

FINITE ELEMENT ANALYSIS OF TRANSMISSION LINE TOWERS

¹Sriranjini.S, ²Shobby Margret Lydia

Department of Civil Engineering

Dhanalakshmi Srinivasan College of Engineering, Tamil Nadu, India

Abstract – The present paper deals with the study of static and dynamic analysis of transmission line tower (X type of bracing system). A typical type of transmission line tower carrying 220KV single circuit conductor is chosen as case study existing in bijapur district. The analysis and modeling of tower is carried out using FE based ANSYS software. The model is created in CATIA and then imported to ANSYS workbench. The loads acting on the tower considered are dead load, live load and dynamic loads (Seismic and wind). The existing tower has height of 40m, which includes ground clearance, maximum sag of the conductor, vertical spacing between conductor wire. Static and dynamic analysis is carried out in detail using FE based ANSYS software. Static, modal, response spectrum and wind analysis is performed. Response spectrum analysis is carried out for 0.3g, 0.4g, 0.5g ground accelerations by considering DBE (Design basis earthquake) for earthquake zone III. Wind zone considered is zone III. The wind pressure depends on the gust response factor, which increases with height. The behavior of existing tower (X type of bracing) is analyzed for different analysis. The maximum deformation, combined stresses, natural frequencies and direct stress are obtained and plotted graphically.

Keywords— Dynamic, Static, Seismic, Tower, Wind

1. INTRODUCTION TO TRANSMISSION LINE TOWERS

The need of electric power consumption has continued to increase in every country, the rate of demand being greater in the developing countries. The transmission line towers are one of most important life-line structures which help in transmitting electric power. The Transmission towers are necessary for the purpose of supplying electricity to various regions of the nation. In present scenario, there is increase in building of power stations and consequent increase in power transmission lines from the generating stations to the different corners. Interconnections between systems are also increasing to enhance reliability and economy. Transmission line should be stable and carefully designed so that they do not fail during natural disaster and should conform to the national and international standard. The planning and designing of a transmission line include a number of requirements of both structural and electrical. From the electrical point of view, the most important requirement is insulation and safe clearances of the power carrying conductors from the ground. The cross-section of conductors, the spacing between conductors, and the location of ground wires with respect to the conductors will decide the design of towers and foundations. Transmission

line is an integrated system consisting of conductor subsystem, ground wire subsystem and one subsystem for each category of support structure. Mechanical supports of transmission line represent a significant portion of the cost of the line and they play an important role in the reliable power transmission. They are designed and constructed in wide variety of shapes, types, sizes, configurations and materials. The supporting structure types used in transmission lines generally fall into one of the three categories: lattice, pole and guyed. The supports of EHV transmission lines are normally steel lattice towers. The cost of towers constitutes about quarter to half of the cost of transmission line and hence optimum tower design will bring in substantial savings.

Objectives of Present Study -The primary objectives of the study are as follows. 1] To Study the linear static analysis of existing transmission line tower (X type bracing system for combined stresses and Deflection). 2] To Study the free vibration analysis of existing tower using FEA. 3] To Study the modes of vibration, natural frequencies and combined stresses. 4] To Study the frequencies, stresses for existing tower subject to earthquake excitation (ground acceleration). 5] To Study the deflection pattern subjected to wind load acting on the tower. 6] To Find dynamic characteristics of present existing tower and finding out dominant load.

Methodology Based on the objectives of present work following methodology has been set. 1] Static Analysis – Dead Load or Self-Weight and live load. 2] Dynamic Analysis – Seismic Analysis, wind analysis 3] ANSYS (work bench) software. [Link element]. Modeling is carried out using CATIA. 4] IS 875 Part III- 1987- Code of practice for design loads (Other than earthquake) for buildings and structures. Wind loads. 5] IS 802 (Part 1)- 1995 section I- Use of Structural steel in Overhead transmission line towers- Code of Practice. IS 1893 part-I (2002) – Criteria for Earthquake Resistant Design of Structures, Part 1: General Provisions and Buildings. IS 1893 part-IV (2005) – Criteria for Earthquake Resistant. Design of structures. Part 4 industrial. Structures.

