International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in

Vol.4., Issue.3., 2016 (May-June)

RESEARCH ARTICLE

ISSN: 2321-7758

DYNAMIC ANALYSIS OF TUBULAR AND LATTICE TYPE WINDMILL TOWER

R.SUSEELA, B.JOSE RAVINDRA RAJ
PRIST UNIVERSITY , THANJAVUR

ABSTRACT

The wind turbine tower is increased very rapidly over recent years for the purpose of electricity..The Dynamic analysis of Lattice and tubular towers has been carried out using software package Ansys. The height of tower is 60m for both lattice and tubular types. The base dimension of lattice tower is 7.5m x 7.5m and top dimension is 1.5m x 1.5m. The tubular tower with base diameter of 3.32m and top diameter of 2.2m with a plate thickness varying along its length were used.

The towers were analysed harmonically and deflections were determined. The maximum lateral displacement of the Lattice tower at the top is 312 mm. The maximum lateral displacement of the Tubular tower at the top is 213 mm.

KEY WORDS: Ansys, wind mill Design, Dynamic

INTRODUCTION

1.1. GENERAL

During the last few decades, the demand for sustainable energy production has led to innovative technological solutions. The forecast of the fuel shortage in the near future combined with the negative environmental impacts caused by the use of the traditional electricity production methods forced all those involved in the energy production field to start exploring new direction in energy production.

1.2. TUBULAR TOWER

Most large wind turbines are delivered with tubular steel towers, which are manufactured in sections of 20-30m length with flanges at either end. The sections are bolted together on the site. The towers are conical i.e., .their diameters increase towards the base , thereby increasing their strength towards the base, where it is needed the most , because this is where the load response owing to the wind loading is largest .

1.3LATTICE TOWER

Lattice towers are manufactured by means of using welded steel profiles or L section steel profiles. Since a lattice tower requires only about half as much material as a freely standing tubular tower with a similar stiffness, the basic advantage of lattice towers is reduced cost.

1.4 DYNAMIC ANALYSIS

Dynamic means time-varying. As such, time is the additional parameter in the dynamic analysis. Dynamic load is that in which its magnitude, direction or position vary with time. Almost all structures are subjected to dynamic load sometime or other during its lifespan.

1.5 FINITE ELEMENT METHOD

Finite Element Method (FEM) is a numerical analysis for obtaining approximate solutions to a wide variety of engineering problems. This has developed simultaneously with the increasing use of highspeed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis.

1.5.1. DESCRIPTION OF FINITE ELEMENT METHOD

In finite element method, a body or a structure is divided into smaller elements of finite dimensions cal or the structure is then considered as an assemblage of these elements connected at a finite number points of points. The properties of the elements are formulated and combined to obtain the solution for the entire body or structure.

1.5.2. ADVANTAGES OF FEM

The Finite Element Method has many advantages of its own. Some of them are given below.

- Various types of boundary conditions are automatically handled in the formulation. They are systematically enforced just before the solution, for the nodal values of the field variables are obtained.
- Material anisotropy and non-homogeneity can be treated without much difficulty.
- Any type of loading can be handled.

1.5.3. DISADVANTAGE OF FEM

The Finite Element Method has its own limitations. They are listed below.

- There are many types of problems where some other method of analysis may prove more efficient than the FEM.
- There are some trouble spots such as longer to smaller directions), which
- may affect the final results.

1.5.4. APPLICATION OF FEM

Applications of Finite Element Method divide into three categories, depending on the nature of the problem to be solved. In this first category are the problems known as equilibrium problems or time dependent problems. The majority of applications of Finite Element Method fall into this category.

1.6. ANSYS

1.6.1 GENERAL

The ANSYS computer program is a general purpose Finite Element Modeling package for numerically solving a variety of mechanical problems. These problems include static and dynamic structural analysis (both linear and non-linear), steady state and transient heat transfer problems, mode-frequency and buckling analyses, acoustic and electromagnetic problems and various types of field and coupled-field applications.

