# FINITE ELEMENT ANALYSIS OF AN EXISTING SEISMICALLY ISOLATED BRIDGE STRUCTURE

University of Tsukuba, Student Member, ○Muhammad Rashid
University of Tsukuba, Regular Member, Mayuko Nishio

## 1. INTRODUCTION

The Finite Element (FE) analysis allows for a more systematic analytical approach to be implemented for the design and assessment of bridge infrastructure for safety and reliability evaluation. The wide applications include, structural health monitoring, repair and strengthening design, up-gradation and maintenance, are among some of them (Nishio et al., 2012). The underlying principles can be used to model analytically the intrinsic non-linear characteristics of a particular structural system subjected to dynamic excitation. A real-time experimental investigation based on large-scale testing is still needed to ensure enough credibility of these models for assessment purposes (Casarotti & Pinho, 2006). However, sometimes due to unavailability or capacity limitations of the available testing facilities and equipment, the problem is tackled by analytical modeling with some numerical approximation and physical assumptions. The accuracy of these models can further be upgraded by incorporating field data for the subsequent structural analysis or health monitoring.

This paper focuses on the development of the FE model to analyze an existing seismically isolated steel box girder bridge structure, subjected to Level-II type lateral loading. Dynamic response of structural components is evaluated, which later on will be used in fragility assessment (not included here). Also, to account for the quantification of performance degradation, the considered models will be updated based on the field monitoring data. This will help to identify the most critical components of the structure based on performance and subsequent maintenance prioritization and design retrofit.

## 2. DESCRIPTION OF THE TARGET BRIDGE STRUCTURE

The target bridge in this study is a steel bridge side of the Byobugaura viaduct in Isogo-ku Yokohama city, Japan. This is a six-span continuous box girder bridge supported by 5 steel and 1 RC pier. The steel framing sections employed are SM 490Y, 400, and SS 400 throughout the structure. The structural details are summarized in Table 1.

Table 1. Structural details of the considered bridge structure

| Description | Superstructure | Substructure | | |
|---|---|---|---|---|
| Geometry | 6 span continuous steel box girders | Piers | Steel pier | P7, P8, P9, P10, P11 |
| | | | Reinforced Concrete (RC) pier | P6 |
| | | Foundation | Shallow foundation | P8, P9, P10, P11 |
| | | | Pile foundation | A2, A3, A4 |
| Concrete | $\sigma_{ck}$ =30 N/mm$^2$ | $\sigma_{ck}$ = 24 N/mm$^2$ | | |
| Structural steel | SM 490Y, SM400, SS400 | | | |
| Steel bar | SD 345 | | | |

## 3. FINITE ELEMENT MODELING AND INPUT EXCITATION

The analytical 3D modelling of the structure was carried out using a finite element software "Engineer's Studio" as shown in Fig. 1. The superstructure and abutments are modeled as linear beam element. The Laminated Rubber Bearings (LRBs) are modeled as multi shear non-linear spring (MSS) elements. Fiber elements are employed for the piers and plastic hinge element for RC P6. Subsequently, the material constitutive models are defined, and section's discretization was carried out. Finally, an equivalent translational and rotational springs are defined and assigned at the foundation interfaces.

Following the 1995 Hyogo-ken Nanbo earthquake, the bridges design guidelines were revised by incorporating Level-II type earthquakes, which are rarely occurring with high intensity and have excessive acceleration vibrations. Based on the seismicity of the region a suit of standard ground motions was selected as input load with PGA of 0.40 - 0.75g (Fig. 1).

## 4. DYNAMIC RESPONSE OF STRUCTURE

### 4.1 Modal Analysis

The modal frequencies and shapes are important parameter in seismic design and assessment of new and existing structures. Change in the modal response provides useful information to assess the performance variation during the service life of a particular structural system. The different mode shapes are shown in Fig. 2 with fundamental frequency of 0.978 Hz.

Keywords: Finite Element analysis, Bridge structure, LRB, Level-II type loading, Non-linear modeling
Contact address: Deptt of Engg Mechanics and Energy, University of Tsukuba, 1-1-1 Tennodai, Tsukuba 305-8573, Japan

 

