



SEISMIC SIMULATION OF AN EXISTING STEEL MOMENT-FRAME BUILDING RETROFITTED WITH EXTERNAL CABLE-STAYED SYSTEM

Limin JIN¹, T. Albert CHEN², Hossein MOZAFFARIAN³ and Atila ZEKIOGLU⁴

SUMMARY

A seismic simulation technique was developed for an existing steel moment-frame building using LS-DYNA computer software [1]. The building was retrofitted by external cable-stayed system that was developed by Black & Veatch [2]. The purpose of the simulation analysis was to obtain better understanding of how the existing steel-framed building and its seismic retrofitting system respond to earthquake ground motions. The simulation technique developed provides design engineers with new ways of creating retrofit strategies to protect buildings from earthquake attacks. The simulation analysis had validated the retrofit approach for the Los Angeles County Department of Public Work Headquarters Building.

INTRODUCTION

Since the 1994 Northridge Earthquake, there has been a great deal of modeling analyses and laboratory tests on seismic behavior of steel moment frame connections. This paper describes an implementation of retrofit strategies for an existing steel moment-frame building retrofitted with external cable-stayed system through seismic simulation technique using LS-DYNA computer software. The detailed seismic simulations were carried out to determine the responses of the existing building and the retrofitting system to earthquake ground motions and hence determine the mitigation benefits. The simulation technique combined with the selected load path approach provides insights to the structural behavior of the building. Three-dimensional seismic simulation models were created directly incorporating the non-linear load-deformation characteristics of individual components of the building. A seismic beam-column element was developed by Arup for the simulation to model the connection fracture behavior identified during the project-specific moment connection testing at the University of California, San Diego (UCSD). An equivalent plastic moment-rotation curve was constructed to incorporate the existing beam and panel zone characteristics. The validation analyses were conducted on the cruciform sub-models. Results from the simulations and tests were compared to verify the behavior of the element.

¹ Ph.D., PE, SE, Ove Arup and Partners, Los Angeles, USA

² Project Manager, Black & Veatch Corporation, Los Angeles, USA

³ Associate, Ove Arup and Partners, Los Angeles, USA

⁴ Principal, Ove Arup and Partners, Los Angeles, USA

The model simulation analyses were employed to validate the retrofit approaches for the Los Angeles County Department of Public work Headquarters Building, a 12-story steel moment-frame structure. The building was originally designed per the 1967 Uniform Building Code, and constructed in 1971. An initial post-earthquake inspection of the moment connections and seismic evaluation of the building did not reveal any significant structural damages caused by the previous earthquakes. However, large areas of low quality welds at the beam-column connections were identified [2]. As a result, certain level of seismic upgrade was necessary to bring the building up to the current standard of the building codes. The seismic retrofit scheme selected was an external cable-stayed system. The performance objectives of the retrofitted building were established based on the seismic rehabilitation guidelines of FEMA-351 and FEMA-356 [2]. The beam-column plastic rotations were extracted graphically from the simulation models for assessment of the individual frame connection performance. The simulation work also included obtaining the force and displacement responses of the building and structural members to the various sets of earthquake records for evaluating the effectiveness of retrofitting.

EXISTING MOMENT FRAME MODELING

The basis of seismic modeling and simulation with LS-DYNA program is a non-linear analysis, either static pushover analysis, or time history analysis. The simulation analysis requires the development of finite element-based models of the building to be investigated. As the full investigation using LS-DYNA program typically involves various sets of analysis with thousands of time steps, computational efforts are always an important consideration in development of the DYNA models.

The building under consideration had four exterior steel moment-resisting frames with typical bay length of 15.0 ft and typical story height of 14.0 ft. Additional welded moment connections were also placed at the interior gravity frames in one of the principal directions of the building to increase the structural stiffness. Figure 1 shows typical exterior frame elevation. The various DYNA simulation models were defined by the column lines, which were located on the plan view of the building, and the floor elevations, which were defined as horizontal levels on an elevation of the building.

The moment frame beams and columns were modeled by pairs of seismic beams with lumped plasticity at one end of the element. This end of the element was intentionally oriented to the beam-column joint such that the member plastic rotation at the joint could be extracted. The yield criterion of the seismic beam element was based on the axial force and moment interaction (P-M-M yield surface). The material properties of the rolled wide flange members were taken from Table 5-2 of FEMA-352 [2], as shown in Table 1. Built-up sections of the beams and columns were assumed having yield stresses based on the average results of low yield at 0.2% offset from test reports [3].

