Simulation of full-scale seismic-resistant structural frame tests using LS-DYNA 960 Implicit Solver

Authors:

CAROLINE J.FIELD
Ove Arup & Partners, California Ltd.

Correspondence:

Caroline Field
Ove Arup & Partners California Ltd.
Suite 260
901 Market Street
San Francisco
California, 94103

Tel: +001-(1)415-9460606
Fax: +001-(1)415-9579096
e-mail: caroline.field@arup.com

Keywords:
Implicit solver, nonlinear pseudo static, validation,
finite element modelling, seismic system, Unbonded Brace

ABSTRACT

This paper focuses on the finite element simulation of two full-scale tests of high performance, seismic-resistant structural frames using the LS-DYNA 960 implicit solver.

The frame was physically tested as part of the design validation for the new Stanley Hall building on the University of California Berkeley Campus.

The pseudo static non-linear analyses, showed excellent correlation with the measured test data. Two sequential tests were performed on the same frame but with different brace configurations, hence residual stresses and strains, and the process of brace replacement were important.

This work illustrates the convenience of implicit LS-DYNA for structural applications – transferring this technology to the built environment. It also provides confidence in and verification of the software.

The construction industry tends to shun non-linear analyses, deeming them too complicated; however it is ideal and indeed, essential for seismic applications. This simulation provides an alternative approach to full-scale testing for the future evaluation of this type of structure. It also provides the opportunity for the development of new and improved structural details as well as the retrofit assessment for existing structures.

INTRODUCTION

Unbonded Brace

Recent research in the US, Japan and elsewhere has led to the design of braces for use in seismic-resisting frames with improved performance characteristics. Generally, these braces employ a variety of techniques to restrain or avoid lateral and local bucking of the brace when it is loaded in compression.

The Unbonded Brace comprises a core of high ductility steel within a concrete matrix confined by a steel tube. The brace exhibits nearly identical properties in tension and compression and has the ability to undergo numerous cycles of inelastic deformations without degradation or fracture.

Analyses and tests of individual braces indicate that most buckling restrained braces are more durable and reliable than conventional braces. Analyses of complete structures suggest that buckling-restrained braces can substantially improve overall system behavior and reliability.

Berkeley Full-Scale Tests

The Seismic Review Committee for the Berkeley campus of the University of California recommended that large-scale physical tests be incorporated as an integral part of the design of the replacement structure for Stanley Hall, in which it is intended to incorporate Unbonded Braced Frames, which are a relatively new system.

The first of the three specimens had a chevron configuration, as shown in Figure 1. Specimens two and three had a single diagonal brace configuration, as shown in Figure 17.

Finite Element Test Simulation

The finite element simulation of the full-scale test was carried out to assess the ability of LS-DYNA to predict the behavior of such a structure. If successful, it could provide additional information on the structural behavior that was not monitored or measured during the test – e.g. the load in the Unbonded Braces and local stresses and strains at the connections.

The validation of this analysis tool also sets a precedent for future design schemes, which can then be analysed and designed with confidence, with reduced physical testing. This would give Arup a very useful tool for the validation and verification of building designs, which is particularly useful during peer and official reviews.

Tests 1 and 2 were simulated and the results are documented in this paper.

UNBONDED BRACED FRAME SIMULATION

Solution Procedure

The numerical simulation was carried out using the non-linear static implicit solution procedure of LS-DYNA 960. The analysis was pseudo static, directly simulating the cyclic pseudo static nature of the tests conducted. The implicit solution procedure is ideal for this application, with reduced analysis run time compared with the explicit time integration procedure.

The Implicit control cards used were:

*CONTROL IMPLICIT SOLUTION

*CONTROL IMPLICIT GENERAL

*CONTROL IMPLICIT AUTO

Default values were used for the above cards except for ITEOPT on the control implicit auto card which was increased to 100 to aid convergence. A load curve was specified for DTMAX.

Geometry

The frame is approximately 6100mm wide and 3600mm high and is shown in Figure 1. The lower storey is the frame that is being tested; the upper storey braces and top beam are a convenient method for applying the load.