INTRODUCTION TO FEM PACKAGE

ANSYS V.15 is an integrated design analysis tool based on the FEM developed by ANSYS, Inc. It has its own tightly integrated pre and post- processor. The ANSYS product documentation is excellent and it includes commands reference; operations guide; modeling and meshing guide; basic analysis procedures guide; advanced analysis guide; element reference; theory reference; structural analysis guide; thermal analysis guide; electromagnetic fields analysis guide; fluid dynamics guide; and coupled field analysis guide. Taken together, these manuals provide descriptions of the procedures, commands, elements, and theoretical details needed to use the ANSYS program. All of the above manuals except the ANSYS theory reference are available online through the ANSYS help system, which can be accessed either as a standalone system or from within the ANSYS program. ANSYS is a complete FEA simulation software package developed by ANSYS Inc-USA. The company was founded in 1970 by Dr. JOHN SWANSON and originally named as Swanson Analysis systems; Inc. ANSYS is the original name for the commercial products. ANSYS have a big family of products, which are developed to deal with special purpose problems.

INTRODUCTION TO FINITE ELEMENT MODELLING

The Basis of the finite element method is the representation of a body or a structure by discretizing and then assembling of subdivisions called finite elements. The Finite Element Method translates partial differential equation problems into a set of linear algebraic equations. The finite element method is a numerical technique of solving differential equations describing a physical phenomenon. It is a convenient way to find displacements and stresses of structures at definite physical coordinates called nodes. The structure to be analyzed is discretized into finite elements connected to each other at their nodes. Elements are defined and equations are formed to express nodal forces in terms of the unknown nodal displacements, based on known material constitutive laws. Many software programs are available in the market for the analysis of structures by this method. In the present study, the computer program ANSYS is used for the analyses. The tower was modeled as a steel structure. For the static analysis, the loads considered were dead load and live load. CATIA and ANSYS software were used for modeling. CATIA was used to draw the line model of tower then the line model is exported to ANSYS, then properties and loads were assigned.

Details of Transmission Line Tower

A Transmission line tower of 40m height is considered for the analysis. Case study is considered and the analysis is carried out using ANSYS Workbench software.

1. Location of tower- Bijapur
2. Zone - Zone III
3. Height of tower- 40m
4. Bracing system-X type bracing system
5. Number of panel -15 number
6. Capacity of tower -220KV
7. Single circuit transmission line tower
8. Importance factor of tower -1.5
9. Reduction factor of tower- 3

Input Geometrical Data

1. Height of tower- 40m
2. Bottom width of tower -10.650m
3. Importance factor - 1.5
4. Response Reduction factor of – 3 [Steel frame with concentric braces].
5. Damping ratio - 2% 6. Zone – Z=0.16.
7. Soil Type- Soft soil.

Manual calculations are carried out for sag tension in conductors, wind load on conductor and ground wire, wind calculation for X type bracing system, sag and tension calculation for 220KV Transmission tower, Response spectrum calculations for 0.3g, 0.4g, 0.5g ground

accelerations. For wind calculations the forces are converted to joint load and are applied on the tower.

