be analysed to investigate whether these techniques can control damage to cause the building to crack or tilt.

The aim of this research is to investigate a dynamic response on cracked and normal residential houses at peat soil area and to compare the performance of both houses by using Finite Element Modelling (FEM). The study was carried out to investigate the dynamic response of cracked and normal residential house buildings in Ayer Hitam, Muar, Johor using Finite Element Modelling. A residential building's natural frequency and mode shape were used to predict the effect of displacement on the structure. The performance of both houses using Finite Element Modelling (FEM) will be compared in terms of natural frequency and shape mode.

2. Literature review

2.1 Finite element modelling

The Finite Element Method (FEM) has evolved into a critical and important tool for modelling and simulating complex engineering systems. FEM is widely employed in engineering because it enables the mathematical modelling and numerical solution of most difficult problems [1].

The fundamental principle behind the finite element approach is to find a solution that may be used to replace a complicated problem with a more straightforward solution [2]. The solution for the finite element approach is to divide the geometrical space into a smaller region called the mesh [3]. The structure of the small region and its related sub-regions is also referred to as an element. The components are joined by specific joints referred to as nodes or nodal points, which often sit on the element boundaries. Each element can be considered to have an approximate solution, and the overall structure's equilibrium condition can be deduced [2].

FEM is a numerical approach that is primarily concerned with applying differential equations to simulate physics and engineering problems [3]. FEM has been effectively used to various other sorts of engineering issues, including heat conduction, seepage flow, fluid dynamics, and electric and magnetic fields [2].

The finite element analysis is a practical application of FEM (FEA). FEA is used in the engineering field as a computing application for completing engineering analyses. FEA is highly beneficial for analysing problems in complex contexts. Because the continuous is unknown, the change of the field variable, such as displacement, temperature, stress, velocity, or pressure, can be assumed by a more straightforward function in the finite element [2]. Thus, FEM is beneficial and vital in the engineering industry since it enables the solution of complex problems in a more straightforward manner.

2.2 Analysis using ANSYS software

ANSYS, Inc. now offers a broad range of computer-aided engineering tools. Some are intended for general usage, while others are applied to specific electronics, turbo engines, and offshore structures. Some can do all stages of finite element analysis, whereas others focus on a single stage, such as solid modelling or meshing [4]. ANSYS software is a finite element analysis software that can be utilised in various fields, including financial structure analysis, magnetic field analysis, fluid flow analysis, electric field analysis, and sound field analysis [5].

FEM is important for predicting a complicated engineering system's physical or structural behaviour. FEM can assist in analysing data, determining performance, and providing information about the structural response to the vibration effect. Users are given three options to select a mode for the analysis process: interactive mode, batch mode or combined mode. The most general approach is interactive, which entails activating a platform called Graphical User Interface (GUI). The graphical user interface is suggested for beginner ANSYS users [6]. ANSYS supports three distinct forms of analysis: transient analysis, modal analysis, and harmonic analysis.

The ANSYS software consists of three parts, which are the:

i. Pre-processing Module
It comes with a powerful set of meshing and modelling capabilities. The users can quickly and easily develop a finite element model of this module [5].

ii. Analysis Module
There are several types of structural analysis, including sound field analysis, fluid dynamic analysis, piezoelectric analysis, electromagnetic field analysis, and multi-physics coupling analysis. The structural analysis comprises three types of analyses: linear, nonlinear, and highly nonlinear. This module can assist in simulating the interaction of several physical materials, and it has features for optimization analysis and sensitivity analysis [5].

iii. Post-processing Module
The colour gradient vector flow, three-dimensional slices, transparent and translucent areas that may be seen in the structure, and graphically shown results are all calculated with this software. The results of the calculations will be displayed in the form of graphs, forms, or outputs [5].

2.3 Natural frequency

The frequency, measured in Hertz, refers to the number of times the building sways from side to side in a second. The natural frequency of a structure is when it sways as it returns to its original position following disturbance. The intensity of the excitation force will impact the acceleration and speed of the building, but the frequency will remain constant regardless of the force. The natural frequency is approximately dictated by its weight, stiffness, and height. Buildings that are heavy and robust will not sway as easily as light and narrow structures. While Eurocode regulates the vertical vibration limit in the floor, horizontal movement is not.