The finite element model was constructed from fully integrated (type 16) shell elements. The typical element size used was 30mm x 30mm.

All connections in the finite element model (both welded and bolted) were fully meshed together. Whilst it is possible to model bolts and frictional interfaces, no slip was observed or measured during the tests, so the assumption of fixed connections was deemed valid.

The Unbonded Brace was simplified by only modelling the steel core explicitly and by simulating the effect of the concrete casing by meshing beams with bending stiffness only to the core to prevent it from buckling.

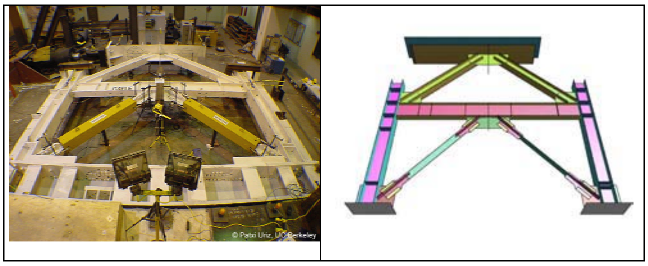

Figure 1 Test 1 Setup

Restraints

The test specimen was restrained at the base by a large steel built-up section, which in turn was anchored to concrete reaction blocks. The finite element (FE) model did not include these beams, as no significant movement was measured at this location during the test. Instead, the baseplates at the bottom of the columns were assumed to have a fully fixed (rigid) connection.

The FE model was restrained out of plane at the centre of the beams and at the top of the columns. The top roller connections (providing rotational restraint) were simulated with a frictionless contact surface between the top of the loading beam and rigid horizontal plane. This provided a rotational and vertical (compression only) restraint, thus allowing the frame to move vertically downwards (in plane of frame), as shown in Figure 2.

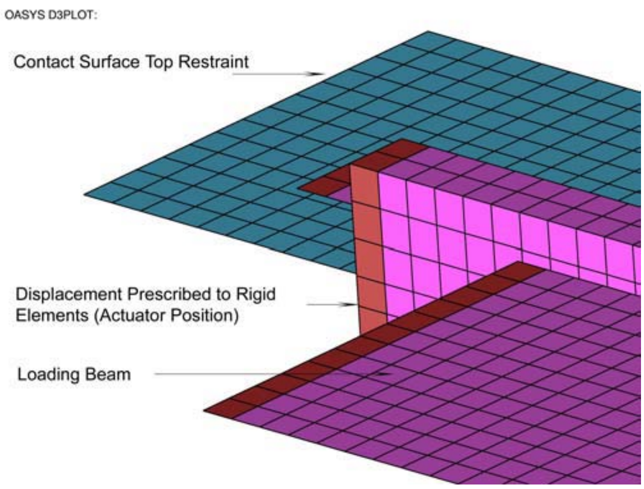

Figure 2 Contact Surface/Loading Beam Detail

Materials

Each Unbonded Brace contains a single interior flat plate. These were oriented perpendicular to each other during Test 1. The simulation used a bilinear steel model to represent the properties of each element. Material properties were taken or calculated from the mill test results and are presented in Table 1. Isotropic strain hardening was assumed. The keyword *MAT_PLASTIC_KINEMATIC was used.

Element	Size (ASTM)	Yield Strength MPa (Ksi)	Tangent Modulus MPa (Ksi)
Unbonded Brace (core plate)	0.75”x 8.5”	282.0 (40.9)	491.9 (71.3)
Column	W14x176	379.2 (55.0)	521.2 (75.6)
Beam	W21x93	372.3 (54.0)	569.0 (82.5)
Loading Beam / Brace	W10x112	379.3 (55.0)	730.9 (106.0)
Plate Steel	Varies	379.3 (55.0)	522.0 (75.7)

Table 1 Material Properties

Loading Protocol

The loading protocol was designed following the AISC/SEAOC Recommended Buckling-Restrained Brace Frame Provisions and is defined in Table 2. The control node was taken at the work point of the southern (left) column panel zone.

