Seismic Analysis of Integral Bridges Research

Krishnendu Panikkavettil R, Gokul Raveendran K

Abstract - Seismic analysis study is considered as a significant parameter especially in case of dams, bridges etc. Integral bridges with its uniqueness to provide structural integrity in the absence of expansion joints and aesthetically pleasing appearance is widely progressing in construction and hence calls for a detailed seismic analysis. As the damage caused during the earthquake is principally concentrated in areas where expansion joints are provided So there has been an urge to build bridges devoid of these joints. Integral bridges are monolithically constructions where substructure and superstructure are connected with moment resisting connections. However, the main purpose of the study is to find out a new design methodology of the unique bridge structure and checking its validity analytically through ANSYS. The study hence focuses on the seismic performance of different span integral bridges by carrying out static analysis, modal analysis and response spectrum analysis in ANSYS2015. Further the effectiveness of bridge with respect to increase in span is studied. A simplified finite-element model of the East Logansport Bridge at West Virginia is used for the investigation. Static analysis modal analysis and response spectrum analysis is performed

Index Terms— Integral bridge, seismic analysis.

◆

IJSER

1 INTRODUCTION

A conventional bridge is usually made up of a girder, simply supported by a pair of abutments via a pair of movable and fixed bearings. Numerous conventional bridges have been seriously damaged or have been completely collapsed during many previous earthquakes. It is proven that thermal and time-dependent effects lead to volumetric changes in bridge super structures. The associated movements have historically been accommodated through the provision of expansion joints and bearings. Joints are susceptible to wear from traffic. Along with this issue, another undesirable outcome is the unsafe driving surface conditions associated with roadway discontinuity at the joint. High costs of expansion bearings are not justified on regards of their potential for failure that is brought on by increased friction due to corrosion or loss of lubrication. Moreover, high costs are associated with manufacturing, installation, and repair and maintenance of expansion joints and bearings. There has been an urge to develop a new type of bridge that is more earthquake resistant yet more cost-effective, here comes the significance of integral bridges(IB). Integral bridges are bridges where the super structure is continuous and connected monolithic-ally with the substructure with a moment resisting connection. They have superior seismic performance compared to the conventional bridges. Here superstructure and substructure move together to accommodate the require translation and rotation and act as a single structural unit. Integral bridges do not use any kind of expansion joints to allow for movement, and have been gaining worldwide popularity as a viable alternative.

2 METHODOLOGY

From the papers referred, a paper was selected to model the integral bridge East Logan Sport situated in West Virginia. The finite element software ANSYS 2015 was utilized. A 3D model of the above-mentioned bridge was created. The preciseness of the model was justified by validating the results given in the journal and that evaluated using the software. Advanced study was performed to acquire the design chart. Modal analysis was performed to find out the different mode shapes or possible modes of the failure. Response spectrum analysis was done to study about the dynamic stability of the bridge and the stresses and strains were analysed.

Having found that the unique system could effectively reduce responses in bridges, the study further extends work to check whether the results are reliable on any other bridge for a different span. Response spectrum and modal analysis in ANSYS2015 gave authentic results during the progress of work and helped to reach the desired conclusions. A new technology oriented post-tensioned bridge system was constructed at East Logan-sport in West Virginia in late 1995 known as East Logan-sport Bridge comprising of a 28.7 m clear span steel superstructure, with four 91-cm diameter galvanized steel tubes, as shown in Figs. 1 and 2. Were created of 13-mm thick steel plate. Each tube is of fabricated three segments of 6.1-m long attached with two segments approximately 5.3-m long. The superstructure is post tensioned with twelve 3.5-cm diameter poly-coated Dywidag rods, six for each couple of tubes. The deck incorporates precast concrete panels of 3.1-m long and 5.3-m wide, that simply recline on the steel tubes.

