Numerical Analysis of Transmission Tower on Pile Foundation

Lakshmi S[1]
[1]Post Graduate Student,
Department of Civil Engineering
Saintgits College of Engineering
(Autonomous)

Pinky Merin Philip[2]
[2]Assistant Professor,
Department of Civil Engineering
Saintgits College of Engineering
(Autonomous)

***Abstract*—Electrical power demand has been increasing around the world, and many large-scale transmission towers have been newly constructed. A Transmission line (TL) tower is supported on a pile foundation, particularly in clayey soils. The differential settlement of the foundation causes failure of the entire structure when subjected to lateral and vertical loads. Before designing and planning in construction, analysis of the structure becomes important in the process. The work aimed to prepare a model of the whole structure and to perform a static analysis of the foundation of the TL tower. In this work, a steel lattice tower structure resting on a group of three piles embedded in weak soil is analyzed to obtain their stresses and deformations. Three analyses were done in the ANSYS workbench. The modal analysis, static analysis, and response spectrum structural analysis have been considered. The modal analysis was applied to verify the model compatibility. The static analysis was applied to find maximal deformations and stresses caused by overhead conductors and self-weight of the structure. The response-spectrum analysis measures the contribution from each natural mode of vibration to indicate the likely maximum seismic response of an essentially elastic structure. The result pointed out that the analysis of the foundation of the structure is important for the realistic understanding of such structures.**

Keywords — Electrical transmission line tower structures; pile foundation; differential settlement; finite element analysis.

I. INTRODUCTION

The transmission line tower structure is an important infrastructure for the electric power supply system. Generally, a transmission tower is a medium to carry power loads from one station to another station. It is usually composed of steel and can run at long distances. Transmission towers consist of several types and are designed by the tower height and capacity of the tower to support the load from the conductor, compression load, wind load, vertical load, longitudinal load, and uplift load. Transmission towers are usually used where a large amount of electrical current is to be distributed often ranging from 115 to 800 kV. Transmission towers are a vital component and management to assess reliability is needed to minimize the risk of power supply disruption that may result from in-service tower failure. It Consists of conductors, overhead power lines, steel-lattice tower & foundation parts. Foundation parts are key components to guarantee sustainability & continuous serviceability of the entire transmission system. The foundations are generally installed at the four corners of the tower structure. The importance of the transmission tower on the national economy and people's living has been well recognized. Unfortunately, natural disasters such as earthquakes and floods come without notice. These natural phenomena caused major damages to the transmission towers.

Before planning and designing in construction, analysis of the design turns into the primary part all the while. Transmission tower structures are generally analyzed by linear static analysis methods but it is also necessary to determine the deformation of the structure as well so that any failure can be avoided. Also, Finite Element Analysis was applied towards the plan of the transmission tower. Be that as it may, utilizing this manual strategy is less productive and less down to earth. In this manner, a program called ANSYS was utilized to figure estimation to make a safe and optimum design. Henceforth the principal destinations of this research are:

- To analyze a TL transmission tower using Finite element analysis - ANSYS Workbench software.
- To determine deflection at critical points.
- To determine stresses on pile foundation.

II. LITERATURE SURVEY

Beilstein [5] *et al.*, carried out Finite element (FE) modeling software to model the tower's observed deflection and the discrepancy in the sag of the supported spans. The FE model served to determine the cause of the deflection and provide a conductor stringing sequence to mitigate it. After the conductor stringing mitigation sequence was completed, the tower and spans moved to their expected positions per the original design. Lattice tower height and overturning moment are significant factors in the possible deflection and rotation experienced both during the stringing and in the final constructed state. However, load development through the lattice structure is the key to maintaining a rigid lattice tower. A good lattice tower design accounts for these factors – in both everyday loads and extreme weather events. It may be necessary to analyze the tower under each loading configuration encountered during stringing and tensioning of the cables. This is especially useful when the stringing plan requires differential tensions on the tower during construction. They concluded that the use of FE modeling software can enable the analysis and consideration of these effects in a cost-effective and timely manner.

Luan [4] *et al*., carried out the nonlinear, explicit, and finite difference program FLAC3D, which considers the mechanical behavior of soil-pile interaction, is used to establish an under consolidated soil-pile mode. The response processes of the pile side friction force, the pile axial force, and the soil response under seismic load are also analyzed. Under the seismic dynamic loads, the distributions of the negative and positive pile side friction forces vary over the seismic duration. The axial force of the pile with seismic load changes over the longitudinal waves in the seismic wave. The compression and stretching effects on the soil occur alternately when the longitudinal waves pass, causing an alternating increase and decrease in the settlement of the soil. Horizontal displacement of the soil around the pile does not occur without seismic load.

