Static and Dynamic Analysis of Electric Tower Structure by FEM

Shubham Kashyap[*1], Prof. Sumit Pahwa[2]

[*1]*Research Scholar, Department of Civil Engineering. Alpine Institute of Technology, Ujjain, M.P, India.*

[2]*Assistant Professor, Department of Civil Engineering, Alpine Institute of Technology, Ujjain, M.P, India.*

Abstract: **The present Study interacts with the investigation of static and dynamic analysis of Electric tower structure. The analysis and modeling of tower is executed the use of FE based ANSYS software program. The model is created in Solidwork and then imported to ANSYS workbench. The loads acting at the tower considered are dead load, live load and dynamic masses (Seismic and wind). The current tower has height of 40m and X type bracing used in designing of tower structure. Static and dynamic analysis is executed in element using FE based ANSYS software program. Static, modal, reaction spectrum and wind analysis is executed. Wind region taken into consideration is region III. The wind strain relies upon at the gust reaction factor, which increases with height. The conduct of current tower is analyzed for one-of-a-kind analysis. The most deformation, combined stresses, natural frequencies and direct pressure are acquired and plotted graphically.**

Keywords: **Electric tower, FEM, ANSYS, Wind, static analysis, modal analysis.**

I. INTRODUCTION

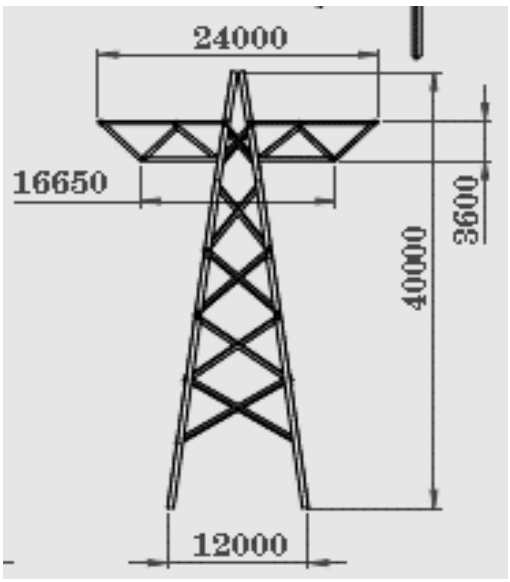
In every country, the requirement of electricity consumption has endured to growth, the rate of demand being greater in the developing nations. The Transmission towers are important for the reason of imparting power to various regions of the nation. This has brought about the increase in the constructing of strength stations and consequent growth in strength transmission traces from the generating stations to the distinct corners in which it's needed. Transmission line need to be stable and punctiliously designed so they do not fail at some point of natural disaster. It should also conform to the country wide and international general. In the planning and layout of a transmission line, a number of requirements ought to be met from each structural and electrical point of view.From the electric factor of view, the most vital requirement is insulation and secure clearances of the electricity carrying conductors from the floor. The pass-phase of conductors, the spacing among conductors, and the vicinity of floor wires with respect to the conductors will decide the design of towers and foundations. The important additives of a transmission line encompass the conductors, ground wires, insulation, towers and foundations. Most of the time transmission traces are designed for wind and ice in the transverse direction. However, the Indian Sub-continent is at risk of mild to excessive earthquakes seismic masses can be crucial due to the fact the transmission line towers and the cables may be subjected to higher pressure and pressured all through floor motion. However, the foremost concern of the transmission line at some stage in excessive earthquakes may be that the massive displacements do no longer reasons the cables to touch every other or any surrounding objects, causing energy failure and accidents. Therefore, earthquake forces may be essential in layout in high earthquake zones of the country. In this undertaking Seismic conduct of transmission line is determined from the dynamics evaluation of the tower and the cable subjected to earthquake floor movement. The protection of these structures is vital in an effort to offer non-stop strength transfer from energy plant life to communities. Due to electric clearances and attachments, transmission line towers regularly have complicated geometries. The towers are a lattice type, consisting of legs, primary, secondary bracings, and cross-arm participants. The structural layout of the tower is ruled by means of wind masses acting at the conductor, tower frame, self-weight of the conductor, tower, and other masses resulting from icing, line deviation, broken twine condition, cascading, erection, maintenance, etc.