Symbol	Definition	Value mm (in)
Δ_b	Deformation quantity used to control loading of the test specimen (total brace end rotation for the sub-assemblage test specimen: total brace axial deformation for the brace test specimen)	
Δ_{bm}	Value of deformation quantity, corresponding to the design story drift.	9.4 (0.37)
Δ_{by}	Value of deformation quantity, at first significant yield of test specimen	44.5 (1.75)

Table 2 Loading Protocol definitions

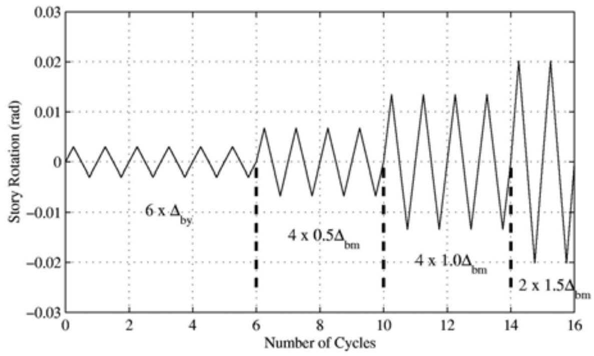

Figure 3 Loading Protocol

The test actuator was attached to a heavy built-up section of steel (the "loading beam") that was to remain elastic during the testing. From this built-up section, two W10x110 sections were welded to the frame via a 1" thick gusset plate to transfer the lateral shear force from the actuator to the sub assemblage.

The actuator was not included in the simulation. The end row of elements in the loading beam were made rigid and given the prescribed displacement taken from the test actuator reading. This is shown in Figure 2 and uses the keyword:

*BOUNDARY_PRESCRIBED_MOTION_RIGID.

TEST 1 RESULTS

Observation Comparison

Set 1 $\Delta_b = \Delta_{by}$

The simulation agreed well with the test observations. The braces yielded first followed by "hotspots" in the external columns stiffener elements and the column/brace gusset plate connections, as shown in Figure 4.

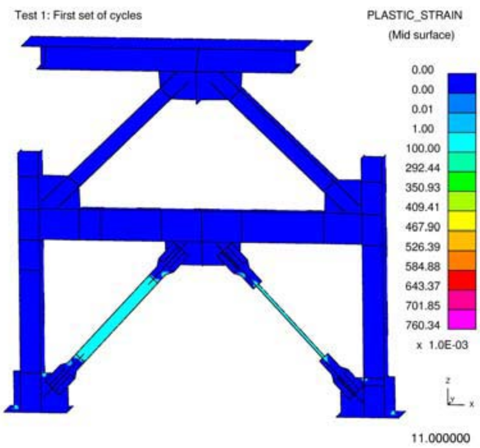

Figure 4 Plastic Strain at end of first set of cycles

Set 2: $\Delta_b = 0.5\Delta_{bm}$

The simulation at this displacement, shown in Figure 5, continued to match the observations of the test.

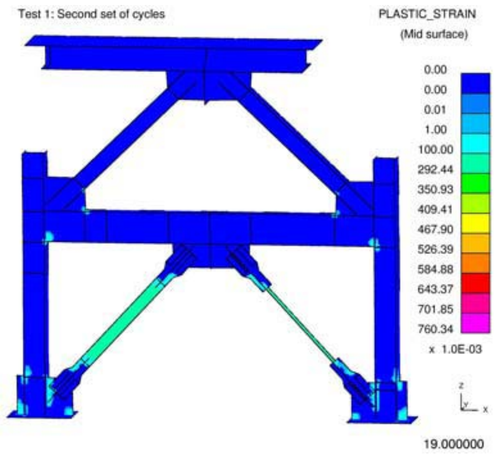

Figure 5 Plastic Strain at end of second set of cycles

Substantial yielding was seen in the column base stiffeners (Figure 6) and column/brace gusset plates. A small amount of shear yielding was indicated at the bottom of the columns above the gusset plate and a small amount of yielding was shown at the bottom of the beam-column connections, which was not noted during the actual test. The simulation results are plotted either for von Mises stress or plastic strain.

