International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 07 | July 2021 www.irjet.net p-ISSN: 2395-0072

ANALYSIS OF A PRESTRESSED BRIDGE DECK IN SEISMIC ZONE BY STRUCTURAL RESPONSES USING FUSION 360 AND WITH FEA SOFTWARE

K.Muthupiriya[1], Dr.G.Lavanya[2]

[1]Post Graduate Student, Department of Structural Engineering, Anna University Regional Campus, Madurai
[2]Assistant Professor, University College of Engineering, Ramanathapuram

---

Abstract – *Now a days, there are many technical varieties of modern concrete, but historic buildings and bridges generally used three main types: Plain or unreinforced concrete, reinforced concrete, and prestressed concrete. Even reinforced concrete has limited capacity to span distances before cracking and failing under stress, a properly engineered prestressed-concrete beam can span longer distances than a reinforced-concrete beam which is thinner, lighter in weight, and uses less concrete without cracking or breaking. This paper provides an opportunity to implement the knowledge gained through the concrete design course on a real-world situation. The problem statement requires group members to divide the tasks such as planning paper schedule, software analysis and preparation of design. The first task is a bridge design with pre-tensioned bridge girders and the second task involves the analysis using Fusion 360 and Ansys. The design and analysis of the given structure analyzed with static and dynamic method with the application of loads on the bridge and obtaining the responses from the result.*

Key Words: Bridges, 3D modeling, Bridge model, dynamic responses**.**

1. INTRODUCTION

Bridge construction today has achieved a worldwide level of importance. Bridges are the key elements in any road network and gaining popularity in bridge engineering because of its better stability, serviceability, economy, aesthetic appearance and structural efficiency. In construction industry Concrete is the material that is used mostly in different forms as different structural components. Bridge structures have traditionally been designed with the primary objective of avoiding failure under static loads. The response of bridge structures to static loading can be determined quite satisfactorily by any of a number of classical analysis techniques. The response to dynamic loading due to moving vehicles, however, is not as easy to predict. It has now become the choice method to analyze concrete structural components. The use of computer software to model these elements is much faster, and extremely cost-effective to fully understand the capabilities of finite element.

1.1 Basic concept of pre stressing

In reinforced concrete members, the pre stress is commonly introduced by tensioning the steel reinforcement. Within certain limits, a permanent dead-load may be counteracted by increasing the eccentricity of the pre stressing force in a prestressed structural element, thus effecting savings in the use of materials. Prestressed concrete members possess improved resistance to shearing forces, due to the effect of compressive pre stress, which reduces the principal tensile stress. The use of curved cables, particularly in long-span members, helps to reduce the shear forces developed at the support sections.

1.2 Type of pre stressing systems

Pre stressing System can be classified by two basic methods viz.,

1. Pre-Tensioning
2. Post-Tensioning

Pre-Tensioning is a method where Pre stressing Steels are tensioned, prior to casting of concrete, against two rigid abutments whereas in Post-Tensioning is a method where Pre stressing Steels are stressed after casting of concrete to attains its preliminary strength. In the long-span range, prestressed concrete is generally more economical than reinforced concrete and steel. The minimum grade of concrete in pre stressing technique is M40 for pre tensioning whereas M35 for post tensioning.

1.2 Objectives

- To ensure safety of structure from external loads.
- To design a prestressed bridge using Fusion 360 software for modeling and Ansys software for analyzing.
- To perform analysis for the bridge with removal of critical columns fully and partially.
- To analyze the structure in economical way.
- To determine the potential for the structure through the static and dynamic loads to find out the responses by applying parameters in the bridge structure.

1.2 Scope of the study

- The scope of this study is limited to the determination of the structural static properties such as deflections and stress distributions.
- For that pre stressed bridge structure is taken for the analysis.
- The Ansys workbench is used as a tool in analyzing the structure with finite element analyzing method whereas Fusion 360 is used for modeling the structure.
- The pre stressed concrete beam is modeled as simply supported.