II. 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 discredited 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. Solidwork and ANSYS software were used for modeling. Solidwork was used to draw the line model of tower then the line model is exported to ANSYS, then properties and loads were assigned.

III. MODELLING OF ELECTRIC TOWER

The tower was modeled as a steel structure. For the static analysis, the loads considered were dead load and live load. Solidwork and ANSYS software were used for modeling. Solidwork was used to draw the line model of tower then the line model is exported to ANSYS, then properties and loads were assigned.



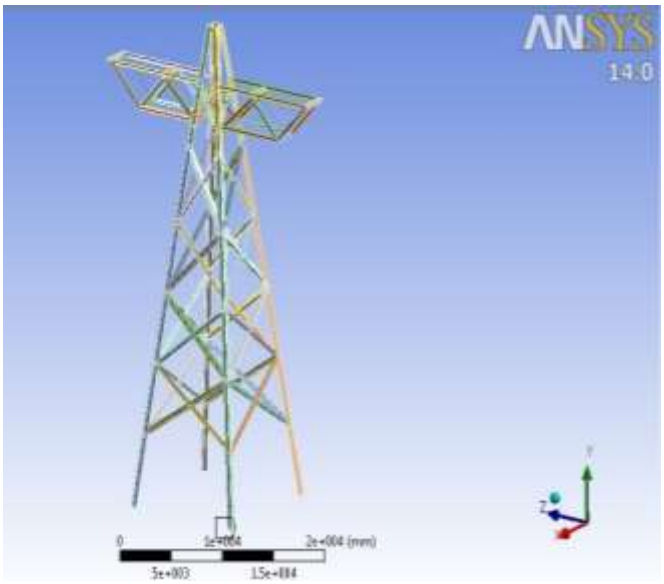


Figure 1:Model of Bridge truss structure

➢ *Input Geometrical Data*

- Height of tower- 40m
- Bottom width of tower -16.650m
- Importance factor - 1.5
- Response Reduction factor of – 3 [Steel frame with concentric braces].
- Damping ratio - 2%
- Zone – Z=0.16.
- Soil Type- Soft soil.

IV. BOUNDARY CONDITIONS

Applying boundary conditions on electric tower structure, Base of structure kept fixed support and Standard earth gravity applying 9.8 m/s on the electric tower structure model. In this structure dead Load and wind load case considered.