Patel [2] *et al*., evaluated the effect of soil stiffness on connected and unconnected structural systems to understand the role of various parameters like skin friction resistance, lateral resistance of pile-raft, and the lateral earth pressure. As the pile cap is placed above ground level and the piled raft is placed on the ground as the top of the raft coincides with the ground surface, the resistance of the piled raft against lateral loading and uplift effect is more as compared to the pile cap. The lateral resistance of the raft occurred due to the passive pressure of the surrounding soil. They observed that, as the number of piles increased in the piled raft, the load sharing by the raft is decreased. Also, in the case of the pile raft system, the lateral resistance provided by the raft was more than the skin resistance of the pile.

Shah [7] *et al*., studied the behavior of steel tower structure resting on a pile and provide an economical design and to understand the performance of such structures is necessary to know their behavior considering soil-structure interaction. They analyzed steel tower structure resting on a single pile at each leg and group of three piles embedded in soft, medium, and stiff soil to obtain their stresses and deformations. The modal analysis, static analysis, and transient structural analysis considering the Kobe earthquake have been considered. The number of piles increases the frequency obtained for soil-structure system increases in both soft, medium, and stiff The maximum stress in the steel structure obtained for a fixed base condition is found to be 198MPa which is lesser than the yield stress by 22.8%. The vertical deformation or settlement decreases respectively for medium and stiff soil to that of soft soil.

Shu [6] *et al.*, designed two 1:2 scaled substructure models for a typical 110 kV transmission tower. The scaled tower substructure models were tested subjected to the stretching movements of the horizontal ground surface under different wind load conditions. The wind speeds were assumed to be 15 m/s and 30 m/s, respectively in this study. The deformations of the tested tower models and the stresses and strains within the different members of the tower were fully measured. Also, a FE model using ANSYS was developed and validated by the test data. With increased wind load, the bottom cross-bracing members were failed at less support's displacement. The peak stress in bracing member F11 was increased considerably with the increased wind speed and the corresponding ultimate support's displacement at the peak stress was decreased. Compared with the case of zero wind speed, the maximum stress in the member F11 was increased by 7.87% and the limit support displacement was decreased by 18.9% for the wind speed of 30 m/s. Hence the wind load has a significant adverse effect on the resistance of the power transmission towers to the ground surface deformation. The FE model developed in this research can be used to study the structural behavior of power transmission towers subjected to the ground surface deformation under different wind loading conditions.

III. DETAILS OF TOWER AND FOUNDATION

A. Pile foundation

Pile foundation belongs to the classification of deep foundation which transmits the loads coming over it safely to the ground. The pile foundation can be constructed in non - cohesive soil by undergoing techniques of displacement and vibration methods. Pile foundation helps in reducing the differential settlement. Piles are usually slender having a high length to width ratio. They are fundamentally designed to resist axial loads. They transmit the load to the underlying stronger soil layer.

B. Finite element model

The pile foundation and tower were modeled using the Ansys workbench. The finite element analyses were planned for 220/110 kV transmission line tower structures. The height of the tower structure is 43 m. The material properties of the tower such as Elastic modulus and Poisson's ratio are 2.01×10^5 N/mm^2 and 0.3 respectively. The sectional properties are shown in table 1.

The foundation of the transmission tower is placed at the four legs of the tower, and the foundation at each side consists of a pile group with three piles. The diameter and length of the piles are 1 and 30 m, respectively, and the pile-to-pile spacing equals 3 m. The depth of the pile cap is 1.5 m. The isometric and front view of the finite element model of the tower with pile foundation is shown in fig. 1. The design was according to IS 2911- (part 1/sec 2) – 2010, IS 456: 2000 cl 26.5.2.1, SP16.

TABLE 1. THE SECTIONAL PROPERTIES OF THE TOWER MEMBERS

Member type	Dimension (mm×mm)	Steel grade N/mm^2
Tower leg	L125×10	500
Horizontal Diaphragm	L90×7	500
Diagonal Member	L90×7	500
Truss member	L90×7	500

The soil is assumed as homogenous and the properties of soil are given in table 2.