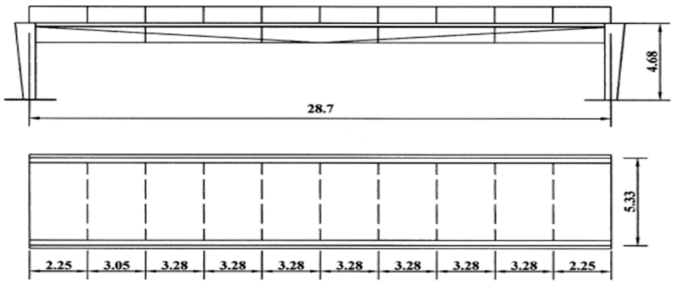

FIG 1: ELEVATION AND DECK PLAN OF BRIDGE

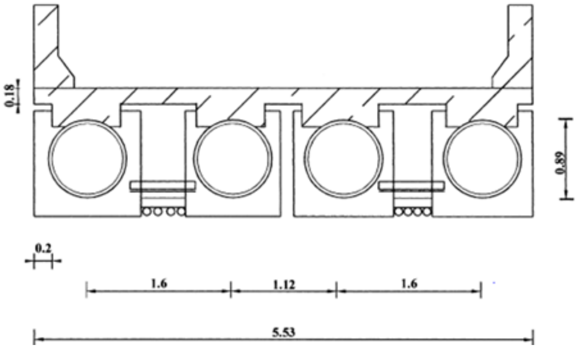

FIG.2 : SECTION DETAILS OF THE BRIDGE DECK

3 DESIGN METHODOLOGY OF THE SELECTED BRIDGE

Since the distinctive bridge structure selected does not have any iterative design procedure of its own and the literature regarding the bridge system is sparse a new methodology has been adopted through meticulous survey of existing literatures. The bridge deck was assumed to be a beam with fixed ends on both ends with the deck slab resting on four steel tubes of 13mm thickness. The thickness of the steel tubes was redesigned according to different span length.

The method of redesigning thickness was chosen because it was told in the journal that it was the moment of inertia of the steel tubes that increases the dynamic stability of the bridge. The post tensioning part in the bridge was not considered during the design it was kept as it is since not much of its details are given.

Initially self-weight of the concrete part of the bridge deck was calculated. Then the self-weight of the steel tubes was calculated. The moment of inertia of the bridge deck cross section was calculated. The deflections corresponding to the self-weight of the structure was calculated, the self-weight was assumed to be uniformly distributed load and the corresponding displacement was noted. The deflection due to the live load was also separately done and the total deflection was calculated. The manual designs were validated through ANSYS to check their accuracy.

Initially centroid of the section was calculated using the formula shown below the section was transformed to concrete using modular ratio

$\bar{y}$ = [A1Y1 + 4(A2Y2)]/(A1 +A2)
$\bar{y}$ = Centroid of the entire section
A1 = Area of the concrete section
A2 = Area of the steel tub

Now with the knowledge of parallel axis theorem moment of inertia of the transformed section was found out

Ixx = BD3/12 +(A1r2)
B = Width of the concrete section
D = Depth of the cross section

Since the steel tubes were assumed to be fixed steel beams. The deflections corresponding to live load and dead load are found out using the below mentioned formulae.IS800 has been used to obtain the deflection limit.

Δ = (W1L4/384EI)+(W2L3/192EI)
W1 = self weight of the bridge
W2 = weight of the truck
E = modulus of elasticity
I = moment of inertia of transformed section

Deflection limit is calculated as a check to ensure the calculated deflection fall within the limit

Δlimit =L/325

TABLE 1
RESULTS OF MANUAL AND ANALYTICAL DESIGN

Length (mm)	Thickness (mm)	Deflection (mm)	Deflection limit	Deflection/ Deflection limit	Deflection From ansys (mm)
30500	13	11.52	93.84	8.14	10.141
35500	15.7	13.51	109.23	8.13	13.35
40500	18.5	15.35	124.615	8.11	12.15
45500	21.4	17.09	140	8.11	14.735
50500	24.2	19.15	155.38	8.11	17.459
55500	27.1	21	170.76	8.11	19.34

4 MODAL ANALYSIS

Modal analysis calculates the frequency modes or natural frequencies of s given system but not necessarily its full-time history response to a given input. The natural frequency of a system is dependent only on the stiffness of the structure and mass which participates with the structure including the self-weight. It is not dependent on the load function. Hence it is good to know that the modal frequencies Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Modal analysis is the field of measuring and analysing the dynamic response of structures. In structural engineering, modal analysis uses the overall mass and stiffness of a structure to find the various periods at which it will naturally resonate. These periods of vibration are very important to note in earthquake engineering, as it is imperative that a structure's natural frequency should not match the frequency of expected earthquakes in that region. Modal analysis is important in structures such as bridges where the engineer should attempt to keep the natural frequencies away from the frequencies of other vibrational loads. Other natural excitation frequencies may exist and may excite a bridge's natural modes. The goal of modal analysis in structural engineering is to determine the fundamental time period and mode shapes of a structure during free vibration. It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analyzed can have arbitrary shape and the results of the calculations are acceptable. Modal analysis of integral bridge modelled in ANSYS2015 yielded a natural frequency which decreased as the span length of the bridge increased. The below given figure are mode shapes of the bridge with respect to different span.