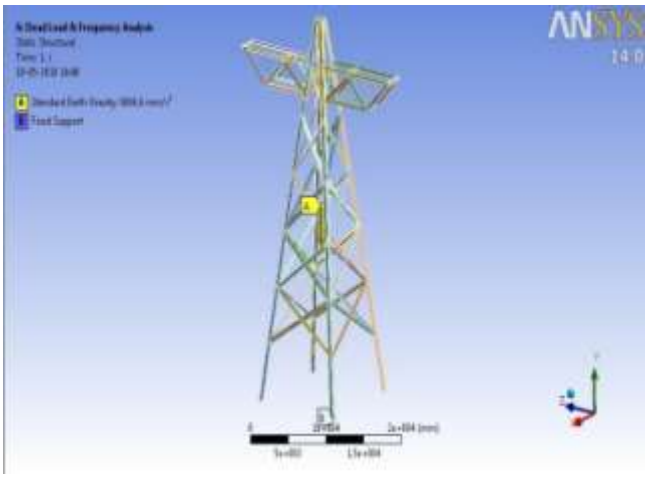


Figure 2: Boundary condition on modal frequency analysis

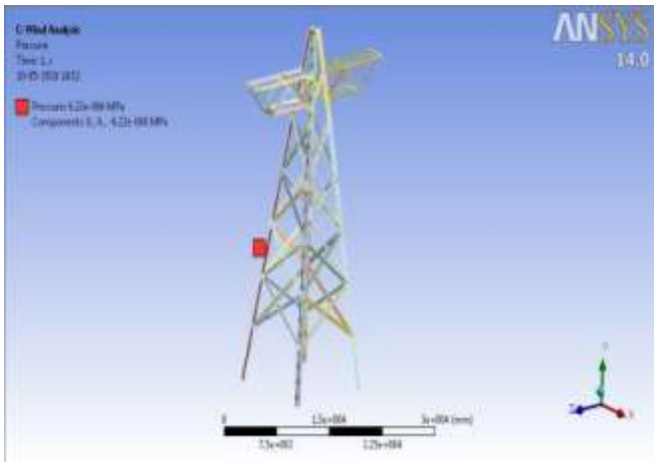


Figure 3: Applying Boundary conditions on Electric tower structure

V. RESULT AND DISCUSSIONS

➢ *Static Analysis [Load Combination DL+WL and self-weight]*

Static analysis is carried out by modeling the transmission line tower in ANSYSusing 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. Directional deformation and equivalent stress is

- *Deformation due to self-weight and dead load in tower structure*

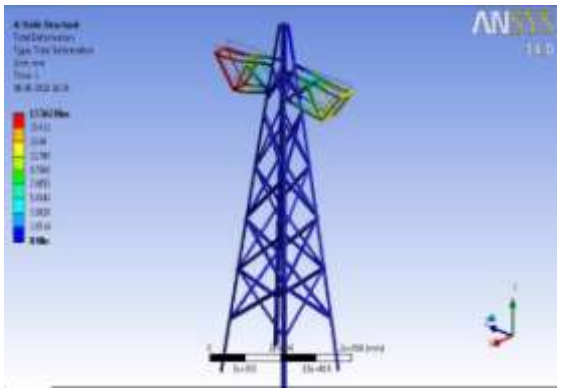


Figure 4: Total deformation

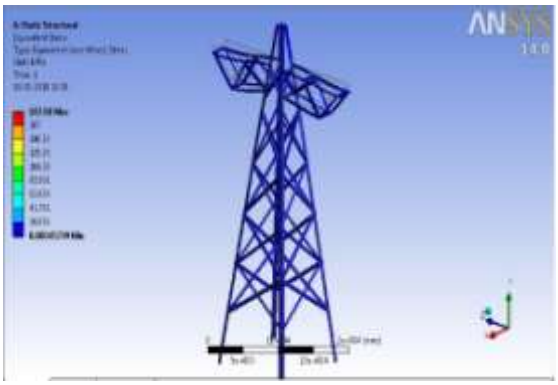


Figure 5: Equivalent stresses

Maximum total deformation found on electric tower structure is 17.56 mm and Equivalent stress found 187.88 MPa due to dead load or self-weight.

- *Deformation due to wind load in tower structure*

The Transmission tower is analyzed statically by consideringthe wind load acting at an instant of time. The result of thestatic analysis is obtained in the form of maximumdeformation, directional deformations, bending moment andshear force. The obtained result is checked for the permissiblelimits. The maximum stresses should be lower than the elasticlimit of the material. There should not be large deflection which may cause extra bending in the transmission tower.