Table 1 Frame Member Material Properties for DYNA 3D Modeling

Section Type	AISC Group	Yield Stress(Fy)	Reference
Rolled Section	ASTM A572, Group 2	58 (ksi)	Table 5-2 of FEMA-352
Rolled Section	ASTM A572, Group 3	57 (ksi)	Table 5-2 of FEMA-352
Rolled Section	ASTM A572, Group 3	57 (ksi)	Table 5-2 of FEMA-352
Rolled Section	ASTM A36, Group 1	51 (ksi)	Table 5-2 of FEMA-352
Rolled Section	ASTM A36, Group 2	47 (ksi)	Table 5-2 of FEMA-352
Built-up section	Columns	41.5 (ksi)	Test Report
Built-up section	Beams	38.3 (ksi)	Test Report

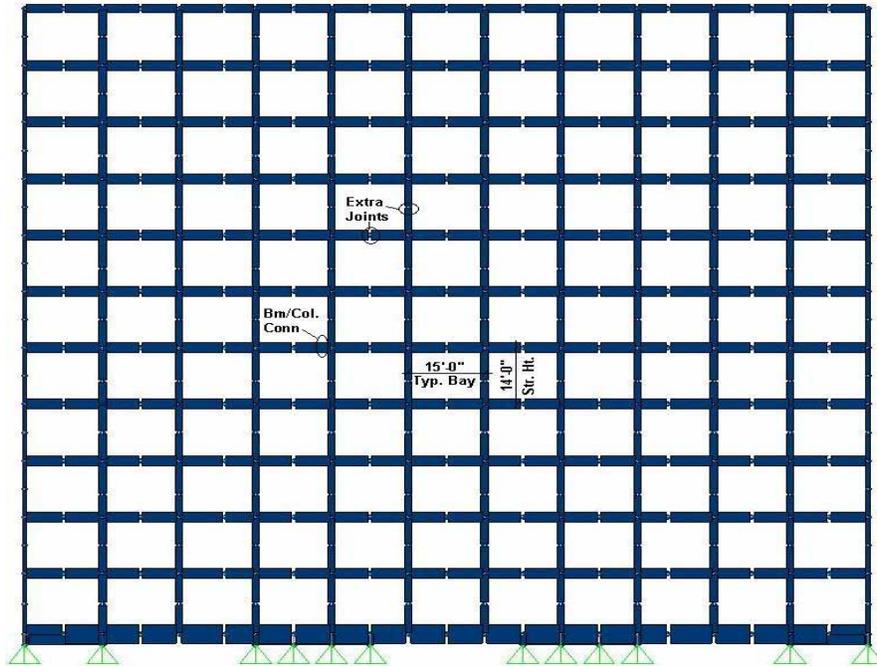


Figure 1 Typical Exterior Moment Frame Elevation

Moment Frame Columns

Columns were modeled as seismic-beam elements with 3% strain hardening plastic behavior beyond yield at the member plastic moment (M_p). Material degradation was not modeled for the columns. In addition, extra joints were created at column splice locations so that forces can be reported to check against existing splice capacities.

Moment Frame Beams

A seismic beam element was developed to simulate the beam-column connection behavior of the steel moment frames of the building. The effects of beam fracturing were considered in the models following the results of cyclic testing of two full scale mock-ups. In the beam plastic moment vs. rotation curves, after yielding, it was assumed that the loading on future plastic excursions would be elastic/perfectly-plastic up to the highest previous plastic moment recorded. Once degradation occurred, it was assumed the loading on future plastic excursions would be elastic/perfectly-plastic up to the degraded moment value.

Figure 2 shows the beam moment vs. equivalent plastic rotation curve developed to model all the built-up exterior frame beams. This curve had taken into account the panel zone plastic deformations so that explicit panel zone modeling was not needed to capture the combined effects seen in the laboratory tests on actual connections. Analytical results taken at the column face were scaled up geometrically to represent the centerline modeling approach that was used. The post-fracture modeling includes an initial negative slope that was used to minimize ringing of the model that would be more pronounced with infinite slope. Values for moment at initiation of plasticity, rotation at fracture, and degraded moment capacity were chosen to match results for the tested joints of LAC-1 and LAC-2 from UCSD report TR-2000/14 [4]. Strain hardening of 3% of the initial elastic slope was used per FEMA guidelines. The elastic rotation at yield was 0.0033 radians and the plastic rotation at plastic moment (M_p) was also 0.0033 radians, so moment (M) increases from M_p to 1.03 M_p for this range of plastic rotations.

Interior moment frame beams were also modeled as seismic-beam elements with 3% strain hardening behavior beyond yield at the member plastic moment (M_p).