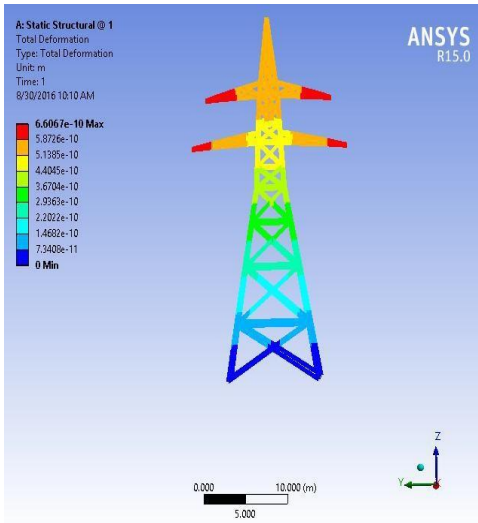


Fig 1: Total Deformation

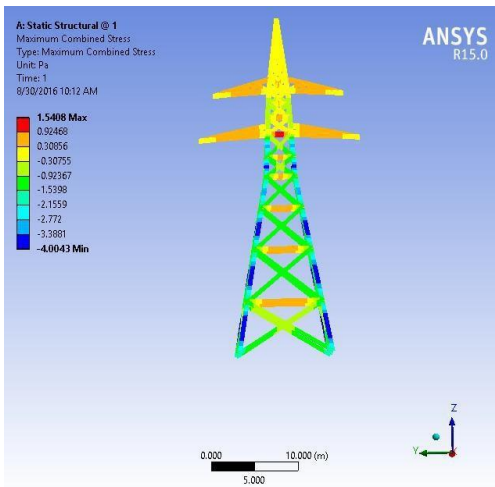


Fig 2: Maximum combined stress

Modal Analysis

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 important parameters in the design of a structure for dynamic loading conditions. They are also required if you want to perform spectrum analysis or a mode- superposition harmonic or transient analysis. It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analyzed can have arbitrary shape and the results of the calculations are acceptable. Procedure: Model is created in CATIA and then imported to ANSYS software. The modal analysis is carried out by extracting 10 modes. The First 10 number of modes is extracted and natural frequencies are obtained. The deformations and stresses are also obtained for 10 numbers of modes. The results are read in post processor.

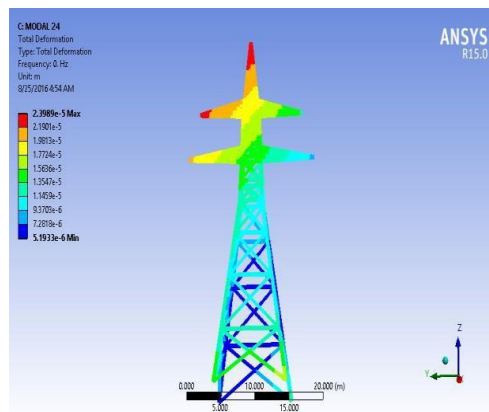


Fig 3: Total Deformation (Mode 1)

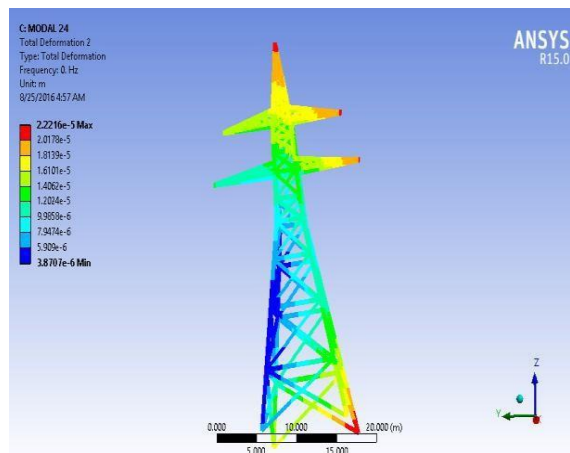


Fig 4: Total Deformation (Mode 2)

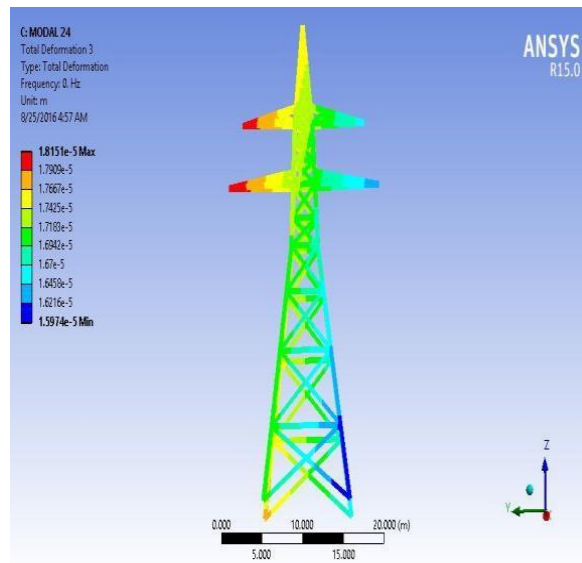


Fig 5: Total Deformation (Mode 3)

