SDM COLLEGE OF ENGINEERING & TECHNOLOGY, DHARWAD

DEPARTMENT OF CIVIL ENGINEERING

Sl.No.	Name of the Faculty Member	Publication Count Jan-Dec 2022	Titles of all publication/IPR/Book	Description	Quality
1	Dr.U.D.Hakari	--	--	--	--
2	Prof. K.V. Pramod	3	Structural Engineering Handbook P 38	New Age Publication Delhi, London	Book
			Calculation and Verification of Zpz Values in IS800-2007 P40	International Journal for Research in Applied Science and Engineering Technology	Non-Indexed
			Plastic Moduli for IS Rolled Steel Beam Sections Zpy about Y-Y Axis P41	International Journal for Research in Applied Science and Engineering Technology	Non-Indexed
3	Dr. D.K. Kulkarni	2	Effect of Wind load on tall structures in different terrain category	International Journal of Engineering Research and Technology.	Non-Indexed
			Parametric Study and Optimization of Steel Dome structure P39	International Journal of Engineering Research and Technology.	Non-Indexed
4	Dr. R.J. Fernandes	3	Bending Analysis of Laminated Composite Plates Using Higher Order Theory of 18 Degrees of Freedom Adopting Finite Element Approach P42	International Journal of Scientific & Engineering Research	Non-Indexed
			Dynamic Analysis Of Laminated Composite Plates Using Higher Order Theory Of 18 Degree Of Freedom Adopting Finite Element Approach P43	IOSR Journal of Mechanical and Civil Engineering	Non-Indexed
			Optimization of Shell Structure Using Genetic Algorithm P44	International Journal of Science and Research	Non-Indexed
5	Shri. B.M. Gudadappanavar	3	Efficiency of Neural Networks in the Analysis of Trusses P45	International Journal of Advanced Research in Computer and Communication Engineering	Non-Indexed
			Performance Analysis of Precast and Cast-Insitu Skew Culverts P46.	International Journal of Engineering Research and Technology.	Non-Indexed
			Performance and comparing RCC Beam with Post tension Beam Using Dynamic Load condition of Larger span Structure	International Journal of Engineering Research and Technology.	Non-Indexed

Sl.No.	Name of the Faculty Member	Publication Count Jan-Dec 2022	Titles of all publication/IPR/Book	Description	Quality
6	Mrs. M.B. Mogali	1	Analysis of 2D Structural Elements using MATLAB P47	International Research Journal of Modernization in Engineering Technology & Science.	Non-Indexed
7	Shri..S.A.Kanalli	1	Pavement Design and cost analysis of Mine Waste stabilized low volume Road" P48	National Conference, SVNIT, Surat.	Scopus Index Book Chapter
8	Shri. M.S. Bandiwad	1	Porous Baffle Performance in a Sloshing tank P49	SSRG International Journal of Civil Engineering	Indexed
9	Shri. P.M. Sakare	1	Studying the influence of Constricted Froude's Number on Energy Dissipators in Open Channel Flow P57/56	IOP Conf. Series: Earth and Environmental Science	Scopus
10	Shri Vishwanatha Bhat	1	Modelling and Analysis of Arecanut collecting Machine P50	Current Agriculture Reserach Journal	UGC CARE List
11	Shri..G.S.Majjagi	--	--	--	--
12	Shri.P.I.Chollappanavar	2	Developping and individual Excel sheet for analysis and design of beam and slab P51	International Journal for Research in Applied Science and Engineering Technology	Non-Indexed
			Developping and individual Excel sheet for analysis and design of Colomn & footing P52		Non-Indexed
13	Shri.S..S.Hubballi	--	--	--	--
14	Shri.S.S.Chitnis	4	Studying the influence of Constricted Froude's Number on Energy Dissipators in Open Channel Flow P56/57	IOP Conf. Series: Earth and Environmental Science	Scopus
			Parametric study of multicell box culverts using STAAD pro P54	IOSR Journal of Mechanical and Civil Engineering	Non-Indexed
			Comparison Of Steel Girder And RC Girder As Peripheral Beam P55 To Study The Effect Of Spacing Of Grid Beams And Its Depth On Peripheral Beams In Grid Floor Frame	IOSR Journal of Mechanical and Civil Engineering	Non-Indexed

Sl.No.	Name of the Faculty Member	Publication Count Jan-Dec 2022	Titles of all publication/IPR/Book	Description	Quality
			Analysis, Design And Structural Estimation Of RCC Structure Using ETABS And Microsoft Excel P53	IOSR Journal of Mechanical and Civil Engineering	Non-Indexed
15	Smt Kushal Kapali	2	Stability of Different Shaped Building in Seismic Zone V and II P59	Science, Technology and Deveopment Journal	Non-Indexed
			Study and Comparison of Conventional Analysis P58 with Construction Sequence Analysis Using ETABS	International Journal of Engineering Research & Technology	Non-Indexed
16	Smt. Pooja Hukkeri	--	--	--	--
17	Smt. Shruti Kambalimath	--	--	--	--
18	Shri. Akhilesh Menasi	--	--	--	--

IQAC Coordinator

HOD, Civil Engg.

Dean (Academic Program)

Principal

Flagged – 015.

05 – INDEXED.

01 – BOOK

18 – NON INDEXED.

24 – TOTAL

SHRI DHARMASTHALA MANJUNATHESHWARA COLLEGE OF ENGINEERING & TECHNOLOGY ★ IQAC DAHARWAD-580 002 KARNATAKA 19/1/23

Effect of Windload on Tall Structures in Different Terrain Category

Nagraj S Airani
PG Student
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Dr. D. K Kulkarni
Professor,
Dept. of Civil Engineering,
SDMCET, Dharwad, India

***Abstract*— Wind is a perceptible natural movement of air relative to earth surface mainly within the shape of air cutting-edge blowing in a specific route. The major dangerous factor which problem to civil engineering systems is that, it will load any and every item that comes in its way. Wind blows with much less speed in rough terrain and better speed in smooth terrain. This paper gives Storey Displacement, Storey Drift, Storey Shear occurring in special storey due to wind in special terrain category. Four models are analysed using of ETABS 2019 package. Present works gives an excellent source of facts about variation in Storey Displacement, Storey Drift, Storey Shear in different terrain categories.**

Keywords*: *Wind Load, ETABS, Terrain, Storey displacement, Storey Shear, Storey Drift.

I. INTRODUCTION

No obvious definition for tall building is there. From the features of the building, it is quite difficult to term it as "Tall Building". Number of floors or height are not the factors for a building to be said as tall. A structure which is bounded by floors, roof top, walls & commonly windows can be named as tall or short building depending on many factors. "A tall building" is a multi-storey arrangement in which maximum of the occupier's rest on elevators to fulfil their purposes.

Maximum noticeable structures are known as high rise buildings in many nations. On the basis of operational approach, it is quite easier to account a structure as high when their plan and structural examination are somewhat exaggerated by the crosswise loads. This can be explained by an example such as sway which it experiences due to wind load or earthquake load which all are horizontal loads. As elevation rises, the wind forces start to govern. Hence, basic outline for high rise buildings is established about ideas related totally to the resistance experienced by turbulent wind. High rise structures, which are commonly planned for workplace or marketable use, are one of the best well-known descriptions in the history of Indian urban development in 12th era.

There are four distinct wind classifications for tall constructions that can be built

- Terrain category 1
- Terrain category 2
- Terrain category 3
- Terrain category 4

Category 1: Exposed open area where there are few or no obstacles and when the average height of any nearby objects is less than 1.5 meters.

Terrain Category 01

Category 2: Open terrain with well scattered obstruction having heights generally between 1.5 to 10m.

Terrain Category 02

Category 3: Terrain with numerous, widely spread obstacles that are the size of buildings and can reach heights of 10 meters, with or without a few lone, tall obstructions.

Terrain Category 03

Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 08, August-2022

Parametric Study and Optimization of Steel Dome Structure

Praveen Belligatti
Student
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Dr. D. K. Kulkarni
Professor,
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Abstract— **Domes are space constructions that offer an affordable way to cover a lot of valuable, column-free space. The utilization of domes can be seen in places like sports arenas, assembly halls, exposition centers, malls, industrial complexes, etc. This feature offers efficiency in the use of building materials and attractive appearance. The dome is used as a space truss in this article, with all joints being pinned to provide a torsion- and moment-free structure. As a result, only tensile and compressive forces are applied to all members. Even though only axial forces considered when designing a dome. So an effort to create software in Octave has been made. The program can be used to configure, analyze, design, and optimize the weight of a steel dome. The effects of the dead load and live load have been taken into account when designing the members, which are made of tubular steel sections.**

To establish the relationship between weight variation and dome height, a parametric analysis was also conducted. For the domes with bracings in one or both directions, this analysis is expanded. It is also done a parametric analysis for the optimization of the dome using various methods, such as with discrete and continuous variables. Weight variation according to the number of plan segments has also been done.

Keywords:- Dome, Optimization, Octave, Bracings, etc..

I. INTRODUCTION

Architects and engineers are constantly looking for new ways to solve the problem of space enclosure. Because of their diversity and flexibility, In their search for novel forms, architects and engineers might use space structures as a useful tool.. A three-dimensional collection of components known as a space structure is capable of withstanding loads that can be applied from any location. The proliferation of the space frame in recent decades has been primarily due to its great structural potential and visual beauty. Sports stadiums, exhibition pavilions, assembly halls, transit hubs, aviation hangars, workshops, and warehouses are just a few of the building types that utilize space frames. They have been utilized for mid- and short-span enclosures in addition to long-span roofs as roofs, floors, exterior walls, and canopies. A well-known illustration of space is a dome.