2. METHODOLOGY

In this study, analysis of bridge deck is carried out for both static and dynamic conditions. A 3D model is prepared for the bridge deck using Fusion 360 and analyzed using Ansys. The steps involved in the study are as follows:

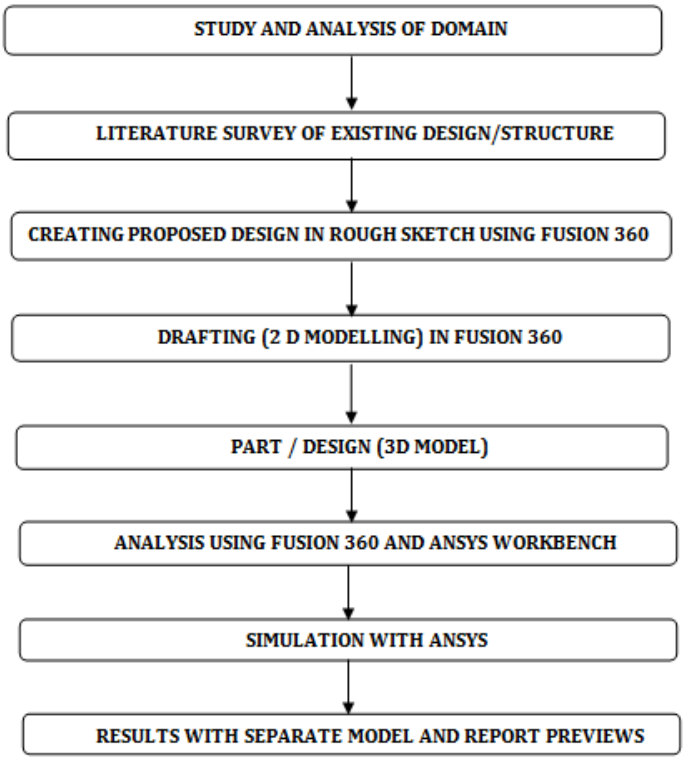

Fig -1: Steps involved

2.1 Layout and models

The 2 – D view of the bridge deck model with sectional view using Fusion 360 and 3-D exploded view along with final assembled view modelled in Ansys are given below.