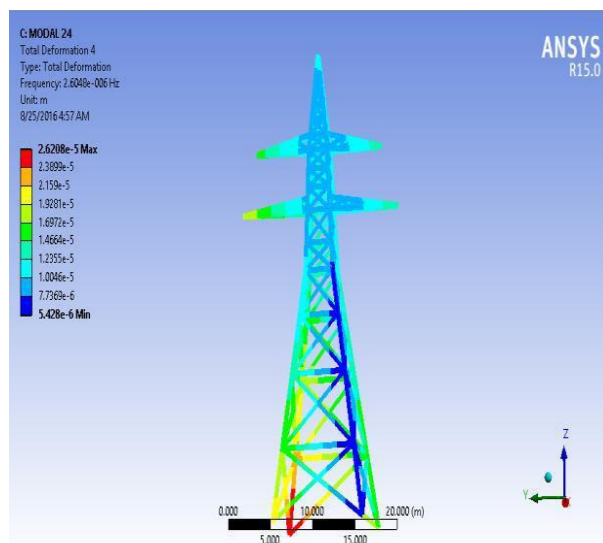
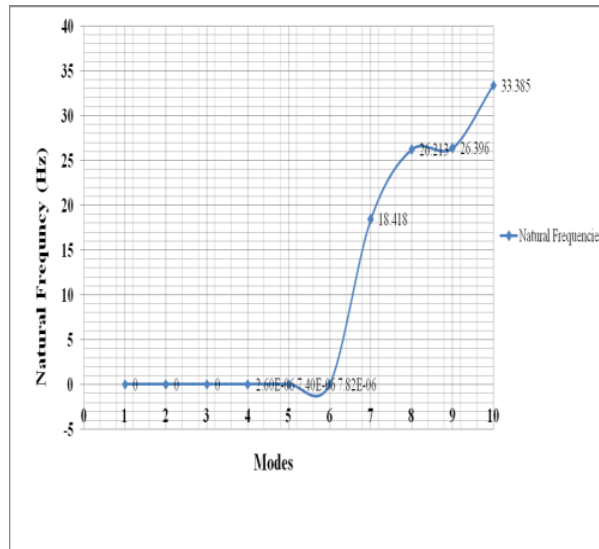
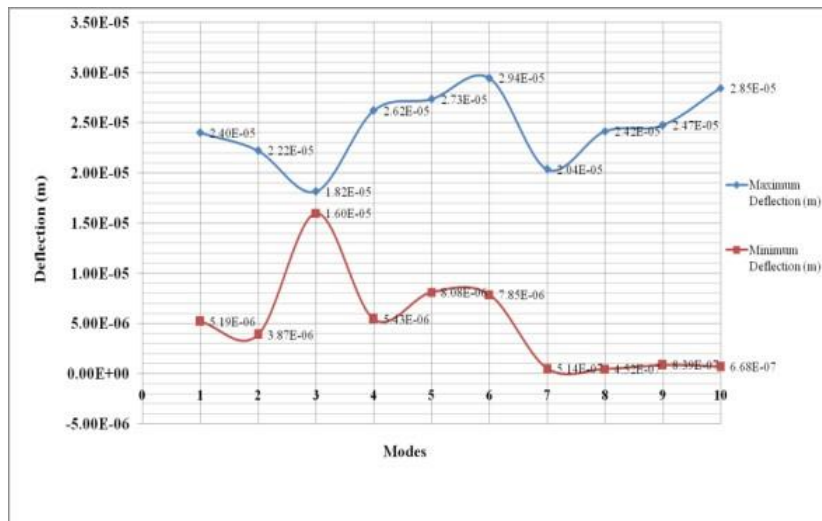


Fig 6: Total Deformation (Mode 4)