Domes have stimulated people's interest because they enclose the most space with the least amount of surface area. This feature saves money by reducing the consumption of building materials. Despite the fact that stone was the only structural material used in ancient times, brickwork gradually replaced stone masonry. Timber was used for the same purpose in Middle Ages. However, significant advancements in dome structures began with the development of the steel industry in the nineteenth century. This allowed the engineers to design large spanned and multi-story steel structures. Steel is now widely used to enclose large spans of length.

Because it depends on an engineer's skill to iteratively consider several feasible shapes and sizes for each component of a certain construction using a computer, structural design has historically been a less-than-ideal procedure. This is just a fast variation of the same computations performed by engineers of the last century, but not a real improvement to the design process. During the last few years, a lot of work has been done to fully automate the design of structures. Nonetheless, most of the new methods developed share a common flaw: they are based on linear programming techniques, and thus treat structural optimization as if the search space is continuous, when it is actually discrete due to the limited number of structural shapes available on the market. These issues can be resolved using mathematical programming techniques for optimizing steel dome structures. Because total weight of structure is easily accessed by economy, an attempt is made here to reduce total weight of structure.
The utilization of a constrained issue search technique via mathematical programming is the main topic of this thesis. With minimal modification, the created code can be applied to a wide range of steel truss dome construction configurations.

II. OBJECTIVES

- To develop software of computer programs in Octave programming language that will generate configuration and loading data, analyze the structural response, design and optimize the weight of domes. Only dead load and live load are considered in the present study. Design of members is done as per IS: 800-2007 code.
- Weight Optimization is done through the process non linear constrained optimization method.
- Parametric study for weight variation of dome considering various parameters like height, number of sectors in plan and different variable approaches.

III. METHODOLOGY

The Octave programming language was utilized to optimize the steel dome. The data is generated, the structure is analyzed and designed using a set of programs developed, and optimization is performed using the same programming language and the precise penalty approach.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue XII Dec 2022- Available at www.ijraset.com

Calculation and Verification of Z_{PZ} Values in IS 800:2007

K.V.Pramod[1], Dr. B. R. Patagundi[2]
[1]*Prof. Civil Engg. Dept., SDMCET, Dharwad, Karnataka, India – VTU, Belagavi, Karnataka*
[2]*Principal, Prof. in Civil Engg. Dept.,, SGBIT, Belagavi, Karnataka, India - VTU, Belagavi, Karnataka*

Abstract: ***Plastic Section Modulus of a section is the first moment of the cross sectional area about an axis which divides the sectional area into two equal halves. In this paper an attempt has been made to calculate and verify the Z_{pz} values of tapered flange I and C Sections. I-Section has been divided into a total of 13 component areas and C-Section into 7 component areas. The area of each component is calculated and the position of centroid of each component area is identified and used in the calculation of the Plastic Section Modulus of the cross section about Z-Z axis. Eleven I-sections and One C-section Z_{pz} values are found to differ from the values given in the IS 800 : 2007[2] by more than 100 mm^3, i.e., 0.1 cm^3 and are reported here. These verifications gain significance due to the fact that, in case of four I-sections, the difference is found to be more than 10000 mm^3, i.e., 10cm^3.***

Keywords: I-Sections, C-Sections, Z_{pz}, IS 800 : 2007[2], Axis of symmetry.

I. INTRODUCTION

IS 800:2007[2] gives the values of Plastic Sectional Modulii, Z_{pz}, of I and C-Sections about z-z axis, in Annexure H, pages 138 to 140. The values given are in cm^3. The values of Z_{pz} are used in the determination of Moment of Resistance of Plastic and Compact sections as per IS 800: 2007[2] and hence play an important role in the design of beams by Limit State Method. For Semi-Compact sections Elastic Section Modulus Z_e is to be used as per the Code, which is the ratio of Moment of Inertia of the section, I, and the distance of extreme fibre from the neutral axis.

Plastic Section Modulus of a section is the first moment of the cross sectional area about an axis which divides the sectional area into two equal halves. It is usually called 'Equal Area Axis'. For symmetrical sections the identification of such axis is very simple – the axes of symmetry represent the Equal Area Axis. But in case of unsymmetrical sections it is necessary to first identify the Equal Area Axis, before proceeding to the calculations of Plastic Section Modulus. In unsymmetrical sections Neutral Axis (Centroidal Axis) and Equal Area Axis do not coincide.

In the present case the I-section has two axes
of symmetry and the Z-Z axis is located at half the depth of the section, while the Y-Y axis is located through the centerline of the web. In case of C-section, the Z-Z axis is the only axis of symmetry and is located at half the depth of the section, while the Equal Area Axis in 'Y' direction is to be worked out considering the geometry of the web, flange, fillet and rounded end of flange. For the calculation of Z_{pz}, the Z-Z axis is only necessary to be identified.

The Plastic Sectional Modulii, Z_{pz}, values for I and C sections have been calculated with higher accuracy, in mm^3, and presented here. A few values are found to differ from the values given in the IS 800:2007[2] and have been reported here.

II. METHOD OF CALCULATION

Typical calculations of Z_{pz} for one I-Section – ISLB 400 @ 558.2 N/m and one C-Section ISMC 300 @ 351.2 N/m have been given hereunder. I-Section has been divided into a total of 13 component areas and C-Section into 7 component areas. The area of each component is calculated and the position of centroid of each component area is identified and used in the calculation of the Plastic Section Modulus of the cross section about Z-Z axis.

A. Plastic Section Modulus, Z_{pz}, of ISLB 400 @ 558.2 N/m:

For ISLB 400@ 558.2 N/m the various geometrical parameters, as per SP: 6(1)-1964,[1] are as follows:

$h = 400$ mm ; $b = 165$ mm ; $t_f = 12.5$ mm ; $t_w = 8$ mm ; $(D)\theta = 98^\circ$; $r_1 = 16$ mm ; $r_2 = 8$ mm ;

In the Figure 1 :

Z-Z represents the horizontal neutral axis

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue XII Dec 2022- Available at www.ijraset.com

Plastic Section Moduli for I.S. Rolled Steel Beam Sections Z_{py} about Y-Y Axis

K.V. Pramod[1], Dr. B. R. Patagundi[2]
[1]*Prof. Civil Engg. Dept., SDMCET, Dharwad, Karnataka, India – VTU, Belagavi, Karnataka*
[2]*Principal, Prof. in Civil Engg. Dept.,, SGBIT, Belagavi, Karnataka, India - VTU, Belagavi, Karnataka*

Abstract: ***There are many situations in which the I-Sections used in construction are subjected to moments about their weaker axis, i.e. the y-y axis. For such purposes the Plastic Section Modulus about y-y axis becomes necessary. In the present paper an attempt has been made to calculate and present the values of Z_{py}, for I.S. Rolled Steel Beam Sections (with tapered flanges). Since IS 800 : 2007[2] has not given the values of Z_{py}, of any section, one tapered flange I-Section, viz., 125 TFB @ 13.1 kg/m, from "Onesteel" (Australia),[4] has been used to ascertain the correctness of calculations. The results are presented in table form in descending order.***
keywords: I-Sections, Z_{py}, IS 800: 2007[2], y-y axis.

I. INTRODUCTION

I.S. Rolled Steel Beam and Channel Sections are used as Beams, Columns, components of Built-up Beams/Columns, Members of Lattice Girder Bridges, Gantry Girders, Crane Girders, etc. There are many situations in which the I-Sections used in construction are subjected to moments about their weaker axis, i.e. the y-y axis, like eccentric loads on columns, members acting as beam-columns, etc. Hence, the knowledge of Plastic Moment of Resistance becomes necessary – especially when site conditions demand the use of smaller sections. Also, the strength of any section about the main axes, both z-z and y-y axes, is of academic interest. For such purposes the Plastic Section Modulus about y-y axis becomes necessary. Leading steel manufacturers and distributors in the world e.g. British Steel, Fletcher Easy Steel (New Zealand), Onesteel (Australia)[4], Nippon steel (Japan) etc., publish the values of Z_{py}, Plastic Section Modulii about y-y axis, in their brochures alongside the values of Z_{pz}, i.e., Plastic Section Modulii about z-z axis. In the present paper an attempt has been made to calculate and present the values of Z_{py}, for I.S. Rolled Steel Beam Sections (with tapered flanges).

II. METHOD OF CALCULATION

Typical calculations of Z_{py} for the I-Section – ISLB 400 @ 558.2 N/m have been given hereunder. The I-Section has been divided into 13 component areas. The area of each component is calculated and the position of centroid of each component is identified and used in the calculation of the Plastic Section Modulus of the cross section about Y-Y axis. Further, the same procedure is applied for calculation of Z_{py} of one tapered flange I-Section, viz., 125 TFB @ 13.1 kg/m, from Onesteel (Australia),[4] to ascertain the correctness of calculations.

A. Plastic Section Modulus, Z_{py}, of ISLB 400 @ 558.2 N/m:

For ISLB 400@ 558.2 N/m the various geometrical parameters, as per SP: 6(1)-1964,[1] are as follows:
$h = 400$ mm ; $b = 165$ mm ; $t_f = 12.5$ mm ; $t_w = 8$ mm ; $(D)\theta = 98^\circ$; $r_1 = 16$ mm ; $r_2 = 8$ mm ;
In the Figure 1 :
Z-Z represents the horizontal neutral axis
Z'-Z' represents the horizontal Equal Area Axis – Z-Z and Z'-Z' axes coincide.
Y-Y represents the vertical neutral axis
Y'-Y' represents the vertical Equal Area Axis – Y-Y and Y'-Y' axes coincide.

