Analysis of Bridge Pier as per Different Seismic Zone of India

Aditi Dubey
Department of Civil Engineering, SSIPMT, Raipur, India
Aditidubey725@gmail.com

Toshan Singh Rathour*
Department of Civil Engineering, SSIPMT, Raipur, India
tsrathour@gmail.com

Menda Babu Rao
Civil Engineering Department , SSIPMT, Raipur, India
mendababurao27@gmail.com

*Corresponding Author

Abstract:

The main objective of this paper is to analyze a bridge pier through finite element method as per different seismic zones of India. In this paper a detail analysis is done of bridge pier by using ANSYS workbench software. This study deals with static structural analysis. A bridge pier model is taken which is built in engapalli nalla on elmidi sankanpalli to maddedroad at ch km 17/10 block- usoor, district- bijapur, C.G. The 3D model is drawn is AUTO Cad Software and six variations are done in model by increasing and decreasing the dimensions of the pier and foundation. Then the seven models are analyzed in ANSYS as per different seismic zones i.e. zone 2, zone 3 and zone 4. The analysis and study is done to evaluate total deformation, directional acceleration, directional velocity, normal stress, shear stress, equivalent stress, normal elastic strain and shear elastic strain for seven different model in three different seismic zones of India.

Keywords*—*pier, ansys, seismic zone, finite element analysis

DOI Number: 10.48047/NQ.2022.20.21.NQ99086 Neuro Quantology 2022; 20(21): 717-722

I. Introduction

Bridges are constructed for a lifetime facility, which should be functional even during major disasters like earthquake shaking, the damage to bridges can cause many difficulties in post-disaster restoration and relief activity. The collapse of a bridge not only causes loss of life, properties and money but it is also a hurdle in recovering from the loss that happened during the natural disaster. Foundation, superstructure, substructure and substructure-foundation connections have failed during or after a major earthquake, and in past years a large number of bridges has collapsed because of the damage in the structure.

During the earthquake the connected part of the bridge to the earth surface is more of substructure then the super structure and so the impact of the earthquake is more and acts firstly on substructure. If the substructure is designed as per the seismic zone and earthquake frequency, then the chances of earthquake failure could be reduced. Through finite element method of analysis, the deformation and effect of bridge pier could be calculated before construction of bridge through which pre knowledge is acquired about the load and frequency of earthquake the structure can bear.

II. Methodology

In this project an attempt is made to check the analysis for seismic zones by taking the actual response spectrum data of land during the earthquake. By taking the actual bridge model seven sub models are created in which one model is with decreased dimensions and the rest five is with increased dimension. Then in these seven model, finite element analysis is done through ANSYS workbench software to evaluate the different total deformation, directional acceleration, directional velocity, normal stress, shear stress, equivalent stress, normal elastic strain and shear elastic strain in three different seismic zones of India.

A. *Planning*

The planning is done for all the process that is to be followed, it is the most important phase of project for proper and systematic conduction and completion of the project. The main job in planning stage is to check all the previous paper and books and create the proper steps for the project.

B. *Data Collection*

For the project the two major data required were the a pre designed bridge dimensions and response spectrum data. Without bridge dimensions the ideal bridge dimensions could not be known and in ANSYS software the seismic analysis is done through response spectrum data in which either frequency vs velocity or frequency vs acceleration curve is required. For the rest data collection, a proper knowledge about the ANSYS software and AUTO Cad software was required the rest data was taken from books and previous papers. For the dimension of bridge pier, the dimension was taken from bridge of engapalli nalla on elmidi sankanpalli to maddedroad at ch km 17/10 block- usoor, district-bijapur, C.G. Then the study of the dimensions of bridge pier was done and through which models were created in AUTO Cad software.

C. Software

For the Finite Element Analysis, a suitable software is ANSYS and as the seven different models calculation was to be done the 3D modeling was done in AUTO Cad software. For the proper and correct analysis, the knowledge of both the software that is AUTO Cad and ANSYS Workbench was required. The 3D modeling was done in AUTO Cad software then it was converted to iges file so that it can be imported to ANSYS software.