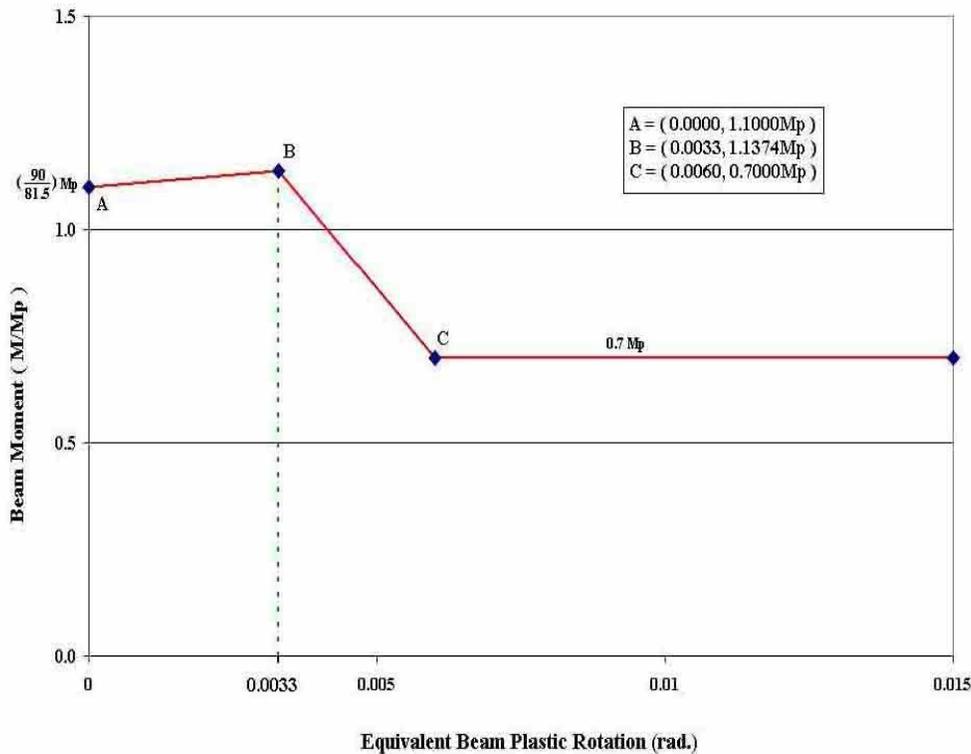


Figure 2 Moment vs. Equivalent Plastic Rotation Curve for DYNA Modeling

Column Panel Zones

Panel zones were not explicitly modeled. Beam and column centerline dimensions were used, and the results for building drift and period matched with SAP2000 [5] results to confirm that this modeling approach did not make the building too soft. However, the panel zone geometry was used to scale member end results. The detailed considerations were given as: (a) exterior moment frame beams were assumed to reach the member plastic moment (M_p) at the face of the column, so centerline results were increased to $1.10M_p$ based on relative geometries; (b) since the exterior moment frame columns did not have continuity plates, taking full benefit of the panel zone depth to allow for larger forces in the columns appeared to be unreasonable, and therefore, half the beam depth was used to scale up the results to the column end so that column plasticity initiated at $1.10M_p$; (c) since the bay length of interior frame beam was longer than that of exterior beam, it was assumed that the beam plasticity initiated at the beam end moments of $1.04M_p$; (d) since the interior frame beams were shallower than the exterior frame beams, it was assumed that the plasticity of the interior columns initiated at the moment of $1.08M_p$.

Validation of Moment vs. Equivalent Plastic Rotation Curve

Two cruciform analytical models were constructed and tested in DYNA matching the properties of the actual cruciform tests of LAC-1 and LAC-2. The displacement records of the test specimen were used as the loading input. Results from tests and DYNA were compared to corroborate centerline modeling and the moment-rotation curve as defined in Figure 2. Figures 3 and 4 show the results from the DYNA cruciform models and test specimens of LAC-1 and LAC-2. It can be seen that the initial elastic stiffness from the models and the test results matches very well.