Modal analysis was conducted in 5 different cases and it was observed that the bridge displays five transverse among six mode shapes obtained and a torsional mode too. The first mode shape in all the cases exhibits transverse mode. The occurrence of torsional mode is lower, this is mainly due to the rigidity of the structure. The first mode shape obtained is similar in all the cases and the natural frequency shows a decreas-

IJSER

ing trend as the span length increases, this phenomenon is shown in the graph depicted above. The difference in natural frequency from the validated 30.5m span and the remaining cases is mainly due to the embedding of the post tensioning rods in concrete to ensure structural integrity.

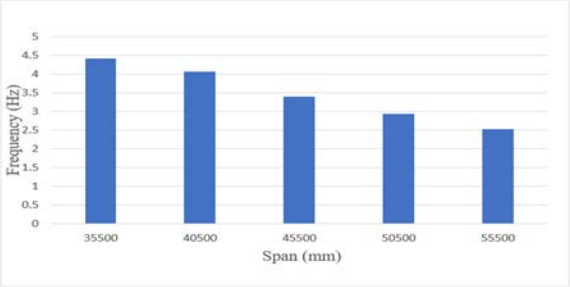

FIG3: FREQUENCY VARIATION WITH SPAN

5 STATIC ANALYSIS

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes) Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

IJSER

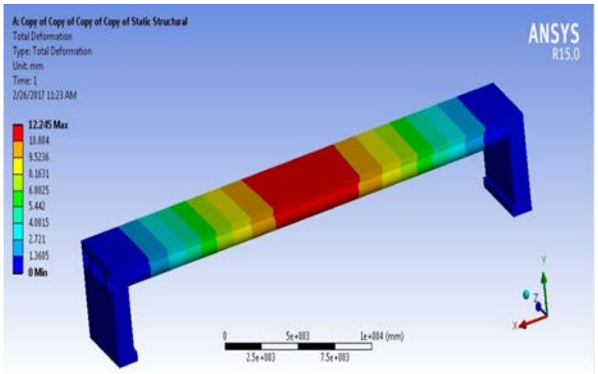

FIG 4: STATIC ANALYSIS CONDUCTED FOR 30.5M SPAN

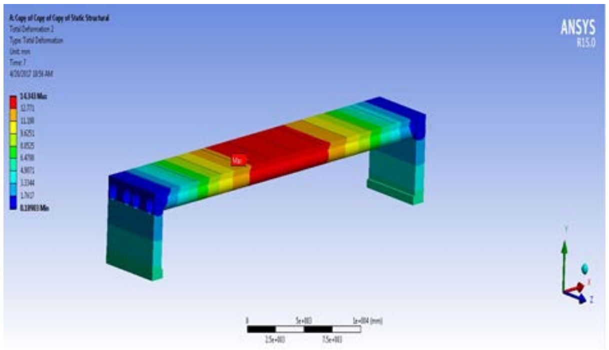

FIG.5 : STATIC ANALYSIS CONDUCTED FOR 35.5M SPAN

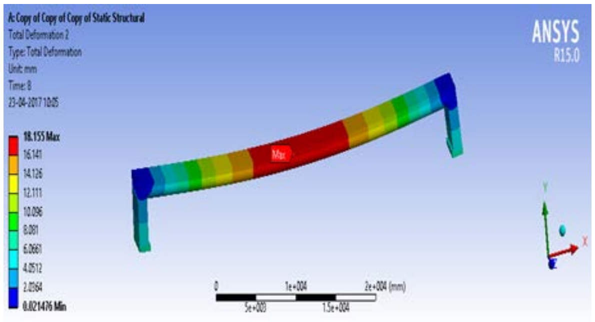

FIG.6 : STATIC ANALYSIS CONDUCTED FOR 40.5M SPAN

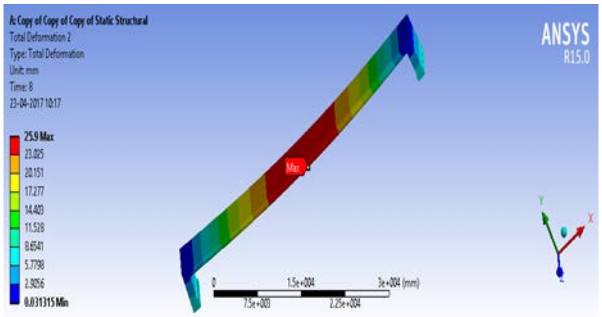

FIG.7 : STATIC ANALYSIS CONDUCTED FOR 45.5M SPAN

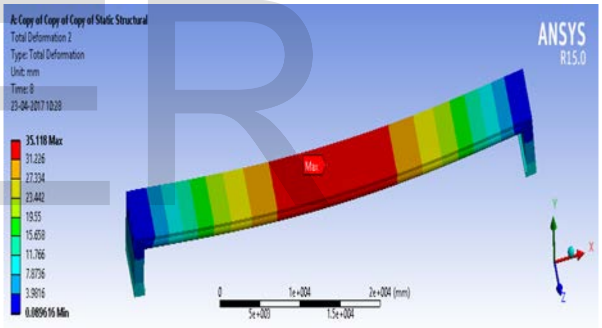

FIG.8 : STATIC ANALYSIS CONDUCTED FOR 50.5M SPAN

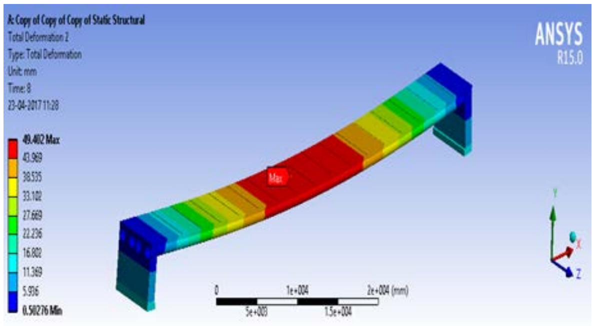

FIG.9 : STATIC ANALYSIS CONDUCTED FOR 55.5M SPAN

TABLE 2

DEFORMATIONS FOR DIFFERENT SPAN

Span (mm)	Thickness (mm)	Deformation (mm)
30500	13	12.3
35500	15.7	14
40500	18.5	18
45500	21.4	25.9
50500	24.2	35
55500	27.1	49

6 RESPONSE SPECTRUM ANALYSIS