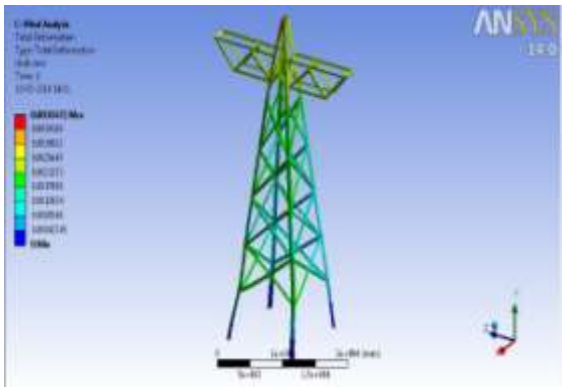


Figure 6: Total deformation due to wind load

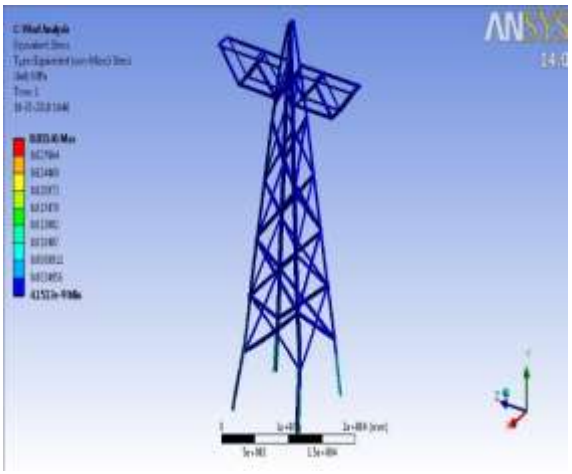


Figure 7: Equivalent stresses due to wind load

- *Modal Analysis of tower structure in Natural frequency modes*

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. 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 Solidwork 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.

Figure shows modes deformations of tower structure due to natural frequency.

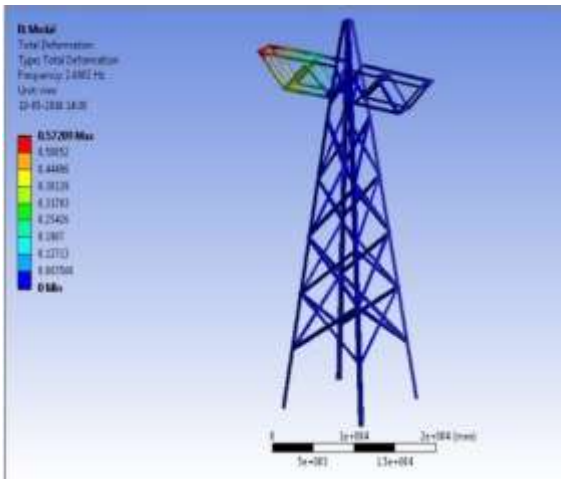


Figure 8: Total Deformation (Mode 1)

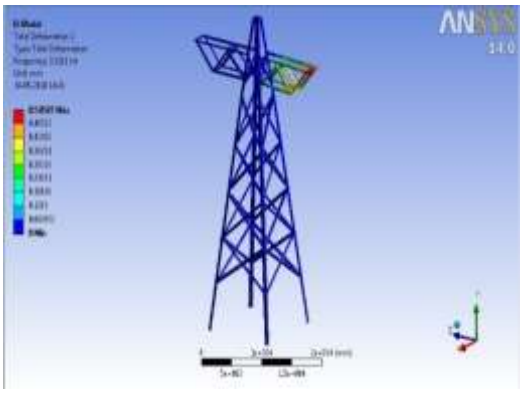


Figure 9: Total Deformation (Mode 2)

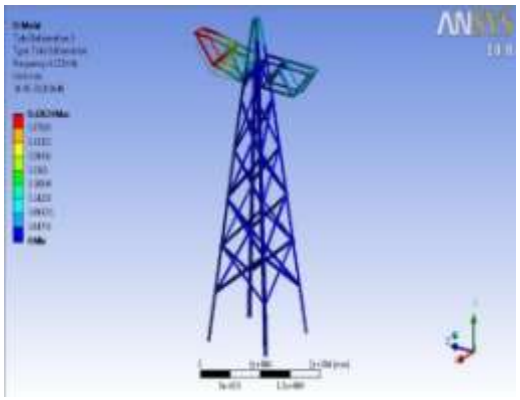


Figure 10: Total Deformation (Mode 3)