TABLE 2. PROPERTIES OF SOIL

Properties	Homogenous soil
Modulus of Elasticity, E_s (kN/m^2)	14,000
Poisson's ratio	0.4
Density (kg/m^3)	1600
Cohesion (kN/m^2)	38
Coefficient of friction	0.95

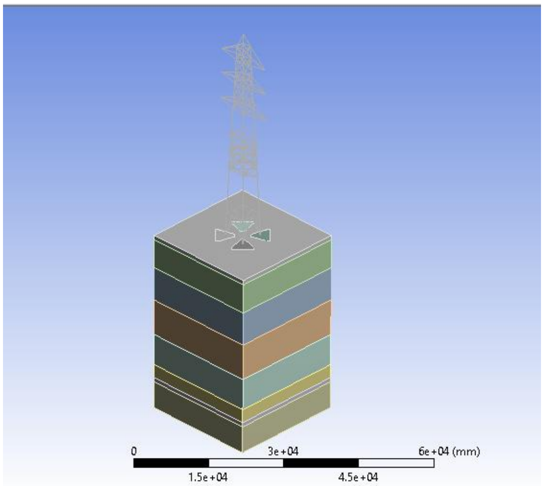

Fig.1. Finite element model of TL Tower with pile foundation

C. Boundary Conditions

The support conditions are shown in fig. 2. Top of the soil model is a free edge without constraints. The bottom is a fixed edge constraining all displacements. The top of the pile is free and unconstrained. The bottom surface is bonded to the soil.

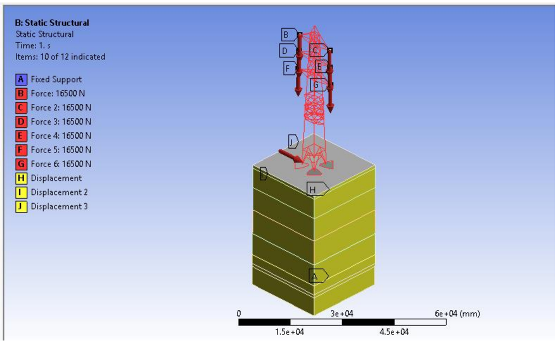

Fig. 2. Supports & Loading

The transferred loads on the foundations were found from the geometric condition of the transmission tower and also foundation is subjected to lateral load and cable load on the tower structure. Seismic condition is also analyzed. A cable load of 17 kN was applied on the cross-arms of the tower. The wind pressure was also calculated based on zone V. Seismic analysis was done by using the frequency response spectrum (FRS) of Zone II. Meshing is also done to improve the accuracy, convergence, and speed of the solution. Fig. 3 shows the meshing diagram of this structure.

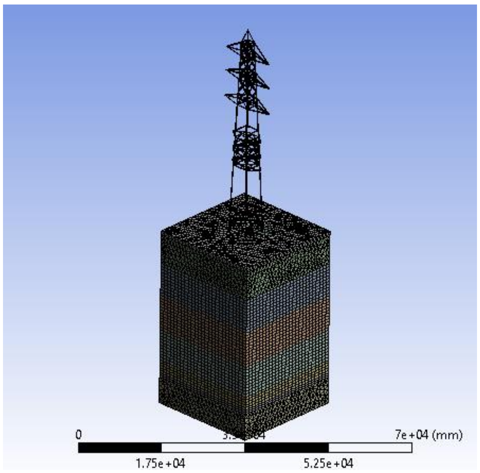

Fig. 3. Meshing Diagram

D. Finite element analysis

Static analysis is carried out by modeling the entire structure in ANSYS Workbench and then material properties, loads are assigned. Modal analysis in structural mechanics is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure. The natural frequencies and mode shapes are the main parameters to be considered for the analysis of the structure under dynamic loading conditions. Modal analysis is performed to distinguish the modes, and the response in that mode can be taken from the response spectrum. These peak responses are then joined to assess a complete response.

IV. RESULTS AND DISCUSSIONS

Fig. 4. shows the deformation of the TL tower. The maximum deformation is at the top portion of the tower. Fig 5. shows the deformation of Pile. The maximum deflection is observed at the pile head. The maximum deformation of the pile cap occurs at the top face of the pile cap as shown in fig. 6. The maximum deformation of the tower leg is observed at the joint portion of the tower leg as shown in fig. 7. Fig. 8 shows the stress in soil which is concentrated at the pile cap portion. The maximum stress in the pile is as shown in fig. 9. Stress is concentrated at the top face of the pile. The stress in the pile cap is shown in fig. 10 and the maximum value is concentrated at the edges of the pile cap.

