International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

Analysis of Seismic Behaviour of a Composite Bridge using ANSYS

Jayakrishnan. T J
M Tech Student
Sree Narayana Institute of Technology
Theppupara, Adoor, Kerala

Lekshmi Priya. R
Assistant Professor, Department of Civil Engineering
Sree Narayana Institute of Technology
Theppupara, Adoor, Kerala

***Abstract*— This paper summarizes the research work on the seismic behavior analysis of composite bridge without cross girders using ANSYS. Composite structures have many useful applications in the field of aerospace, civil infrastructure and construction. Seismic behaviour of composite bridges is studied by response spectrum method. Response of the structure depends on the geometry, material, configuration, response spectrum selected and construction details of the system. In this paper, finite element tool ANSYS Workbench is used for the study of seismic behavior of composite bridge**

Keywords*— Bridges, Composite Bridges, Modal Analysis, Response Spectrum Method

I. INTRODUCTION

Bridge is a structure that is used to cross some form of barrier, making it easier to get from one place to another. Barriers, such as rivers, have always caused problems to travellers and traders who wanted to take the shortest, quickest and safest route to complete their journeys. Composite bridges are that type of bridges combines of more than one material such as concrete, steel, timber or masonry in any combination. In recent days, the common usage of composite construction is meaning either steel -concrete construction or in-situ concrete or precast concrete bridges. Steel concrete composite bridges became commonly used all over the world, because of their aesthetic appearance and economically. Their capability of covering long spans without requiring of false work make them more desirable in metropolitan areas. In addition, there is a significant difference in the weight of steel plate girder bridges and concrete box girder bridges have an important effect on the seismic design.

The inertia forces generated by an earthquake in steel plate are significantly less than the forces generated by concrete bridges. Steel bridge superstructures are susceptible to damage even during low-to-moderate shaking, and appear to be more fragile than structural concrete superstructures in this regard if not designed properly. Typical damage includes unseated girders and failures in connections, bearings, cross-frames, and expansion joints. Steel plate girder bridges have generally suffered minor/moderate damage in past earthquakes compared to the significant damage suffered by structural concrete bridges. Figures 1 shows the composite bridges

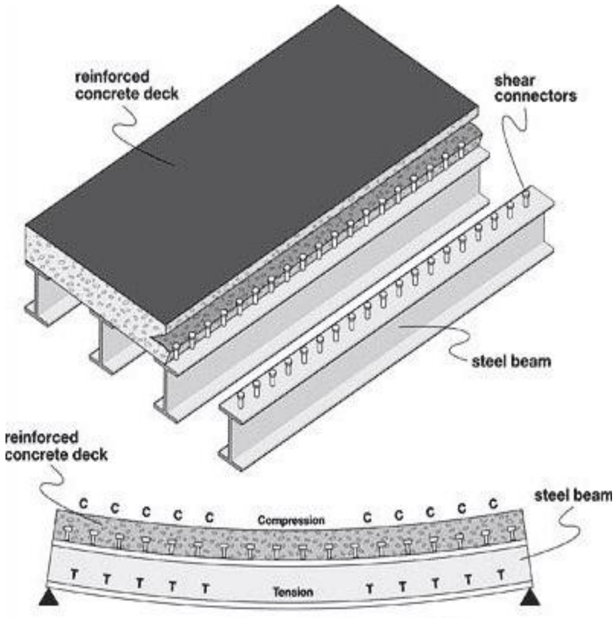

Fig. 1. Composite Bridge

II. FEATURES OF COMPOSITE BRIDGE

A. Modelling and loading of conventional bridge

It is a single span roadway bridge of 25m length. Deck slab is of 9.9m width and 300mm thickness. Footpath of 1.2m is provided on both sides. Two cases are considered. In the first case 5 longitudinal girders are provided at a spacing of 2m c/c distance and in the next case 3 girders are provided at a spacing of 3.75m. Material used are M30 grade concrete and Fe415 grade steel for both cases. Composite bridge is modeled as solid 186, solid 187, surf154, targe170, conta174 element in ANSYS Workbench. It is a three dimensional twenty nodded solid element. This solid has the capacity of crushing in compression and cracking in tension. And also it has special features like plasticity, creep, cracking, crushing, large deflection and large strain. Translations in the nodal X, Y and Z direction.