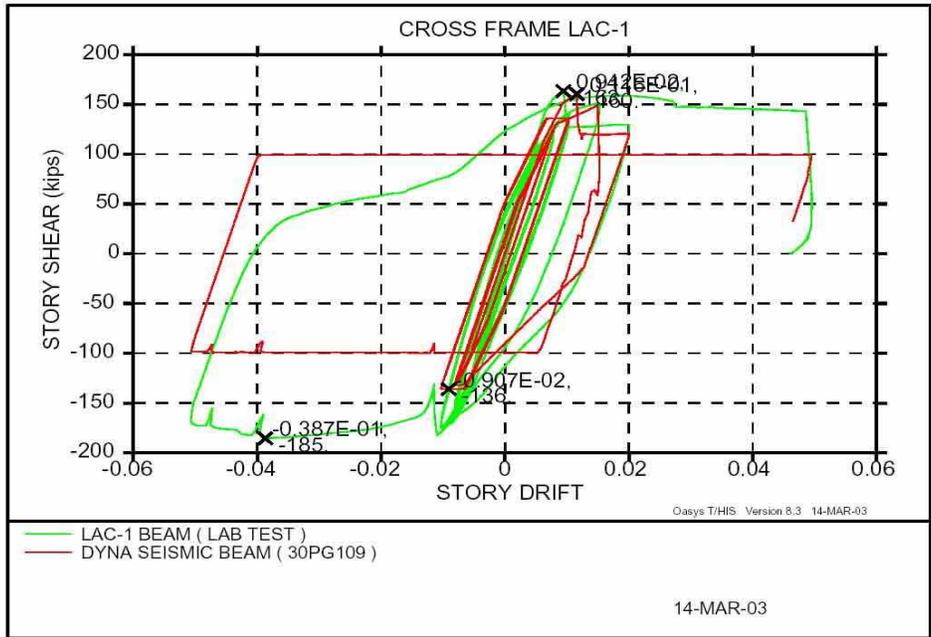


Figure 3 LAC-1 Laboratory Test vs. DYNA Cruciform Model Results

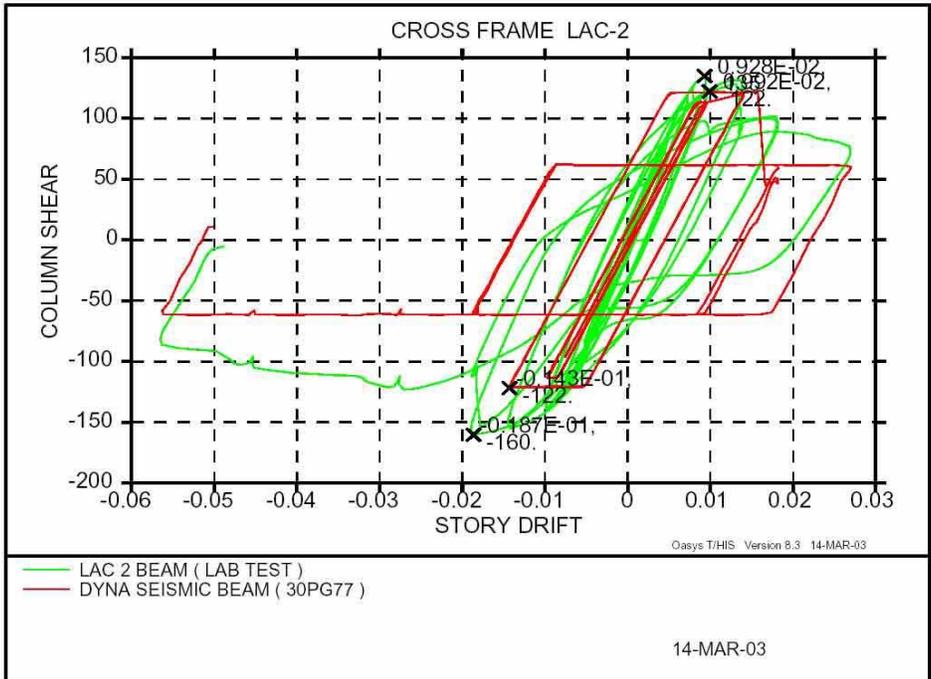


Figure 4 LAC-2 Laboratory Test vs. DYNA Cruciform Model Results

MODELING OF CABLE-STAYED RETROFITTING SYSTEM

The cable-stayed retrofitting system consisted of 16 high-strength main cables, placed at each of the four building faces, and 4 new pylons of steel pipe encased in reinforced concrete. The pylons were located away from the corners of the existing building. One end of the main cable was connected to the pylon at each floor level, and the other end was connected with a yield element and then anchored at the ground level. The pylons were further tied back to the existing floor diaphragms at the fifth floor up to roof level using buckling-restrained braces. Additional roof cables and steel beams were placed slightly above the existing roof diaphragm to drag the roof seismic forces back to the pylons. Figure 5 shows an isometric view of the DYNA model for the retrofitting system.