1) Calculations Of Areas, Centroids And Plastic Section Modulus

Refering to Figure 2 –
Entire Web is taken as a Rectangle of -- $(h \times t_w) = 400 \times 8.0 =$ **3200.0000 mm^2**

Bending Analysis of Laminated Composite Plates Using Higher Order Theory Of 18 Degree Of Freedom Adopting Finite Element Approach

R J Fernandes, Megha H Koppad

Abstract— Laminated composite plates(LCP) are extensively used to solve special problems in engineering applications so bending, dynamic and stability behaviors are important to the designers. The paper aims at bending analysis of these plates with higher order theory. The application of higher-order theory that accounts for the realistic variation of in-plane and transverse displacements through the thickness for the static response analysis of thick multi-layered composite plates shall be studied. Code is developed using MATLAB with finite element formulation for 18 degrees of freedom with good agreement.

Index Terms— Laminated composite plates, bending analysis, transverse displacement, higher order theory, finite element, degrees of freedom, MATLAB.

1 Introduction

Composite plates are made by joining same or different material plates together in layers and laminated to fulfill the required properties. Laminated composite plates have unique properties than when compared to its constituent materials such as high stiffness to weight ratio, high strength to weight ratio, low maintenance, high corrosion resistance, durable, low specific weight, high specific strength and stiffness properties. Deformation of laminated plates is defined by coupling between bending and shear deformation.

2 Method

2.1 FEM

Finite element method is a mathematical method used to determine boundary value problems. In FEM the major element is divided into smaller ones called finite elements followed by solving them and then combining those to one major original initial problem. The division of an element is made by creating a mesh by joining certain number of nodes with each other in which that mesh constitutes the whole element. This project involves finite elements of mesh created by using nine nodes.

2.2 Displacement model

Based on the assumptions of displacement model, a higher order shear deformation theory (HSDT) is developed to analyze the stresses. The displacement model with EIGHTEEN degrees of freedom is in the following form:

The displacement model for unsymmetrical laminates,

$u(x,y,z)=u_o(x,y)+z\Theta_x(x,y)+z^2u_o^*(x,y)+z^3\Theta_x^*(x,y)+z^4u_o^{**}(x,y)+z^5\Theta_x^{**}(x,y)$

$v(x,y,z)=v_o(x,y)+z\Theta_y(x,y)+z^2v_o^*(x,y)+z^3\Theta_y^*(x,y)+z^4v_o^{**}(x,y)+z^5\Theta_y^{**}(x,y)$

$w(x,y,z)=w_o(x,y)+z\Theta_z(x,y)+z^2w_o^*(x,y)+z^3\Theta_z^*(x,y)+z^4w_o^{**}(x,y)$

Stress-Strain Relationship

$$\begin{Bmatrix}\epsilon_1\\ \epsilon_2\\ \epsilon_3\\ \gamma_{12}\\ \gamma_{23}\\ \gamma_{13}\end{Bmatrix}=\begin{bmatrix}1/E_1 & -\gamma_{21}/E_2 & \gamma_{31}/E_2 & 0 & 0 & 0\\ 1/E_{22} & -\gamma_{32}/E_3 & -\gamma_{12}/E_2 & 0 & 0 & 0\\ 1/E_3 & -\gamma_{13}/E_1 & -\gamma_{23}/E_2 & 0 & 0 & 0\\ 0 & 0 & 0 & 1/G_{12} & 0 & 0\\ 0 & 0 & 0 & 0 & 1/G_{23} & 0\\ 0 & 0 & 0 & 0 & 0 & 1/G_{13}\end{bmatrix}\begin{Bmatrix}\sigma_1\\ \sigma_2\\ \sigma_3\\ \tau_{12}\\ \tau_{23}\\ \tau_{13}\end{Bmatrix}$$

The strain corresponding to displacement model can be written as,

$$\epsilon_x = \epsilon_{xo}+ zk_x + z^2\epsilon_{xo}^* + z^3k_x^* + z^4\epsilon_{xo}^{**} + z^5k_x^{**}$$

$$\epsilon_y = \epsilon_{yo}+ zk_y + z^2\epsilon_{yo}^* + z^3k_y^* + z^4\epsilon_{yo}^{**} + z^5k_y^{**}$$

$$\epsilon_z = \epsilon_{zo}+ zk_z^* + z^2\epsilon_{zo}^* + z^3k_z^{**} + z^4\epsilon_{zo}^{**}$$

$$\gamma_{xy} = \epsilon_{xyo}+ zk_{xy} + z^2\epsilon_{xyo}^* + z^3k_{xy}^* + z^4\epsilon_{xyo}^{**} + z^5k_{xy}^{**}$$

$$\gamma_{yz} = \phi_y+ zk_{yz} + z^2\phi_y^* + z^3k_{yz}^* + z^4\phi_y^{**} + z^5k_{yz}^{**}$$

$$\gamma_{xz} = \phi_x+ zk_{xz} + z^2\phi_x^* + z^3k_{xz}^* + z^4\phi_x^{**} + z^5k_{xz}^{**}$$

Shape function for nine node –element

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 19, Issue 4 Ser. II (Jul. – Aug. 2022), PP 15-21
www.iosrjournals.org

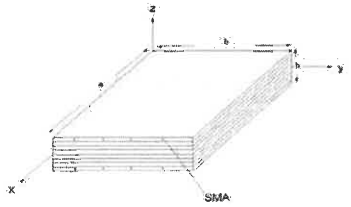
Dynamic Analysis Of Laminated Composite Plates Using Higher Order Theory Of 18 Degree Of Freedom Adopting Finite Element Approach

Reginald Fernandes[1], Priyanka Kambali[2],
[1](Shri Dharmasthala College of Engineering and Technology/ VTU, INDIA)
[2](Shri Dharmasthala College of Engineering and Technology/ VTU, INDIA)

Abstract: *Laminated composite plates (LCP) are extensively used to solve special problems in engineering applications so static, dynamic and stability behaviors are important to the designers. The project aims at dynamic analysis of these plates with higher order shear deformation theory (HSDT18). The application of higher-order theory that accounts for the realistic variation of in-plane and transverse displacements through the thickness for the dynamic response analysis of thick multi-layered composite plates also shall be studied. Finite element formulation for 18 degree of freedom which was be extended to examine the static responses of the laminated composite plates.*

Date of Submission: 21-07-2022 Date of Acceptance: 05-08-2022

Background: A plate is a thin and flat structural element. The word "thin" suggests that the plate is transverse. When compared to its other two dimensions, the laminated composite plate's thickness is quite modest. i.e. the dimensions of its length and width.



Where,
h- Thickness of the plate,
a/b - Length or width dimension.
Composite plates have been widely used in a variety of fields, including structural, marine & aerospace engineering, due to their high strength to weight ratio and corrosion resistance.

Method: Mathematical formulation to obtain the equations of equilibrium. Governing differential equations by using HSDT18. Stress-strain relationship by Simple Deformation Theory. Formulation of the solution of above shear differential equation using finite element method. Finite element formulation and writing codes for above formulations to obtain frequency of laminated composite plate. To validate obtained results with available literature.

Results: For comparison, the dimensionless natural frequencies of Laminated Composite Plate (LCP) with simply supported are taken into account. To verify the effectiveness and accuracy of the current work, the results are compared with those from a number of other previously published works. The frequency is obtained for the problem one and problem two for 2 layers (0/90),3 layer (0/90/0) and 4 layers (0/90/90/0) for different a/h ratio which was carried out.

I. Introduction

With creation of higher-order terms in Taylor's expansions of the displacement in the thickness coordinate, this theory overcame the limitation of First Order Shear Deformation Theory. Laminated composite plates are analysed using HSDT for redirection, stresses, natural frequencies, and buckling loads. With this

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

Optimization of Shell Structure Using Genetic Algorithm

Dr. R. J. Fernandes[1], Sushmita Yadahalli[2]

[1]Assistant Professor SDM college of Engineering and Technology, Dharwad, Karnataka, India

[2]PG Scholars, SDM College of Engineering and Technology, Dharwad, Karnataka, India

Abstract: ***Shell structures are used efficiently and also economically in many fields of architecture and engineering. Shell roofs are made from structural 'skins' where the shell material is thin in section relative to the other dimensions of the roof and undergoes relatively little deformation under load. They are commonly used where a building interior needs to be free from intermediate walls or columns that might support a more conventional flat or pitched roof. Shells provide an aesthetic view. In this paper, optimization is carried out for different grades of concrete keeping thickness, chord length, and semi-central angle variable, whereas in a shell roof with an edge beam width of the beam is constant and the depth of the beam varies. A computer program is written in MATLAB software for the optimization process using the genetic algorithm method, and analysis results are retrieved from SAP2000 software to carry out the optimization process.***

Keywords: shell structure, genetic algorithm, shell roof, edge beam, concrete

1. Introduction

Large span constructions with a single storey, such as assembly halls, amusement centers, theatres, factories, research labs, etc., are covered with shell roofs. Because shell roofs are aesthetically pleasant, they are frequently utilized in public buildings. The shell roof is far more practical than a flat or even a pitched roof when the interior of the structure must be open with no walls or pillars. The roofs made of shells are lighter and require less concrete and steel for reinforcement.

The two forms of shell roofs are single-curved shell and doubly-curved shell. Single-curved shells must have a minimum thickness of 5 cm. Structure-wise; double curved shells are superior to single curved shells.

In shell roof construction, the roof's slope and arc's curvature must be properly planned in order to keep the building stable. As the entire roof structure must be constructed at one level, shell roofs are not suited for situations where the roof must be at various levels. Lighting and ventilation setups are challenging to establish in shell roof construction.

The design of a shell roof is challenging, expensive, and requires knowledgeable personnel. A shell roof can be built without support beams and is quite thin. Materials for these kinds of roofs are less expensive than those needed for flat or pitched roofs.

2. Modeling and Analysis of Shell Structure

The analysis of shell structure is carried out by using CSI SAP2000 software.