To perform the seismic analysis and design of a structure to be built at a particular location, the actual time history record is required. However, it is not possible to have such records at each location. Further, the seismic analysis of structures cannot be carried out simply based on the peak value of the ground acceleration as the response of the structure depend upon the frequency content of ground motion and its own dynamic properties. To overcome the above difficulties, earthquake response spectrum is the most popular tool in the seismic analysis of structures. There are computational advantages in using the response spectrum method of seismic analysis for prediction of displacements and member forces in structural systems. The method involves the calculation of only the maximum values of the displacements and member forces in each mode of vibration using smooth design spectra that are the average of several earthquake motions. This chapter deals with response spectrum method and its application to various types of the structures. The codal provisions as per IS:1893 (Part 1)-2002 code for response spectrum analysis of multi-story building is also summarized. 4.2 Response Spectra Response spectra are curves plotted between maximum response of SDOF system subjected to specified earthquake ground motion and its time period (or frequency). Response spectrum can be interpreted as the locus of maximum response of a SDOF system for given damping ratio. Response spectra thus helps in obtaining the peak structural responses under linear range, which can be used for obtaining lateral forces developed in structure due to earthquake thus facilitates in earthquake-resistant design of structures. Usually response of a SDOF system is determined by time domain or frequency domain analysis, and for a given time period of system, maximum response is picked. This process is continued for all range of possible time periods of SDOF system. Final plot with system time period on x-axis and response quantity on y-axis is the required response spectra 103 pertaining to specified damping ratio and input ground motion. Same process is carried out with different damping ratios to obtain overall response spectra. [NPTEL].