Graph 1: Graphical representation of natural frequency v/s modes

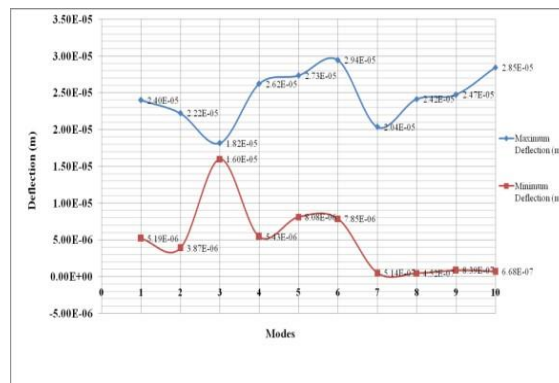


Graph 2: Graphical representation of maximum and minimum deflection v/s modes

LOADS CONSIDERED

The transmission line towers are subjected to the following loads:

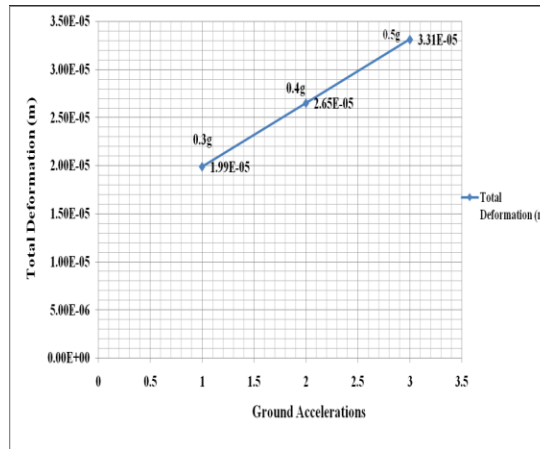
- a) Vertical Loads b) Transverse loads c) Longitudinal loads
- d) Torsional loads
- a) Vertical loads :
 - i) Weight of tower structure
 - ii) Weight of insulator strings and fittings
 - iii) Weight of power conductors
 - iv) Weight of ground wire
 - v) Weight of ice coatings(if any)
 - v) Weight of the maintenance crew with tool (1.5 kN)



Graph 2: Graphical representation of maximum and minimum deflection v/s modes

Response Spectrum analysis

A Spectrum analysis is one in which the results of a modal analysis are used with a known spectrum to calculate displacements and stresses in the model. It is mainly used in place of a time-history analysis to determine the response of structure to random or time-dependent loading conditions such as earthquakes, wind loads, ocean wave loads, jet engine thrust, rocket motor vibrations, etc. The spectrum is a graph of spectral value versus frequency that captures the intensity and frequency content of time-history loads. Procedure: Model is created in CATIA and then imported to ANSYS software. The response spectrum analysis is carried out for 0.3g, 0.4g and 0.5g ground accelerations. The response spectra curves are plotted for 0.3g, 0.4g, 0.5g considering DBE (Design basis earthquake). The values of frequency and accelerations of corresponding response spectra curves are calculated and then assigned in response spectrum method. By SRSS method of mode combination, the problem is run and then output is read in post processor.



Graph 3: Graphical representation of total deformation v/s ground acceleration

Wind analysis

Procedure: Model is created in CATIA and then imported to ANSYS software. The wind load calculations are carried out for existing tower. The wind zone considered is zone III. The calculated loads are assigned on each node from top to bottom of the tower. The analysis is run and output is reading post processor.