1.6.2. PROGRAM OVERVIEW

The ANSYS element library contains more than sixty elements for static and dynamic analyses, over twenty for heat transfer analyses and numerous magnetic field and special purpose elements. This variety of elements allows the ANSYS program to analyze two and three dimensional frame structures, piping systems, two dimensional plane and axis symmetric solids, flat plates, axis symmetric and three dimensional shells and non-linear problems including contact, interface and cables

Pre Processor: This builds the model.

- Solution Processor: This is for assigning loads, constraints and finally to get Finite Element Solution.
- General Post Processor: This is for further processing and viewing the results over the entire model at specific time points.
- Time History Post Processor: Reviews results at specific points in the model as a function of time.
- Topological optimization:Execute several topological optimization iterations.
- ROM Tool.
- Design Optimization: This improves an initial design.
- Probabilistic Design: This accounts for the inaccuracies and uncertainties influencing the outcome of an analysis by the use of a random input variable.
- Radiation Matrix: This calculates radiation view factors and generates radiation matrix for thermal analysis.
- Run Time Statistics: Predicts CPU Time, Wave front Requirements etc for an analysis.
- Session Editor: Allows the user to modify or save commands issued since the last RESUME or SAVE command.

1.7 SCOPE

In this paper, dynamic analysis and design of for the Lattice type and Tubular type windmill towers are presented. The dynamic analysis of Lattice and tubular towers has been carried out using software package Ansys.

1.8 METHODOLOGY

Both the towers have been modeled using ANSYS.

The natural frequency is calculated manually. The pile foundation for the tower has been designed according to IS 2911 -1979.

REVIEW OF LITERATURE

2.1 DEVELOPMENT AND RESEARCH

The analysis and design of windmill towers have been studied by many researchers and designers. Following are the few which gives information about analysis and design of the towers.

1. Non-linear behaviour of lattice panel of angle towers: Lattice microwave towers and transmission towers are frequently made of angles bolted together directly or through gussets. Such towers are normally analysed to obtain design forces using the linear static methods, assuming the members to be subjected to only axial loads and the deformations to be small.

2. **Static, seismic and stability analyses of a prototype wind turbine steel tower:** Selected results of a study concerning the load bearing capacity and the seismic behavior of a prototype steel tower for a 450 kW wind turbine with a horizontal power transmission axle are presented. The main load bearing structure of the steel tower rises to almost 38 m high and consists of thin-wall cylindrical and conical parts, of varying diameters and wall thicknesses, which are linked together by bolted circular rings.

2.**Analysis and design of the prototype of a steel 1-MW wind turbine tower:** In the present paper, some basic features of the analysis and the design of the prototype of a steel 1 MW wind turbine tower are presented. The structure is 44,075 m high and has a tubular shape with variable cross section and variable thickness of the wall along its height. The steelwork has been manufactured by steel quality S355J2G3.

3.Simple models for natural frequencies and mode shapes of towers supporting utilities: P.J. Murtagh, B. Basu *, B.M. Broderick (2004) Department of Civil, Structural and Environmental Engineering, Trinity College, Dublin 2, Ireland Computers and Structures

The objective of this paper is to demonstrate simple approximate methods of obtaining natural frequencies and mode shapes of towers supporting utilities. The structural system considered is a lattice tower with a mass at the top, the mass representing a utility mounted on the tower.

4.**Structural assessment of current steel design models for transmission and telecommunication towers :** J.G.S. da Silvaa, *, P.C.G. da S. Vellascob, S.A.L. de Andradeb, c, M.I.R. de Oliveirab (2005) A Mechanical Engineering Department, State University of Rio de Janeiro, UERJ, Brazil, structural Engineering Department, State University of Rio de Janeiro, The usual methods of structural analysis involved in the design of steel telecommunication and transmission towers tend to assume a simple truss behaviour where all the steel element connections are considered hinged.

5.**Elastoplastic large deformation analysis of a lattice steel tower structure and comparison with full-scale tests:** Phill-Seung Lee Ghyslaine McClure(2007) Samsung Heavy Industries, Kangnam center building 11th fl., 825-13 Yeoksam, Kangnam, Seoul 135-080.

