Abstract

The aim of this study is to numerically model a cable stayed bridge and then study its dynamic characteristics which could be used later on in determination of dynamic response or behavior of the structure. The bridge adopted for this study is Garigliano cable-stayed bridge (Italy) that crosses the Garigliano River, along the coast road between Naples and Rome. A finite element model of the bridge has been made with the available geometrical and material properties. The cable-stayed bridge has been modelled with different types of elements and boundary conditions. In this study the modelling and study of the dynamic properties of the cable stayed bridge have been done using engineering tool MATLAB®. Then comparison has been drawn from the results obtained using the engineering tool with the experimental analysis. The cognizance is obtained in finite element formulation of complicated structures such as a cable stayed bridge. It is observed that the present numerical model captures the experimental data well with minor discrepancies, which can be attributed to lack of proper data in numerical modelling. The model can be further improved with access of more structural data and can be effectively used for determination of dynamic response due to earthquake and wind.

I. INTRODUCTION

The bridge considered here for study is a two span cable stayed bridge over the Garigliano River, along the coast road between Rome and Naples. The structural details including material or sectional properties and experimental responses is provided here, adopted from the detailed description given in P. Clemente et.al. (1998)[3]. The length of the bridge is 180 m, with each span being 90 m. The width of the deck is 26.1 m. The pre-stressed pre-fabricated co
Concrete box girder is simply supported at both ends and fully constrained with the tower. The girder is suspended by 18 couples of cables with 9 couples of cables for each span, starting at different section of the pylon but not evenly spaced along the deck. The cables are spaced 0.85 m from the centerline of the cross-section of the bridge. The height of the pylon is 10.85 m from the foundation to the deck extrados and 30 m from the deck. The lower part, from the foundation to 5 m above the deck is made of concrete, and the remaining part is a steel box beam.

The view and plan of the bridge is shown in Fig. 1.1 & Fig. 1.2 respectively.

1.1 Geometrical Properties
Geometrical properties of the structure are adopted from P. Clemente et.al (1998). The cross-sectional properties of the deck, tower and cables is listed in Table 1.1 through Table 1.3.

<table>
<thead>
<tr>
<th>Cross-sectional area, A (m²)</th>
<th>Iₓ (m⁴)</th>
<th>Torsion moment of inertia, J (m⁴)</th>
</tr>
</thead>
<tbody>
<tr>
<td>16.52</td>
<td>13.3</td>
<td>650.7</td>
</tr>
</tbody>
</table>

1.2 Cross-sectional properties of the tower

<table>
<thead>
<tr>
<th>Y</th>
<th>A (m²)</th>
<th>Iₓ (m⁴)</th>
<th>Iᵧ (m⁴)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 &lt; y &lt; 10.85</td>
<td>23.66</td>
<td>178.37</td>
<td>11.68</td>
</tr>
<tr>
<td>10.85 &lt; y &lt; 15.85</td>
<td>10.8</td>
<td>16.7</td>
<td>5.6</td>
</tr>
<tr>
<td>15.85 &lt; y &lt; 25.85</td>
<td>0.75</td>
<td>1</td>
<td>0.8</td>
</tr>
<tr>
<td>y &gt; 25.85</td>
<td>0.81</td>
<td>0.833</td>
<td>0.677</td>
</tr>
</tbody>
</table>

1.3 Material Properties
Material properties of the structure are adopted from P. Clemente et.al (1998). The material properties of the deck, tower and cables i.e. elastic longitudinal and tangential moduli of materials are listed in Table 1.4.