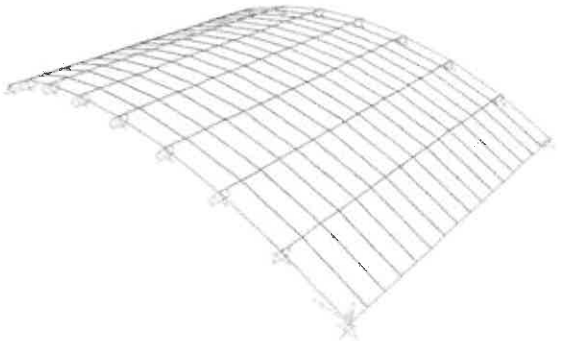


Figure 1: Shell structure model

Data used in the analysis is as follows

Chord length – 8m to 12m

Span – 8m

Thickness – 75mm to 200mm

Semi central angle (T) – 30^0 to 45^0

$$\text{Radius} = \frac{\text{Chord length}}{2*\sin T}$$

Width of beam - 230mm

Depth of beam - 230mm to 450mm

3. Results

In this chapter we study on process of optimization of shell structure of M30 grade of concrete is carried out and keeping length 8m and the following inputs are listed below and results obtained for the same are as follows.

a) Objective function = minimization of weight for the shell structure
b) No of variables 4 =X1, X2, X3,X4
c) No of constraints

IJARCCE ISSN (O) 2278-1021, ISSN (P) 2319-5940

International Journal of Advanced Research in Computer and Communication Engineering

ISO 3297:2007 Certified ⁜ Impact Factor 7.39 ⁜ Vol. 11, Issue 7, July 2022

DOI: 10.17148/IJARCCE.2022.11730

EFFICIENCY OF NEURAL NETWORK IN THE ANALYSIS OF TRUSSES

Deepika Sambrani[1], Basavaraj.G[2], Dr. R. J. Fernandes[3]

Student, Department of Civil Engineering, SDMCET, Dharwar-580008, India[1]

Assistant Professor, Department of Civil Engineering and Engineering, SDMCET, Dharwar-580008, India[2]

Professor, Department of Civil Engineering and Engineering, SDMCET, Dharwar-580008, India[3]

Abstract: Neural networks are simply known as the biological nervous system. An Artificial Network (ANN) is an information processing system that is inspired by the way biological Nervous System, such as the brain, process information. The key element of ANN is the novel structure of the information processing system. It is composed of a large number of highly interconnected processing elements (neurons) working in unison to solve specific problems. ANNs, like people learn by example. They can be trained with a known example of a problem. Once trained, the network can be put to use in solving unknown and untrained problems. An ANN is configured for a specific application , such as pattern recognition or data classification, through a learning process. Learning in biological system involves adjustment to the synaptic connection that exist between the neurons. This examines the efficiency of neural networks. Taking into consideration type of ANNs such as Generalized Regression (GR) neural network, Radial Basis Function (RBF) neural networks, Linear Layer (LL) neural network efficiency of ANNs is checked in the design of trusses. The neural networking tool available in MATLAB is used. To train ANNs, various input and output data are provided using an analysis and design package STAAD PRO. The ANNs are trained with some values and are tested for both interpolation and extrapolation Then percentage error is calculated in all three ANN. Based on percentage error, the efficiency of each ANNs is compared in the design of trusses. The study is made by increasing the number of training, by increasing the number of input and output variables, by training in the matrix form, etc. From these results the suitability of each ANN is studied and conclusions are drawn.

Keywords: Neural networks, Truss, General Regression Neural Network, Generalized Regression

I. INTRODUCTION

ANNs are great at solving algorithmic and math problems, but often the world can't easily be defined with a mathematical algorithm. Facial recognition and language processing are an example of problems that can't easily be expressed into an algorithm, however, these tasks are of importance to humans. The key to Artificial Neural Networks is that their design allows them to process information in a similar way to our own biological brains, by drawing the idea from how our own nervous system functions. This makes them useful for solving problems like facial recognition, which our biological brains can do easily. Dr. Robert Hecht-Nielsen was the inventor of one of the first neuro-computers. ANN is defined as a computing system made up of a number of simple, highly interconnected processing elements, which process information by their dynamic state repose to external inputs. Neural networks are typically arranged in layers. Layers are made up of a number of interconnected 'nodes'. These interconnected nodes contain an 'activation function'. Patterns are sent to the network through the input layer, which Communicates to one more 'hidden layer' here the actual processing is done through a System of weighted connections. The hidden layers are then connected to an 'output Layer' where he answer is output.ANNs are used in the optimization of structures and in the field of concrete technology.

Optimization of structures is the analysis and design procedure and it requires a lot of time. These ANNs save time in concrete technology, it is difficult to conduct several sets of experiments for finding the best admixtures and concrete with different strengths. ANN model can be developed and best admixture and concrete strength can be known and verified experimentally.Character recognition has become very important as handheld devices like the palm pilot are becoming increasingly popular. Neural networks can be used to identify handwritten characters. Neural networks can receive and process large amounts of information at once, making them useful in image compression. With the Internet explosion and more sites using more images on sites, using neural networks for image compression is worth. Business of the stock market is extremely complicated. Many factors weigh in whether a given stock will go up or down on any given day. Since neural networks can examine a lot of information quickly and sort it all out, they can be used to predict stock price.

Performance Analysis of Precast and Cast-Insitu Skew Culverts

Laxmi
PG Student
Dept. of Civil Engineering,
SDMCET, India

Basavraj Gudadappanavar
Asst. Professor,
Dept. of Civil Engineering,
SDMCET, India

Abstract— **In past, limited study has been carried out on Precast culverts. In this study the behaviour of precast culverts with hinge joints and cast-in situ culverts (fixed joint) are compared using MIDAS CIVIL software, parametric studies is carried out on 36 models considering different skew angles varying from 0° to 45° with 15° interval, span varying from 3m to 6m and hinge position at 1/3rd from bottom at 2/3rd from bottom are considered to arrive at most optimum hinge position. Analysis is carried out using IRC:6 2016, Models are compared for bending moments and shear forces. The results obtained are given in form of graphs**

Keywords *— Box culvert; precast culverts; skew; midas civil*

I. INTRODUCTION

Culverts are the tunnel like structures which allow water to flow under a road, railway or any structures. Culverts are designated as important medium of transport of economical value. These are largely used in roadways, railways, sewage conduits, monsoon runoff, pathways of telephone and electrical lines and stock or pedestrian underpass. A culvert is enclosed in soil and it is exposed to earth and vehicular loads. Culverts can either have singular cell or multiple cells. And it can be classified by the construction type as cast-in-place culverts or precast culverts. Advanced technology indicates that the precast concrete culverts are in use last few decades.

A. Precast culverts

Precast concrete box culverts are the most versatile and cost-effective solution for quick-build projects. Cost savings are realised as a result of design flexibility and ease of installation on-site. The applications for pre-cast concrete culverts are limitless. They can be used for service tunnels, underpasses, bridges, stream culverts, cattle passes, and other similar purposes. A Precast Concrete Box Culvert (PCBC) is a simple structure that allows for the passage of roads, pathways, and flowing water (e.g., streams, storm water, or drains) beneath roads, railways, or embankments. Many countries, including India, have used precast concrete box culverts. With innovative and mechanised box culvert production facilities, over 40 - 60 m of culvert section can be produced per day.

B. Shapes of precast culverts

Precast concrete box culvert are manufactured and transported in required shape and size. Bigger box culverts where transporting is difficult as a one unit, so they are constructed in from two 'U' sections and connected on-site. Even they are manufactured as two 'L' shapes and in between 'T' shapes are also been used. These can be provided with rebated joints/V notched to allow sections to be laid open or sealed. They can even be manufactured with precast wing walls and head walls.

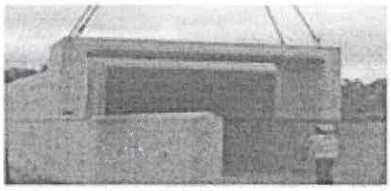


Figure No. 01 – Culvert with Hinge Joint at Mid of Wall

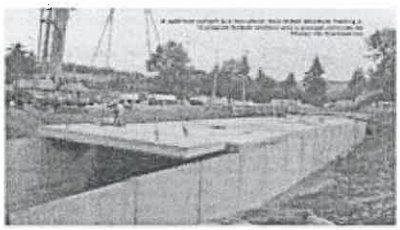


Figure No. 02 – Culvert with Hinge Joint at Top

II. OBJECTIVES

- Study of structural behavior of Precast and cast in situ Box culverts with varying degree of skew using Midas civil software.
- Parametric studies of precast and cast-insitu culverts using different span and skew.
- Analysis of Precast culverts with different position of hinges and optimum position of hinge will be studied.

Performance and Comparing RCC beam with Post Tension beam using Dynamic Load Condition of Larger Span Structure

Vijay K More
PG Student
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Basavaraj Gudadappanavar
Asst. Professor,
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Abstract:- **In ordinary Reinforced cement concrete beam, compressive stresses are taken up by concrete and tensile stresses by steel alone. The concrete below the neutral axis is ignored since it is weak in tension. Although steel takes up the tensile stresses, the concrete in the tensile zone develops minute cracks. The load carrying capacity of such concrete sections can be increased if steel and concrete both are stressed before the applications of external loads. This is the concept of prestressed concrete. Internal prestressing can be done by two methods Pre-tensioning and Post-tensioning. In Pre-tensioning system, the tendons are first tensioned between rigid anchor blocks cast on ground or in a column or unit-mould type pre-tensioning bed, prior to the casting of concrete in the moulds. In Post-tensioning, the concrete units are first cast by incorporating ducts or grooves to house the tendons. When the concrete attains sufficient strength, the high-tensile wires are tensioned by means of jack bearing on the end face of the member and anchored by wedges or nuts. The space between the tendons and the duct is generally grouted after the tensioning operation. Referring particularly to post tensioning applications, it is generally recognized how it opens the possibility to improve economy, structural behaviour and aesthetic aspects in concrete solutions. As in modern days post tensioning has been most economical method when compared to the RCC works. The study is subjected to evaluation of performance of RCC deep beam and PT beam slab with multi-storey building system with seismic loading performance using analysis tool ETABS.**

Keywords ***RCC Beam, ETABS, PT Beam, Storey displacement, Storey Shear, Storey Drift.***

I. INTRODUCTION

RCC Structures are commonly utilized for residential and industrial buildings in Asian countries. For small span buildings, PT beams are rarely used. There was a huge disadvantage of expert staff for Pre-Stressing job two decades ago. However, there are currently a significant number of agencies for the execution of a comparable work. Due to deflection limitations in RCC Beams, the depth of the beam increases as the span increases. The depth of the beam is reduced in pre-stressed sections, therefore pre-stressed beams are less expensive for long spans.