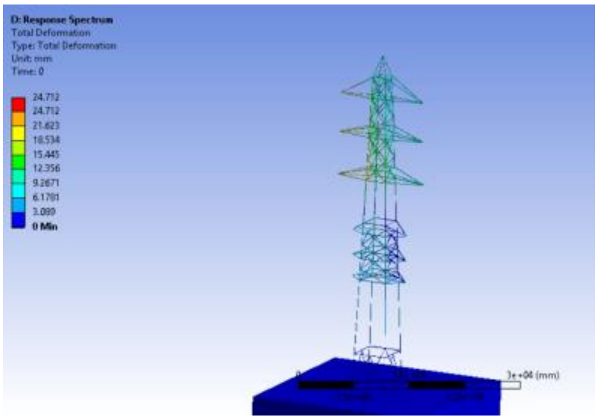

Fig. 4. Deformation of TL Tower

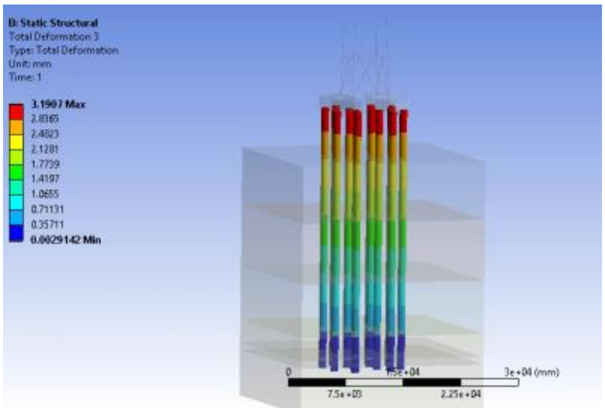

Fig. 5. Deformation of Pile

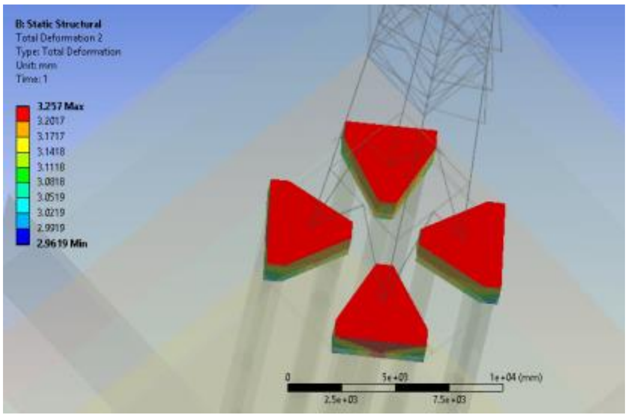

Fig. 6. Deformation of pile cap

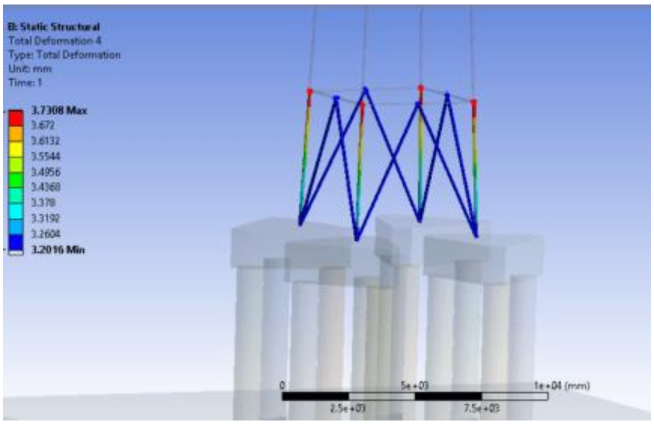

Fig. 7. Deformation of Tower leg

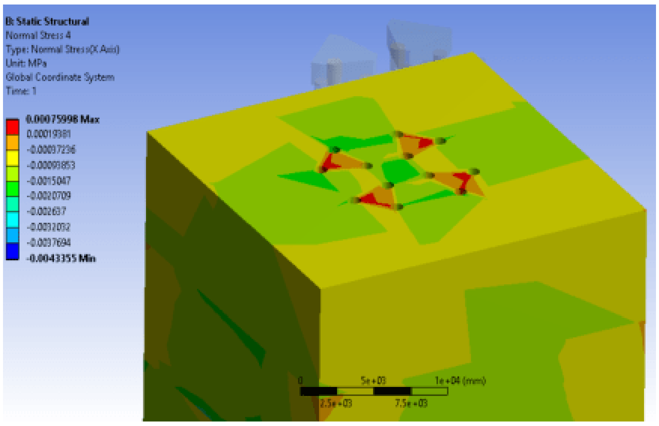

Fig. 8. Stress in soil

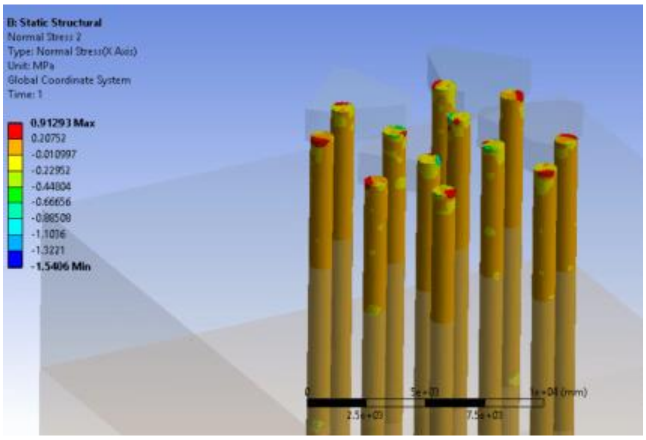

Fig. 9. Stress in pile

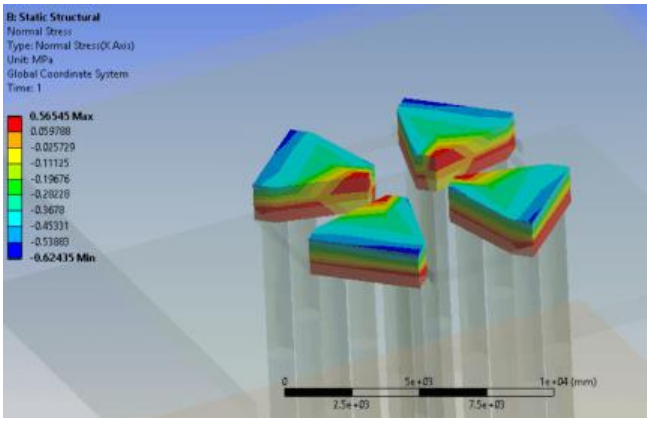

Fig. 10. Stress in Pile cap

V. CONCLUSIONS

The work aimed to prepare a model and to perform a static analysis of the transmission tower with a foundation structure that is fixed to the earth. We made three types of different analyses. The conclusions were:

- The maximum deflection and maximum stress at critical points of the structure can be measured by using the ANSYS workbench.

- Results obtained by the finite-element program Ansys Workbench can be considered approximate.
- An advantage of the process of a solution is its speed and flexibility. In case of a change of applied profiles, we do not need to define the whole geometry from the ground, we just need to allocate a new profile to a selected body.
- The FE model elaborated in this research can be used to study the structural behavior of transmission towers subjected to lateral loading and vertical loading condition.

REFERENCES