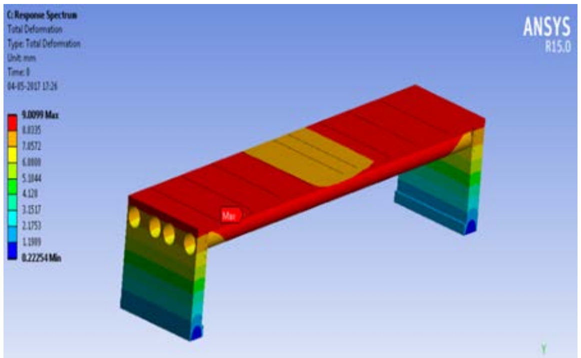

FIG.11 :RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 30.5M SPAN

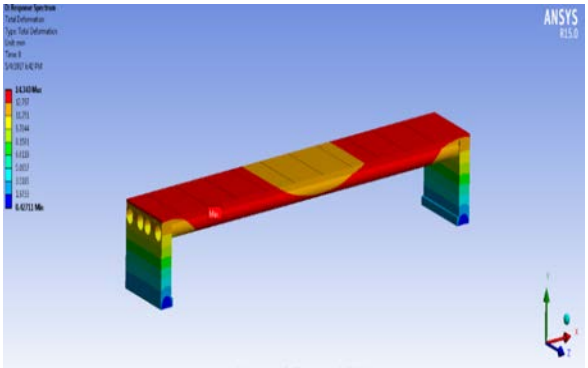

FIG. 12:RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 35.5M SPAN

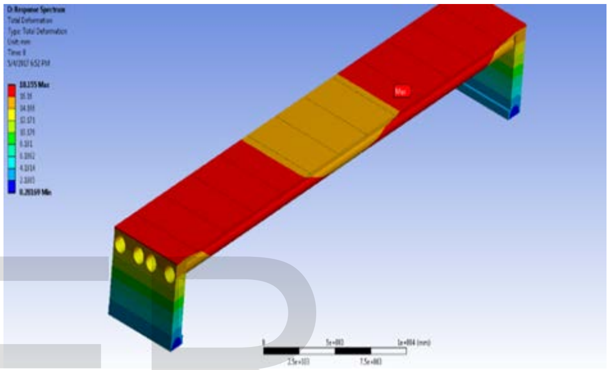

FIG. 13: RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 40.5M SPAN

IJSER

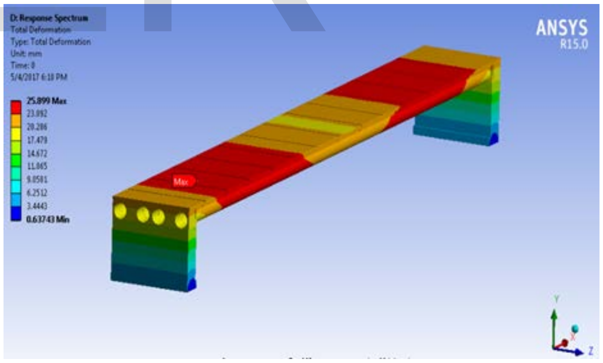

FIG. 14 : RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 45.5M SPAN

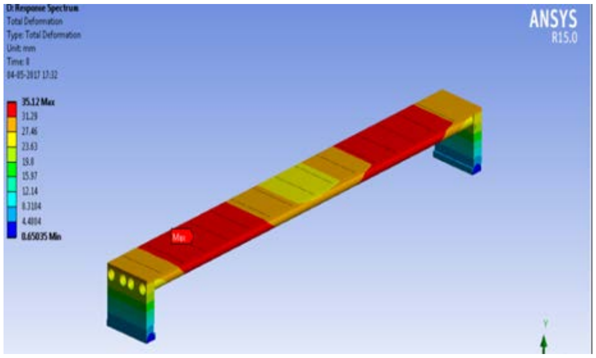

FIG.15 : RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 50.5M SPAN

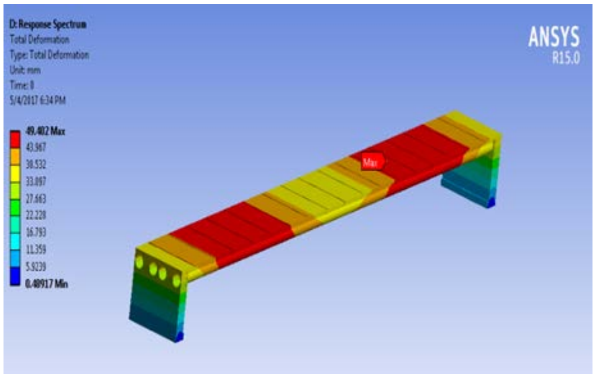

FIG.16: RESPONSE SPECTRUM ANALYSIS CONDUCTED FOR 55.5M SPAN

TABLE 3
RESULTS OF RESPONSE SPECTRUM ANALYSIS

Span (mm)	Thickness (mm)	Deformation (mm)
30500	13	12.3
35500	15.7	14
40500	18.5	18
45500	21.4	25.9
50500	24.2	35
55500	27.1	49

Response spectrum analysis was performed for five different cases the results from response spectrum signifies that in all the five cases the maximum deformation is concentrated in the same place for all the cases. the deformation obtained by performing response spectrum seems to vary significant as the length increases up to double the span.