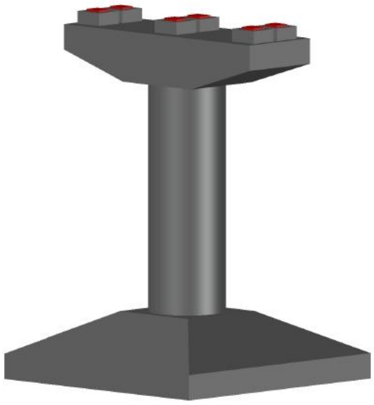

FIG I. PIER ORIGINAL MODEL IN AUTOCAD SOFTWARE

D. Material Properties

Material properties is to be added in ANSYS Software and as per out requirement the materials were reinforced cement concrete and elastomers. As per the requirement of the place and the structure the specific properties of concrete and elastomers were selected. The concrete of bridge pier was of M30 grade and for elastomeric bearing the specifications were taken from IRC:83 (part II) 2015 standard

E. Checking

After the complete analysis the value of total deformation, directional acceleration, directional velocity, normal stress, shear stress, equivalent stress, normal elastic strain and shear elastic strain for seven different model in three different seismic zones of India is putted in tabular for and accordingly the results are compared.

The results are compared of bridge pier under static loading condition and due response spectrum in different dimensional structure in order to improve the existing design and possibly reduce cost in bridge pier construction without reducing strength of bridge pier.

III. FINITE ELEMENT ANALYSIS

Material model by computer or stressed design and then analyzed for specific area is present in FEA that is finite element analysis. Finite element analysis is used in new product design or in existing product refinement. With finite element analysis a company is able to verify the proposed design and also able to perform according to the client's specifications before starting the process of manufacturing or construction. A new service condition is required while the process of modification of an existing product or structure utilizations.

There are mainly two types of analysis which are used in industries for calculations of finite element analysis, they are as follows:

1. 2-D modeling
2. 3-D modeling.

TABLE I. DIMENSIONS OF PIER MODEL

Model No.	B1xB1	B2xB2	B3xB3	B4xB4
1	6x6	6x6	2.2x2.2	6x2
2	5.5x5.5	5.5x5.5	2x2	6x2
3	6.25x6.25	6.25x6.25	2.5x 2.5	6x2
4	6.5x 6.5	6.5x 6.5	2.75x2.75	6x2
5	6.75x6.75	6.75x6.75	3x3	6x2
6	7x7	7x7	3.25x3.25	6x2
7	7.25x7.25	7.25x7.25	3.5x3.5	6x2

TABLE II. DIMENSIONS OF PIER MODEL

Model No.	H1	H2	H3	H4	H5
1	600	1	5.4	600	600
2	500	0.8	5.7	600	600
3	700	1.1	5.2	700	700
4	800	1.2	5	800	800
5	900	1.3	4.8	900	900
6	1000	1.4	4.6	1000	1000
7	1100	1.5	4.4	1100	1100

TABLE III.

TABLE IV. DIMENSIONS OF PIER MODEL

Model No.	L1	L2	L3	D1	T1
1	6	2.2	60	1.8	2
2	6	2.2	60	1.6	2
3	6	2.3	70	1.9	2.1
4	6	2.4	80	2	2.2
5	6	2.5	90	2.1	2.3
6	6	2.6	100	2.2	2.4
7	6	2.7	110	2.3	2.5

TABLE V. DESCRIPTIONS

Descriptions of Abbreviations used	
Dimension of Square base of footing (meter)-	B1xB1
Height of Square base of footing (mm)-	H1
Dimension of Trapezoidal base of footing (meter)-	B2xB2
Dimension of Trapezoidal top of footing (meter)-	B3xB3
Height of Trapezoidal part of footing (meter)-	H2
Diameter of circle part of the bridge pier (meter)-	D1
Height of circle part of the bridge pier (meter)-	H3
Top Length of upper trapezoidal part of bridge pier (meter)-	L1
Bottom Length of upper trapezoidal part of bridge pier (meter)-	L2
Thickness of upper trapezoidal part of bridge pier (meter)-	T1
Height of upper trapezoidal part of bridge pier (meter)-	H4
Dimension of top rectangular part of bridge pier (meter)-	B4xB4
Height of top rectangular part of bridge pier (mm)-	H5
Distance of elastomer from the center of pier (mm)-	L3

IV. RESULT AND DISCUSSION

After the Manual calculation of bridge, the following loads were calculated in pier-

Dead Load=4069.8 Kilo Newton

Live Load=2955.45 Kilo Newton

Total Load=Dead Load+Live Load=7025.25Kilo Newton

After analysis in ANSYS Workbench Software following results were obtained-

The model 2 has least total Deformation before seismic analysis the model 7 has most with the increase in dimension firstly the total deformation increased then it started decreasing with increase in dimension then again it increased. Thus in case of total deformation before seismic analysis it is clear that model 7 will have most deformation.