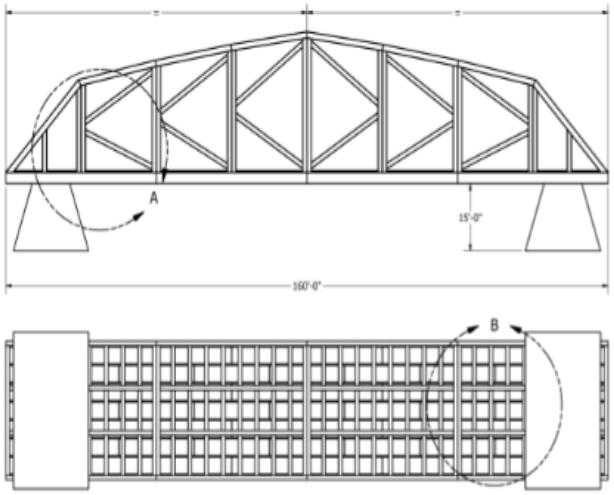

Fig -2: 2D View of the bride deck model

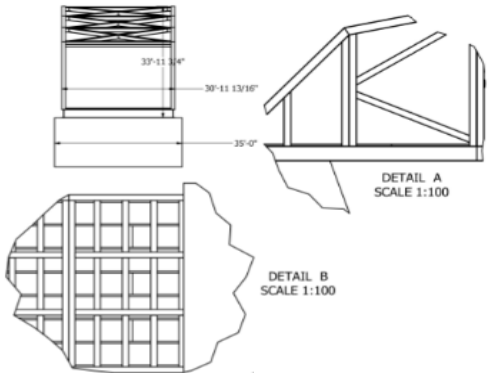

Fig -3: Sectional view of the model

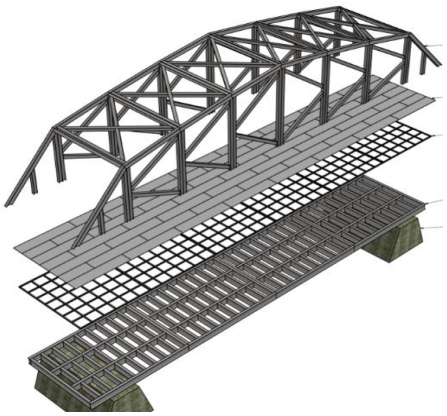

Fig -4: Exploded view

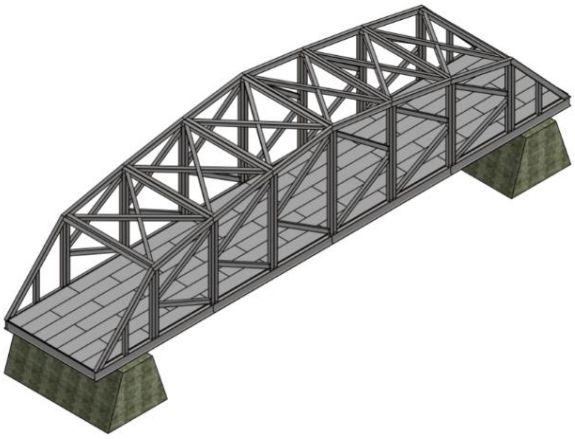

Fig -5: 3D rendered view

3. MATERIALS AND PROPERTIES

Concrete of M40 grade is adopted. The below engineering data are used from the ANSYS library sources.

Table -1: Material properties of concrete

Characteristic strength	40Mpa
Density	2300 kg/m^3
Co efficient of thermal expansion	1.4 e^{-005} C^{-1}
Specific Heat	780 J/Kg.^{0} C

4. MODELLING AND ANALYSING

As mentioned earlier, Fusion 360 software is used for modeling and Ansys is employed in analyzing the structure.

4.1 Modeling in Fusion 360

The model is designed as 3 – Dimensional structure and further analyzed in Static condition using modal frequency method for the initial load applied which is done after the meshing.

Table -2: Initial force

Type	Force
Magnitude	150 N
X Value	-106.1 N
Y Value	1.619E-14 N
Z Value	106.1 N

The following figure provides a general shape of the buckling and informs that where maximum and minimums occur.

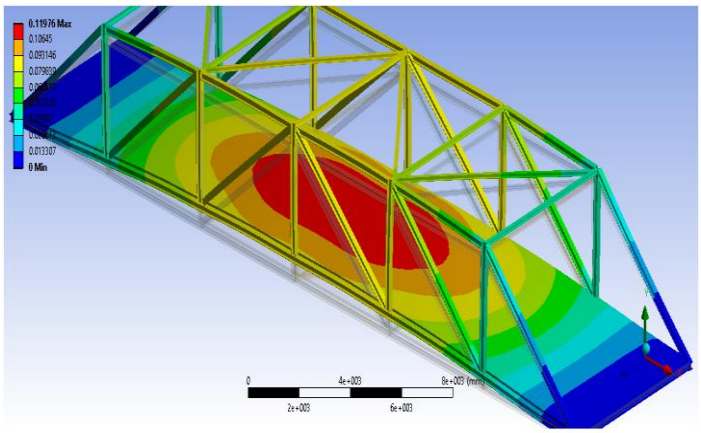

Fig -6: Mode Displacement

4.2 Meshing

The mesh selected is triangular mesh .The mesh smoothing adopted is medium. An element triangular mesh consists of three nodes. The reason behind adopting triangular mesh is to obtain precise results.

Table -3: Nodes and Elements

Types	Nodes	Elements
Solids	1103258	2766331

4.3 Analysing in Ansys

As mentioned earlier, with the following values the model is imported from Fusion 360 and analyzed further for dynamic conditions for the following load condition and results were obtained.