Additionally, numerous studies have been conducted on this topic, most of which focus on buildings with a natural frequency of 0 – 1.0 Hz. The studies are conducted in that range since low frequencies cause the most concerns in terms of motion sickness for the building's occupants [7]. Because skyscrapers are tall and slender, their inherent frequencies are low, and their deflections are large. This is because the building sways over a long distance, and if the acceleration of the movement is high, the occupants may feel queasy.

Calculating a building's natural frequency, even with the assistance of FEM (Finite Element Approach), is hard, and it is, therefore, vital to apply an approximation method early in the design process. Later, when additional parameters are known, it may be beneficial to develop a FEM model of the structure to check the frequency. Finite element software can compute systems with several degrees of freedom and simulate the motion caused by external forces such as wind [7].

Moreover, this research, numerical programming, and analysis using universal software are effective in determining structural behaviour. Researchers simulate structures subjected to a variety of loading conditions using a variety of computer programmes and software. For example, Siddika et al., (2019) conducted a free vibration analysis using the boundary element method, where a non-singular approach was used to analyse a thin plate and investigated the variation in natural frequency with varying numbers of floors in an RCC building, where STAAD-pro software was used to conduct the analysis. [8] conducted a modal analysis of a titan cantilever beam using ANSYS and SolidWorks software. Thus, dynamic analysis of structures utilising a variety of universal software packages and a variety of factors is beneficial and contributes to the field of dynamic study.

3. Methodology

3.1 Site investigation

A site investigation was conducted at Buildings 1 and 2 to gain a thorough understanding of the materials used in the building structures and the dimensions of the building components. The dimensions of the building structures were determined using measuring tapes.

This study is divided into two buildings, defined as Building 1 and Building 2. Building 1 is 16m x 6m in size, while Building 2 is 16m x 10m in size. Building 1 is estimated to be 3metres high from ground floor to roof beam, while Building 2 is estimated to be 3.5metres high from ground floor to roof beam. The building's frame is constructed of reinforced concrete. The building's wall system is non-load bearing. The wall system is constructed entirely of masonry block units.

3.2 Modal analysis

Modal analysis is a part of procedures used to determine the mode shapes of a building structure. Modal analysis requires the creation of five pre-processing command codes that will be used to model the specified building in ANSYS software. The supervisor had already prepared all of the fundamental command codes. Several modifications to the current command codes were developed to accommodate the specifications and requirements of Buildings 1 and 2. The following five command codes were included in this phase:

i Parameter Geometry
ii Meshing
iii Degree of freedom (DOF)
iv Run modal analysis

3.3 Parameter

Firstly, the parameter was one of the essential commands that needed to be constructed because it contains all the critical building structure's features. The parameters defined in these commands included the beam and column diameters, slab thickness, and the building's width and length. Additionally, it contains the density, elastic modulus, and poison ratio of the materials used to construct the building structures. It will generate all the necessary details for usage in the geometry. The parameters of the building structure used are shown in Table 1, while the size of the beams and slab thickness are listed in Table 2.

Table 1: Properties of the building structure

Model	Materials	Density (kg/m^3)	Elastic Modulus (GPa)	Poisson Ratio
Building 1	Reinforce concrete	2500	34	0.2
	Concrete	2400	38	0.2
Building 2	Reinforce concrete	2500	34	0.2
	Concrete	2400	38	0.2

Table 2: Sizes of beam and thickness of slab

Model	Floor	Beam size (mm)		Slab thickness (mm)
Building 1	Ground Floor	Primary beam	150 x 500	150
		Secondary beam	150 x 450	
	Roof	Primary beam	150 x 400	150
		Secondary beam	150 x 350	
Building 2	Ground Floor	Primary beam	150 x 350	150
		Secondary beam	150 x 350	
	Roof	Primary beam	150 x 350	150
		Secondary beam	150 x 350	

3.4 Geometry

The second command code that needed to be generated was the geometry command code. This command will produce the essential points, lines for horizontal and vertical structural elements, as well as the surface area for the building's flat surface. All major points, lines, and areas were built using the building's original dimensions as depicted in Appendix.