[1] A. Jendoubi and F. Legeron, "Effect of the dynamic soil-structure interaction on rigid transmission line tower subjected to wind and impulse loads", Electrical transmission and substation structures, 7, pp. 324-335, 2013.

[2] A. S. Patel, U. K. Patel, M.V. Shah, "Effect of Pile Group and Pile-Raft System on Uplift Capacity of Cable-Stayed Transmission Tower through Soil-Structure Interaction", International Research Journal of Engineering and Technology, 2020.

[3] E. Savory, G. A. R. Parke, P. Disney, N. Toy, "Wind-induced transmission tower foundation loads: a field study-design code comparison", Journal of Wind Eng. Indian Aerodynamics, pp. 1103–1110, 2008.

[4] L. Luan, Y. Liu and Y. Li, "Numerical Simulation for the Soil-Pile-Structure Interaction under Seismic Loading", Mechanical problems in engineering, pp.1-7, 2015.

[5] N. Saumya, S. Beilstein, L. Jessen, J. Frantz, M. Nicholson, K. Kator and K. Heller, "Lattice Tower Deflection and Modeling of the Structure and Spans in Practice", Electrical Transmission and Substation Structures, 2018

[6] Q. Shu, Z. Huan, G. Yuan, W. Ma, S. Yea and J. Zhou "Impact of wind loads on the resistance capacity of the transmission tower subjected to ground surface deformations", Thin-Walled Structures,13, pp. 619-630, 2018.

[7] S. J. Shah and Sreerag, "Assessment of Soil-Structure Interaction and Vibration Analysis of a Tower Structure", International journal of scientific & engineering research, http://www.ijser.org, 2016.

.