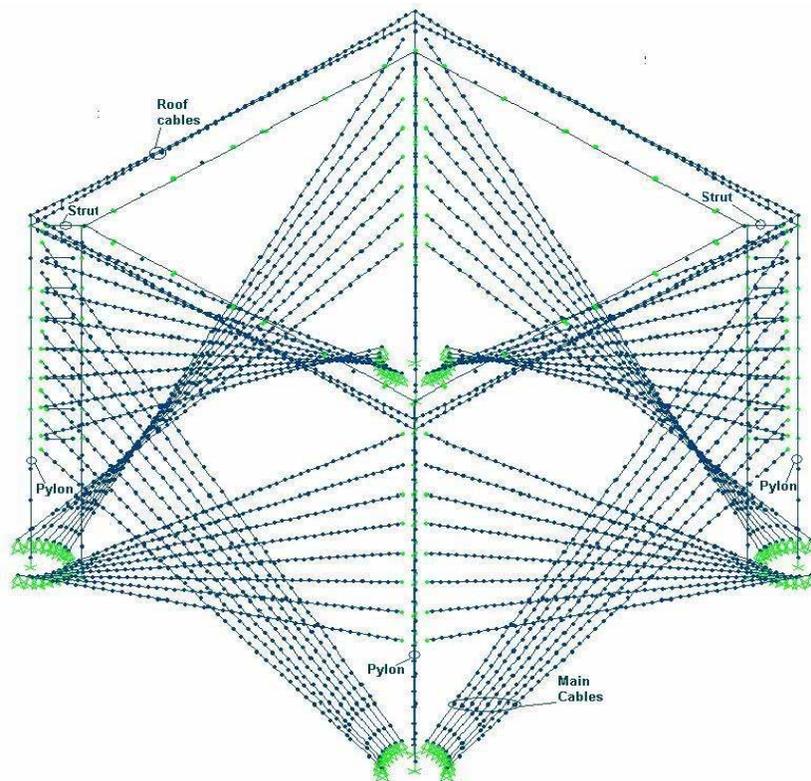


Figure 5 DYNA Models of Cable-Stayed Retrofitting System

High Strength Cables

The main cables were modeled using a material type of "MAT_CABLE_DISCRETE_BEAM". Elements using this type of material are able to transmit tensile forces, but not compressive. A separate test model of the cables on one face of the building was created to validate the modeling approach on sags in the cables. An initial slack was applied to the models, and the resulting sag and tension force were compared with the theoretical values that were calculated by assuming that the cable had a catenary curve. The initial slack was defined as the difference between the manufactured cable length and the cord length. It was assumed that the target pre-tension force in the cable was 70 kips. Table 2 lists the comparison of the sags that were calculated and the sags that were obtained from the test models.

In the full DYNA analysis models, the required value of slack from the test model analysis was applied to each of individual cables. The resulting tension forces in the cables were checked against the target pretension force of 70 kips.

The top roof horizontal cables were modeled with initial sag and a pre-tension force of 150 kips while the bottom roof cables were modeled with initial inverse sag and no pre-stress forces. The forces and resulting geometry were checked. The pre-tension forces were iterated until desired sag and force level were obtained.

The cable yielding elements were modeled using a non-linear spring element. This element can be defined to take either tension or compression or both, and can also be given yielding properties, which were approximated to bilinear elastic-plastic relationship with 3 percent of strain hardening. However, the cable yield elements in the simulation analysis were modeled as tension-only elements.

Table 2. Cable Theoretical Sag vs. DYNA Sag

Cable Location	Section Distance (in.)	Cable Slack (in.)	T' (TOP) (kips)	T'' (BOT) (kips)	THEORY SAG (in.)	DYNA SAG (in.)	THEORY/DYNA (%)
ROOF	2739.820	0.344	76.18	70	25.24	25.25	100.0
12 th	2617.590	0.330	75.55	70	25.19	25.20	99.9
11 th	2519.813	0.328	74.99	70	25.25	25.30	99.8
10 th	2433.643	0.337	74.43	70	25.46	25.50	99.8
9 th	2360.797	0.343	73.87	70	25.81	25.80	100.0
8 th	2303.486	0.364	73.31	70	26.33	26.40	99.8
7 th	2265.450	0.389	72.75	70	27.11	27.10	100.1
6 th	2256.021	0.439	72.19	70	28.38	28.40	99.9

New Pylons

The new pylons were attached to the existing structure through the corner struts and were further connected by the main cables. The elements selected in the analysis models can account for the interactions of the axial loads and bi-axial bending (P-M-M yield surface). Cracked section properties of the pylons were used in the models, where the flexural and shear stiffness properties were taken as one half of the gross section properties.

Buckling-Restrained Struts

The struts connecting the existing building to the pylons were also modeled as non-linear elements. The struts were made from buckling-restrained braces and treated as a gap/hook element with the material property type of "MAT_SPRING_GENERAL_NONLINEAR". The element did not have an initial clearance in the compression side ("gap" action). However, the element was assigned with a certain amount of initial clearance in the tension side ("hook" action). The initial clearance was implemented in the tension loading and the unloading part of the force-displacement relationship curve.

DYNA SIMULATION MODELS

Various DYNA simulation models were created using the given geometries and selected sets of material properties of the building. Figure 6 shows the DYNA model of the retrofitted building.

DYNA Modeling Features

In addition to the capabilities of general non-linear analysis programs, such as SAP2000, DYNA can account for the geometric non-linearity of the structure. This is particularly helpful in analyzing the larger displacements of the cable-stayed retrofitting system. DYNA can capture local instability of the structure and identify non-performing structural members of the building. In the analysis of moment frame connections, DYNA has the capability of modeling strength and stiffness degradation with seismic beam elements to simulate the FEMA-type beam-column connections. Usually, the seismic beam elements have one node at one end of the element that reports the plastic rotation and moment for the members. If a member is required to output the plastic rotations at both ends of the member, an additional joint can be created to split the member into two elements.