III. SOFTWARE USED

A. CATIA V6

CATIA (an acronym of computer aided three-dimensional interactive application,) is a multi-platform software suite for computer aided design (CAD) developed by the French company Dassault Systèmes. In this paper, the composite Bridge is modelled in CATIA and it is shown in Fig.2

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

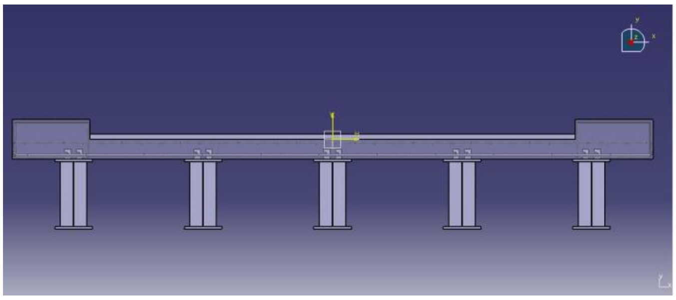

Fig. 2. Bridge modeled in CATIA

B. ANSYS WORKBENCH

ANSYS stands for Analysis Systems. It offers a suite of engineering simulation software for engineers and designers. ANSYS virtually analyze how their products work in real world environment, at an early stage of product design. It is a comprehensive FEA tool for structural analysis, including linear, nonlinear, dynamic, hydrodynamic and explicit studies. It provides a complete set of elements behavior, material models and equation solvers for a wide range of mechanical design problems. In this paper the bridge model is imported to ANSYS Workbench. It is shown in Fig.3. ANSYS Workbench, which is written for high level compatibility with especially personal computer, is more than an interface and anybody who has an ANSYS license can work with ANSYS Workbench. Workbench provides a single interface to all of ANSYS" tools. The goal is to provide a single platform that allows users to take advantage of a simpler, schematic style approach to build simulation tasks. It's driven by building up and connecting different building blocks. Each of these blocks
allows us to take inputs and outputs from one stage and feed them into the next or indeed, multiple processes.

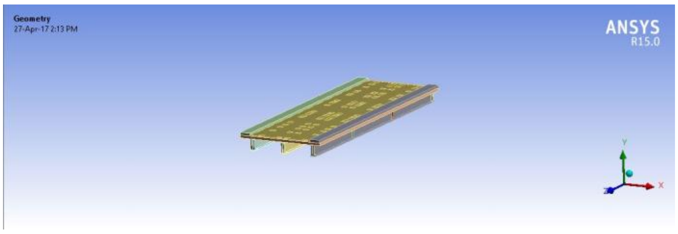

Fig.3. Imported model in ANSYS

IV. MODAL ANALYSIS OF COMPOSITE BRIDGE

Vibration can be an undesired side effect of poor product design or the environment in which the product is operating. It can have a big impact on durability and fatigue, leading to a shorter service life. We need to understand how our designs will respond to vibrations from phenomena such as brake squeal, earthquakes, transport, and acoustic and harmonic loads to predict the behavior of products and components. ANSYS Workbench simulations can provide this understanding and helps to overcome toughest vibration challenges.

A. Material Properties

The Engineering Data Manager provides a powerful tool for defining, organizing, and storing material properties. Material properties of structural steel are already available in ANSYS engineering data. Material properties of steel, concrete and bitumen are as shown in Table 1.

TABLE 1. Material Properties

SL.NO	Material	Young's modulus (pa)	Poisson's ratio	Density (Kg/m^3)
1	Concrete	4.5E+10	0.18	2500
2	Steel	2E+11	0.3	7850
3	Bitumen	6400	0.4	2243

B. Section Properties

The thickness provided for wearing coat is 80mm. Based on the codal provisions, Girder dimensions of the flange plate is 580mm x 40mm, web plate is 1000mm x 10mm for case1, and girder dimensions of flange plate is 600mm x 40mm, web plate is 1200mm x 30mm for case 2.

