Strength and Ductility Evaluation of Rigid-Frames Using Finite Element Analysis

K.A.S. Susantha[1*] and T. Aoki[2]

[1]*Department of Engineering Mathematics, Faculty of Engineering, University of Peradeniya*
[2]*Department of Civil Engineering, Aichi Institute of Technology, Toyota, Japan*

Introduction

Improvement of highway network of Sri Lanka includes construction of motorways, flyover crossings, various bridge structures, etc. These structures can have sub-structure components such as cantilever type columns, inverted L-shaped piers, and rigid-frames, which are primarily constructed using concrete. However, steel is also a very useful construction material since it offers a great flexibility in space utilization, has speedy construction time and high earthquake resisting characteristics. In particular, steel rigid-frames supporting highway bridge systems are popular in many countries due to their effectiveness in overcoming space restrictions. Introduction of rigid-frames in large-scale motorway improvement projects in Sri Lanka would be very helpful when densely constructed areas are concerned. Meanwhile, there is a considerable interest among professionals involving infrastructure projects to ensure adequate earthquake resistance in view of many recent earthquake events reported in the region. Therefore, an analytical investigation that involves a large displacement finite element analysis procedure is carried out to examine the performance of rigid-frames subjected to earthquake-induced forces.

Analytical model

Finite element analysis method (FEM) is known to be accurate in analyzing steel structures provided that realistic material model and proper elements with optimum element mesh are employed. In this study, numerical model of a steel rigid-frame pier was made using general purpose finite element analysis program Abaqus (Abqus, 2003). The model consisted of a one-story, rigid-frame pier with a beam length of 5.0m, a column height of 5.8m, and a $600 \times 600 \times 6$ mm rectangular cross section that has 12 longitudinal stiffeners (sizes: 50×6 (column), 80×8 (beam flange), 60×8 (beam web)), as shown in Figure 1. More details of the model can be found in Nishikawa *et al.*, (1998).

There are eight diaphragm plates in one column and six in the beam. Yield strength of steel was 298 MPa. The finite element mesh of the model is shown in Figure 2. The model consists of four-nodes doubly curved shell element (S4R) at column bases and adjacent panels of beam column joints, four-node linear beam-column element (B31OSH) at mid segments of columns and the beam, and rigid beam elements (R2D2) at the beam column joints and the interface between shell and beam-column elements. The load bearing capacity beyond the peak load occurs mainly due to the local buckling deformation at column bases and probably at beam-column joints. Thus it is essential to use shell element at those locations in order to capture the effects of the local buckling deformations. The lengths of the segments where the beam-column elements were assigned were decided so that the elements undergo elastic or very little inelastic deformation. Therefore the use of beam-column elements would not lessen the accuracy but increases the computational efficiency. In order to simulate the proper boundary condition at the interface of beam-column and shell elements and column and beam joints rigid beam elements were used. Only a half of the frame was modeled considering the symmetry in loading and the geometry. Mesh size was decided based on a trial and error method. The length between the base and the first diaphragm is divided into 18 segments while subsequent lengths are divided into 6 segments. Each sub-panel between longitudinal stiffeners consists of 4 columns of shell elements. Three columns of elements were assigned in longitudinal stiffeners. The material behavior of shell elements was simulated using the modified two surface plasticity model (2SM) (Shen *et al.*, 1995) externally linked to the Abaqus program. The beam-column elements were simulated using a bilinear elasto-plastic material model available in the program. The 2SM has been extensively checked and found to be very accurate in inelastic cyclic analysis of steel structures. A subroutine has been developed for 2SM and is connected to the ABAQUS through the user defined material option. Since the beam-column elements undergo elastic or very little inelastic deformations the use of bilinear model would not affect the accuracy. When loading procedure is concerned, first the vertical load P given by 0.15 times the yield load of the column was applied at columns top and then the incremental cyclic lateral loads were applied in terms of multiples of yield displacement δ_y (i.e., $\pm\delta_y$, $\pm 2\delta_y$, etc.) at the top left corner, as indicated in Figure 2. The displacement corresponding to the yield load was taken to be the yield displacement.

Result and discussion

The strength and ductility of the frame can be evaluated using the envelope curve of the lateral load-lateral displacement hysteretic curve shown in Figure 3. It is seen here that the maximum load occurred at displacement level of $2\delta_y$ and beyond that a gradual decrease in load carrying capacity was observed. Normally, the strength is given in terms of the ratio of maximum lateral load H_{max} to H_y (i.e., H_{max}/H_y). The ductility (μ) is given by either the ratio of lateral displacements at H_{max} to δ_y (i.e., $\mu_m=\delta_m/\delta_y$) or lateral displacement at $0.95H_{max}$ beyond the peak to δ_y (i.e., $\mu_{95}=\delta_{95}/\delta_y$) (Aoki and Susantha 2005). Above two definitions are graphically shown in Figure 4. The values of H_{max}/H_y, δ_m/δ_y, and δ_{95}/δ_y of the frame are 1.30, 2.0, and 2.70, respectively.