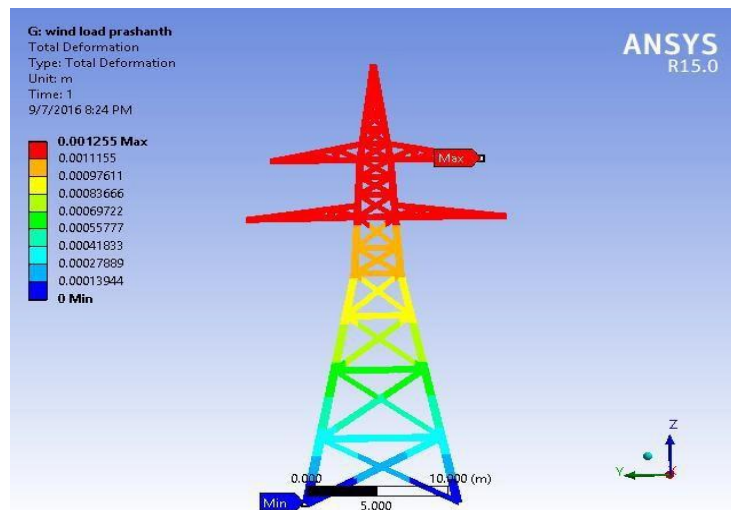


Fig 10: Total Deformation

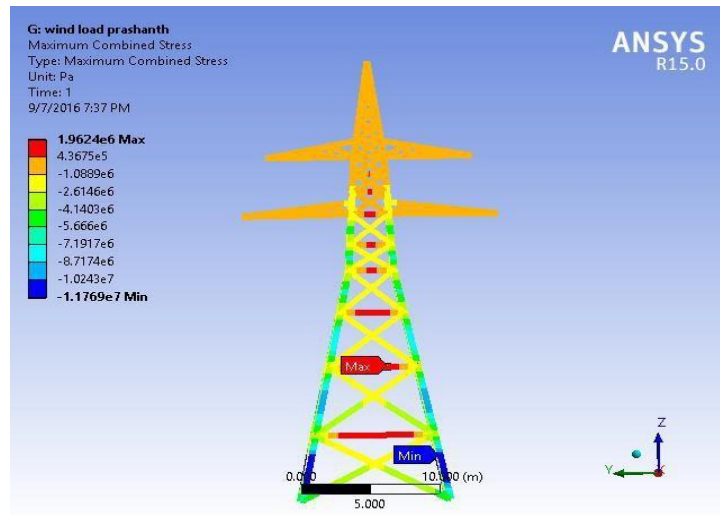
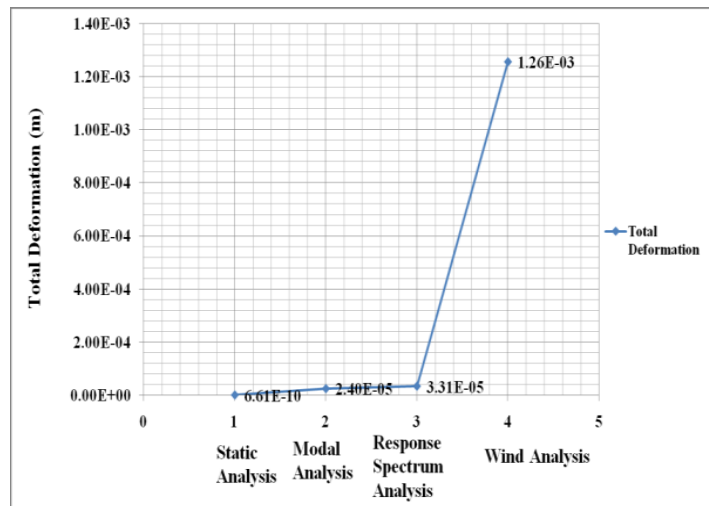


Fig 11: Maximum combined stress



Graph 4 Graphical representation of total deformation for static, modal, response spectrum, wind analysis

RESULT AND DISCUSSIONS

Static Analysis [Load Combination DL+LL]

Static analysis is carried out by modeling the transmission line tower in CATIA using dimensions. The prepared model is imported to ANSYS Workbench and then material properties, loads are assigned. Dead load and live load are acted on the tower and solution

is run. The results are read in post processor

SUMMARY & CONCLUSIONS

The present paper deals with the study of static and dynamic analysis of transmission line tower (X type of bracing system) of 220KV single circuit. The present tower is chosen as case study existing in Bijapur district. An attempt has been made in analysis and modeling of transmission line tower using CATIA and Finite element based ANSYS software. The model is created in CATIA and then

imported to ANSYS Workbench. The link element is used for the modeling. In the present study the behavioral changes due to deflection and stresses of transmission line tower against static and dynamic loadings is carried out. Following are the conclusions drawn from the analysis. 1] The Value of Maximum deformation obtained in the case of static analysis is 6.6067×10^{-10} (m) and maximum combined stress is 1.5408 Pa. 2] In free vibration analysis of existing transmission line tower of X type of bracing system.

a) As the number of mode is increased, the value of natural frequencies gradually increases. For first three modes, the natural frequency remains almost constant and thereafter increases in higher modes. (Refer graph no 1)

b) It is observed that for increasing number of modes, the value of maximum deformation increases in the same line the stresses also increase (Refer graph no 2).

c) As the number of modes is increased, the effect of deformation shifts from top to bottom region of the tower. The natural frequency for fourth mode attains maximum value and thereafter gradually increases for higher number of modes.