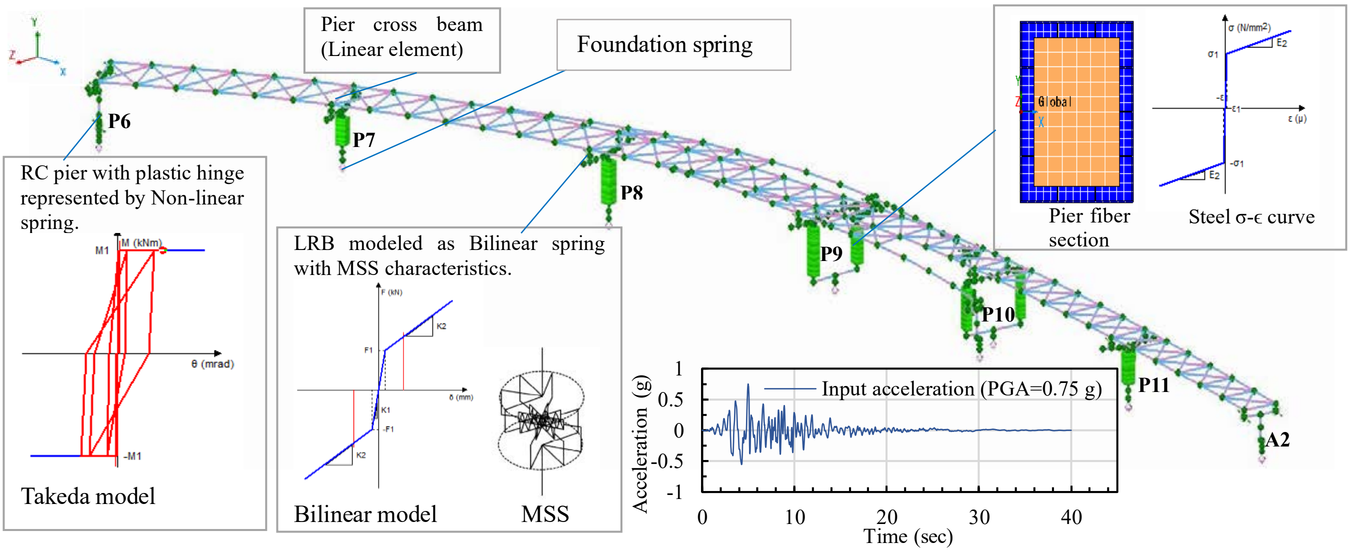


Fig. 1 Bridge Non-linear model and Level-II type accelerogram applied in longitudinal ($x$) direction

Fig. 2 First 3 Mode shapes with $f_1$= 0.978 Hz, $f_2$= 1.064 Hz and $f_3$= 1.108 Hz, respectively.

### 4.2 Structural Components Response

For the same input excitation, the response of the piers and superstructure are considerably varying, as shown in Fig 3. For instance, the peak displacement and acceleration at the selected nodes are compared in Table 3. The relative change in displacement is ranging from 21.11 % for A3 to 64.08 % for P9. The LRBs are highly effective in mitigating the inertial forces by reducing the relative acceleration from 29.71 % at P6 to 94.64 % at A4. The hysteresis response of the bearings showed good performance in term of energy dissipation at displacement below the ultimate capacity of 250%.

Table 2. Peak response values at selected locations

| | Location ID | P6 | P7 | P8 | P9 | P10 | P11 | A2 | A3 | A4 |
|---|---|---|---|---|---|---|---|---|---|---|
| **Displacement (mm)** | Super-structure | 58.06 | 51.06 | 68.62 | 46.83 | 29.48 | 29.89 | 61.33 | 37.55 | 96.33 |
| | Pier | 40.81 | 11.89 | 23.04 | 14.97 | 19.20 | 13.51 | 14.70 | 7.82 | 5.16 |
| **Acceleration (g)** | Super-structure | 0.81 | 0.77 | 0.77 | 0.77 | 0.77 | 0.78 | 0.74 | 1.45 | 1.10 |
| | Pier | 1.41 | 1.39 | 1.59 | 2.15 | 1.89 | 1.42 | 1.51 | 1.84 | 1.43 |

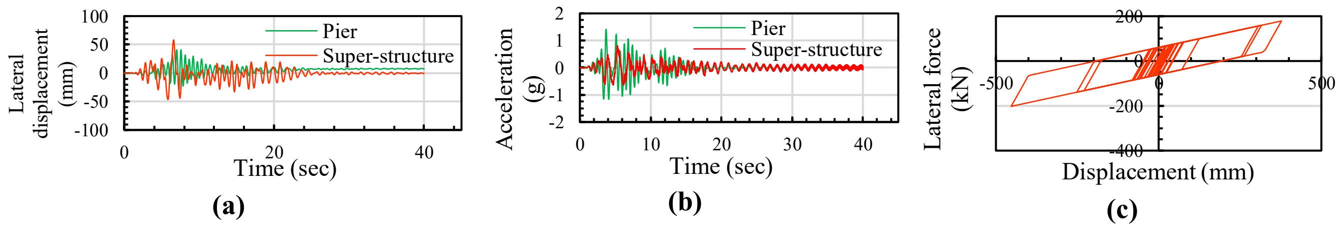


Fig. 3 Dynamic response at P6 (a) Displacement Time History (b) Acceleration Time History (c) Bearing hysteresis

## 5. CONCLUSIONS

A finite element analysis was carried out for an existing bridge structure considering Level-II type lateral loading. Based on the design guidelines, material and geometrical nonlinearities were considered. The analysis result showed that LRBs are extremely capable of reducing the acceleration demand on superstructure, while remaining within the capacity limits. The analysis results can further be extended in assessing the structural seismic fragility and will be updated for considering the performance deterioration during structure's service life for maintenance, strengthening and seismic retrofit design.

## REFERENCES

Chiara, C. and Rui, P., 2006. Seismic response of continuous span bridges through fiber-based finite element analysis. Earthquake Engineering and Engineering Vibration, 5(1), pp.119-131.

Nishio, M., Marin, J. and Fujino, Y., 2012. Uncertainty quantification of the finite element model of existing bridges for dynamic analysis. Journal of Civil Structural Health Monitoring, 2(3-4), pp.163-173.