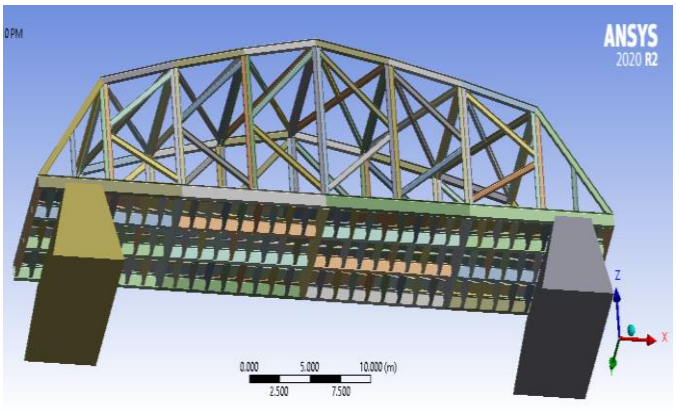

Fig -7: Imported Model

Table -4: Considerations and values

Length X	48770 mm
Length Y	10668 mm
Length Z	2400 mm
Volume	479.07 m^3
Mass	3.7607 e+006 kg
Centroid X	24385 mm
Centroid Y	5300.1 mm
Centroid Z	1201.5
Moment of Inertia I_p 1	2350.9 Kg .m^2
Moment of Inertia I_p 2	2449.6 Kg. m^2

5. RESULTS AND DISCUSSIONS

The pre stressed bridge deck is analysed in Fusion 360 and with Ansys, the following results are obtained.

5.1 Total Deformation

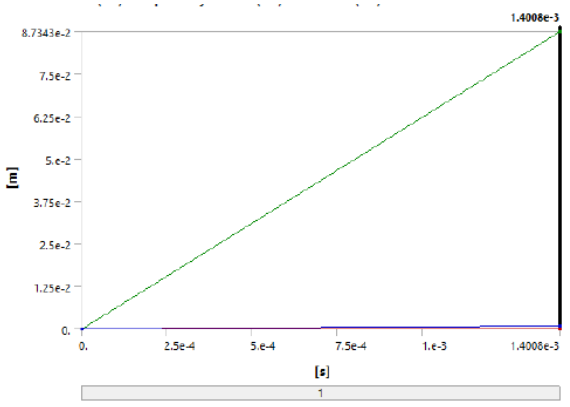

Chart -1: Total Deformation

Chart -1 gives the graphical representation of the total deformation values of the structure with respect to time.

From the graph, it shows the following results with minimum, average and maximum value.

Table - 5: Total Deformation

Time [s]	Minimum [m]	Maximum [m]	Average [m]
1.1755e-038	0	0	0
1.4008e-003	0	8.7343e-002	9.7921e-004

5.2 Equivalent Elastic Strain

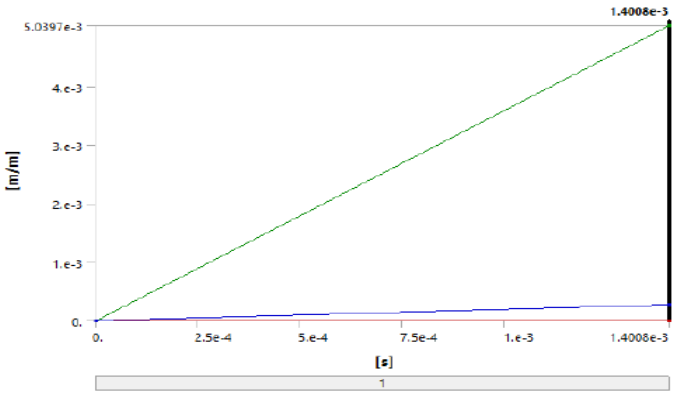

Chart -2: Equivalent Elastic Strain

Chart -2 gives the graphical representation of the equivalent elastic strain values of the structure with respect to time.

From the graph, it shows the following results with minimum, average and maximum value.