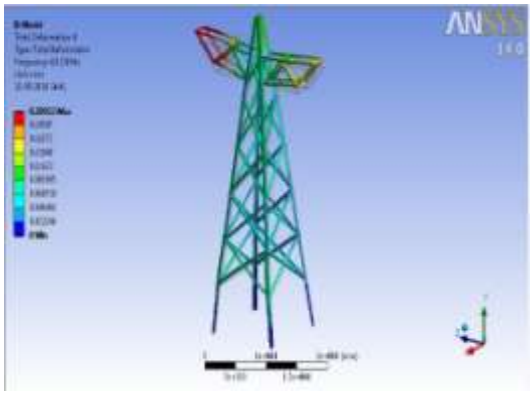


Figure 11: Total Deformation (Mode 4)

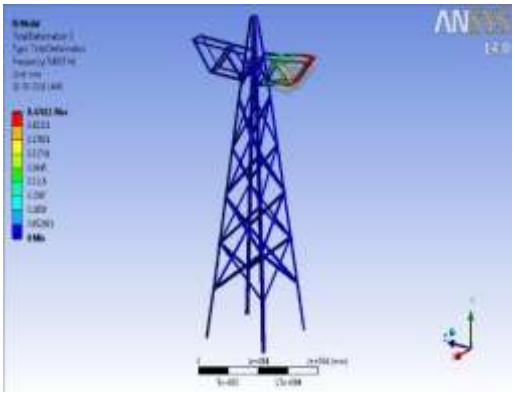


Figure 12: Total Deformation (Mode 5)

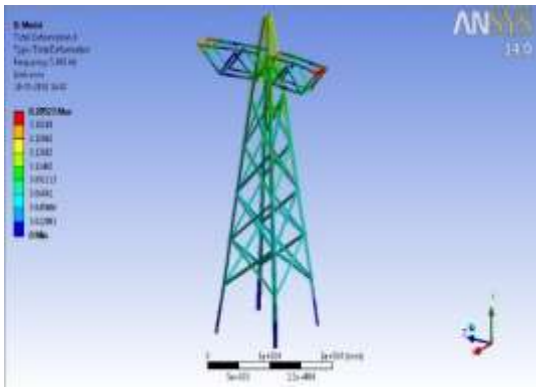


Figure 13: Total Deformation (Mode 6)

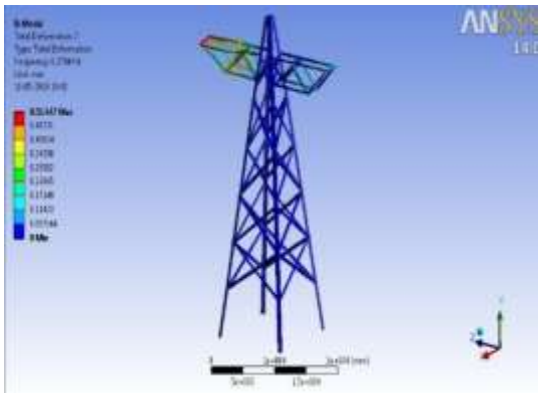


Figure 14: Total Deformation (Mode 7)

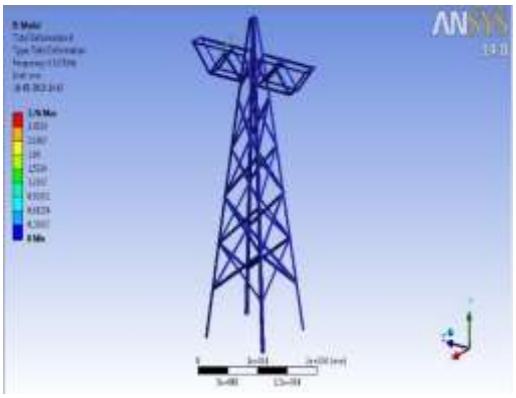


Figure 15: Total Deformation (Mode 8)