PSC is the most recent main type of structural engineering construction introduced. Because the technology is currently available on the market in both developed and developing countries, it has become a well-established construction technique. Today, prestressing is employed in buildings, subterranean structures, communication towers, floating storage and offshore structures, power plants, apparatus boats, and a variety of bridge systems.

The primary style goals for structural engineers are safety, practicability, economy, and current lawfulness of style. Engineers and designers must comprehend the proper use of posttensioned concrete as well as the effects that will result when choosing a structural construction system. If properly evaluated and constructed, concrete buildings composed of high-quality components will offer a superior combination of durability, sound management, and fireplace safety in today's construction market. Given the current market aspects of value options, material availability, and lower floor-to-floor heights, as well as market developer finance, Concrete is generally regarded as a more cost-effective material than steel. Concrete that generates internal stresses of sufficient magnitude and distribution to significantly offset the stresses caused by a given external force.

II. OBJECTIVES

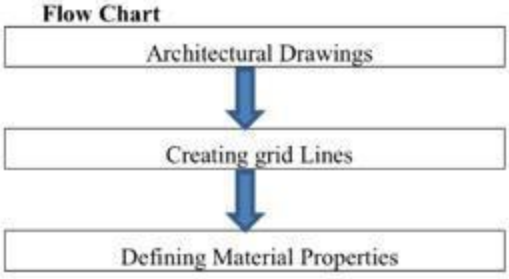
- To analyze the dynamic performance of RCC beam and PT beam of multistorey building system.
- To compare the results of Base Shear, Storey Displacement, Storey drift of RCC beam with PT beam of the multistorey building system.

III. METHODOLOGY

This chapter describes the standard step-by-step method for modelling the two different regular structure with RCC Beam and PT Beam

- Model Type 1- RCC Structure
- Model Type 2- Post-Tension Structure

Flow Chart



e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:04/Issue:08/August-2022 Impact Factor- 6.752 www.irjmets.com

ANALYSIS OF 2D STRUCTURAL ELEMETS USING MATLAB

Soumya Meti*1, Mamta Mogali*2

*1Student, Civil Engineering, SDMCET, Dharwad, Karnataka, India.

*2Assistant Professor, Civil Engineering, SDMCET, Dharwad, Karnataka, India.

ABSTRACT

In the subject of civil engineering, analysis of structural components is usually an importantstage. There are a variety of techniques for analysing structural components, including the limit state approach and the finite element method, as well as software for doing so, like Staad pro, SAP, etc. In this project, 2D structural elements are taken into account, and MATLAB software is used to analyse the structural elements, as there is a gap that was discovered while conducting the literature review: more emphasis has been given to space structures, and there is a lack of information available in regards to 2D structures. To analyse structural elements including beams, trusses, and frames, MATLAB software was used to write the project program.

Keywords: Structural Analysis, MATLAB, Stiffness Matrix, Nodal Loads, Beam, TrussAnd Frame.

I. INTRODUCTION

1. GENERAL

A fundamental and essential component of every structural engineering project, structural analysis is the process of estimating how a particular structure will perform under specific loading conditions. Shears, Axial forces, bending moments, support responses and deflections are the performance parameters that typically prompt questions concerning structural design. The calculation of these parameters as a result of the specified loads and outside influences is typically a part of a structure's examination. This study focuses on the analysis of 2D structural elements, especially beam, truss, and frame. A beam is an extended, straight construction. that is loaded and maintained in such a way that all external forces acting on the beam are parallel to the centroidal axis and exist on the cross-sectional symmetry plane. Beams can be broadly categorized based on their size, how they are supported, and where they are located within a specific structural system. They are Cantilever Beam, Overhanging Beam, Continuous Beam, Fixed Beam, and Simply Supported Beam. A truss is a structure made up of members joined at the ends by pin joints and solely susceptible to load and response at the joints. The truss is divided into two categories: plane truss and space truss. A truss isclassified as being a plane truss when all of its members and external loads are located in the same plane.The truss members and the loading, however, all employ the three-coordinate system in space trusses. Frame structures are those that combine a beam, a column, and a slab to withstand lateral and gravitationalloads. The large moments that emerge from the imposed loading are often overcome by these structures.At each node, there are three degrees of freedom in a frame structure. Since there are three times as manynodes as degrees of freedom, this is the case.

In this project analysis of beam, truss and frame is performed using Finite element method (FEM) of approach and MATLAB program has been developed for all the three structures and results are obtained.When the task involves analysis of frame for applied loadings, resolving it analytically becomes tedious and unreliable, especially when the frame structure is subject to many degrees of freedom. This program's development was motivated by the need for an effective, accurate, and quick approach to examine such frames. Finite element analysis is one of the best suited, most popular approaches for solving a frame with such a high degree of freedom.

2. FEM APPROACH

Instead of solving challenging partial differential equations, the finite element method is a tool for approximative analysis of one-dimensional, two-dimensional, and three-dimensional structures. The main advantage is that it quickly solves each design step's optimization and validation problems and derives the structure into manageable chunks. The intricacy of the solution is caused by the finite element method's usage of matrix calculations to solve such small elements.

3. DIRECT STIFFESS METHOD

The classical slope deflection method, also known as displacement or the classical method, is the ancestor of the

SARDAR VALLABHBHAI NATIONAL INSTITUTE OF TECHNOLOGY SURAT, GUJARAT

DEPARTMENT OF CIVIL ENGINEERING

CERTIFICATE OF PRESENTATION

This is to certify that

Shravan A Kanalli

has presented paper entitled

"Pavement Design and Cost Analysis of Mine Waste Stabilized Low Volume Roads"

co-authored by ***Dr. Sureka Naagesh and Dr. Ganesh K.*** *in the*

4th National Conference on ***Recent Advances in Traffic Engineering-2022*** *held at Sardar Vallabhbhai National Institute of Technology (Surat) during 11th - 12th November 2022.*

Dr. Shriniwas S. Arkatkar
Chairman

Dr. Ashish Dhamaniya
Chairman

Prof. Gaurang J. Joshi
Head of Department

12.11.22

48

SSRG International Journal of Civil Engineering
ISSN: 2348-8352 / https://doi.org/10.14445/23488352/IJCE-V9I12P101

Volume 9 Issue 12, 1-6, December, 2022

Original Article

Porous Baffle Performance in a Sloshing Tank

M. S. Bhandiwad[1*], B. M. Dodamani[2]

[1,2] *Department of Water Resources and Ocean Engineering, NITK Surathkal, Karnataka, India.*

Received: 13 October 2022 Revised: 19 Novemer 2022 Accepted: 02 December 2022 Published: 15 December 2022

Abstract *- The sloshing phenomenon is the participation of liquid in the partially filled vessels or tanks, which is a primary consideration for designing and constructing many structures in offshore, offshore, and space engineering. Employing porous baffles of varying porosities is an effective way to minimize or/and control wave elevations in the sloshing tank. In the present study, analytical investigations were carried out for the sloshing motion in the clean tank (tank without baffles) and the tank with baffles. The two porous baffles of three different porosities consider in the rectangular sloshing tank. The porous baffles place at equal intervals in the sloshing tank. The analytical simulations were carried out under a range of swaying motions. During simulation, the sway excitation frequencies cover the clean tank's first five resonant sloshing modes. The two porous baffles of lower porosity levels in the sloshing tank are beneficial in suppressing the wave elevations in the sloshing tank near the first resonant mode.*

Keywords *- Sloshing motion, Porous baffle, Analytical investigation, Rectangular tank, Resonant frequency.*

1. Introduction

Earthquakes, wind, and wave action are among the most destructive natural disasters, causing significant loss of life and livelihood. The load characteristics and the structure itself mainly govern the response of a structure during these actions. The land, ocean, and space structures are often susceptible to seismic, wave, and wind-induced vibrations due to low levels of inherent damping. The use of Dynamic Vibration Absorbers (DVAs) to reduce excessive vibrations caused by various types of dynamic actions has grown in the last few decades. A DVA improves damping by dissipating some energy generated when a structure moves under both regular and irregular excitations. As a result, the displacement of primary structures coupled with the liquid tank (DVAs) is reduced, and the structures become more stable. Various systems propose to mitigate the structural vibration against loads. Different categories of protection systems have been implemented in the last few decades based on structural conditions, configuration, and applications. The sloshing tank was found to be more effective.