C. Meshing

It is important to correctly select the mesh size and layout in finite element analysis. A good mesh means accurate results with better convergence but also has time consideration. A very fine mesh model will always provide accurate results but will require excessive computer time. The nodes and elements representing the geometry model make up the mesh. A default mesh is automatically generated during initiation of the solution. The user can generate the mesh prior to solving to verify mesh control settings. A finer mesh produces more precise answers but also increases CPU time and memory requirements. In this analysis, suitable numbers of elements were carefully chosen for the models based on convergence studies in order to obtain accurate results without excessive use of computer time. Figure 4 and 5 shows the meshing of model

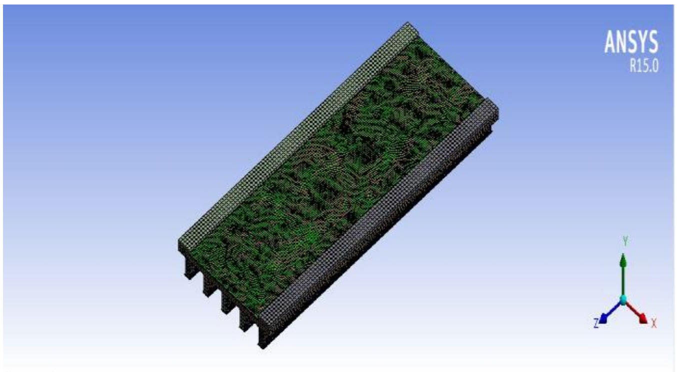

Fig. 4. Meshed Model of case1

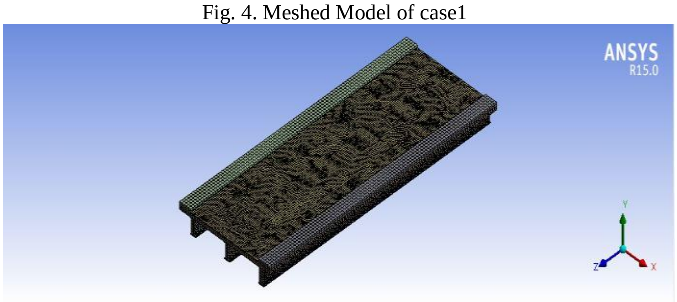

Fig. 5. Meshed Model of case2

D. Results and Conclusions

Modal analysis is done and different modes of vibration and fundamental frequencies are determined. In this analysis, first six modes of vibrations at different frequencies are considered for each cases. Figure 6 and 7 shows the maximum deformation of bridges in each cases.

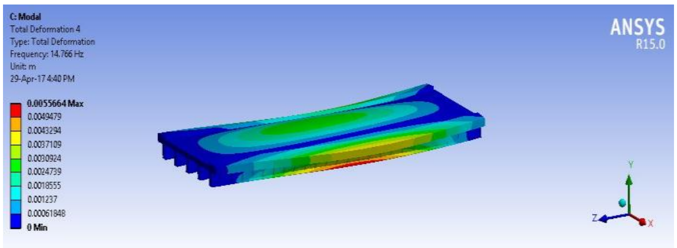

Fig. 6. Total Deformation of case1

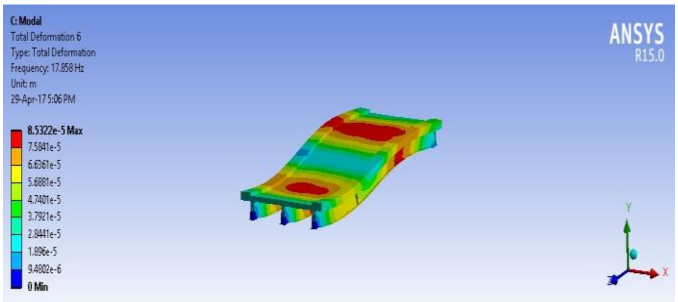

Fig. 7. Total Deformation of case2

After Modal analysis, total deformation values of all the two bridge structures are compared and as shown in table 2. In case of 3 Girder Bridge and 5 Girder Bridge, the 3 Girder bridge show less deformation than 5 Girder Bridge. That is, 3 Girder Bridge shows better performance than 5 Girder Bridge