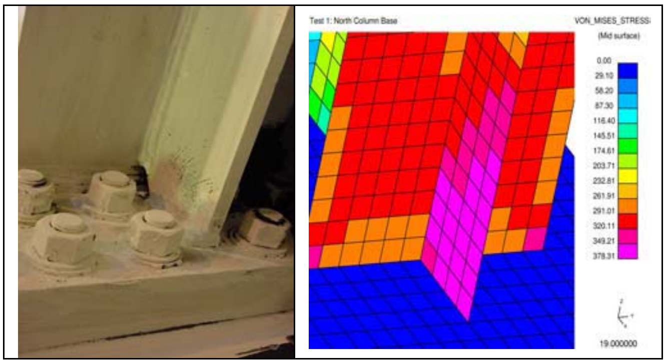

Figure 6 Column Stiffener Plate Yielding

Set 3: $\Delta_b = 1.0\Delta_{bm}$

The simulation agreed well with the test observations. High strain levels occurred in the braces, as shown in Figure 7.

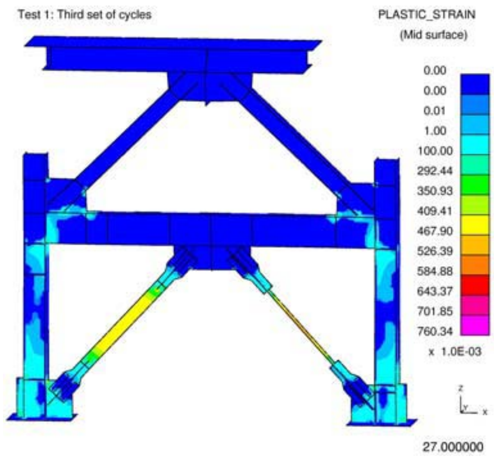

Figure 7 Plastic Strain at end of third set of cycles

Extensive yielding occurred in the column/brace gusset plates and column base stiffener and increased shear yielding was shown throughout the length of the columns. Substantial yielding occurred at the beam-column connections (Figure 8).

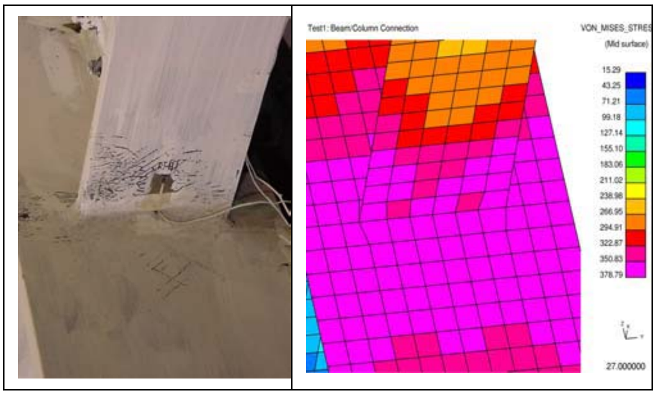

Figure 8 Beam-Column Connection

In addition to the observations during the test, some yielding occurred in the top of the beam flange and web at the loading brace gusset connection. No yielding occurred at the central beam/brace gusset plate connection.

Set 4: $\Delta_b = 1.5\Delta_{bm}$

The simulation again agreed well with the test observations. Plastic Strain results are shown in Figure 9 below.

