May 2003

LEAP -Australia

GISSETA - India

KOSTECH - Korea

CAD-FEM -Germany

Dr. Bhavin V. Mehta - US

Prof. Ala Tabiei - US

ALTAIR - Western Region US

MFAC -Canada

ERAB - Sweden

Numerica SRL - Italy

Dr. Taylan Altan - US

Dr. Ted Belytschko - US

DYNAMAX - US

DYNA*more* - Germany

THEME - Korea

ANSYS - China

Dr. David Benson - US

Cril Technology Simulation - France

FLOTREND - Taiwan

ALTAIR - Italy

Prof Gennaro Monacelli - Italy

Dr. Alexey I. Borovkov - Russia

FEA Information Inc. Worldwide News

May 2003

LS-DYNA Article

Simulation of full-scale seismic-resistant structural frame tests using LS-DYNA 960 Implicit Solver – Caroline J. Field – Ove Arup & Partners, California Ltd.

Fender Musical Instruments Fine Tunes Guitar Design by Coupling MSC Software and Silicon Graphics Hardware

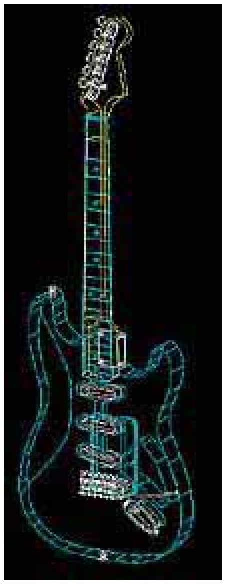
Reprinted from www.mscsoftware.com

March 1996 Model of the Month

Created by Mark Carlson
Fender Musical Instruments

In a stellar example of defense technology being converted to a non-military application, Fender Musical Instruments Corporation (www.fender.com), known worldwide for creating the first solid-body, Spanish-style electric guitar in 1948, is fine-tuning the design of its largely wooden bass guitars, using the same finite element software that was instrumental in developing the arresting gear that keeps jet fighters from plunging off aircraft carriers.

> "As a classical guitarist who studied finite element analysis (FEA) and solid modeling in college, applying the technology to the development of musical instruments was a natural progression for me," explained Mark Carlson, a young mechanical design engineer in Fender's research and development department,. Carlson holds a mechanical engineering degree with an emphasis in vibration and acoustics from Michigan Technological University in Houghton, Michigan.
>
> Unlike earlier generations of engineers who compartmentalized the use of FEA technology into discrete functions, Carlson is representative of today's innovators who are exploring the full potential of predictive engineering, in which analysis and design take place simultaneously and are performed by the same person.
>
> "The integration of analysis, solid modeling, and pre- and postprocessing makes it possible to create better designs more quickly, while maintaining greater control over the development process," he says. "This approach not only ensures the quality of the designs, but reduces the time required to get a product to market. It also lowers production costs by reducing prototype testing and minimizing the waste of material."



FEA for Wood

> Unlike typical FEA applications, in which the products being modeled are made of metal or a composite material, a guitar made of wood has a number of unusual forces acting on it that must be stabilized. Carlson spent a great deal of time studying *The Wood Engineering Handbook* in order to understand the properties of wood.
>
> "Wood can behave very strangely. It has a lot of variables and odd quirks. As a consequence, we needed extremely flexible solver software that could handle a very long solution

process, which is why we selected MSC.NASTRAN for the FEA calculations. The software is ideal for modeling wood because it can simulate what's called the orthotropic characteristics, or complex growth patterns, of trees (in a cylindrical fashion, depending on how the log was cut). I can't think of any other analysis method for something this complex except physical testing. It works perfectly for our application," Carlson explains.

UNIX-based MSC.Aries software was chosen for the pre- and postprocessing stages of the design cycle, providing interactive solid modeling, parametric modification, and automatic meshing. "In addition to rapidly building solid models of the guitar neck, the software makes it very easy to interpret the results and then quickly make any necessary modifications," noted Carlson. "MSC.Aries is an easy tool to use and is geared to engineers who have design responsibilities rather than to specialists in analysis."