The objective of this paper is to develop a numerical model for simulating ultimate behavior of lattice steel tower structures. We present the elastoplastic large deformation analysis of a lattice steel tower structure using finite element analysis and we compare the numerical results with full-scale destructive tests.

6.**A peak factor for non-Gaussian response analysis of wind turbine tower:** Luong Van Binh, Takeshi Ishihara, Pham Van Phuc, Yozo Fujino(2008) Department of Civil Engineering, The University of Tokyo, 7-3-1, Hongo, Bunkyo-ku, Tokyo 113-0033, Japan Institute of Engineering, Innovation School of Engineering.

Equivalent static wind load evaluation formulas considering the dynamic effects based on peak factor were proposed to estimate the design wind load on the wind turbine tower in complex terrain.

7.Estimation of the maximum allowable lack of penetration defects in Circumferential butt welds of structural tubular towers: S. Cicero R. Lacalle, R. Ciceroe (2009)Dpto. Ciencia e Ingeniería del Terrenoy delos Materiales, Universidad de Cantabria, Santander, Cantabria, SpainEngineering Structures.

This paper analyses the structural integrity of structural tubular towers (i.e., towers of wind turbines and floodlight towers) with lack of penetration defects on their circumferential butt welds. The methodology presented is particularised to the analysis of the lack of penetration defects detected in certain sections of several wind towers after the construction process.

DYNAMIC ANALYSIS

3. DYNAMIC ANALYSIS

3.1. Harmonic Analysis: Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system. Harmonic response analysis gives you the ability to predict the sustained dynamic behavior of your structures, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

3.1.1 Uses for Harmonic Response Analysis: Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency.

3.2. harmonic analysis of lattice tower: The lattice tower of height 60m and base dimensions of 7.5mx7.5m, top dimensions of 1.5m x 1.5m were modelled in ANSYS package. Using line element the tower was modelled. The material property is defined using link 8elements, Young's modulus, Poisson ratio, link 8 element.

3.2.1. LINK8 Element Description: LINK8 is a spar which may be used in a variety of engineering applications. This element can be used to model trusses, sagging cables, links, springs, etc. The 3-D spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions.

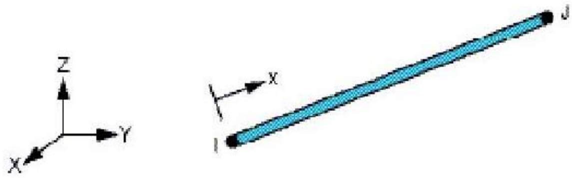


Fig 3.2.1 Link 8 Element

LINK8 Assumptions and Restrictions

The spar element assumes a straight bar, axially loaded at its ends and of uniform properties from end to end.

The length of the spar must be greater than zero, so nodes I and J must not be coincident.

The area must be greater than zero.

The temperature is assumed to vary linearly along the length of the spar

ANSYS Professional.

The DAMP material property is not allowed.

Fluence body loads are not applicable.

The only special features allowed are stress stiffening and large deflections.

Material Property

Young's modulus of steel EX=2x10 5N/mm2

Poisson's ratio for steel µ= 0.3.

Density for steel =7850 kg/m3

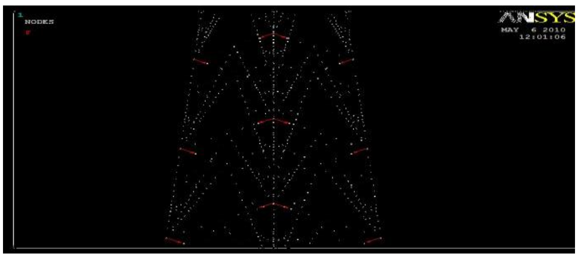
3.2.2. FINITE ELEMENT DISCRETIZATION

The first step in finite element analysis after the creation of the model is meshing. In other words, the model is divided into number of finite elements, and after loading, the deflection patterns of the model are obtained.