<table>
<thead>
<tr>
<th>Cable</th>
<th>A (cm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A and B</td>
<td>67.5</td>
</tr>
<tr>
<td>C to H</td>
<td>82.5</td>
</tr>
<tr>
<td>I</td>
<td>70.5</td>
</tr>
</tbody>
</table>
Table 1.4 Elastic moduli of the materials

<table>
<thead>
<tr>
<th>Structural element</th>
<th>E (MPa)</th>
<th>G (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Girder</td>
<td>42200</td>
<td>17 400</td>
</tr>
<tr>
<td>Cables</td>
<td>200000</td>
<td>77 000</td>
</tr>
<tr>
<td>Tower</td>
<td>36000</td>
<td>15 600</td>
</tr>
<tr>
<td>Concrete</td>
<td>200000</td>
<td>77 000</td>
</tr>
</tbody>
</table>

1.4 Boundary Conditions

Boundary conditions of the structure are adopted from P. Clemente et.al (1998). The girder is fully fixed to the tower. The girder is simply supported at two other ends where torsional rotation isn’t allowed. The tower is fixed at the foundation.

II. MODEL ANALYSIS

The dynamic response characteristics depend upon the vibration characteristics of a structure or proper study of the dynamic behavior of a cable stayed bridge, a modal analysis of the bridge is essential. The cable stayed bridge structure consists of a system with continuous distribution of stiffness and mass. In structural analysis of any cable stayed bridge modelling is important as the model if can simulate or reflect the stiffness and mass of the structure accurately, might seriously affect the accuracy of the calculation. Mass, structural stiffness and boundary conditions consideration is essential in model analysis as because these are related to the structural characteristics of the cable stayed bridge. Thus the structure has to be divided into finite elements for its better and comprehensive study. The results are not only reliable but also easier to determine than by using traditional empirical formulas. The finite elements such as beam elements, truss elements are used in this study for modelling the cable stayed bridge. The basic unknowns are determined at the nodes or at any point inside the element in terms of the nodal values of the element. The boundary conditions of the model are defined by restraining the corresponding nodes. The dynamic equilibrium equation for the model is as follows:

\[
[M]{\ddot{U}} + [C]{\dot{U}} + [K]{U} = [P(0)]
\]

where,
- \([M]\) = mass matrix of the structure
- \([C]\) = damping matrix of the structure
- \([K]\) = stiffness matrix of the structure
- \({U}\) = displacement vector of each node
- \({\dot{U}}\) = velocity vector of each node
- \({\ddot{U}}\) = acceleration vector of each node

Ignoring the resistance we get the equation as below:

\[
[M]{\ddot{U}} + [K]{U} = 0
\]

Taking \([U(t)] = \{\Phi\} \sin \omega t\), then solving the differential equation we get,

\[
([K] - \omega^2[M]) \{\Phi\} = 0
\]
Thus we get natural frequencies of the system $\omega_i$ (i= 1, 2, 3,......N) and the vibration modals \{${\Phi}_i$\}(i= 1, 2, 3,......N) of the structure from the above equations.

### III. MODELLING GARIGLIANO CABLE STAYED BRIDGE USING MATLAB®

#### 3.1 About Toolbox

MATLAB® stands for MATRIX LABORATORY as the basic building block here is MATRIX. MATLAB® is a software package, a fourth generation programming language. It provides an interactive environment with built-in functions for technical computation, graphics, processing, optimization and animation. The modelling and analysis of the cable stayed bridge in MATLAB® is done using a toolbox. Since Finite Element Method package is not provided by MATLAB® so to model and analyze the bridge, the method adopted in this regard is from a formulation i.e. the toolbox given by Dr. Juan Martin Caicedo, Washington University in Saint Louis Structural Control and Earthquake Engineering Laboratory. It was created in January 2002.

#### 3.2 Steps involved in modelling and studying the dynamic characteristics of the bridge using Matlab®

A 3-D finite element model of Garigliano cable- stayed bridge is developed in MATLAB® as shown in Fig.3.1 using the toolbox. The bridge is modelled based on the information obtained regarding geometrical properties, material data and support conditions from P. Clemente et.al (1998) [3] and line diagram of the bridge with cable type is plotted using MATLAB® as shown in fig.3.2. The deck is modelled as single spine employing beam elements for girder. The nodes are taken at certain intervals along the spine as well as at the cable ends that are connected to the spine through rigid links where cable ends are at a distance of ± 0.85m from the centre line. The spine nodes are considered as the master nodes and the nodes connecting the cables to the spine via rigid links are considered as the slave nodes for the evaluation of stiffness. The distribution of lumped masses both translational and rotational mass is done along the spine.