Water storage tanks are liquid sloshing tanks for land structures and buildings, so-called tuned liquid dampers (TLD). The TLDs are a type of passive mechanical damper and are designed as a sloshing tank. These partially filled rigid tanks are located in the structure at or near the location of maximum modal displacement for suppressing structural vibrations. For ocean structures, William Froude proposed in 1862 that a tank partially filled with water may minimize the roll motions of cargo if the frequency and phase of the oscillating water were appropriately tuned. The idea of employing liquid-filled tanks to damp motions first emerged in the late 1800s. In the 1960s, this concept was implemented to reduce the wobbling movement of satellites in space. Sloshing dynamics in the tank provide an effective and easy technique to enhance the system's damping and rely on liquid sloshing inside the tank to dissipate energy from the viscous action and wave breaking. The sloshing in the tank functions similarly to a tuned mass damper, with the main difference being that the liquid tank relies on the sloshing fluid's inertial force to reduce the vibratory motion. The efficiency of sloshing in the tank mainly depends on the fluid height in the tank, the tank length, and the type of structure that couples with the sloshing tank.

2. Literature

In the structures-TLD and liquid storage systems, sloshing in the tank is a highly complex phenomenon to achieve the required level of damping in the tank. It may not control the vibration of land structures and the stability of ocean vehicles effectively. Thus, controlling the liquid sloshing in the tank is essential for safety and operation.

As a result, numerous researchers conducted experimental, analytical, and numerical tank studies using various flow-dampening devices. The devices mainly include; Solid baffles and porous baffles. Damping in the tank-fluid system and system coupled with the fluid tank to enhance the perforated screens is a feasible technique that can work more effectively. However, they also introduce complex nonlinear behaviour in the sloshing environment. The design of TLDs benefits from a deeper understanding of the effects and performance of perforated screens on sloshing in the tank. Hence, recent efforts have concentrated on designing and modelling the sloshing in tanks with the varying number, positions of placement, and varying percentages of perforated

ISSN: 2347-4688, Vol. 10, No.(3) 2022, pg. 300-306

Current Agriculture Research Journal

www.agriculturejournal.org

Modelling and Analysis of Areca Nut Collecting Machine

JAYARAM BHAT[1]*, VISHAWANATH BHAT[1], GIRISH KUMAR[2] and B L SHIVAPRASAD[1]

[1]Shri Dharmasthala Manjunatheshwara College of Engineering and Technology, Dharwad Karnataka, India.

[2]Shri Dharmasthala Manjunatheshwara Institute of Technology, Ujire, Karnataka, India.

Abstract

Areca is a prominent commercial crop in coastal regions of Karnataka, a state of India. Major difficulties faced by areca growers include the shortage of labor and intensive time required for collecting areca nuts from ground during the drying process.To address the above problem, an area collecting machine is modelled. For design optimization of machine components, stress analysis on chassis, impact analysis on feeder ramp and deflection analysis on conveyor beltare assessed using solid edge tool. Design modifications incorporated from the above Finite Element based analysis are increasing the thickness of feeder ramp from 3 mm to 6mm along with the provision of ribs and providing support plates for conveyor belt. Modelled machine can bear 100 kg load with bagging efficiency of 900 areca nuts for per revolution of the belt.

Article History

Received: 07 September 2022

Accepted: 15 November 2022

Keywords

Areacanut;
Chassis Design;
Feeder Mechanism;
Material Handling System.

Introduction

Areca nut is being used in food and beverage, cosmetics, textiles and coir industries.[1] India is the largest producer of Areca nuts in the world and ranks first in both area of cultivation (58%) and production (53%) of Areca nuts. Areca nut cultivation forms the major livelihood of farmers of costal and malnad parts of Karnataka as shown in the map (Fig.1) covering around 2.15 lakh hectors of land on and annual production of 12375 metric tons of areca.[2]

Areca nut tree belongs to a palm family Aceraceae. The tree can grow up to 30 meter high, which has adventitious root system and leaves are of paripinnate in structure and emerges from crown (Fig.2). Humid tropic climate and loamy soil with uniformly distributed rainfalls are best suitable for high yields. It takes about six to seven years for a plantation to start yielding arecanuts. Areca nut takes 40 to 45 weeks to get fully ripe and then it has to be harvested and dried in the sun for about 40 to 45 days by spreading on the ground evenly as shown in the Figure 3.

The process of manual collection of areca nut from ground during drying, is time consuming and

CONTACT Jayaram Bhat jmb.bhat@gmail.com Shri Dharmasthala Manjunatheshwara College of Engineering and Technology, Dharwad Karnataka, India.

Doi: http://dx.doi.org/10.12944/CARJ.10.3.12

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IX Sep 2022- Available at www.ijraset.com

51

Developing an Individual Excel Sheet for Design and Analysis of Footing and Column

Ankita Dhotre[1], Sri. P. I. Cholappanavar[2]
Asst. Prof., Department of Civil Engineering, SDM College of Engineering and Technology, Dharwad-580002

Abstract: Analysis and design of RC structures using FEM based software package needs basic knowledge of that software. This project focuses on developing a excel sheet with simple user interface to analyze and design RC structures without any prior knowledge of software. This excel sheet considers only the gravity loads. Grids in X and Y directions are defined in the sheet. User can enter distance between grids. This sheet provides design of footing and column.

I. INTRODUCTION

Analysis and design of multi-story structure is a tedious task. FEM based software are used to carry out the same. However, use of these software needslot of experience and expertise. Lack of exposure and expertise in these software leads to inaccurate designs. To ease the design of structures, a excel sheet with simple user interface is developed which can be used by almost anyone for building design. The excel is designed to take minimum input and provide maximum output. Building structures are usually divided by grid systems depending on their column location, which denotes centre to centre distance. This excel sheet takes centre to centre distance as input. Load on the column is calculated using the tributary area defined by the centre-to-centre distance. This load includes live load, floor finish, dead load of slab, beam, column and wall load. Depending on number of floors the total load is calculated and the same is used to design footings. Similarly, columns are designed for axial load and moments using SP16. Design of beam is carried outusing bending moment and shear force factors for continuous beams as specified in the IS456:2000. Slab panels are designed using the centre-to-centredistance provided as per the support condition. All the footings, columns, beams and slabs can be independently designed. To validate the results, the same structure was analysed and designed in ETABS, which gave results withaccuracy up to 95%. The accuracy decreases as the size of the structure increases.

This excel sheet is very useful in designing regular structures till G+3 and up to span 7m under gravity loading. Lateral loads are not considered for design.

II. LITERATURE REVIEW

A. "Development of Design Spreadsheet Tool for R.C.C. Beam Design using V.B.A."Author – Vijay Srivastav

The outline manages a complicated and repeated task because the design technique in light of the Limit State Method includes different mathematical statements and parameters. In addition, outlining is a time-consuming and extremely repeated trailing technique. Therefore, using spreadsheets can significantly cut down on a planner's or builder's time and effort. Despite the availability of numerous standard configuration programming packages, spreadsheets have emerged as one of the best options for designers due to their minimization and compliance. Themain goal of this project is to create an MS Excel Spreadsheet with VBA Programming that will enable users to

1) Analyse a beam for Shear Force, Bending Moment, Slope, and Deflection for a variety ofend Conditions and for a variety of load patterns.
2) To determine the safe load carrying capacity of a beam
3) To Create RCC Beams An outline equipment that can be usefully used by a professional toanalyse and design an RCC beam will be the project's output.

B. "Automated Excel Sheets for Various RC Elements."Author – Nitin Tiwari, Rashmi Sakalle

In this work, the rebars of various RC elements, such as beams, columns, and slabs, have been calculated and analysed using the EXCEL spreadsheet programme. This project work has calculated five different types of EXCEL spreadsheets, including one-way and two-way slabs,short columns and long columns, cantilever beams, simply supported beams, and short columnsand long columns. Effective span, nominal cover, and effective length of compression members were only a few of the distinctive factors that were considered in our analysis. The RC elementshave been assigned several checks in addition to having a variety of distinguishing attributes. RCC code IS 456:2000 has been used as a source of inspiration.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IX Sep 2022- Available at www.ijraset.com

Developing an Individual Excel Sheet for Design and Analysis of Beam and Slab

Sachin Tattimani[1], Sri. P. I. Cholappanavar[2]
[1]Student, [2]Asst. Prof., Department of Civil Engineering, SDM college of Engineering and Technology, Dharwad-580002

***Abstract:** Analysis and design of RC structures using FEM based software package needs basic knowledge of that software. This project focuses on developing a excel sheet with simple user interface to analyze and design RC structures without any prior knowledge of software. This excel sheet considers only the gravity loads. Grids in X and Y directions are defined in the sheet. User can enter distance between grids. This sheet provides design of footing and column.*

I. INTRODUCTION

Analysis and design of multi-story structure is a tedious task. FEM based software are used to carry out the same. However, use of these software needs lot of experience and expertise. Lack of exposure and expertise in these software leads to inaccurate designs. To ease the design of structures, a excel sheet with simple user interface is developed which can be used by almost anyone for building design. The excel is designed to take minimum input and provide maximum output. Building structures are usually divided by grid systems depending on their column location, which denotes centre to centre distance. This excel sheet takes centre to centre distance as input. Load on the column is calculated using the tributary area defined by the centre-to-centre distance. This load includes live load, floor finish, dead load of slab, beam, column and wall load. Depending on number of floors the total load is calculated and the same is used to design footings. Similarly, columns are designed for axial load and moments using SP16. Design of beam is carried out using bending moment and shear force factors for continuous beams as specified in the IS456:2000. Slab panels are designed using the centre-to-centre distance provided as per the support condition. All the footings,columns, beams and slabs can be independently designed. To validate the results, the same structure was analysed and designed in ETABS, which gave results with accuracy up to 95%. The accuracy decreases as the size of the structure increases.

This excel sheet is very useful in designing regular structures till G+3 and up to span 7m under gravity loading. Lateral loads are not considered for design.

