Finite Element Analysis Based Vibration behavior on Warren Truss Bridge

Ashuvendra Singh  
Assistant Professor, Department of Civil Engineering  
Dev Bhoomi Institute of Technology, Dehradun, Uttarakhand

Faraz Ahmad  
Assistant Professor, Department of Mechanical Engineering  
Dev Bhoomi Institute of Technology, Dehradun, Uttarakhand

Nitish Kumar Saini  
Assistant Professor, Department of Mechanical Engineering  
Dev Bhoomi Institute of Technology, Dehradun, Uttarakhand

Abstract

A bridge is an important structure in the road transportation network. Its performance during and after an earthquake is quite crucial to provide relief as well as for security purposes. It is also subjected to vibration during the movement of vehicle. In present work vibration behavior was carried out on warren truss bridge using finite element analysis. The CAD model of bridge was designed according to Indian standard with the help of CATIA. In present analysis bridge was designed like a composite structure to provide better structural properties. Vibration behavior of this composite structure was carried out by model analysis using ANSYS. Furthermore the CAD model was tested under static load of 500 KN. The simulation result of vibration and deformation was under satisfactory condition.

Keywords: Bridge, vibration, earthquake, warren truss, finite element, CAD, CATIA, ANSYS.

Introduction

Bridge is the most widely used in transportation engineering. To get from one place to another place many Obstructions like ridge, valley, and river are hurdle in path of human travelers. To overcome this or to make journey more comfortable a structures made up of concrete, steel, masonry and timber are made. In today world composite bridges like steel-concrete, timber- masonry etc are generally used. Steel concrete bridges are favorite to Engineers because of their aesthetic appearance, load carrying capacity, long span, economically and their reduced weight by using steel plate girders and concrete box girders which aid in seismic design.

Nowadays rolling load Due to increase in loads and speed problem are primary in transportation engineering. Especially railway track and bridge structures are severely affected by these growing requirements by increased deformations. For existing structures the load carrying capacity is a serious concern. Solutions to which are: rebuilding the structure or apply control system (with active, passive or semi-passive vibration damping). These solutions are not fully practical. Practical extent can be increased by computational part of the project. In the present study the vibration behavior of warren type composite structures bridge was calculated. Furthermore the Bridge was tested for a static load of 500 KN. Vibration of the composite structure bridge depends on the geometry, materials, arrangement, design and construction. An analytical solution is done using a numerical problem with the help of ANSYS software which is based on finite element analysis. With the help of ANSYS software we have found deformation on different vibration frequency as well as stress strain behavior under static loading.

Material Properties and CAD Model
This warren type composite bridge was constructed with the help of structural steel and concrete. Structural steel property is already available in ANSYS database software. Girder dimensions of the flange plates are 600mm x 40mm and web plate are 550mm x 20 mm. Thickness of concrete slab is 200mm which transfer the vehicle load to supported girders. The thickness provided for wearing coat is 100 mm. Mechanical properties of the materials are given in table 1.

<table>
<thead>
<tr>
<th>Materials</th>
<th>Density (Kg/m³)</th>
<th>Young’s Modulus (GPa)</th>
<th>Poisson ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural steel</td>
<td>7850</td>
<td>200</td>
<td>0.3</td>
</tr>
<tr>
<td>Concrete</td>
<td>2500</td>
<td>31.26</td>
<td>0.18</td>
</tr>
<tr>
<td>Bitumen</td>
<td>2250</td>
<td>2.944</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Design details are given below and shown in figure 3.
- Types of bridge: warren truss
- Length of span: 30m
- Carriage Width: - 7.5m
- Height: - 5m
- Wear course: - 0.10 m
- Steel section: - ISWB550

The CAD model of the considered bridge was design in CATIA and ANSYS was used as analysis software. CATIA has power full tool for designing complex geometry and ANSYS reduce the problem by providing the user friendly analysis interface. Now days ANSYS is more popular in optimizing the engineering problems like: structural, Model, thermal and CFD etc. The figure 1 and 2 shows the 3-D CAD model and mesh model of considered bridge. The bridge consists of 82840 numbers of nodes and 39573 numbers of elements.

It can be seen in the figure 3 that bridge consist of five structural steel section which support the concrete slab and wear course. The trusses were also made up of structural steel which helps in load distribution. For analysis the steel section was hold on expansion support from both the end as shown in figure 4.
The CAD model was imported to ANSYS workbench; material properties and expansion support boundary condition were used for vibration analysis. Furthermore, first 6 mode shape and the deformation on corresponding frequency were calculated for the considered bridge problem. Figure 5 shows mode shape and deformation under different vibration and loading condition and figure 6 shows graphical representation of all six modes with vibration frequency. The variations of deformation with respect to different frequency were represented in tabular form as shown in Table 2.

![Figure 4: Expansion support](image)

**Result and Model Analysis**

**Table 2: vibration Frequency and corresponding deformation**
<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Vibration Frequency (HZ)</th>
<th>Deformation (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7.6142</td>
<td>0.04997</td>
</tr>
<tr>
<td>2</td>
<td>8.1949</td>
<td>0.09045</td>
</tr>
<tr>
<td>3</td>
<td>10.482</td>
<td>0.07754</td>
</tr>
<tr>
<td>4</td>
<td>13.232</td>
<td>0.08648</td>
</tr>
<tr>
<td>5</td>
<td>13.636</td>
<td>0.11081</td>
</tr>
<tr>
<td>6</td>
<td>15.098</td>
<td>0.10464</td>
</tr>
</tbody>
</table>

**Figure 6:** Frequency variation in first six modes

**Static Structural Analysis**

The CAD model of bridge was again analyzed for structural deformation under a loading of 500 KN. The load was applied on mid span which was shown in figure 7. The maximum total deformation is 0.7862 mm and Y directional deformation is 0.0149 mm shown in figure 8 and 9 respectively. Stress and strain distribution is shown in figure 10 and 11 respectively.

**Figure 7:** Loading condition

**Figure 8:** Total Deformation

**Figure 9:** Directional Deformation in Y axis

**Figure 10:** Stress Distribution

**Figure 11:** Strain Distribution

**Conclusion**
The bridge model was designed according to the Indian standard provisions. Frequency-based analysis was performed to find the resonance frequency of the considered bridge. In figure 5 we can see, in first 6 modes of failure, we have the range of frequency 7.6142 to 15.098 Hz. So the 7.6142 Hz was selected for the fundamental operating frequency of the bridge. Furthermore, the result of the static structural analysis shows the deformation, stress, and strain are under safety limits. In future, it will be interesting to analyze this model with different materials and designs to find out the best suited structure for serviceability purposes.

Reference

10. IRC:6 – 2000