3.2.3. LOADING AND BOUNDARY CONDITIONS

The finite element models were loaded at every 4 m interval. Wind load and earthquake loads are applied at the 4 m interval of the model.

The model of the tower is shown in Figure 3.2.3.1. The tower after applying constraints is shown in Figure 3.2.3.2. The meshed tower is shown in Figure



3.2.3.3. The wind loads on the tower are shown in 3.2.3.4. The earthquake loads on the tower are shown in 3.2.3.5.The deformation of meshes of tower is shown in Figure 3.2.3.6. The frequency – response graph obtained from the analysis is shown in figure 3.2.3.7.

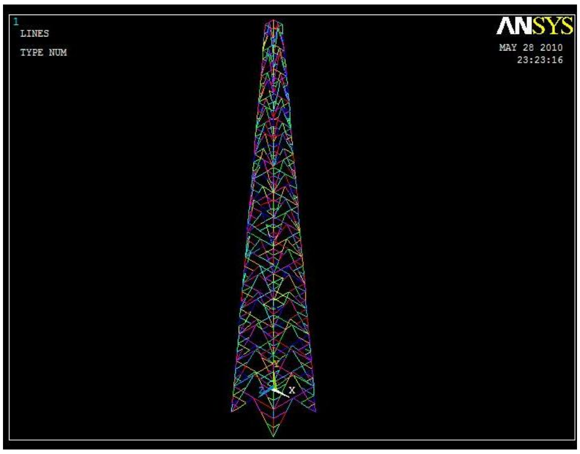


Fig 3.2.3.1 Model of lattice tower

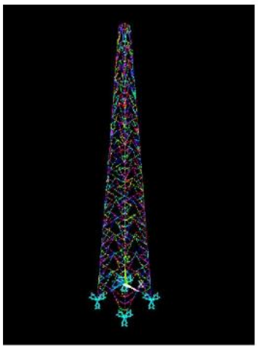


Fig 3.2.3.2 Support condition

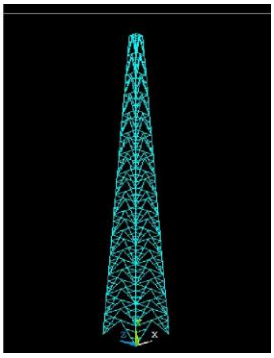


Fig 3.2.3.3 Meshed model

Loads

The loads applied on the tower are rotor load, wind load and earthquake load. The wind load and earthquake loads are calculated according to the relevant codes

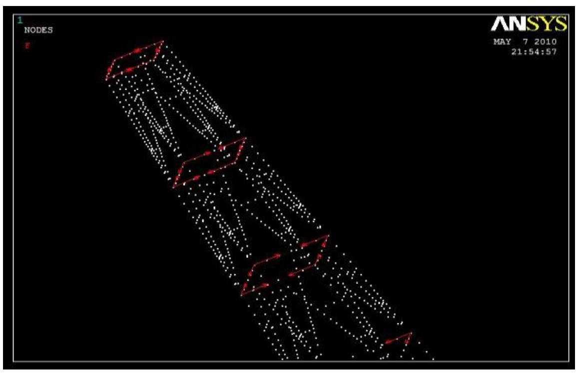


Fig 3.2.3.4 Wind load on lattice tower

Earthquake loads are calculated according to code IS: 1893-2002

Table 3.3.3.1 wind force on tubular tower IS 875 part 3-1987

S.No	Height	Forces
1	0	4.452
2	4	4.019
3	8	2.759
4	12	2.954
5	16	3.159
6	20	3.366
7	24	3.576
8	28	3.788
9	32	4.002
10	36	4.219
11	40	4.437
12	44	4.657
13	48	4.88

After applying the loads, the towers natural frequency of 0.45Hz to 0.9Hz are given as input data of the tower, then it is analysed using the analysis option harmonic analysis, the total maximum deflection of the tower is 312mm