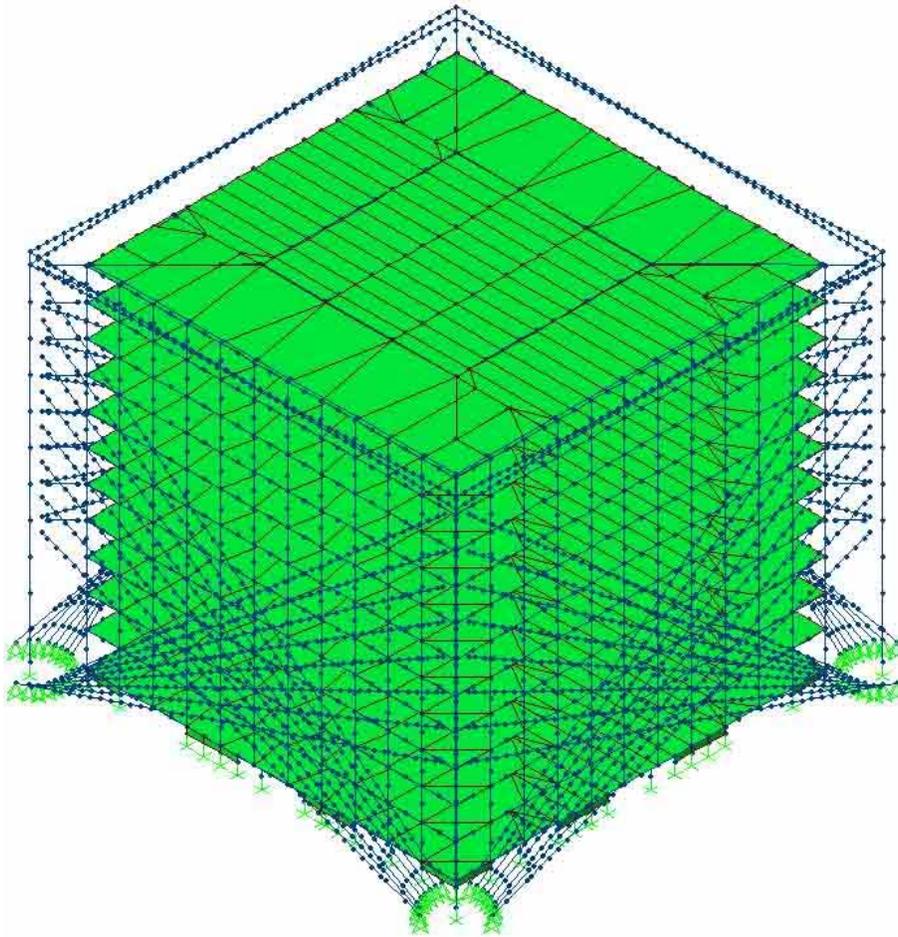


Figure 6 DYNA Simulation Model of Retrofitted Building

Floor and Roof Diaphragms

The existing floor and roof diaphragms were explicitly modeled with 2D membrane shell elements in the DYNA model. The shell element thickness was taken as equal to the thickness of concrete fill over the metal deck. The existing roof along the building perimeter areas does not support mechanical equipment, and hence was designed as architectural instead of structural topping slabs. The structural stiffness of this portion of the roof diaphragm was modeled as having 0.286 times the stiffness of the light-weight concrete of the typical floor. The model mass was spread over the elements to match the overall diaphragm mass. The diaphragm mass density was varied to produce 5 percent of floor mass eccentricity. The eccentricity was placed in the +x and +y directions

All new steel roof diaphragm elements were modeled as elastic elements. The steel members along the perimeter of the building as well as the diagonal members spanning between the perimeter and the interior structural slab were modeled as continuous elements. These steel members were connected to the existing roof diaphragm, but were not constrained in the axial deformation of the element except at the corners and core area. This would allow for direct transmitting of the roof seismic forces from the existing roof core areas to the pylons.

Ground Motions and Damping

Tri-directional ground motions were applied to the building supports in the DYNA models. The earthquake ground motions were represented by total of seven sets of acceleration time history records, each consisting of a pair of horizontal ground motion components and a vertical component, associated with the seismic hazard levels of Operational Level Earthquake (OLE) and Basic Safety Earthquake 2 (BSE-2). Acceleration records provided had orientations measured in degrees clockwise from North (+y axis). The DYNA model was also oriented with North as +y axis and East as +x axis.

Modal damping of 2% in the analysis was assumed over the frequency range of 0.333 to 10 Hz. Global damping was only used during the application of the cable pretension force and the gravity loads to damp out initial vibrations.

DYNA Results

To reduce the computational efforts, a representative set of ground motion records was selected for the preliminary DYNA analysis. The results from the DYNA run with the representative set of records were thoroughly reviewed and the modeling assumptions were evaluated. With the representative set of records, parametric studies were also performed on the retrofitting system. Design of the retrofitting system was then iterated based on the results from the DYNA analysis. After the final set of retrofit design parameters was determined, further DYNA analyses were carried out using the remaining six sets of ground motion records. The final retrofitting system and the building behavior were evaluated based on the median value of the responses from all seven sets of records.

The DYNA program can generate graphical time history results for many design parameters of interest. However, a certain type of responses such as actual time histories of forces and displacements must be flagged for reporting before proceeding with a run. Because of the large volume of results, it was not feasible to request all results, and therefore selective sets of results were pre-coded in DYNA's input file for the desired response time histories. When the maximum value of the results was requested, the time step at which it occurred was also extracted from the model.