7 REVALIDATION PROCEDURE

This procedure was adopted so as to test the correctness of the design chart obtained. Hence forth a random span length was chosen from the design chart and the corresponding deformation was found out from the graph. For the convenience, a span of 42m was selected from the graph and its deformation was noted as 19mm.The bridge for the span was also manually designed and the obtained total deformation was found to be 18.279mm which is almost equal to that validated in ANSYS.

This procedure was adopted so as to test the correctness of the design chart obtained. Hence forth a random span length was chosen from the design chart and the corresponding deformation was found out from the graph. For the convenience, a span of 42m was selected from the graph and its deformation was noted as 19mm.The bridge for the span was also manually designed and the obtained total deformation was found to be 18.279mm which is almost equal to that validated in ANSYS.

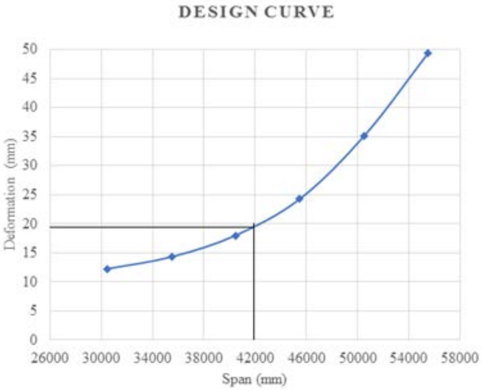

FIG.17: DESIGN CURVE FROM RESPONSE SPECTRUM ANALYSIS

1.Find out the centroid of the section

ȳ = [A1Y1 + 4m(A2Y2)]/(A1 +(4mA2))
ȳ = Centroid of the entire section
A1 = Area of the concrete section
A2 = Area of the steel tubes
m = modular ratio
A1 = 1642410 mm2
A2 = 53978.2014 mm2
Y1 = 148.5mm
Y2 = 752mm
ȳ = 865.175mm

2.Find out the moment of inertia of the entire section

Ixx =BD3/12+(A1r12)+(((πxd14/64)-(πxd24/64))+mA2r22)
B = Width of the concrete section
D = Depth of the cross section
Ixx = 12072945308+(1642410x513624.1833) +(3.05x1011)
= 1.182 x 1012 mm4

3.Find out the self-weight of the section

W1 = self weight of concrete and self-weight of steel
= 45.613 KN/m

4.Find out the deflection

Δ = (W1L4/384EI)+(W2L3/192EI)
W1 = self weight of the bridge deck
= 45.613KN/m
= 1368.39KN
W2 = weight of the truck
= 267.62KN
E = modulus of elasticity
I = moment of inertia of the transformed section
Δ = 16.0004mm

5.Deflection limit is calculated as a check to ensure the calculated deflection fall within the limit

Δlimit = L/325
= 129.23mm

6.To obtain the thickness of tube Δlimit/ Δ is calculated it

IJSER

should be 8 and the corresponding value of thickness is obtained from the design made manually 19.3mm is obtained.

Now the structure is modelled as a fixed beam in ANSYS. Later static analysis, modal analysis and response spectrum analysis was performed to check the correctness of the design methodology. The results are summarized in the table given below.

TABLE 4

REVALIDATION DETAILS

Span length (mm)	Deformation from manual design (mm)	Deformation from ansys (mm)	Deformation from static analysis (mm)	Natural frequency from modal analysis (mm)	Deformation from response spectrum (mm)
42000	16.0004	15.656	18.279	3.9628	18.3

The below shown figure signifies that the deformation from manual design and ANSYS are almost same. The deck of the bridge is assumed to be as a fixed beam by providing fixed supports in the ends. The deformation pattern obtained in the randomly selected span is same as the previous cases.