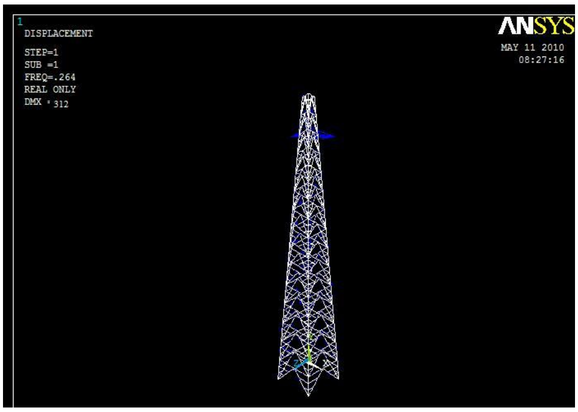


Fig 3.2.3.5 Maximum Deflection of lattice tower

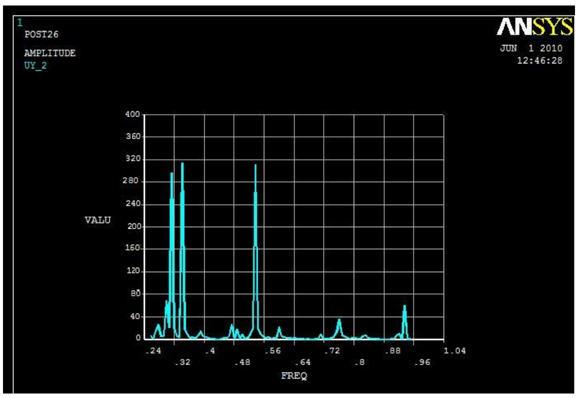


Fig 3.2.3.Frequency-Response graph for lattice tower

3.3. HARMONIC ANALYSIS OF TUBULAR TOWER

The tubular tower of height 60m and base diameter is 3.2m; top diameter is 2.22m were modelled in ANSYS package. Using volume element the tower was modelled. The material property is defined using shell 63 elements, Youn modulus, Poisson ratio, density, thickness are defined for shell 63 elements.

3.3.1. SHELL63 ELEMENT DESCRIPTION

SHELL63

SHELL63 has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses.

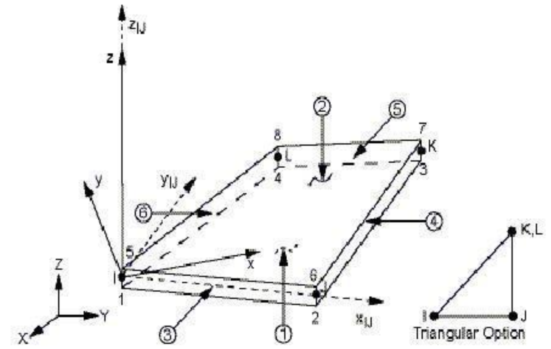


Fig 3.3.1 SHELL63 Geometry

x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL63 Assumptions and Restrictions

Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.

- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness and vary bilinearly over the shell surface.
- An assemblage of flat shell elements can produce a good approximation of a curved shell surface provided that each flat element does not extend over more than a 15° arc.
- A triangular element may be formed by defining duplicate K and L node numbers as described in Triangle, Prism.

SHELL63 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional.

The DAMP material property is not allowed.

The only special features allowed are stress stiffening and large deflection.

Material Property

Young's modulus of steel EX=2x10 5N/mm2

Poisson's ratio for steel µ= 0.3.

Density for steel =7850 kg/m3

3.3.2. FINITE ELEMENT DISCRETIZATION

The first step in finite element analysis after the creation of the model is meshing. In other words, the model is divided into number of finite elements, and after loading, the deflection patterns of the model are obtained. The accuracy of the results is directly proportional to the number of elements chosen.

3.3.3. LOADING AND BOUNDARY CONDITIONS

The finite element models were loaded at every 3m interval. The wind and earthquake loads are applied at the every 3m interval.