![Figure 3.1: Finite Element Model using MATLAB®](image_url)
The trapezoidal cross-section of the girder is divided into two rectangular and triangular hollow sections each of which length and depth dimensions are mentioned in P. Clemente et al. [3]. The mass of each spine nodes for the two rectangular and triangular section of the trapezoid is calculated individually. There is no data available regarding the thickness of the entire section since it is a hollow trapezoidal cross-section, therefore we obtain approximate values of mass, mass moment of inertia of the individual element and finally rotational mass moment of inertia by assuming the thickness between outer and inner section to be 0.5m. The length of different intervals at which the spinal nodes are taken is also considered while calculating the mass.

The mass moment of inertia of each nodal element of the trapezoidal section is then calculated for the three axis i.e. x, y and z axis.

The cables are modelled using linear elastic truss elements where cable sag is considered negligible. Each cable is modelled using single truss element. The mass density of cables and cable tension data is not available. The mass density of cable is assumed as that of steel i.e. 7850 kg/m³. Cables are not considered to be pretension. The tower is modelled using three dimensional beam elements with nodes introduced at locations of cable attachments and sectional or material changes. The torsional moment of inertia of tower is not available and thus the modelling is done by assuming a value which is actually the summation of Iₓ and Iᵧ.

The boundary conditions are taken as the girder is fixed fully to the tower and the two ends of the girder are simply supported where torsional rotation is not allowed. The tower at its base is fixed at the foundation w.r.t all displacement components. The model has 93 nodes, 52 beam elements, 36 cable elements, 40 rigid links and 41 lumped mass objects.
IV. RESULT AND OBSERVATIONS

Figure 4.1 Mode 1 (Frequency = 1.02 Hz)  
Figure 4.2 Mode 2 (Frequency = 1.07 Hz)  
Figure 4.3 Mode 3 (Frequency = 1.71 Hz)  
Figure 4.4 Mode 4 (Frequency = 1.99 Hz)
V. CONCLUSION AND DISCUSSION

5.1 Conclusion
The finite element model of the bridge which is modelled in MATLAB® and analysed, though shows a good agreement with that of the experimental values yet cannot accurately predict the frequencies and vibration mode shapes of the structure. This discrepancy is attributed to the insufficient data of geometrical and material properties of the bridge structure. MATLAB® takes into account the rotational masses so mode shapes and frequencies related to torsion are obtained in the fifth mode which matches with the experimental results. Thus it can be concluded that finite element model modelled in MATLAB® can very well predict the dynamic characteristics of a cable stayed bridge showing good agreement with that of the experimental results taking into account all the flexural and torsional behaviour of the structure, only if all the geometrical and material property data along with boundary conditions is available in order to simulate the full scale structure.

5.2 Discussions
In this work a cable stayed bridge, Garigliano cable-stayed bridge (Italy) that crosses the Garigliano River long the coast road between Naples and Rome, is numerically modelled and studied for its dynamic characteristics. A finite element model of the bridge has been made with the available geometrical and material properties from standard literature. The cable-stayed bridge is modelled with different types of elements and boundary conditions. The modelling and study of the dynamic properties of the cable stayed bridge have been done here using MATLAB®. A comparison also has been drawn from the results obtained using the engineering tool with those from the experimental analysis of established
literature. An insight is obtained in finite element formulation of complicated structures such as a cable stayed bridge. It is observed that the present numerical model captures the experimental data well with minor discrepancies, which can be attributed to lack of proper data in numerical modelling. The model can be further improved with access of more structural data and can be effectively used for determination of dynamic response due to earthquake and wind.

VI. SCOPE OF FUTURE WORK

The model in MATLAB® can be used for determination of dynamic response due to wind and earthquake. The model can further be used to find a control strategy against such excitations.

VII. REFERENCES


TO CITE THIS PAPER