II. LITERATURE REVIEW

A. *"Development of Design Spreadsheet Tool for R.C.C. Beam Design using V.B.A."*

Author – Vijay Srivastav

The outline manages a complicated and repeated task because the design technique in light of the Limit State Method includes different mathematical statements and parameters. In addition, outlining is a time-consuming and extremely repeated trailing technique. Therefore, using spreadsheets can significantly cut down on a planner's or builder's time and effort. Despite the availability of numerous standard configuration programming packages, spreadsheets have emerged as one of the best options for designers due to their minimization and compliance. The main goal of this project is to create an MS Excel Spreadsheet with VBA Programming that will enable users to

1) Analyse a beam for Shear Force, Bending Moment, Slope, and Deflection for a variety of end Conditions and for a variety of load patterns.
2) To determine the safe load carrying capacity of a beam
3) To Create RCC Beams An outline equipment that can be usefully used by a professional to analyse and design an RCC beam will be the project's output.

B. *"Automated Excel Sheets for Various RC Elements."*

Author – Nitin Tiwari, Rashmi Sakalle

In this work, the rebars of various RC elements, such as beams, columns, and slabs, have been calculated and analysed using the EXCEL spreadsheet programme. This project work has calculated five different types of EXCEL spreadsheets, including one-way and two-way slabs, short columns and long columns, cantilever beams, simply supported beams, and short columns and long columns.

Journal of Science and Technology
ISSN: 2456-5660 Volume 7, Issue 05 (JULY 2022)
www.jst.org.in *DOI:https://doi.org/10.46243/jst.2022.v7.i05.pp60 - 72*

53

ANALYSIS, DESIGN AND STRUCTURAL ESTIMATION OF RCC STRUCTURE USING ETABS AND MICROSOFT EXCEL

Kartik Ravikumar Kajagar, Sameer Chitnis

Department of Civil Engineering, SDMCET, Dharwad,
Department of Civil Engineering, SDMCET, Dharwad,

To Cite this Article

Kartik Ravikumar Kajagar, Sameer Chitnis, **ANALYSIS, DESIGN AND STRUCTURAL ESTIMATION OF RCC STRUCTURE USINGETABS AND MICROSOFT EXCEL"**, *Journal of Science and Technology, Vol. 07, Issue 05,-July 2022, pp60-72*

Article Info
Received: 13-05-2022 *Revised: 2-06-2022* *Accepted: 10-07-2022* *Published: 18-07-2022*

Abstract

☐ Since India is a developing country, many new projects are being started on a daily basis that must be finished within a certain time frame. The main factors in project completion are design and implementation, which aid in the stability of the structure. In this dissertation, we describe our novel method to design utilising ETABS and Microsoft Excel. We start by designing the structure and simulating it in ETABS software using various loads and load combinations according to the IS codebook, then exporting to MS Excel and additional processing delivers the entire detailed and schedule drawings of the RCC structure. Following the design of structural elements, we must acquire the bar bending schedule in order to
achieve structural estimation. So, after we have the bar bending schedule, we can calculate the quantity of steel and concrete required in the building, and then we can estimate the cement, sand, coarse aggregate, and bricks based on the concrete given. The outcome of our proposed strategy demonstrates that the procedure is very efficient, saving time and money. Normally, the design process takes many hours to complete, but with the aid of this proposed approach, efficient solutions may be obtained in just a few minutes.

INTRODUCTION

☐ Every structure has two parts: the superstructure and the substructure foundations. The superstructure is the upper component of the building that serves the function of its intended use, whereas the substructure is the lower portion of the building that transmits the load of the superstructure to the sub soil, and the footing is the lower portion of the foundation that has direct contact with the sub-soil. The foundation is the most fundamental component of any structure. In India, structural designers use ETABS to perform analysis and design by going through several processes such as modelling, material properties, support conditions, materials and their mixtures, analyses, design options, and so on. ETABS is a comprehensive structural engineering software system that covers all phases of structural engineering, including model generation, verification, analysis, and design. With ETABS, we can simply analyse diverse materials such as concrete, steel, reinforced concrete, and so on, and the results may be exported without any errors. It can automatically generate gravitational loads, lateral loads, and self- weights.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 19, Issue 4 Ser. II (Jul. – Aug. 2022), PP 36-42
www.iosrjournals.org

54

Parametric study of multicell box culverts using STAAD Pro

Sachin Honwad[1], Sameer Chitnis[2]
[1](Shri DharmasthalaCollege of Engineering and Technology/ VTU, INDIA)
[2](Shri DharmasthalaCollege of Engineering and Technology/ VTU, INDIA)

Abstract: *The present Paper presents the behavior of culverts with varying skew angles, for this study 3 cell culverts are studied with skew angles varying from 0 to 70 degree, with 10-degree interval. STAAD Pro software isused for analysis, skeweffect on bottom slab, sidewall, top slab of culverts isstudiedusingvariousloads (dead load, live load, live load surcharge and earth pressure) and load combination using IRC 6 :2017. Parameters used for this study are shear force, longitudinal moments, transverse moments and torsional.*
Results*: While designing box culvert extreme caution and measures must be taken even at lower and also try to minimize the skew where it's possible, the box culverts with higher skew angles greater than 20 degrees must be examined properly so that the higher bending moments and shear forces shouldn't go un noticed or neglected. This behaviour is unlike bridge structure where the change in moments and forces doesn't change abruptly even at lower angles.*
Key Word: Culverts, Box culvert, skew,

Date of Submission: 06-08-2022 Date of Acceptance: 21-08-2022

I. Introduction

A culvert is a structure used to direct underground waterways or to divert water past obstructions. A culvert is often built of a pipe, reinforced concrete, or another material that is surrounded by earth and is immersed in the ground. Culverts are frequently used to transport water beneath a road at natural drainage and stream crossings as well as crossdrains to ease drainage of ditches along the roadside. Oftentimes, they are empty when discovered beneath highways. A culvert may also resemble a bridge and be built to let vehicles or pedestrians cross over a canal while providing the water with enough room to move through. The top, bottom, and two vertical side walls of a reinforced concrete box culvert are all built in one piece to create a closed, hollow rectangle or square single cell or several cells. Currently, in order to build and pass roads or trains underneath earth embankments, culverts must be constructed to allow water to cross from both sides of the embankment. Fish, animals, and other aquatic life that needs in-stream passage can move more freely when culverts that allow for proper aquatic creature transit are present. Replacement of culverts that allow for the passage of aquatic organisms is a common method in stream restoration. This method has enhanced fish passage and a decreased likelihood of catastrophic failure as long-term advantages. The aquatic biology will not be significantly affected in the short term if optimal management measures are used.

II. Methodology

I. Choosing culvert dimensions based on prior research and the load cases to be utilized for modeling and analysis in the STAADPRO program.
II. Culvert modeling and analysis with various skew angles.
III. Comparing the outcomes based on the moment and the shear force.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 19, Issue 4 Ser. II (Jul. – Aug. 2022), PP 43-48
www.iosrjournals.org

55

Comparison Of Steel Girder And RC Girder As Peripheral Beam To Study The Effect Of Spacing Of Grid Beams And Its Depth On Peripheral Beams In Grid Floor Frame

Megha Saboji[1], Sameer Chitnis[2]
[1](ShriDharmasthalaCollege of Engineering and Technology/ VTU, INDIA)
[2](ShriDharmasthala College of Engineering and Technology/ VTU, INDIA)

Abstract:
The terms "Grid slab" and "Waffle slab" refer to an arrangement of intersecting beams that are spaced regularly and joined to form a slab of nominal thickness. For buildings like hotels, porches, airport terminals, banquet halls and parking garages, grid slabs are a fairly common structural form. In contrast to a conventional concrete slab, the Grid construction is monolithic in nature, stronger, and capable of supporting heavier weights. This study suggests a brand-new type of composite Grid slab made of flat RC slabs and orthogonal steel peripheral girders. Because it fully utilizes both the compressive resistance of concrete slabs and the tensile resistance of steel, composite slab structures have a strong load-bearing capacity. In this study, a non-linear static analysis is conducted using the E-Tabs 2016 program to examine the impact of grid beam spacing and depth in the grid floor frame. The findings are tabulated using MS Excel. The comparison of such results with the available results obtained from the study carried out by Prof.Dr.S.A.Halkude and S.V.Mahamuni for halls with reinforced concrete peripheral beams is done by plotting graphs.
Conclusion: *For a fixed depth of steel Peripheral beams, the maximum bending moment can be reduced by decreasing the depth of Grid beams. For L /B ratios less than 1.2, the maximum bending moment in peripheral beams may be attained. The maximum bending moment in RC peripheral beams in halls grows with increasing L/B ratio for the specified depth of peripheral beams.*
Key Word:Grid slab, Waffle slab, steel peripheral girders, RC slabs, E-Tabs 2016.

Date of Submission: 06-08-2022 Date of Acceptance: 21-08-2022

I. Introduction

A grid slab system comprises of beams or ribs that are monolithic with the slab and spaced at regular intervals in a perpendicular direction. These sorts of slabs have a nominal thickness and are supported by a grid or mesh of beams that run in both directions. It has been discovered that grids (beams) are particularly effective in transferring weight. Grids that run in perpendicular directions often maintain the same size. For architectural reasons, grid slabs are typically used in big rooms like auditoriums, vestibules, theatre halls, and retail display rooms where column free space is frequently the primary need. As a result of their greater rigidity and reduced deflection, they are employed for heavy loads and long span constructions. The ceiling void that is created reduces dead load and is useful for architectural lighting that is hidden. Although they are most frequently utilised in commercial or industrial buildings, these slabs are also employed as the foundation for many various kinds of buildings and structures. Grid foundations can support a lot more weight than conventional concrete slabs and are resistant to cracking and drooping.