Figures 7 through 10 show the base shears, roof displacements and beam-column plastic rotations of both the existing and retrofitted buildings for the Sylmar time-history records.

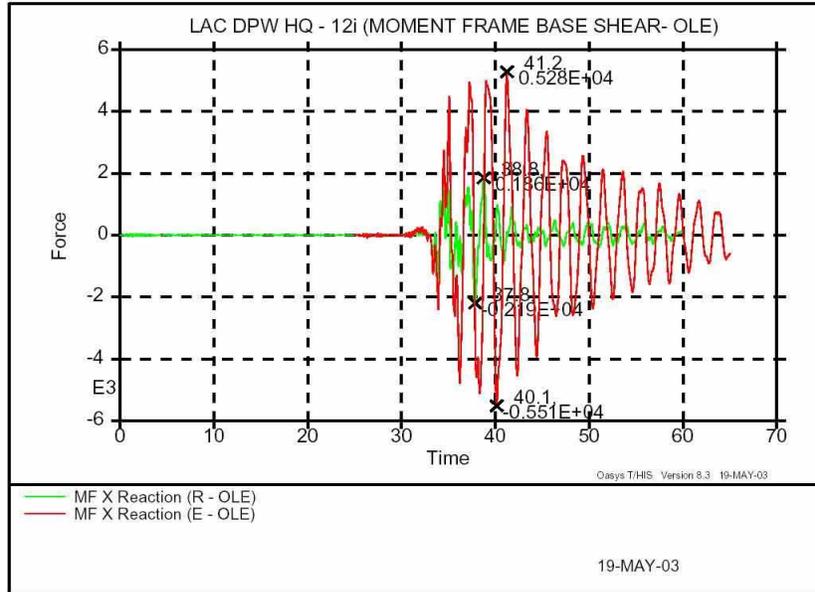


Figure 7 Base Shear of Existing and Retrofitted Buildings at OLE

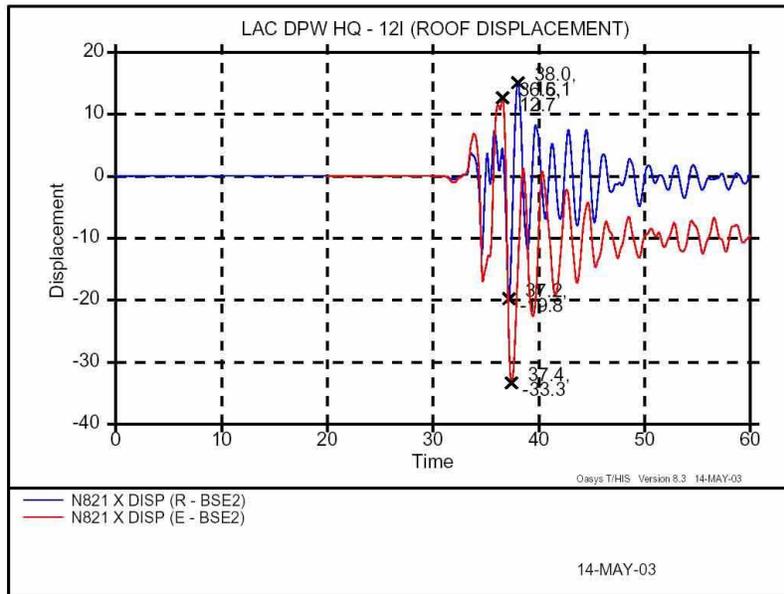
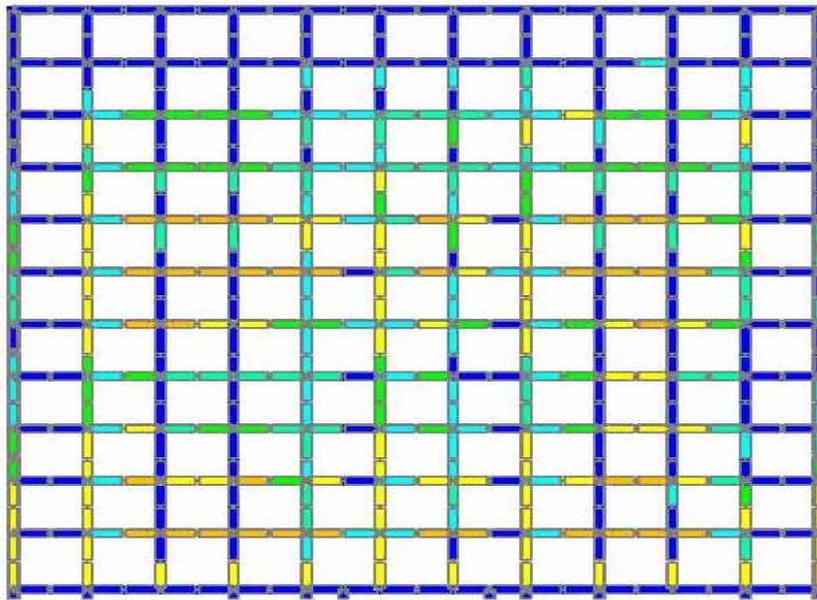