After the response spectrum in taken then it is seen that total deformation decreases with increase in dimension but if the dimension are increased more i.e. after model 5 it starts increasing.

Directional acceleration kept on increasing with increase in dimensions of structure.

Directional velocity increased then decreased i.e. in model 5 then again it started increasing with increase in dimensions.

In case of normal stress, the max normal stress kept on increasing with increase in dimension but it was least in model 6 and then again started increasing and in case of min normal stress it was maximum in model 3 and least in model 4.

In case of shear stress, the max shear stress kept on increasing with increase in dimension but it was least in model 6 and then again started increasing and in case of min normal stress it was maximum in model 2.

In case of Equivalent stress, it kept on increasing and decreasing as per the dimension and was maximum in model 7.

In case of normal elastic strain, the value kept on decreasing with increase of dimension and at model 7 it increased.

In case of shear elastic strain, the value kept on increasing with increase of dimension then started decreasing and at model 7 it increased.

In case of analysis of seismic zone, it was seen that with increase of zone the values of total deformation, directional acceleration, directional velocity, normal stress, shear stress, equivalent stress, normal elastic strain and shear elastic strain also increased.

It could also be seen that model 4,5,6 is the ideal model in which the all the value decreases and model 7 in which the maximum dimension is the negative part of structure which is making it weak.

TABLE VI. TOTAL DEFORMATION AS PER DIFFERENT SIESMIC ZONES

Total Deformation in meter			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	1.020E-04	3.697E-04	5.437E-04
Model 2	1.326E-04	2.532E-04	6.502E-04
Model 3	9.293E-05	2.460E-04	5.069E-04
Model 4	7.036E-05	2.963E-04	4.557E-04
Model 5	4.616E-05	2.760E-04	2.793E-04
Model 6	5.238E-05	3.069E-04	3.540E-04
Model 7	3.774E-05	2.270E-04	3.535E-04

TABLE VII.

TABLE VIII. DIRECTIONAL ACCELERATION AS PER DIFFERENT SIESMIC ZONES

Directional Acceleration in meter/second²			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	3.476E-02	4.928E-02	1.058E-01
Model 2	3.350E-02	3.396E-02	1.014E-01
Model 3	3.740E-02	4.333E-02	1.139E-01
Model 4	4.151E-02	6.828E-02	1.264E-01
Model 5	4.855E-02	8.282E-02	1.467E-01
Model 6	4.863E-02	8.673E-02	1.473E-01
Model 7	3.724E-02	1.136E-01	1.771E-01

TABLE IX. DIRECTIONAL VELOCITY AS PER DIFFERENT SIESMIC ZONES

Directional Velocity in meter/second			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	1.098E-03	3.982E-03	5.855E-03
Model 2	1.082E-03	2.066E-03	5.306E-03
Model 3	1.148E-03	3.039E-03	6.264E-03
Model 4	9.978E-04	4.203E-03	6.463E-03
Model 5	7.463E-04	4.462E-03	4.515E-03
Model 6	8.298E-04	4.862E-03	5.608E-03
Model 7	7.888E-04	4.746E-03	7.388E-03

TABLE X. MAXIMUM NORMAL STRESS AS PER DIFFERENT SIESMIC ZONES

Maximum Normal Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	7.065E+04	2.561E+05	3.766E+05
Model 2	8.217E+04	1.568E+05	4.027E+05
Model 3	6.190E+04	1.637E+05	3.375E+05
Model 4	8.430E+04	3.548E+05	5.457E+05
Model 5	7.136E+04	4.261E+05	4.312E+05
Model 6	3.832E+04	2.244E+05	2.589E+05
Model 7	9.819E+04	5.896E+05	9.180E+05

TABLE XI. MINIMUM NORMAL STRESS AS PER DIFFERENT SIESMIC ZONES

Minimum Normal Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	8.869E+00	3.210E+01	4.727E+01
Model 2	4.821E+00	9.197E+00	2.362E+01
Model 3	1.190E+01	3.147E+01	6.487E+01
Model 4	2.218E+00	9.129E+00	1.407E+01
Model 5	4.565E+00	2.629E+01	2.684E+01
Model 6	5.058E+00	2.893E+01	3.336E+01
Model 7	4.216E+00	1.976E+01	3.215E+01