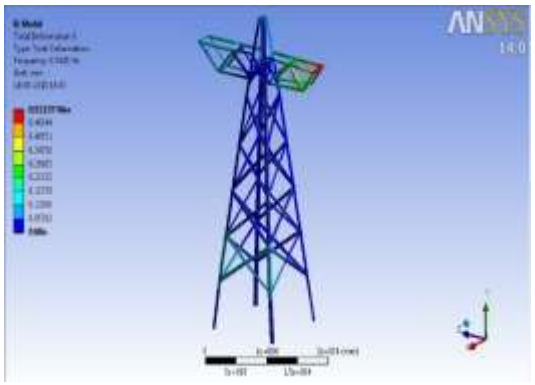


Figure 16: Total Deformation (Mode 9)

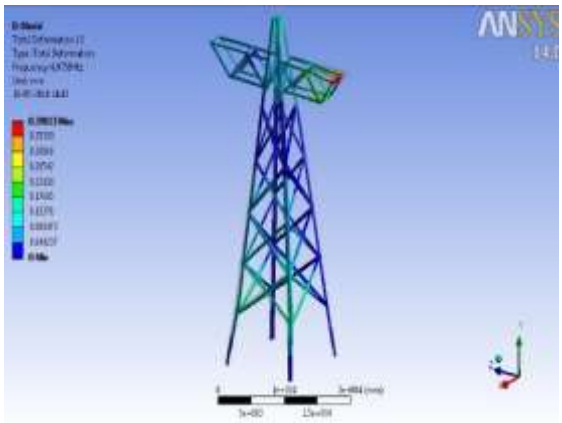
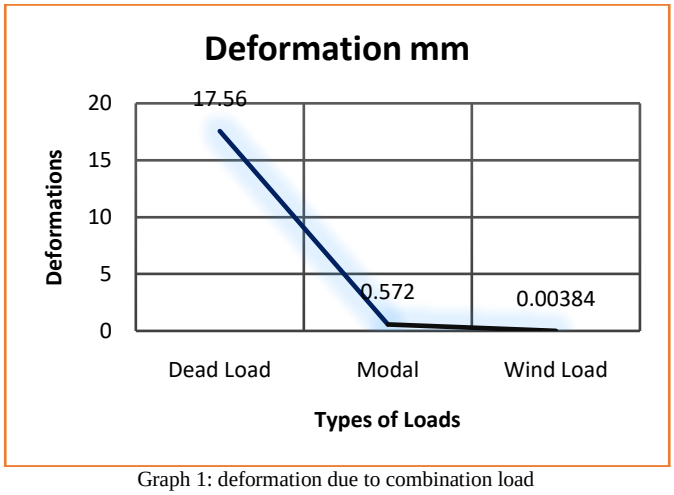


Figure 17: Total Deformation (Mode 10)

Table 1: Shows the deformation due to combination load (deal load+ wind load+ self-weight)

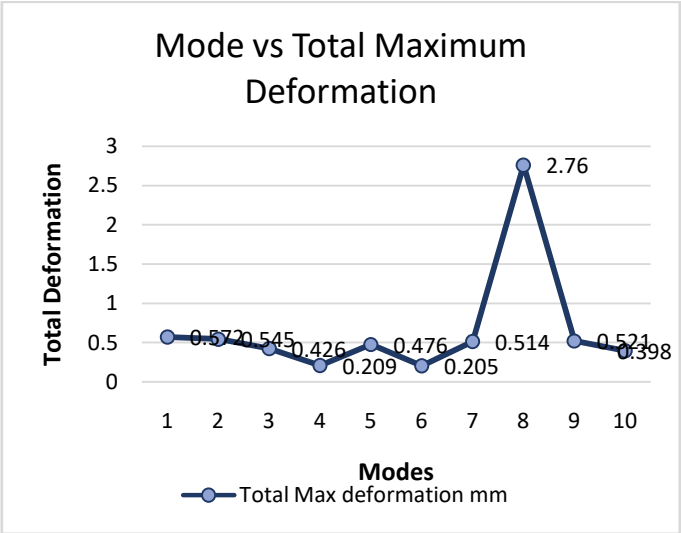
LOADS	Deformation mm
Dead Load	17.56
Modal	0.572
Wind Load	0.00384