3.5 Meshing

The meshing command code was used to define the mesh element sizes for the building structure section of the model. Meshing is critical to assuming the approximate solution and deriving the overall structure's equilibrium condition for beams, columns, and slabs. The meshing commands enable the creation of the mesh as a single part or as a group. The user-supplied mesh element size will be further analysed in the software to determine the most appropriate mesh element sizes for analysis. If the user enters an incorrect mesh element size, the software will display a warning message during the analysis.

3.6 Degree of freedom (DOF)

In structural analysis, the degree of freedom (DOF) is critical. In mechanical systems, the degree of freedom is utilised to specify the constraint on the structure, which includes the roller, pin, and fixed supports. The constraint type includes translation in the X direction (UX), translation in the Y direction (UY), translation in the Z direction (UZ), rotation around the X axis (ROTX), rotation around the Y axis (ROTY), and rotation around the Z axis (ROZ) (ROTZ).

In the modal analysis, the DOF used was the totally fixed on the ground slab. The ground slab was completely constrained in all directions, both translationally and rotationally.

3.7 Run modal analysis

The final command code produced for performing the modal analysis was the modal analysis command. This command code was created to determine the building structure's dynamic response. By inputting this command code into the ANSYS software, the natural frequency and deformation of buildings 1 and 2 in the form of a mode shape can be obtained.

4. Results and Discussions

4.1 Description of mode shape

The mode shape data in ANSYS represent the form of the entire building. The definition of each colour indicated on the mode shape is shown in Table 3.

Table 3: definition of colours in mode shape

Colour	Definition
Blue	No displacement occurs
Green	Minimum displacement occurs
Yellow	Moderate displacement occurs
Red	Maximum displacement occurs

4.2 Analysis of mode shape of Building 1

According to the ANSYS analysis, the fundamental mode shapes of residential Buildings 1 have natural frequencies of 5.15Hz respectively, swayed into y-direction. The structure exhibited no discernible displacement under this mode form. Under mode shape 1, only the most minor displacement occurs for the entire structure.