The classic RC grid slab comprises of a plate (concrete slab), grillage of beams (ribbed), and shell behavior (diaphragm stiff and ribbed). It benefits from two-way load-bearing capacity, and the demand for engineering remains high.In case of composite grid slab,the RC ribs are swapped out for steel girders which are attached to the flat slab by shear studs. The compressive strength of concrete slabs and the tensile strength of steel girders may both be used in the design of composite slabs. By employing the right studs to stop slide between the RC slab and steel girders, a nice composite action may be achieved.

PAPER • OPEN ACCESS

Studying the influence of Constricted Froude's Number on Energy Dissipators in Open Channel Flow

To cite this article: Sameer Chitnis *et al* 2022 *IOP Conf. Ser.: Earth Environ. Sci.* **1086** 012033

Studying the influence of Constricted Froude's Number on Energy Dissipators in Open Channel Flow

Sameer Chitnis [1],Shravankumar S M [2], Praveen Kumar Sakare[3], Mohammed Faheemuddin [4] and Kollozu Sree Harshini [5]

[1, 3]Department of Civil Engineering, Department of Civil Engineering, Shri Dharmasthala college of Engineering and Technology Dharwad, Karnataka-580002, India
[2, 4, 5] Department of Civil Engineering, Sreenidhi Institute of Science and Technology, Hyderabad-501301, India

Abstract. Model experimentation was used to examine the flow properties above the perpendicular descents with a grid roof, or the "grid drop-type dissipators". The software ANSYS FLUENT R 15 is used to analyse the fluid flow above the impediment and research of the flow arrays and hydraulic characteristics. The purpose of the flow analysis is to examine how subcritical flow develops upstream and to let the water flowing downstream to slow down the flow of water to prevent erosion downstream. The pool formation, mixing zone, pressure distribution, and velocity distribution were all analysed using the Ansys software. The findings show that, in comparison to a simple vertical drop, the suggested hydraulic structure removes undesirable flow situations and serves as the foundation for a more operative flow rheostat arrangement. It is shown that, in comparison to a straight vertical drop, the scattering/circulation process in the pool surges the velocity grade, the impetus altercation, and, as a result, the energy dissipation.

Keywords: Energy Dissipation, ANSYS Simulation, Open channel flow, Froude's Number, Hydraulic Jump.

1. INTRODUCTION:

Froude's number is a dimensionless value that defines diverse flow systems of exposed channel flow. It is a quotient of inertial and gravitational forces. It is a dimension of bulk flow characteristics such as surfs, sand crib forms, and flow/gravity at a cross-section or between boulders. The Froude's numeral is defined as the proportion of Inertial forces to gravitational forces, the flow is called as subcritical flow if the Froude's number is lesser than 1 and super critical flow if greater than 1. Froude's number speaks about the influence of gravitational flow on the fluid which gives a fairer idea regarding the influence of flow over the hydraulic structures on the upstream and downstream[1]. The Froude's number has important role in increasing or decreasing the efficiency of the hydraulic structures, the low Froude number on the upstream increases the length and height of the hydraulic jump which results in more and uneconomical energy dissipators[2]. Many hydraulic structures contribute in reducing the energy at the point of high energy but there are methods wherein we can control the problems caused due to hydraulic jump by managing and controlling the Froude's number in the flow [3]. Further the problem is aggravated when the slope of the ground changes abruptly with the respect to the slope of the channel, due low Froude's number the pitting and heavy erosion takes place at the downstream[4]. Maintaining an acceptable Froude's number in irrigation channel is challenging especially when the channel is moving through the rugged terrain, generally a drop structure is used to overcome the excessive potential energy due to change in the datum head in order to maintain the Bernoulli's principle in the flow[1]. The energy loss is achieved mostly through the turbulent mixing which is also responsible for pitting

Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 09, September 2022

Study and Comparison of Conventional Analysis with Construction Sequence Analysis using ETABS

Tirumalrao Mutalikdesai
Student
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Kushal Kapali
Asst. Professor
Dept. of Civil Engineering,
SDMCET, Dharwad, India

Abstract— **Some of the columns in multistorey buildings are designed to be floating columns with transfer girder support. These columns are being analysed in a single step under the presumption that the frame will be subjected to design loads. This is necessary because architectural requirements dictate that certain columns be designed as floating columns. In point of fact, loads are applied at various points during the construction process as the frames of the building are built storey by storey. Within the scope of this study, we have looked at two different instances. Whereas in Case 1 the multistoreyed building with floating columns and transfer girder with 16 stories will be analysed as a whole for the subjected loading, in Case 2 the multistoreyed building with floating columns and transfer girder with 16 stories will be analysed with reference to the construction sequence or staged construction. Both cases involve a multistorey building with floating columns and transfer girders. For the purpose of illustrating the floating column on girders, a comprehensive analysis and comparison of the change in deformations, bending moment, shear force, and axial force will be shown for the transfer girders. The ETABS software is used for both the analysis of the building**

Keywords—Floating columns; Construction sequence analysis; Conventional analysis; Staged construction; ETABS

I. INTRODUCTION

The multistorey building frames have been studied for a very long time under the presumption that the entire load is applied to the finished frame structure with all loads acting on the building—self-weight, superimposed load, live load, and lateral loads—applied on the finished frame at a specific instant as a single step analysis. But in reality, when the building structure is built storey by storey in a sequential manner, the dead load owing to each structural component and finishing item is imposed separately. When different loads are imposed all at once, a building structure performs quite differently than when the stresses are applied gradually. Construction sequence analysis refers to the process of analysing a structure in accordance with real construction methods (CSA). Construction sequence analysis, commonly referred to as staged construction analysis, is a static non-linear method of analysis that takes the idea of incremental loading into consideration.

One of the topics that has drawn a lot of engineering research efforts and designers' attention is the structural analysis of multistorey structures. However, there is one area that has received little attention from earlier researchers: the implications of building sequence analysis in a multistorey frame. The structural components are added gradually while a building is constructed, and as a result, their dead load is supported by the portion of the structure that is complete at the time of their installation. As a result, the qualities of the members that have not yet been built have no impact on how displacement and stresses are distributed within a given story. By adding together, the outcomes of the study of each step of the building frame structure, it is possible to determine the right distribution of the displacement and stresses of every part.

Construction sequence analysis is becoming a crucial component of analysis since so many well-known analysis software packages now incorporate this feature. However, due to a lack of understanding of its value and use, nonlinear static analysis is not as well known. Construction sequence analysis, like many other analyses, had a role throughout the structure's design phase. As was already noted, it deals with nonlinear behaviour under static loads in the form of sequential load increase and how it affects structures when structural elements begin to respond to loads before the entire system is complete. ETABS (Extended 3D analysis of building systems), one of the top analysis programmes, is used for finite element analysis. All displacement outcomes are recorded in meters, whereas moments and axial loads are quantified in KN-m and KN, respectively

II. OBJECTIVES

- To use construction sequence analysis to get an analytical understanding of the behaviour of the high-rise building during the construction process at its various stages.
- To study of the similarities and differences between the conventional method and the Construction Sequence analysis.
- To Determine the percentage of change in deflection, bending moments, shear force, and axial force, of the structural elements using the Conventional analysis and the construction sequence analysis

STABILITY OF DIFFERENT SHAPED BUILDING IN SEISMIC ZONE V AND II

Rohan A Bujurke[1], Kushal Kapali A[2]

[1]*Rohan A Bujurke Civil Engineering SDMCET, Dharwad*
[2]*Kushal Kapali A Civil Engineering SDMCET, Dharwad*

Abstract - Multi-storey different shaped building usually become a landmarks in such scenario the stability of structure becomes utmost concerns. For multi-storey building the lateral forces are the major factor which will concern the stability of structure. The benefits of constructing earthquake resistance buildings are the savings in property damage, injuries and life that occurs in the event of an earthquake. This work is carried out to present a methodology that infers upon which shape(L H C shapes) of building is stable in which seismic zone. With the aid of the computer programmed Etabs, the analysis is carried out using the design parameters specified in IS-456-2000 for structural design and IS-1893-2002-Part-1 for seismic loading. Software called Etabs is used to mathematically model and physically analyze civil structures, such as concrete barriers and cable-stayed bridges. Outcome of this study is H shaped building has high Story displacement, story drift and more story stiffness in Y-direction, on other hand C-shaped is stable in X-direction.

Key Words: multi-storey, seismic, earthquake, stability, shape, building

1.INTRODUCTION

In order to achieve the design and ensure that the structural integrity of the structure is not jeopardized, building that are earthquake resistant require meticulous design and high costs. However, in recent years there has been continuous study into ways to lower the cost of earthquake-resistant structures without sacrificing the building's structural integrity or efficiency. The methodology is based on modern technical and geological research of earthquakes and the damage they inflict, as well as the economic approach to evaluating hazards to human life. Reductions in the likelihood of earthquake-related property damage, injuries, and fatalities are advantages of creating earthquake-resistant structures. The main scope of the present study is to analyze the stability of multi-storey building with shape "C, H and L" in seismic zone II and V that is in moderate and severe earthquake zones respectively. Outcome may help in building landmark multi-storey structures which are stable in both moderate and severe earth quake zone

2. Modeling

A 10-storey, regular-plan building with a 3.0-meter height for each story is modeled for this study. According to the Indian Code of Practice for Seismic Resistant Design of Buildings, these structures were modeled. The height of the floors of the buildings is assumed to be constant, including the ground floor. The buildings are modeled using software ETABS. L , H , C models were studied in severe and minor seismic zones. Models are studied by comparing joint displacement, story drift, frame element joint forces, stiffness, joint reaction.

1	Number of stories	10
2	Column size	700mm × 700mm
3	Beam size in longitudinal and traverse direction	350 mm x 350 mm
4	Thickness of slab	150mm **Ground to 2nd** 125mm **3rd and 4th** 100mm **5th to 10th**
5	Floor to floor height	3 m
6	Grade of concrete and steel	M25 and Fe415
7	Column to column distance	5 m