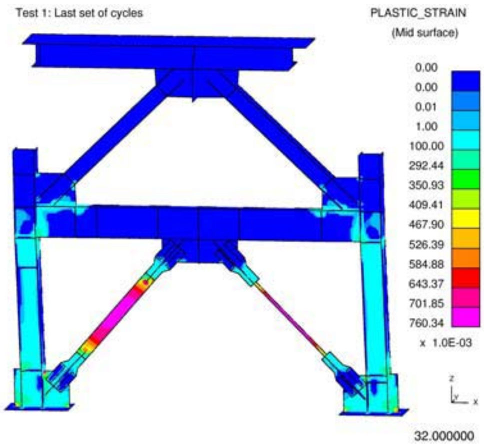

Figure 9 Plastic Strain at end of fourth set of cycles

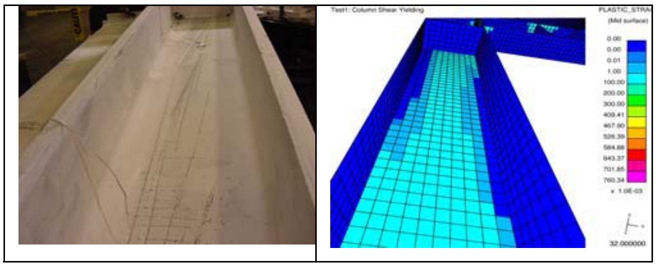

Figure 10 Column Shear Yielding

Shear yielding propagated throughout the entire length of the column web (Figure 10). The majority of the column/brace gusset plate yielded along with the column base stiffener (Figure 11). A few extreme "hotspots" of very high strain were shown at the bottom of the column base stiffener and at each corner edge of the column/brace gusset plate. These indicate serious problem areas, which were observed during the test and included stiffener fracture at the base of the columns.

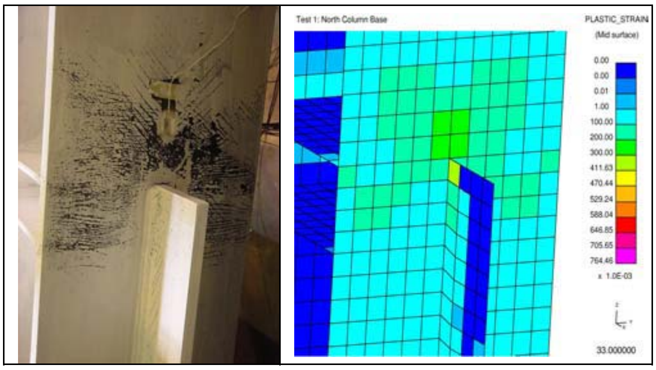

Figure 11 Column Stiffener Plate Yielding

Comparison of Force and Displacement

The control node displacement from the simulation is compared to the test displacement in Figure 13. The horizontal reactions from the simulation are compared to the test actuator horizontal force in Figure 14 and the peak force-displacement comparison is shown in Figure 15.

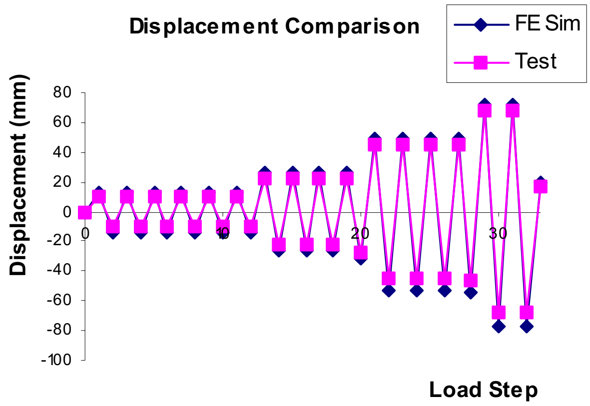

Figure 13 Displacement Comparison

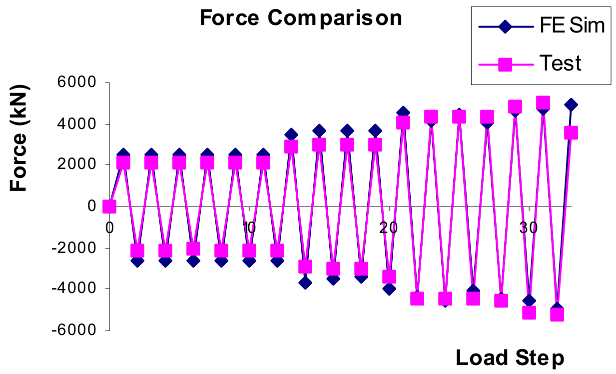

Figure 14 Force/Reaction Comparison

It can be seen from Figure 13 that the finite element simulation predicts slightly higher displacements at the control node than those from the test. The horizontal reactions (in Figure 14) are also predicted to be higher in the simulation, although both parameters show good correlation between analysis and test.