TABLE 2. Comparison of Results

BRIDGE WITH 5 GIRDERS		BRIDGE WITH 3 GIRDERS	
Natural Frequency(Hz)	Total Deformation(m)	Natural Frequency(Hz	Total Deformation(m)
4.602	$2.60*10^{*3}$	6.067	$9.66*10^{*5}$
6.655	$4.07*10^{*3}$	6.183	$8.15*10^{*5}$
8.741	$2.40*10^{*3}$	7.650	$12.67*10^{*5}$
14.766	$5.56*10^{*3}$	13.713	$11.23*10^{*5}$
15.228	$2.93*10^{*3}$	14.996	$28.82*10^{*5}$
15.586	$3.11*10^{*3}$	17.858	$8.53*10^{*5}$

V. RESPONSE SPECTRUM ANALYSIS OF COMPOSITE BRIDGE

A. General

A response spectrum is simply a plot of the peak or steady-state response (displacement, velocity or acceleration) of a series of oscillators of varying natural frequency that are forced into motion by the same base vibration or shock. Response spectra can also be used in assessing the response of linear systems with multiple modes of oscillation (multi-degree of freedom systems), although they are only accurate for low levels of damping. Modal analysis is performed to identify the modes, and the response in that mode can be picked from the response spectrum. These peak responses are then combined to estimate a total response. A typical combination method is the square root of the sum of the squares (SRSS) if the modal frequencies are not close. The result is typically different from that which would be calculated directly from an input, since phase information is lost in the process of generating the response spectrum.

B. Spectrum Details and Results

Here the Ahmedabad frequency response spectrum is selected for the seismic analysis of bridge, it is in Zone III. Response Spectrum analysis is done and total deformation, equivalent stress and the normal elastic strain are determined and it is shown in table 3.

TABLE 3. Comparison of Results

5 GIRDER BRIDGE			3 GIRDER BRIDGE		
Deformation (m)	Equivalent Stress(Pa)	Strain	Deformation (m)	Equivalent Stress(Pa)	Strain
$5.533*10^{-5}$	$4.23*10^{6}$	$5.658*10^{-6}$	$11.70*10^{-5}$	$1*10^{7}$	$1.1*10^{-5}$

VI. CONCLUSIONS

Following are the findings from the study

I. From the Modal Analysis, it is found that 3 Girder bridge suffers less deformation than 5 Girder Bridges.

II. On Response Spectrum Analysis, the deformation value of 3 Girder Bridge is slightly higher than 5 girder bridge is seen.

III. The equivalent stresses formed in both the bridges are observed to be within the permissible limit.

ACKNOWLEDGMENT

The author(s) wish to express their gratitude to Dr. P.G. Bhaskaran Nair, PG. Dean, Sree Narayana Institute of Technology, Adoor for his valuable suggestions, encouragement and motivation. Above all we thank God Almighty for His grace throughout the work.

REFERENCES

[1] Muhammad Hasan S, Sakthieswaran N (2016), “Dynamic Response of Skewed Multi-Span Steel Composite Bridge”, International Research Journal of Engineering and Technology (IRJET), Volume: 03 Issue: 04, 2316-2320.

[2] Praveen Singh Bhil (2015), “Vibration Analysis Of Deck Slab Bridge”, International Research Journal of Engineering and Technology (IRJET), Volume: 02 Issue: 04, 667-670.

[3] Ravi kiran, Avinash A. SrinivasKiran (2014), “Abutment-pile-soil interaction of a psc bridge under seismic loading”, International Journal of Research in Engineering and Technology , Volume: 03 Special Issue: 03, pp856-861.

[4] Johnson Victor, D., Essentials of Bridge Engineering (Fifth Edition), Oxford & IBH Publishing Co. Pvt. Ltd., New Delhi, 2001, p.260.

[5] Krishna Raju, N., Design of Bridges (Fourth Edition), Tata McGraw Hill Publishing Co. Ltd., New Delhi, 2007, p.183.

[6] Jagadeesh, T.R., Design of Bridge Structures (Second Edition), PHI Learning Private Limited., New Delhi, 2016, p.203.