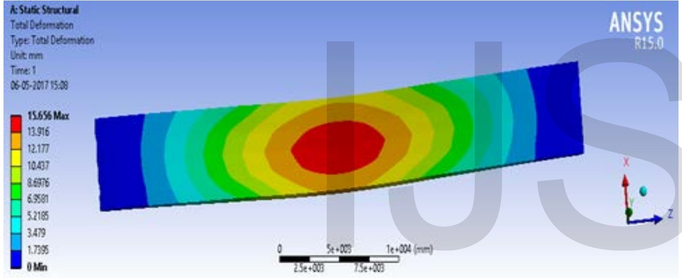

FIG.17 :TOTAL DEFORMATION OF BRIDGE ANALYZED AS A BEAM

Static analysis is performed for the bridge. Maximum deformation was seen to be concentrated in the middle portion of the bridge. The bridge has deformed up to a value of 18. 279mm.The deformation pattern is as shown in the below given figure.

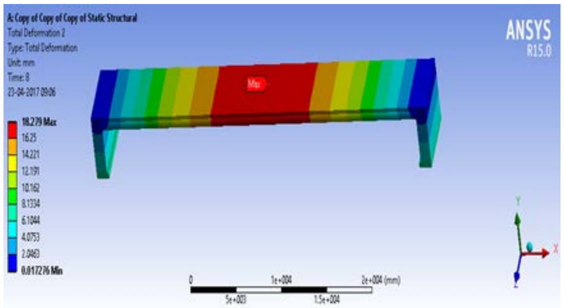

FIG.18 :TOTAL DEFORMATION OF BRIDGE AFTER DOING STATIC ANALYSIS

Modal analysis has been performed for this span and the six mode shapes were computed this is shown in the figures given below. The mode shapes are mainly of transverse mode and one torsional mode. The mode shapes are like the other span and the shape of the first mode shape is similar in all the cases which is a transverse mode. The natural frequency of the bridge is found to be 3.9628Hz.

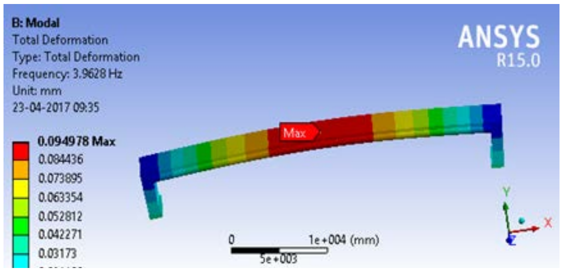

FIG.19 : NATURAL FREQUENCY OF THE BRIDGE

Response spectrum analysis was carried out to understand the behaviour of the bridge under the situation of an earthquake and to know about the maximum deformation the bridge can undergo during the ground shaking.

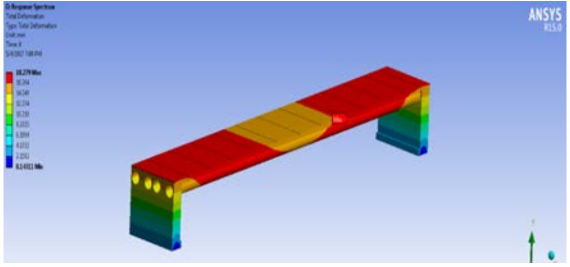

FIG. 20 : TOTAL DEFORMATION OF THE BRIDGE AFTER DOING RESPONSE SPECTRUM ANALYSIS

8 RESULTS

The investigations conducted on the seismic analysis of integral bridges has made a detailed study on integral bridge structures from the literature reviews. A 3D finite element model was successfully made for the static analysis, modal analysis and response spectrum analysis.

- ➢ The exclusive bridge structure selected has several advantages over conventional bridge.
    - o The bridge deck composed of steel tubes, post tensioning rods and concrete panels alone which increases the eases of construction.
    - o As the concrete panels are simply placed on the steel tubes its easier for maintenance
    - o Seat type concrete abutments used in the structure increases the seismic resistance of the bridge.

- ➢ Having obtained knowledge about different layouts of an integral structure, the first part of the study was basically meant to find a design procedure for the selected bridge.
- ➢ The adopted design procedure can be extended to any length and its validity is checked by modelling he bridge as a fixed beam in ANSYS
- ➢ Static analysis was performed to understand maximum possible deformation that the bridge can undergo
- ➢ Modal analysis is carried out to understand the natural frequency of the bridge for different spans, it is noteworthy that the findings in modal analysis are fa-

IJSER

vourable since the natural frequency seems to decrease as the span increases.