INTERMEDIATE STAGE

The second test on the Unbonded Brace frames at UC Berkeley was conducted on the Test 1 frame. The gusset plates and Unbonded Braces were removed and replaced, but the frame elements remained the same. The majority of these elements experienced substantial yielding during Test 1, hence the residual stresses and strains in the frame were likely to have a significant effect on the results of the second test.

In order to apply the correct stresses and strains to the start of Test 2 an intermediate analysis was required to simulate the removal of the Test 1 braces and subsequent relaxation of the frame, prior to the installation of the Test 2 brace. This frame is shown in Figure 15 below.

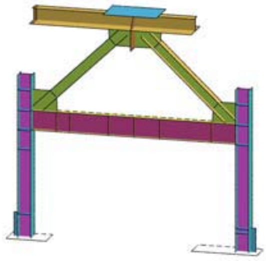

Figure 15 Intermediate Stage: Frame with braces removed

The Test 1 simulation was rerun with an additional end displacement added to get an unloaded condition to apply to the start of this intermediate analysis. This end displacement corresponded to zero (or near zero) reaction force. The stresses and plastic strains at this displacement are shown in Figure 16.

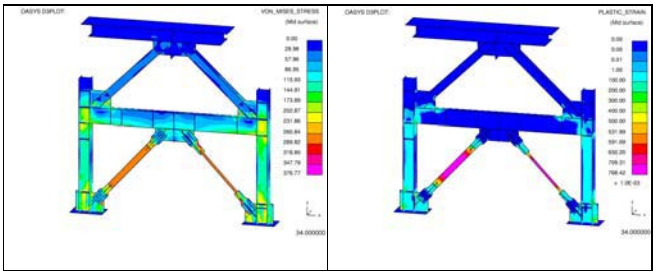

Figure 16 Stresses and Plastic Strains at end of Test 1

These final stresses and strains were output by part from test 1 using the following keyword:

*INTERFACE_SPRINGBACK_DYNA3D_THICKNESS

and applied to the model without braces shown in Figure 16 using:

*INITIAL_STRESS_SHELL.

The frame was allowed to settle and the final stresses and strains from this intermediate analysis were output for application to the start of Test 2.

TEST 2 SIMULATION

Test 2 Setup

The second test specimen had just a single diagonal unbonded brace rather than a chevron arrangement, as shown in Figure 17.

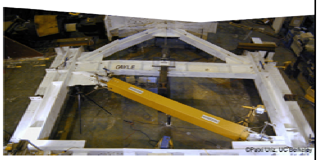	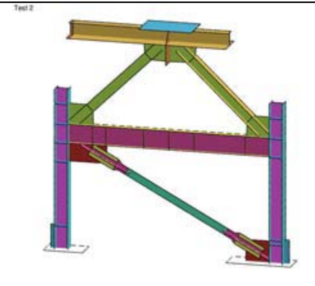
Test 1 Specimen	**Finite Element Model**

Figure 17 Test 2 Setup

The initial stresses and plastic strains for test 2 (output from the intermediate stage) are shown in Figure 18.

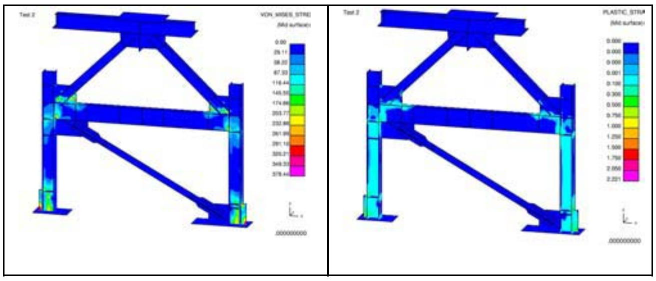

Figure 18 Initial Stresses and Plastic Strains for Test 2

TEST 2 RESULTS

Observation Comparison

The observation comparison for this test is a little more difficult, as the physical test frame was re-whitewashed prior to this test, so only indicates yielding during test 2, whereas the simulation shows cumulative yielding from test 1 and 2. However, effects such as buckling can be compared, as shown in Figure 19.