Loads

The loads applied on the tower are rotor load, wind load and earthquake load. The wind load and earthquake loads are calculated according to the relevant codes

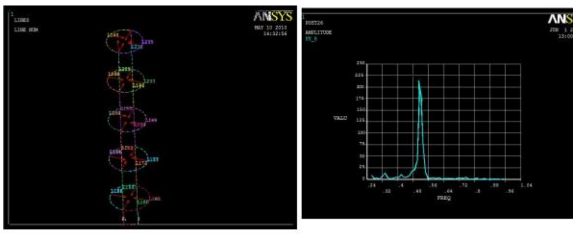


Fig 3.3.3.5 Maximum Deflection of tubular tower

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in
Vol.4., Issue.3., 2016 (May-June)

CALCULATION OF NATURAL FREQUENCY

4. Calculation of Natural Frequency

One of the most useful approximate methods relates the natural frequency of a system to its static deflection under gravity load, δ the natural frequency of a single lumped mass system is

$f_n = 1/2\pi\sqrt{k/m}$

This may be rewritten, replacing the mass term M by the corresponding weight M_g,

$f_n = \sqrt{g}/2\pi k/M$

Since M_g/k is the deflection under gravity load $f_n = \sqrt{g}/2\pi\sqrt{\delta}$

$= 15.\delta 76/\sqrt{\delta}$

δ is measured in mm.

This formula is exact for any single lumped mass system.

For disturbed parameter system a similar correspondence is found, although the numerical factor in eqn 1 varies from case to case, generally between 16 and 20. for practical purposes a value of 18 will give results of sufficient accuracy.

$18/\sqrt{\delta}$> tower natural frequency.

Tower natural frequency range is 0.25 Hz to 0.9 Hz

Lattice

$18/\sqrt{\delta}$> 0.25 Hz to 0.9 Hz δ=312mm

$18/\sqrt{312}$ = 1.01Hz > 0.25 Hz to 0.9 Hz

Tubular

$18/\sqrt{\delta}$> 0.25 Hz to 0.9 Hz δ=213mm

$18/\sqrt{213}$ = 1.23Hz > 0.25 Hz to 0.9 HZ

DESIGN OF PILE FOUNDATON

5.1 . PILE FOUNDATION DESIGN FOR LATTICE TOWER

Self Weight of Tower	= 2103.6kN
Rotor load	= 150kN
Maximum wind load on tower	= 10.25 kN
Total load	= 2236 kN
Factored load	= 2716 kN
Load on one leg	= 679 kN

Assumptions

Cohesion of soil = 30kN/m2

Unit weight of soil = 18kN/m3

Diameter of pile = 500mm

Length of pile = 8m

Angle of shear = 20^o

=1148.13KN

No .of piles required

Q= ncπdl

n= 679x3/30xπx0.5x8

=5.4 say 6 piles

Provide 6 piles for each leg

Spacing between piles = 3d

= 3x500

=1500mm

Group action of piles Wg=ηg (nW)

ηg=1-θ/90x (((n-1) m+ (m-1) n)/mn)

m- no. of rows = 2

n- no. of piles per row = 3

d- diameter of pile = 0.5m

s- spacing of pile = 1500mm

θ=tan-1 d/s

θ=tan -1 (500/1500) =18.43^o

ηg = 1- 18.43/90((3-1)2+(2-1)3))/3x2)