Table - 6: Equivalent Elastic Strain

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
1.1755e-038	0	0	0
1.4008e-003	2.0837e-008	5.0397e-003	2.7388e-004

5.3 Equivalent Stress

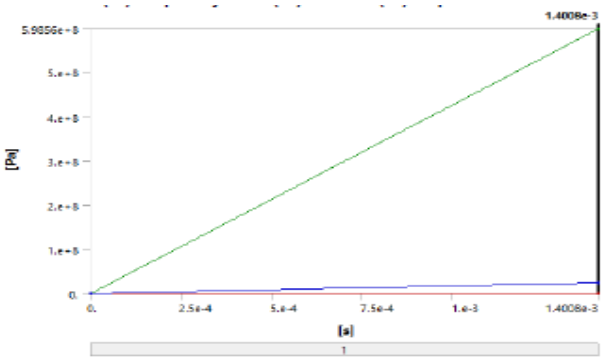

Chart -3: Equivalent Stress

Chart -3 gives the graphical representation of the equivalent stress values of the structure with respect to time.

From the graph, it shows the following results with minimum, average and maximum value.

Table - 7: Equivalent Stress

Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1.1755e-038	0	0	0
1.4008e-003	2436.1	5.9856e+008	2.4358e+007

6. CONCLUSIONS

The conclusions of the study are as follows:

- In this paper, a bridge deck model is taken for its behavior analysis on the application of external loads.
- A 3D model of the building is modeled in Fusion 360 and imported to ANSYS for the analysis process.
- Based on the input values the resultant value of total deformation, equivalent elastic strain and equivalent stress with respect to time is obtained.
- The results show the deformation in the structure is increasing with the load and time and the analysis was done in both static and dynamic conditions.

REFERENCES

[1] Viqarnazir& Mr. Sameer Malhotra (2019), "Design of a Prestressed Concrete Bridge and Analysis by CSi Bridge", - International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 - Volume: 06 Issue: 09 | Sep 2019 www.irjet.net p-ISSN: 2395-0072- 2019, IRJET | Impact Factor value: 7.34 | ISO 9001:2008 Certified Journal.

[2] NajlaYas V & PriyankaDilip P (2018), "Performance Analysis of Composite Box Girder Using Ansys.Pro", International Journal of Engineering Science Invention (IJESI) ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726 www.ijesi.org ||Volume 7 Issue 5 Ver. 1 || May 2018 || PP 19-24 www.ijesi.org 19.

[3] ShubhamLandge, UmeshBhagat, ShubhamBhaisare, VedPrakash Dr. I. P. Khedikar (2018), "Analysis and Design of Pre-stressed Concrete I-Girder Bridge" International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056, p-ISSN: 2395-0072 , Volume: 05 Issue: 05 - 2018.

[4] ArshadShaik Mohammed (2017), "Finite Element Analysis of Prestressed Bridge Girder Using ANSYS", International Journal of Civil Engineering and Technology – 2017.

[5] AlperenOzel (2016), "Seismic Design of a Prestressed Concrete Bridge", University of New Orleans Theses and Dissertations- Spring 5-13-2016.

[6] Khairmode A. S. &Kulkarni D. B (2015), "Analysis of Prestressed Concrete Multi-cell Box Girder Curved Bridge",International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Index Copernicus Value (2013): 6.14 | Impact Factor (2015): 6.391.

[7] Abhinav S. Kasat1 &Valsson Varghese (2012), " Finite Element Analysis of Prestressed Concrete Beams" International Journal of Advanced Technology in Civil Engineering, ISSN: 2231 –5721, Volume-1 – 2012.

[8] MyrtoAnagnostopoulou, Andre Filiatrault ,AmjadAref (2011), "Seismic Design and Analysis of a Precast Segmental Concrete Bridge Model ",State University of New York , Supported by Federal Highway Adminsitration under contract number – DTFH61-07-C-00020.

[9] Huili Wang, H. Wang, S.F. Qin, W.J. Xu (2011), "Seismic Analysis of Prestressed Bridge Pier Based on Fiber Section" International Conference on Green Buildings and Sustainable Cities – 2011.

[10] Hari P. Pokharel (2011), "Seismic Analysis of Precast Segmental Bridge Piers" Sustainable Bridges: The Thread of Society AP-G90/11_038,ABC – 2011.