3] In Response spectrum analysis for 0.3g, 0.4g, 0.5g ground acceleration

a) The variation of deformation and stresses increases as ground acceleration is increased. (Refer graph no 3).

b) The value of deformation and stresses obtained for different ground accelerations corresponds to the response spectra curves plotted for those ground accelerations.

4] In Dynamic analysis, wind loads are dominating as compared to earthquake forces in zone III. 5] On comparing all the analysis i.e. Static, Modal, Response spectrum, wind. The deformation value is maximum in case of wind analysis. Thus wind load prove to be dominant among all loads for present existing tower. (Refer graph no 4) 6] The analysis carried out using finite element analysis (ANSYS software) gives appropriate solutions including nodal, element, and member solutions.

REFERENCES

[1] Karthik S. Sowjanya G.V [2015] "Static and dynamic analysis of transmission line tower under seismic loads", International Journal of Engineering Research & Technology (IJERT), Vol. 4 Issue 08, pp no 29-33.

[2] Mujamil, Banulatha G. N, Narayan. G, Rajeeva

S.J [2015] "Dynamic analysis of transmission line tower for different sectional properties with

fluid viscous dampers” International Journal of Advanced Research in Education Technology (IJARET), Vol. 2, Issue 2, pp no 134-137.

[3] Sakthivel. T and Sanjeevi R [2015] “Analysis and Design of multi circuit transmission line tower” International Journal of Emerging Technology in Computer Science & Electronics (IJETCSE), Volume 13 Issue 1, pp no 56-61.

[4] Gopi Sudam Punse [2014] “Analysis and design of transmission line tower” International OPEN ACCESS Journal of Modern Engineering Research (IJMER), Vol. 4, Issue.1, pp no 116-138.

[5] Srikanth L, Neelima Satyam D [2014] “Dynamic analysis of transmission line towers” International Journal of Civil, Environmental, Structural, Construction and Architectural Engineering Vol:8, No:4, pp no 434-437.

[6] Pahwa S, Tiwari V, Jatwa H [2014] “Analytical study of transmission tower subjected to wind and seismic loads optimization technique” International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 9, pp no 375-383.

[7] Panchal D. R. [2014] “Various transmission tower analysis and design using professional software” International journal of innovative research & development (IJIRD), Vol 3 Issue 7, pp no 107-112.

[8] McClure G, Lapointe M [2003] “Modeling the structural dynamic response of overhead transmission lines” Computers and Structures 81, pp no 825–834.

[9] Raghavendra T [2012] “Computer aided analysis and structural optimization of transmission line towers” International Journal of Advanced Engineering Technology, (IJAET), Vol.3, Issue 3, pp no 44-50.

[10] Addala G, Satyam D. N and Kumar R. P [2010] “Dynamic Analysis of transmission towers under strong ground motion” Proceedings, 3rd International Earthquake Symposium, Bangladesh, Dhaka.

[11] Sudheer Ch, Rajashekar K, Reddy P. P, Bhargava Gopi Krishna Y [2013] “Analysis And Design Of 220kv Transmission Line Tower In Different Zones I & V With Different Base Widths – A Comparative Study”, International Journal of technology enhancements and emerging engineering research, vol 1, issue 4, pp no 35-43.

[12] Chandran R, Mathew L. A [2016] “Model and Seismic Analysis of Transmission Tower with I, C and Circular Sections Using Finite Element Analysis”, International Journal of Science and Research (IJSR), Volume 5 Issue 7, July 2016, pp no-1297-1300.

[13] Varakavi V. T. R, Arunakanthi E [2016] “Dynamic Analysis of Electrical Transmission Line Towers”, International Journal of Engineering Science and Computing (IJESC), Volume 6 Issue No. 7, pp no 1739-1744.

[14] ANSYS Mechanical APDL Structural Analysis guide & ANSYS Reference Guide.