= 0.761

W g= 0.761x6x679.11kN = 3100kN

Check for group action

Qug = 4BLC +B29C B=3s+d

= 3x1500+500=5m

Qug=4x5x8x30/3+52x9x30/3

= 3850kN

This is greater than 679kN.Hence safe

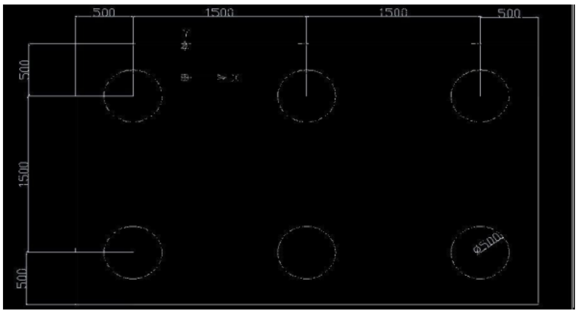


Fig 5.1.1 Arrangement of piles for Lattice tower

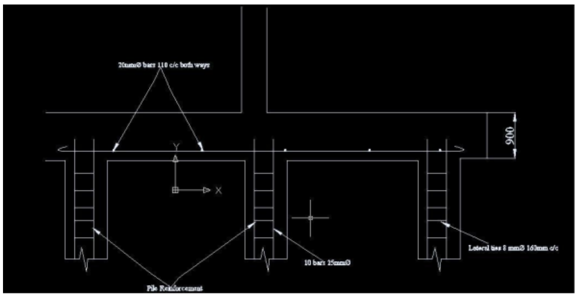


Fig 5.1.2 Reinforcement details for lattice tower foundation

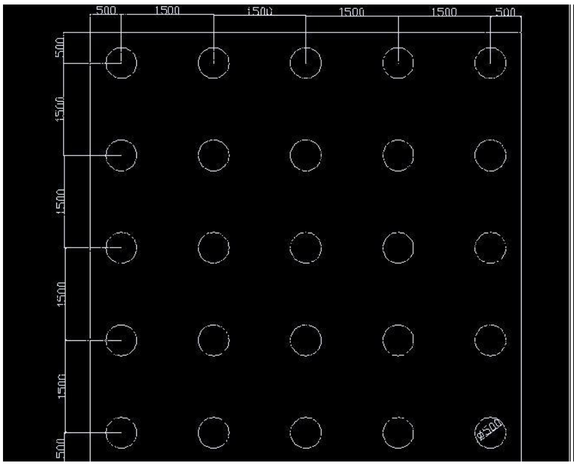


Fig 5.2.1 Arrangement of pile for Tubular tower

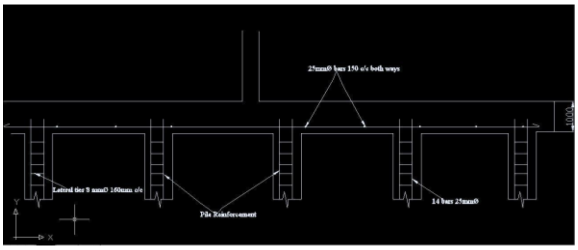


Fig 5.2.2. Reinforcement details of tubular tower foundation

REFERENCES

[1]. N. Prasad Rao , V. Kalyanaraman , (2001) "Non-linear behaviour of lattice panel of angle towers" Journal1337–1357.

[2]. N. Bazeos, G.D. Hatzigeorgiou, I.D. Hondros, H. Karamaneas, D.L. Karabalis, D.E.BeskosStatic,Seismicandstabilityanalyses(2002)ofa " prototype wind turbine steel tower"–1025. Eng

[3]. I. Lavassas, G. Nikolaidis, P. Zervas, E. Efthimiou, I.N. Doudoumis,C.C. Baniotopoulos (2003) "Analysis and design of t steel 1-MW wind turbine tower", Engineering–1106.

[4]. P.J. Murtagh, B. Basu, B.M. Broderick (2004) "Simple models for frequency and mode shapes of towers supporting utilities", Com and Structures 82 .

[5]. J.G.S. da Silvaa, , P.C.G. da S. Vellascob, S.A.L. de Andradeb, , M.I.R. de Oliveirab (2005) "Structural assessment of current steel design models for transmission and telecommunication Constructional Steel Research 61 1108–1134.

[6]. Phill-Seung Leea,_, Ghyslaine McClureb (2007) "Elastoplastic deformation analysis of a lattice steel tower structure and comparison with full-scale tests" Journal of Construction–717.

[7]. Luong Van Binha,_, Takeshi Ishiharab, Pham Van Phuca,Yozo Fujinoa (2008)," A peak factor-Gaussianresponsefor analysisnon ofwindTurbine tower" Journal of Wind Engine96 2217–2227.

[8]. S. Cicero, R. Lacalle, R. Cicero (2009), "Estimation of the maximum allowable lack of penetration defects in circumferential butt welds of structural tubular towers"