TABLE XII. MAXIMUM SHEAR STRESS AS PER DIFFERENT SIESMIC ZONES

Maximum Shear Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	2.714E+04	9.836E+04	1.447E+05
Model 2	2.027E+04	3.866E+04	9.930E+04
Model 3	2.144E+04	5.672E+04	1.169E+05
Model 4	3.191E+04	1.343E+05	2.066E+05
Model 5	2.746E+04	1.639E+05	1.659E+05
Model 6	1.670E+04	9.766E+04	1.126E+05
Model 7	3.740E+04	2.246E+05	3.497E+05

TABLE XIII. MINIMUM SHEAR STRESS AS PER DIFFERENT SIESMIC ZONES

Minimum Shear Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	6.485E-06	2.350E-05	3.454E-05
Model 2	4.827E-05	9.229E-05	2.367E-04
Model 3	5.427E-04	1.435E-03	2.958E-03
Model 4	4.294E-04	1.803E-03	2.776E-03
Model 5	4.136E-04	2.465E-03	2.496E-03
Model 6	5.337E-04	3.116E-03	3.589E-03
Model 7	6.458E-04	3.878E-03	6.039E-03

TABLE XIV. MAXIMUM EQUIVALENT STRESS AS PER DIFFERENT SIESMIC ZONES

Maximum Equivalent Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	1.437E+05	5.211E+05	7.663E+05
Model 2	1.527E+05	2.914E+05	7.483E+05
Model 3	1.219E+05	3.227E+05	6.650E+05
Model 4	9.529E+04	4.013E+05	6.172E+05
Model 5	7.210E+04	4.305E+05	4.357E+05
Model 6	7.201E+04	4.219E+05	4.866E+05
Model 7	9.808E+04	5.889E+05	9.170E+05

TABLE XV. MINIMUM EQUIVALENT STRESS AS PER DIFFERENT SIESMIC ZONES

Minimum Equivalent Stress in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	3.004E+01	1.089E+02	1.602E+02
Model 2	1.126E+01	2.148E+01	5.517E+01
Model 3	6.435E+01	1.702E+02	3.508E+02
Model 4	3.476E+01	1.459E+02	2.247E+02
Model 5	4.742E+01	2.824E+02	2.860E+02
Model 6	5.050E+01	2.946E+02	3.407E+02
Model 7	5.455E+01	3.276E+02	5.099E+02

TABLE XVI. MAXIMUM NORMAL ELASTIC STRAIN AS PER DIFFERENT SIESMIC ZONES

Maximum Normal Elastic Strain in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	1.104E-06	4.002E-06	5.884E-06
Model 2	1.583E-06	3.020E-06	7.756E-06
Model 3	1.099E-06	2.909E-06	5.995E-06
Model 4	9.621E-07	4.050E-06	6.228E-06
Model 5	8.137E-07	4.858E-06	4.917E-06
Model 6	8.610E-07	5.034E-06	5.808E-06
Model 7	9.509E-07	5.710E-06	8.890E-06

TABLE XVII. MINIMUM NORMAL ELASTIC STRAIN AS PER DIFFERENT SIESMIC ZONES

Minimum Normal Elastic Strain in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	2.725E-16	9.874E-16	1.452E-15
Model 2	3.171E-16	6.051E-16	1.554E-15
Model 3	1.615E-14	4.271E-14	8.803E-14
Model 4	1.265E-14	5.306E-14	8.173E-14
Model 5	1.208E-14	7.194E-14	7.286E-14
Model 6	1.574E-14	9.183E-14	1.056E-13
Model 7	1.903E-14	1.141E-13	1.778E-13

TABLE XVIII. MAXIMUM SHEAR ELASTIC STRAIN AS PER DIFFERENT SIESMIC ZONES

Maximum Shear Elastic Strain in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	1.125E-06	4.078E-06	5.997E-06
Model 2	9.252E-07	1.766E-06	4.534E-06
Model 3	1.687E-06	4.462E-06	9.196E-06
Model 4	1.435E-06	6.040E-06	9.289E-06
Model 5	1.392E-06	8.312E-06	8.413E-06
Model 6	1.314E-06	7.682E-06	8.860E-06
Model 7	1.779E-06	1.068E-05	1.663E-05