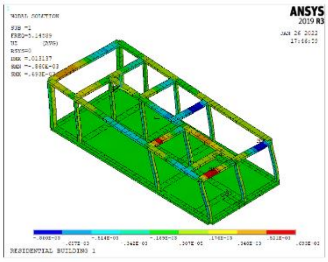
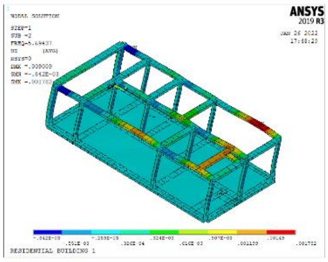
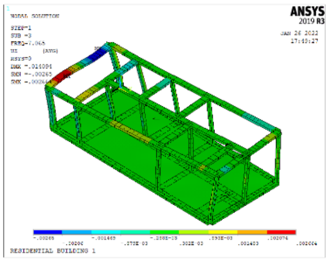
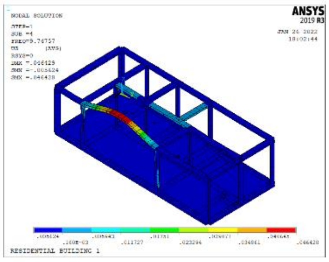
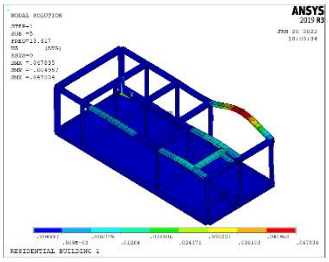
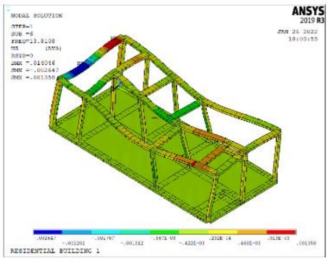
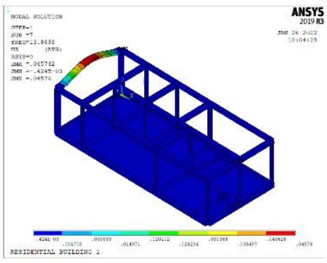
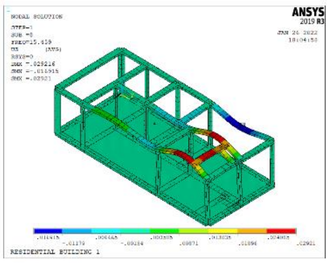
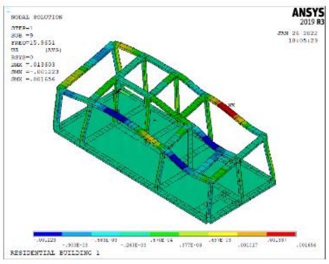
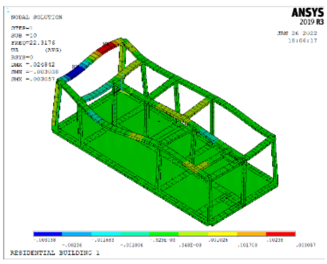
The natural frequency of the mode 2 is swayed, but into x-direction with 5.69Hz, almost similar frequency with the first mode. There was no significant displacement associated with this mode form. While the natural frequencies of Buildings 1 increased slightly to 7.07Hz for mode 3, respectively, front and back buildings swayed to the x-direction in the opposite direction. The roof beam of Building 1 experienced the most significant displacement in this mode. The natural frequency of Building 1 gradually increases to 9.75Hz for mode 4, and the roof beam have a maximum displacement is represent by the red area. This deformation may have an impact on the equipment and occupants.

Additionally, the natural frequency of Building 1 gradually increases to 13.42Hz for mode 5 and the natural frequencies of building 1 increase to 13.81Hz for mode shape 6. For mode 6, the whole of Building 1 was subjected to a serious torsional mode, resulting in the building deforming. This is because the vibrations borne by the building at a rate of 13.96Hz of natural frequency show a significant deflection on the roof beam. The building shows no obvious deformation at 13.97Hz for mode shape 7 but only have maximum deflection at the middle of roof beam as shown in table 4.2. this occurs due to vibrations that cause the building to sway and at the same time, the long span cannot withstand the movement and causes deflection in the centre of the beam.

Moreover, the building shows serious deformation 15.46Hz, 15.97Hz and 22.32Hz. Where the building has deflection on the roof beam. This mode shape applies to the action of natural frequency on the building structure and the occurrence of deformation. In general, the building experiences a minor displacement possible when operating in mode shape 10. The model of the first ten mode shape's natural frequencies is shown in Table 4.

As conclusion, the fundamental mode has natural frequency 5.15Hz of the building is too closed with risk frequency range value by BS5400 and EN1995 standards. It is likely that the closest to the risk of the natural frequency range will be the problem of the effect from wind's dynamic component or human movement. Therefore, the fundamental mode 1 of the building is not acceptable because to avoid the occurrence of future problems.

Table 4: Model of the ten modes and its natural frequencies of Building 1

ANSYS Mode Shape	
	
Mode 1, 5.15Hz	Mode 2, 5.69Hz
	
Mode 3, 7.07 Hz	Mode 4, 9.75 Hz
	
Mode 5, 13.42Hz	Mode 6, 13.81Hz
	
Mode 7, 13.97Hz	Mode 8, 15.46Hz
	
Mode 9, 15.97Hz	Mode 10, 22.32Hz

4.3 Analysis of mode shape of Building 2

The mode shapes of the residential building were presented in three dimensional shapes. The vertical displacement of the floor structure was represented by the colour appeared on the mode shapes.

From the analysis using ANSYS software, the fundamental mode 1 of residential Buildings 2 have natural frequencies of 9.42 Hz, respectively. In shape mode 1 shows that building 2 has all the colours in shape mode including blue, green, yellow and red. The most notable is the roof beam of the building because it has maximum displacement by represent with red colour.