- Large differences in deformations exist in the seismic response of IBs during response spectrum analysis in the last three cases this is mainly due to large increase in length of the bridge without any supports in between.
- In IBs, the monolithic construction of the seat type abutments with the superstructure engages both abutments in resisting the seismic force in the longitudinal direction.
- The design chart made helps to extend the design of the bridge to any span length and the validity of the chart is also tested by ANSYS.

9 FUTURE SCOPE

Future scope of the study aims in varying post tensioning force and designing the post tensioning required different span length Since only sixty percent post tensioning is provided in all the five cases and the design of post tensioning has not been changed from the validated model further studies can be done to redesign the post tensioning tendons to considerably reduce the deformation.

REFERENCES

[1] Spyrakos CC. Assessment of SSI on the longitudinal seismic response of short span bridges. *Engineering Structures* 1990;12:60–6.

[2] AASHTO. (2010). AASHTO LRFD bridge design specifications, *4th Ed. American Association of State Highway and Transportation Officials, Washington, D.C.*

[3] Spyrakos CC. Seismic behaviour of post tensioned integral bridges including soil structure interaction. *Soil Dynamics. Earthquake Engineering* 23 (2003) 53–6

[4] Spyrakos CC. Seismic behaviour of bridge piers including soil structure interaction. *Comput Struct* 1992;4(2):373–84.

[5] Spyrakos CC. Dynamic load factor in composite highway bridge. *Proceedings of Structures Under Shock and Highway* Impact V.

[6] Arsoy, S., Barker, R., and Duncan, J. (1999). “The behaviour of integral abutment bridges.” *Report No. FHWA/VTRC 00-CR3, Virginian Transportation Research Council*

[7] Burke, M. P. Jr. (2009). “Integral and semi-integral bridges.” *Integral Bridges 1st ed., Wiley-Blackwell, Iowa, 1-20*

[8] Roman, E., Khodair, Y., and Hassiotis, S. (2002). “Design details of integral bridges”, *Stevens Institute of Technology, Hoboken, N.J.*

[9] Rodriguez, L. (2012). “Temperature effects on integral abutment bridges for the long-term bridge performance”, *Utah State University, Logan.*

[10] Olson, S., Long, J., Hansen, J., Renekis, D., and LaFave, J. (2009). “Modification of IDOT integral abutment design limitations and details”, Illinois Centre for Transportation, Research Report ICT-09-054, University of Illinois at Urbana-Champaign, Illinois.

[11] Khodair,Y., Hassiotis, S. “Numerical and experimental analyses of an integral bridge”, *International Journal of Advanced Structural Engineering* 2013, 5:1

[12] Dicleli, M. and Albhaisi, S. “Analytical formulation of maximum length limits of integral bridges on cohesive soils”, *Canadian Journal of Civil Engineering 2011*

[13] Barker, R.M., Duncan, J.M.K., Rojiani, K.B., Ooi, P.S.K, Kim, S.G., 1991. “Manuals for the design of bridge foundations”. *Transportation Research Board, National Research Council, Washington, D.C., U.S.A. NCHRP Report 343.*

[14] Burdette, E.G., Ingram, E.E., Goodpasture, D.W., and Deatherage, “Behaviour of concrete integral abutments”. *Concrete International* July: 59–63.

[15] Dicleli, M., and Albhaisi, S.M. 2004a. “Effect of cyclic thermal loading on the performance of steel H-piles in integral bridges with stub-abutments”. *Journal of Constructional Steel Research,* 60(2): 161–182.

[16] Greimann, L. F., Yang, P. S., and Wolde-Tinsae, A. M. “Nonlinear analysis of integral abutment bridges.” *J. Struct. Eng., 1986.*

[17] Civian, S. A., Bonczar, C., Breña, S. F., DeJong, J., and Crovo, D. _2007_. “Integral abutment bridge behavior: Parametric analysis of a Massachusetts bridge.” *Bridge Eng.,* 64–71.

[18] Fennema, J. L., Laman, J. A., and Linzell, D. G. _2005_. “Predicted and measured response of an integral abutment bridge.” *J. Bridge Eng.*,666–677.

[19] Paul, M. D., Laman, J. A., and Linzell, D. G. _2005_. “Thermally Induced superstructure stresses in prestressed girder integral abutment bridges.” *Transportation Research Record. CD 11-S, Transportation Research Board, Washington, D.C.*, 287–297.

[20] ANSYS Inc. _2005_. ANSYS release 10.0, *ANSYS University Advanced, Canonsburg, Pa.*

IJSER