The failure modes such as local buckling at column bases and in adjacent panes of beam-column connections could be observed from this analysis as shown in Figure 5. The local buckling patterns at the column bases were similar to those normally observed in cantilever type steel columns subjected to cyclic lateral loads.

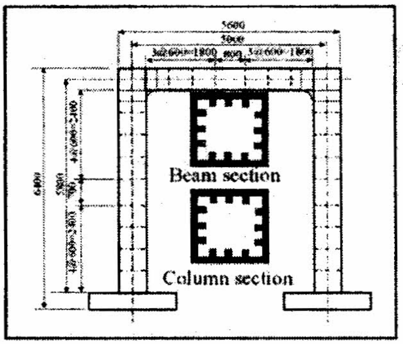

Figure 1. Rigid-frame model

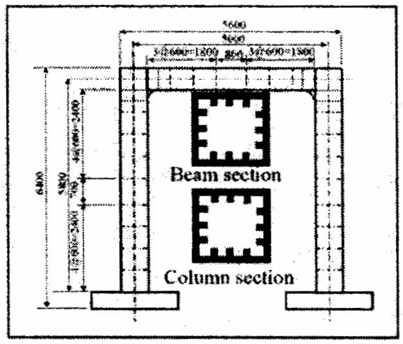

Figure 2. FEM model

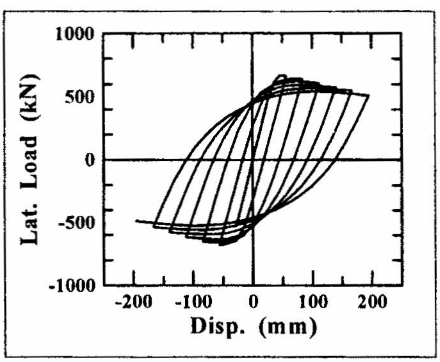

Figure 3. Lateral load-lateral displacement curve

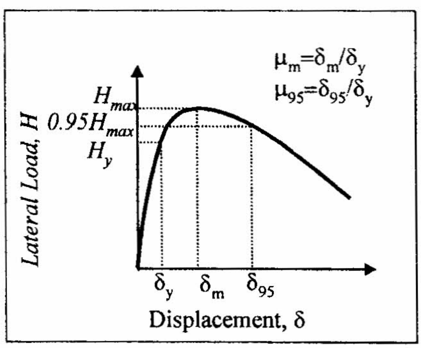

Figure 4. Ductility indices μ_m and μ_{95}

Conclusions

Large displacement finite element analysis procedure was used to evaluate seismic resisting characteristics of rigid-frame bridge piers. The maximum load was found to be 30 percent higher than the yield load of the frame. The ductility was around 2.7. The decrease in load bearing capacity occurred as a result of local buckling deformation at column bases and beam column connections.

References

ABAQUS User's Manual-version 6.4 (2003) ABAQUS, Inc., Pawtucket, R. I.

Aoki, T. and Susantha, K.A.S. (2005) Seismic performance of rectangular-shaped steel piers under cyclic loading, *Journal of Structural Engineering, ASCE*, 131(2), 240-249.

Nishikawa, K., Josen, Y., Takahashi, M., and Takaku, T. (1998) Experimental study on failure and collapse of rigid-frame piers of highway bridges, *Proceedings of the 14th US-Japan Bridge Engineering Workshop, PWRI, Pittsburgh.*

Shen, C., Mamaghani, I.H.P., Mizuno, E. and Usami, T. (1995) Cyclic behavior of structural steels, *Journal of Structural Engineering, ASCE*, 121(11), 1165-1172.

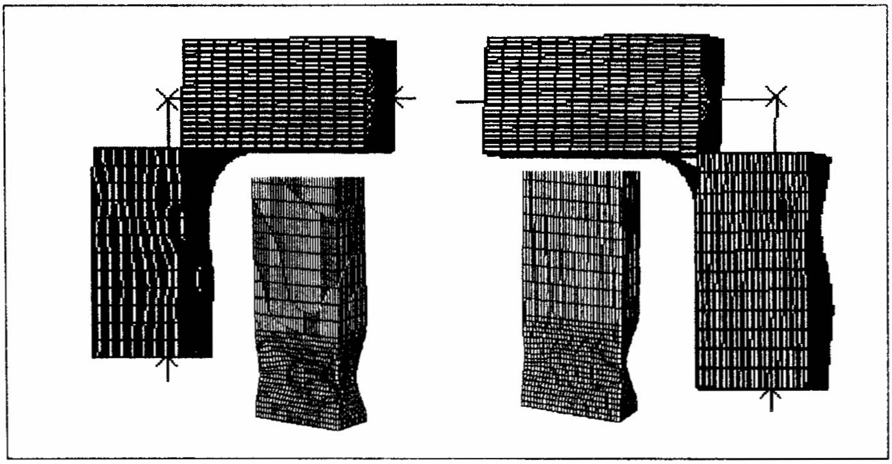

Figure 5. Deformed shapes at $7\delta_y$