The mode 2 of Building 2 has a frequency range of 9.60 Hz, swayed into x-direction. Mode 3 is also swayed front and back buildings to the x-direction in the opposite direction with natural frequency of Building 2 increases slightly to 11.07 Hz. The roof area of Building 2 experienced a moderate displacement to the right of the roof structure. The natural frequency of Building 2 increased to 11.60 Hz, and there was a bulge on the roof structure in mode shape 4. It occurs in the middle of a building that does not receive vibration and the occurrence of compression on the side of the building due to the frequency present in the building or insufficient strength to support the beam.

In addition, the building show is serious for other modes. Mode 5 swayed at angle of building in x-direction with 13.05 Hz for. The mode 6 is also swayed to y-direction encounter to mid of the building and the natural frequency increased to 15.16 Hz. For mode 6, the roof beam of Building 2 became bending, resulting in the roof beam becoming a curve shape. Mode shape 7 and 8 is swayed in one direction to y-direction with 17.94 Hz and 20.30 Hz. The mode 7and 8indicates the condition of the corrugated roof beam due to the presence of a maximum displacement.

In addition, the mode 9 not swayed in any direction but only have minor maximum displacement and the natural frequency is 22.12 Hz. This deformation may have little an impact on the occupants. Building 2 suffered severe deformation, with the building bulging in all roof areas. Finally, and perhaps most importantly, the natural frequency of the mode shape 10 is 24.06 Hz for Building 2. Under this mode, the shape shows the bending mode on the roof and columns. The natural frequency model of the first ten mode shape is shown in Table 5.

Table 5: Model of the ten modes and its natural frequencies of Building 2

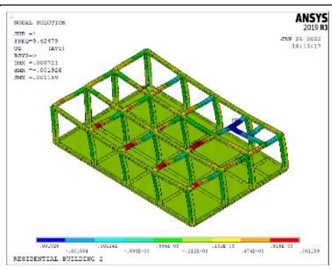
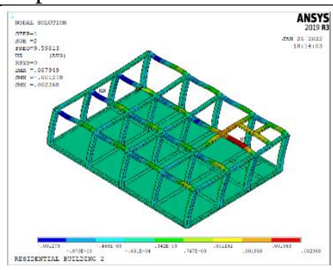
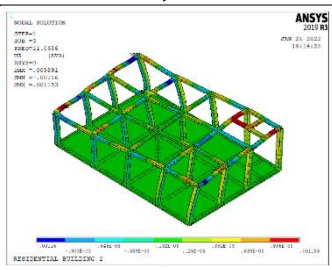
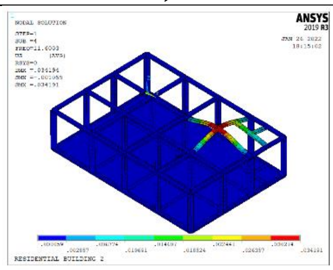
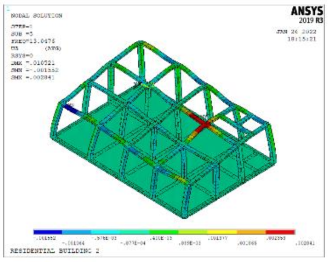
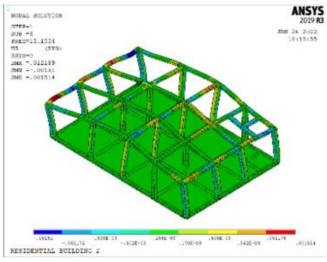
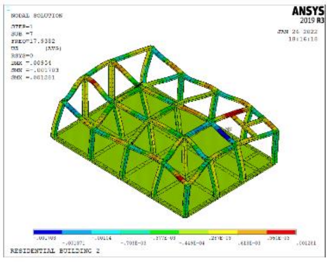
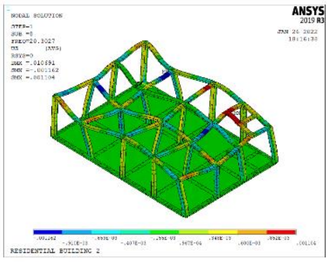
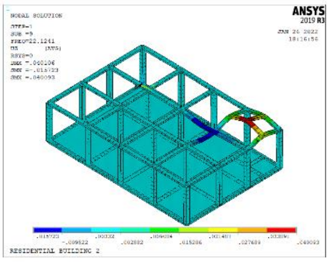
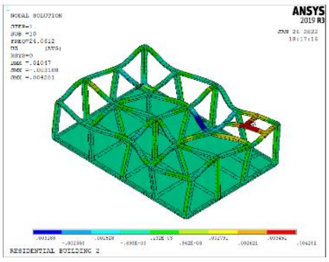
<table>
<tr><td colspan="2">ANSYS Mode Shape</td></tr>
<tr><td></td><td></td></tr>
<tr><td>Mode 1, 9.42Hz</td><td>Mode 2, 9.60Hz</td></tr>
<tr><td></td><td></td></tr>
<tr><td>Mode 3, 11.07Hz</td><td>Mode 4, 11.60Hz</td></tr>
</table>