Silicon Graphics Platform for MSC Tools

At Fender, MSC.Nastran and MSC.Aries run on a Silicon Graphics Indy R4600SC workstation with three gigabytes of hard disk, 32 megabytes of RAM and 24-bit graphics.

"We looked at three hardware systems, but after we saw the Silicon Graphics demonstration there was really no competition," said Carlson. "Silicon Graphics' reputation for excellent graphics capabilities is well deserved. I deal with a lot of people who aren't engineers, so being able to graphically display my work is very important. It's so much easier to explain something that people can visualize. I also know that whenever Silicon Graphics comes out with a new chip or hardware upgrades, MSC is ready to support it. Besides being very price competitive, the Silicon Graphics system came with a lot of bundled software that turned out to be very useful."

Using Showcase, a presentation tool from Silicon Graphics' Mindshare (family of collaboration software), Carlson is able to take snapshots out of MSC.Aries and quickly turn them into a first-rate presentation. "It's like cutting a picture out of a magazine and pasting it wherever I need it." With few clicks of the mouse, he can add text, move objects around, place arrows to highlight critical areas and even animate the demonstration to show vibration.

Merging Wood and Graphite

At the time Carlson joined Fender, the design was already in place for a graphite-reinforced neck for its new line of American Standard bass guitars, which were about to be introduced. It was an intuitive design that incorporated two strips of graphite running the length of the neck. Using MSC.Nastran to solve for the deflections, Carlson studied how much the neck moved when string forces were applied and moisture content changed. According to Carlson, wood soaks up moisture and shrinks differently in each cylindrical direction of the grain, which can vary greatly from neck to neck, so stabilization is very important. This is of particular significance to professional musicians whose instruments are subject to a great many temperature and humidity changes when they travel by air.

Using the MSC.Nastran calculations, Carlson could try different reinforcement scenarios to increase neck stability. MSC.Aries was used to generate the stress contour plots that compared critical pressure points, enabling him to determine inadequacies in the two-strip system and subsequently refine the design for optimum performance under all loading conditions.

"Computer tools like these allow us to validate a design long before it goes to manufacturing," Carlson says. "We can determine where stress is concentrated, where the design is going to fail, and what the safety factors are. In this case, we looked at how graphite would absorb the load and how much was needed to stabilize the neck."

MSC.Nastran was also used to analyze the truss rod that runs through the neck of the guitar. Even though the basic design has remained relatively unchanged since its invention, the rod can be hard to adjust and has some limitations in terms of twisting. "The truss rod is basically a long, thin piece of steel running down a curved channel in the neck," Carlson says. "A screw at the end of the rod allows for the tensioning of this cable, and when you turn the screw, the rod, which is in contact with the wood, forces the neck to move. Through nonlinear slideline contact analysis, we can optimize the slot to determine what yields the best adjustment."

Fender's R&D department is also looking at ways to eliminate "dead spots"—notes that sound dead or don't resonate as well as others. "We're examining a number of scenarios comparing the effects of vibration to determine how we can keep frequencies from interfering with each other. One way we've found effective is to solve for the next natural frequencies and mode shapes, then tune any detrimental modes away from musical notes in the bass range.

"We're also studying fret buzz. If guitar strings are too high off the fret board, the performer has to press too hard. If they're too close, you get a buzz. The range of playability is quite small and reflects the quality of an instrument in many players' eyes. This is why precision analyses, such as those done with MSC.Nastran, have become so vital in our goal to create good-playing, high-quality instruments at all price levels."

Listening for the Future

Some of Carlson's future plans include acoustic analysis to understand better how sound waves travel through the body of an acoustic guitar, and modeling composite materials for a possible solution to the inconsistency of wood. "A guitar is a high-precision tool with a lot of strange forces acting on it, so we're always looking at ways for improvement," he concluded.