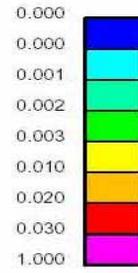
Figure 8 Building Drifts of Existing and Retrofitted Structures at BSE-2

OASYS D3PLOT: LAC DPW HQ-15 (EXIST-SYLAMR-OLE)



Z_PLAST_ROT_END1

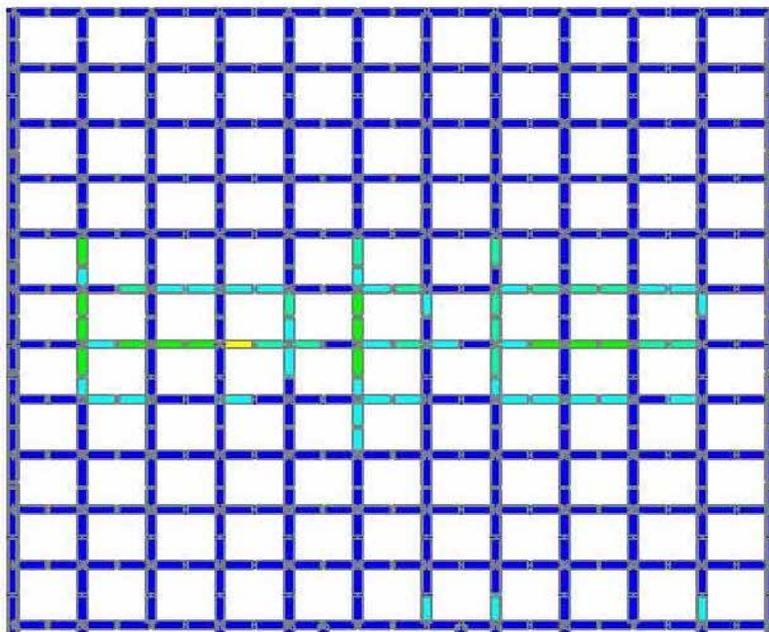
Beam results



35.000038

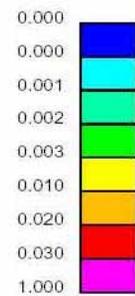
Figure 9 Beam and Column Plastic Rotation of Existing Building at OLE

OASYS D3PLOT: LAC DPW HQ-9B (RETRO-SYLAMR-OLE)



Z_PLAST_ROT_END1

Beam results



39.549637

Figure 10 Beam and Column Plastic Rotation of Retrofitted Building at OLE

CONCLUSIONS

The DYDA analysis models were created using the given geometries of the building and the design parameters of the retrofit scheme. A special seismic beam element was developed to simulate the beam-column connection behavior. The element considered the effects of steel moment frame beam fracturing following the observations made during the connection tests. The moment and rotation values at all the controlling points of the degrading curves were based on the test results. Modeling of column panel zones was simplified using the beam moment vs. equivalent plastic rotation curve. As the simulation always involves many analyses with thousands of time steps, the program run time is an important consideration in development of the model. However, the DYNA simulation models employed to assess the existing and retrofitted buildings were able to represent all the significant non-linear behaviors as expected in the retrofit design.

The seismic simulation approach described in this paper has been used to evaluate the building performance of the existing and retrofitted structures using non-linear time history analysis. The simulation analysis of the existing steel moment frame building retrofitted with the external cable-stayed system indicates that seismic performance was significantly improved, particularly in reducing the building base shear and the number of connection fractures in the operational level earthquake (OLE).

ACKNOWLEDGEMENTS

The assistance from Los Angeles County Department of Public Work, Black & Veatch Coporation, Arup Advanced Technology Group, and Henry Medina of Arup Los Angeles is greatly appreciated.

REFERENCES

1. LS-DYNA with Civil Engineering Application Extensions (version 8.0e), OASYS Limited, 13 Fitzroy Street, London, W1P6BQ.
2. Black & Veatch Corporation, "Basis of design report: seismic hazard mitigation for Los Angeles County Department of Public Works Headquarters Building", 2003, 1-1.
3. FEMA-352, Recommended post-earthquake evaluation and repair criteria for welded steel moment-frame buildings, prepared by the SAC Joint Venture for the Federal Emergence Management Agency, Washington, DC, 2000, 5-9.
4. Chi,B. and Uang, C.-M., "Seismic retrofit study on steel moment connections for the Los Angeles Department of Public Works Headquarters Building." Report No. TR-2000/14, Final Report Prepared for Black & Veatch, University of California, San Diego, La Jolla, CA, 2000.
5. SAP2000 nonlinear version 7.44, General 3-D static/dynamic finite element analysis and design, Computers and Structures, Inc., Berkeley, California.