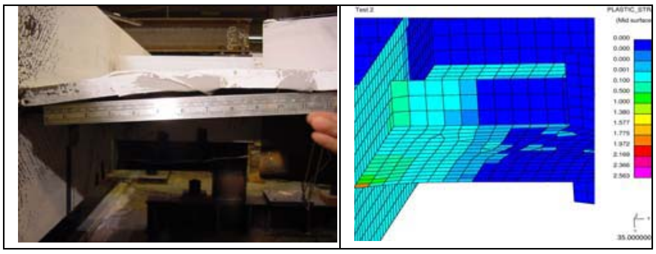

Figure 19 Buckling of gusset plate

This occurred when the brace was in tension, due to the crushing action of the frame.

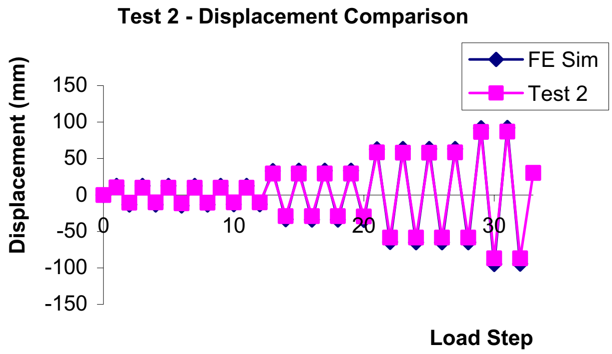

Figure 20 Displacement Comparison

The simulation closely matches the deformation observed in the test. The gusset plate fractured in the area of highest strain, shown by the pink area in the simulation plot.

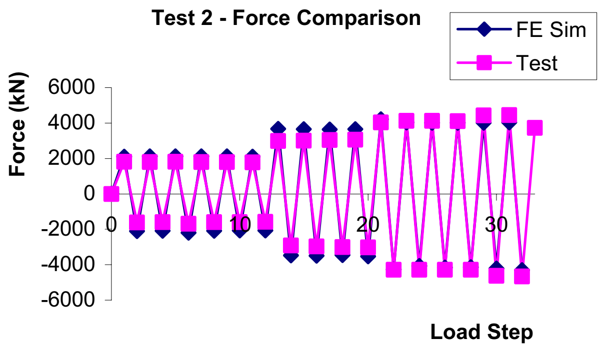

Figure 21 Applied Force/Reaction Comparison

The simulation agrees very closely with the test, as shown in Figures 20 & 21. Again, the displacements and reactions from the simulation are slightly higher than the test control node displacement and applied (actuator) force, respectively.

CONCLUSIONS

The validated simulations give confidence in the implicit finite element solution procedure within LS-DYNA. This application enabled pseudo static non-linear analysis simulations to be completed quickly, compared to more time-consuming explicit time-history analyses.

The ability to include residual stresses and plastic strains from previous/historic loading in subsequent analysis simulations is very valuable and has been shown to be successful.

This validated analysis methodology can now be used to design and verify similar structures. Connections and sections can easily be modified and re-assessed to improve the performance of the frame and to investigate the behaviour of new details.

This techique can now be used to complete virtual tests of similar structures with confidence, and can be used to develop more resistant, economical design solutions for both new and existing buildings.

ACKNOWLEDGEMENTS

Prof. Steve Mahin, UC Berkeley
Dr Patxi Uriz, UC Berkeley
Eric Ko, Arup
David Gration, Arup
Andrew Mole, Arup
Richard Sturt & Brian Walker, Arup
Neil Noble & Jack Pappin, Arup
Brad Maker, LSTC

REFERENCES

1. MAHIN, S. & URIZ, P. (2002) ”Full Scale Brace Frame Tests Containing Unbonded Braces”.
2. AISC (American Institute of Steel Construction) (1997) “Seismic Provisions for Structural Steel Buildings”, Chicago.
3. SEAOC/AISC (2001), “Recommended Buckling-Restrained Braced Frame Provisions”
4. LSDYNA Reference Manual