TABLE XIX. MINIMUM SHEAR ELASTIC STRAIN

Minimum Shear Elastic Strain in Pascal			
	Zone 2	*Zone 3*	*Zone 4*
Model 1	5.102E-16	1.848E-15	2.717E-15
Model 2	3.797E-15	7.260E-15	1.862E-14
Model 3	4.269E-14	1.129E-13	2.327E-13
Model 4	3.378E-14	1.482E-13	2.184E-13
Model 5	3.254E-14	1.939E-13	1.963E-13
Model 6	4.199E-14	2.452E-13	2.824E-13
Model 7	5.080E-14	3.050E-13	4.751E-13

TABLE XX. TOTAL DEFORMATION BEFORE ADDING RESPONSE SPECTRUM DATA

S No.	Model	Total Deformation Before Response Spectrum	
		unit	*max*
1	Model 1 (original)	m	1.327E-03
2	Model 2 (less)	m	1.317E-03
3	Model 3	m	1.326E-03
4	Model 4	m	1.329E-03
5	Model 5	m	1.326E-03
6	Model 6	m	1.301E-03
7	Model 7	m	1.322E-03

V. CONCLUSION

By the analysis of the bridge pier it is clear that with increasing values of dimensions as listed in table the strength of the bridge increases at some extent but after a limit if the dimension exceeds it is the drawback behind the failure of the structure. Analysis has been carried out from actual design to six modified designs. The results such as total deformation, directional acceleration, directional velocity, normal stress, shear stress, equivalent stress, normal elastic strain and shear elastic strain are taken out for each modal of bridge pier from actual design to six modified designs are determined. Comparing the results of all modified design the total deformation value the least at model 5 and then after it again increases, so the analysis has been stopped here. Hence it is concluded that fifth model is suitable for bridge pier for seismic zone 2, 3, 4.

REFERENCES

[1] Parisa HosseiniaS, Hooman Ghasemibc, Mohammad Jalayera Andrzej S.Nowakc "Performance-based reliability analysis of bridge pier subjected to vehicular collision: Extremity and failure." Engineering Failure Analysis, Volume106, December 2019

[2] Lingkun Chen, Lizhong Jiang, Peng Liu "Seismic response analyses of high-speed railway bridge round-ended piers using global bridge model." International Journal of Materials and Product Technology,July 2012

[3] Rashmi R.Vanahalli, Dr. S.V.Itti "study on RCC bridge pier using ansys" ,International Journal of Engineering and Innovative Technology,August 2013

[4] Hayder Ala'a Hasan1, Ammar A. Abdul Rahman1 and Hani Aziz Ameen "Finite element analysis of reinforced concrete bridge pier subjected to seismic loading", American Journal of Scientific and Industrial Research,2012

[5] Amit Katkar1, Prof. P. M. Kulkarni "Parametric Study of Bridge Piers" International Research Journal of Engineering and Technology (IRJET), January 2018

[6] Digambar B. Patil, Prachi Sohoni and Rushikesh P. Jadhav "Evaluation of Assorted Profiles in Bridge Pier Exposed to Exciting Flood Loading", Jordan Journal of Civil Engineering, Volume 15, No. 4, 2021

[7] Rupen Goswami and C. V. R. Murty "seismic strength design provisions for reinforced concrete (RC) bridge piers" given in Indian codes in 2002.

[8] T.Subramani, B.Subha , T.Anbu , S.Sasitharan , M.Seetharaman "Design and Analysis of Flyover Bridge By Using SAP",

International Journal of Application or Innovation in Engineering & Management (IJAIEM), 2009

[9] Siva Avudaiappan, Kinson Prabu, Deban Selvaraj, Kiran Raja, Paul Oluwaseun Awoyera , and Erick,"Performance of Pier-to-Pier Cap Connections of Integral Bridges under Thermal and Seismic Loads ",Advances in Civil Engineering Volume 2021.

[10] Prasanna and Suresh Kumar,"Simulation of Flow Behaviour around Bridge Piers Using ANSYS – CFD." International Journal of Engineering Science Invention (IJESI), September 2018

[11] Reshma Babu and Jobil Varghese, "Dynamic Analysis of Metro Rail Supporting Structure.", International Journal of Engineering Science Invention (IJESI), April 2018

[12] Dhruv Saxena, "Computational Analysis for Predicting Scouring Around Cuboid Shaped Pier in a Stream", International Journal of Engineering Trends and Technology (IJETT), October 2017

[13] Premsai.T "Structural Analysis and Optimization of Bridge Pier Using ANSYS", International Journal of Engineering and Innovative Technology, March 2015