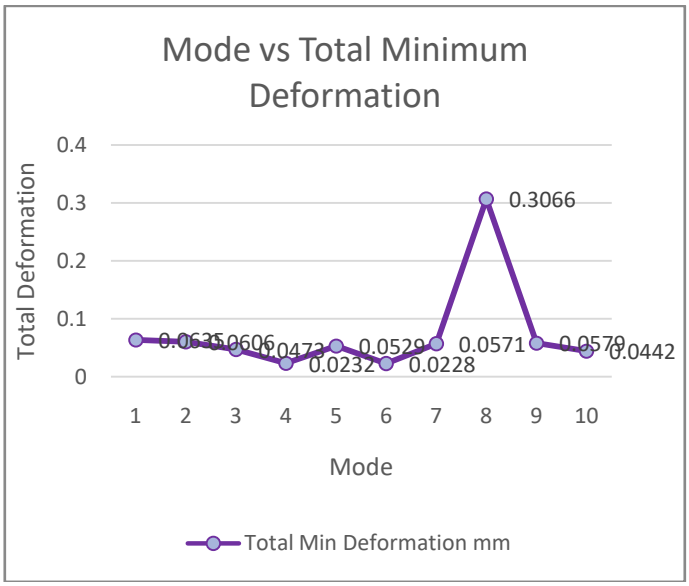
Graph 1: deformation due to combination load

Table 2: Shows the deformation due to natural frequency with various modes

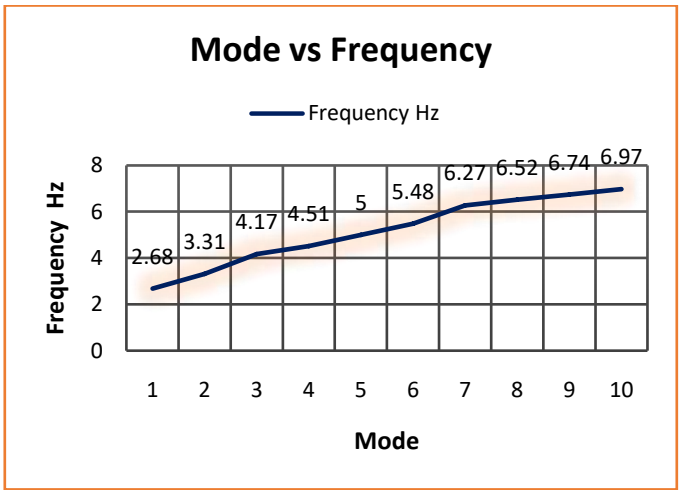
Frequency Hz	Total Max deformation mm	Total Min Deformation mm
2.68	0.572	0.0635
3.31	0.545	0.0606
4.17	0.426	0.0473
4.51	0.209	0.0232
5	0.476	0.0529
5.48	0.205	0.0228
6.27	0.514	0.0571
6.52	2.76	0.3066
6.74	0.521	0.0579
6.97	0.398	0.0442



Graph 2: Maximum deformation of various modes due to Natural frequency



Graph 3: Minimum deformation of various modes due to Natural frequency



Graph 4: Frequency with respect to modes

VI. CONCLUSIONS

The present tower is designed for studying failure due to dead load, wind load or natural frequencies. An attempt has been made in analysis and modeling of transmission line tower using Solidwork and Finite element based ANSYS software. The model is created in Solidwork 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.