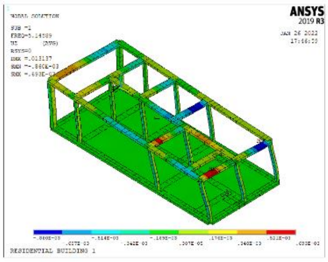
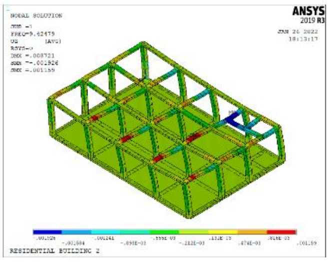
Table 5: Model of the ten modes and its natural frequencies of Building 2 (continued)

ANSYS Mode Shape	
	
Mode 5, 13.05Hz	Mode 6, 15.16Hz
	
Mode 7, 17.94Hz	Mode 8, 20.30Hz
	
Mode 9, 22.12Hz	Mode 10, 24.06Hz

4.6 Compare the performance Building 1 between Building 2

The data recorded from ANSYS software were used to obtain the natural frequencies and mode shapes of both buildings that were analysed. According to Table 6 and Table 7, both buildings' natural frequency of the first ten mode shapes and Model of first modes and its natural frequencies of both building were obtained. Therefore, to compare the data, the results are presented in values of frequencies for better understanding. Besides, only one data of mode shape 1 will compare building 1 with building 2. The natural frequency for Building 1 has been obtained as 5.15 Hz. While the frequencies of building 2 are 9.42 Hz. From the frequencies data for mode shape 1, Building 1 has the lowest value compared to Building 2. The possibility of lower natural frequency causes building 1 to suffer severe damage, including cracks on non-structural and structural cracks. While building 2 in theory, has been added mitigation to the building to strengthen the structure of the building further. However, the value of natural frequency of building 2 is more acceptable than building 1 to be a reference source for engineers in the future. According to [9], these low natural frequencies present a problem regarding the effects of wind's dynamic component or human movement.

Table 6: Model of the ten modes and its natural frequencies of both building

ANSYS Mode Shape	
Building 1	Building 2
	
Mode 1, 5.15Hz	Mode 1, 9.42Hz

In conclusion, risk frequency range values by all standards had been shown in Table 4.5 that Building 2 exceeds the indicated range and the fundamental natural frequency of Building 2 is acceptable compared to Building 1. Additionally, the roof region of both buildings has the most displacement for most of the mode shape. The roof has the highest displacement compared to the ground floor and the most significant distance from the ground level. If seen the natural frequency for building 1 exceeds all standards but the value issued is close to range of all standards. With this evidence, Building 1 is not acceptable in terms of building stability for the safety of future users.