- The Value of Maximum deformation obtained in the case of static analysis is 17.56 mm due to dead load and 0.00384 mm obtained due to wind load.
- As the number of mode is increased, the value of natural frequencies gradually increases. For first three modes, the natural frequency almost lower and thereafter increases in higher modes.
- It is observed that for increasing number of modes, the value of maximum deformation increases in the same line the stresses also increase.
- As the number of modes is increased, the effect of deformation shifts from top to bottom region of the tower. The natural frequency for eight mode attains maximum value and thereafter decreases on 9th and 10th mode of frequency.
- In Dynamic analysis, wind loads are dominating as compared to earthquake forces in zone III.
- 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.
- The analysis carried out using finite element analysis (ANSYS software) gives appropriate solutions including nodal, element, and member solutions

REFERENCES

[1]. Falguni Patel, Unnati Bhagat, Dimple Desai, "A Review on Buckling Analysis of Lattice Transmission Tower", International Journal of Advance Engineering and Research Development, Volume 4, Issue 11, 2017, pp.494-498.

[2]. ParashuramHadimani, Sachin Kulkarni, "Static and Dynamic Analysis of Trasmission Line Tower", International Journal of Emerging Research in Management &Technology, Volume-6, Issue-2, 2017.

[3]. J. Vinotha Jenifer, Priya A. Jacob, "A comparative study on analysis of Telecommunication tower with different member cross section", International Journal of Civil Engineering and Technology (IJCIET), Volume 8, Issue 4, April 2017, pp. 1002-1011.

[4]. Harshil Patel, SatyenRamani, "Finite Element Analysis of Monopole Tower for Domestic Wind Turbine", Journal of Emerging Technologies and Innovative Research (JETIR), Volume 4, Issue 04, 2017.

[5]. Emil Veg, MladenRegodić, Aleksandra Joksimovi, "Development of the Transmission Tower Virtual 3D Model for Structural Analysis in ANSYS", FME Transactions, VOL. 45, No 2, 2017.

[6]. ZarinaItam, SalmiaBeddu, Mohd Kamal, "Finite Element Analysis of the Maximum Stress at the Joints of the Transmission Tower", International Conference on Advances in Renewable Energy and Technologies, 32, 2016.

[7]. ShivamPanwar, Yogesh Kaushik, Anubhav Singh, "Structural Analysis and Design of Steel Transmission Tower in Wind Zones II and IV- A Comparative Study", International Journal of Engineering Technology, Management and Applied Sciences, Volume 4, Issue 5, 2016.

[8]. Pasupuleti. Muraharikrishna, G.GaneshNaidu,"dynamic analysis of electrical transmission tower usingFinite element Technique", International Journal of Engineering Research, Vol.4., Issue.5., 2016.

[9]. Raju Varakavi, E.Arunakanthi, "Dynamic Analysis of Electrical Transmission Line Towers", International Journal of Engineering Science and Computing, Volume 6 Issue No. 7, 2016.

[10]. Lin Wang, Athanasios Kolios, Martinez Luengo, "Structural optimisation of wind turbine towers 1 based on finite element analysis and genetic algorithm", Wind Energy Science Discussions, 2016, doi:10.5194/wes-2016-41.

[11]. BoshraEltaly, Amen Saka,KamelKandil, "FE Simulation of Transmission Tower", Hindawi Publishing Corporation Advances in Civil Engineering, 2014, doi:dx.doi.org/10.1155/2014/258148.

[12]. Huang, Li-Jeng, Lin, Yi-Jun, "Free Vibration and Seismic Responses of Power Transmission Tower Using ANSYS and SAP2000", International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 8, August 2014.

[13]. Magalhaes Junior, Rios, T. S. Ferreira, "Structural analysis and structural optimization of self-supporting truss towers to support a large Wind turbine", international journal of optimization in civil engineering, Vol.4, 2014.