Table 7: Risk frequency range value by all standards [9]

Standards	Risk frequency range value
EN 1992	1.60 Hz – 2.40 Hz
EN 1995	0.00 Hz – 5.00 Hz
BS5400	0.00 Hz – 5.00 Hz
ISO/DIS 10137	1.70 Hz – 2.30 Hz
CEB 209 Bulletin	1.65 Hz – 2.30 Hz

5. Conclusion

In conclusion, this study accomplished both of its objectives. The first purpose of this project is to perform a dynamic response on crack and normal residential houses at peat soil areas. Additionally, ANSYS software was used to do a modal analysis. Modal analysis can determine the natural frequencies and geometry of the structure's modes. Additionally, the modal analysis results can be used as a starting point for dynamic analyses such as transient dynamic analysis. Moreover, modal analysis can determine the mode shapes of a structure. This phase includes the following five commands: parameter, geometry, meshing, degree of freedom (DOF), and modal analysis.

The second purpose of this study is to compare the performance of both building by using Finite Element Modelling (FEM). For both buildings, the modal analysis determined the first 10 mode shapes and natural frequencies. Both buildings' natural frequencies are very different. Meanwhile, the mode shape indicates that some modes have a maximum displacement. Additionally, the roof region experiences the most displacement as compared to the ground floor, column, and so on.

However, this study found that structures with high natural frequencies showed the best performance. It is supported because a low building will be safer with a high natural frequency because a high natural frequency will not allow the building to move much and can avoid significant damage to the building structure. Compared to tall buildings, it is not safe with high natural frequency because the building will be erratic and not static at one point, this will cause the building to be damaged and collapse.

Acknowledgement

The author would like to give her deep appreciation to Faculty of Civil Engineering and Built Environment, Universiti Tun Hussein Onn Malaysia for the endless support and complete facilities for her being able to conduct the study.

References

[1] G. R. Liu and S. S. Quek, "Finite Element Method: A Practical Course," *Finite Elem. Method A Pract. Course*, pp. 1–348, May 2003, doi: 10.1016/B978-0-7506-5866-9.X5000-2.

[2] S. S. Rao, "The finite element method in engineering," *Finite Elem. Method Eng.*, pp. 1–763, Oct. 2017, doi: 10.1115/1.3167179.

[3] T. N. Tuan Chik, R. A. Asiew, M. H. Wan, and N. A. Yusoff, "Dynamic Performance on Multi Storey Structure Due to Ground Borne Vibrations Input from Passing Vehicles," vol. 5, no. 2, pp. 51–58, 2013.

[4] M. K. Thompson and J. M. Thompson, "ANSYS Mechanical APDL for Finite Element Analysis.," p. 467, 2017, Accessed: Jan. 03, 2022. [Online]. Available: https://books.google.com/books/about/ANSYS_Mechanical_APDL_for_Finite_Element.html?id=qbT5DQAAQBAJ.

[5] J. Ji, W. Zhang, W. Zhao, C. Yuan, and Y. Liu, "Research of seismic testing and dynamic character of high-rise building structure based on ANSYS," *Int. J. Digit. Content Technol. its Appl.*, vol. 6, no. 8, pp. 63–71, May 2012, doi: 10.4156/JDCTA.VOL6.ISSUE8.8.

[6] E. Madenci and I. Guven, "The finite element method and applications in engineering using ANSYS®, second edition," *Finite Elem. Method Appl. Eng. Using ANSYS, Second Ed.*, pp. 1–657, Jan. 2015, doi: 10.1007/978-1-4899-7550-8.

[7] J. Eriksson, "Horizontal Natural Frequency In A 10 Story Building A Comparison Between Clt And Horisontell Egenfrekvens I Ett 10- Våningshus," 2018.

[8] P. Lengvarský, J. Bocko, and M. Hagara, "Modal Analysis of Titan Cantilever Beam Using ANSYS and SolidWorks," *Am. J. Mech. Eng.*, vol. 1, no. 7, pp. 271–275, 2013, doi: 10.12691/ajme-1-7-24.

[9] P. Hradil, V. Salajka, and J. Kala, "Requirements of technical standards for the dynamic analysis of the load-bearing structures of footbridges," *MATEC Web of Conferences*, vol. 107. 2017, doi: 10.1051/matecconf/201710700010.