The corporate offices of Fender Musical Instruments are in Scottsdale, Arizona. Its United States manufacturing headquarters are in Corona, California, which is also the site of the Fender Custom Shop, where the world's greatest guitarists have instruments created to their own specifications. Fender manufactures and distributes acoustic and electric guitars, amplifiers, professional sound equipment, stage lighting, and accessories. The company's legendary Stratocaster®, introduced in 1954, became the industry standard in the hands of rock and roll icons such as Buddy Holly and Jimi Hendrix. The company's Signature Series honors world-class musicians such as Eric Clapton, Jeff Beck, Robert Cray, Stevie Ray Vaughan, Bonnie Raitt, and James Burton.

California Department of Justice Bureau of Criminal Identification and Information

Reprinted from www.necsolutions-am.com

California needed an automated system to accurately identify criminal suspects and maintain arrest records. The NEC Automated Fingerprint Identification System (AFIS) enabled the Department of Justice to compile a database of more than 15 million fingerprint records that can be instantly queried.

Challenge

Tasked with managing a massive database of criminal and state employee fingerprints, the California Department of Justice (DOJ) required a solution that could automate and expedite criminal identification.

The California Department of Justice turned to NEC Solutions America to implement their best-of-breed identification solution.

Solution

To provide the reliability and scalability demanded by the DOJ, the NEC AFIS21™ (Automated Fingerprint Identification System) was chosen. More than any other system of its kind, AFIS21™ satisfies a broad range of capacity, performance and functional needs.

AFIS is the only proven technology that can verify and identify a person from a database of more than one million records. This feature was crucial to the DOJ, as their criminal database was rapidly growing. "The system we had before contained only 250,000 records," says Gary Cooper, assistant chief of Criminal Identification. Using the AFIS system, the Bureau has compiled a database of more than 15 million fingerprint records. This figure represents the largest regional fingerprint database in the world.

Benefit

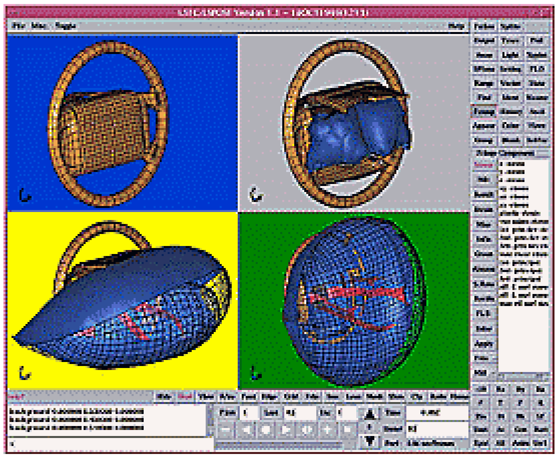
According to Cooper, the implementation of the NEC AFIS21™ created new identification abilities as well as increased performance in tracking criminals. "The previous system was strictly for crime scene work. NEC Solutions America helped us develop an identification system for all subjects, not just for the crime scene, so it has been a big improvement, mainly because the capabilities and database are much broader," says Cooper. After converting from a manual fingerprint identification system, Cooper has realized a 25% increase in identification accuracy rates and a tenfold improvement in the time needed for fingerprint record processing.

"This automation has been a great improvement for us right from the beginning," comments Cooper. The first search using AFIS produced the identity and subsequent arrest of the infamous "Night Stalker," who had remained at-large for several years. Cooper also cites AFIS as the key to unraveling more than a quarter of the Bureau’s unsolved homicides, some over 15 years old. In total the NEC system has helped to identify over 50,000 individuals since 1985.

Gary Cooper sees many ways for AFIS technology to assist the California Department of Justice in the future. "We're interfacing so we can get immediate real-time identification. We are incorporating the live-scan electronic fingerprinting with AFIS to provide identification at the point of arrest while suspects are still in custody. Rather than having it done over a two-week period, we can do it in two hours."

Larger databases, higher accuracy rates and quicker searches are just a few ways that NEC Solutions America helped the California Department of Justice keep pace with criminal identification demands in the state of California.

Pre Post Processing Product Information
For complete product information
please visit the company's website



Livermore Software Technology Corporation

LS-PrePost

The graphic user interface was carefully crafted to create a user friendly environment. It supports the latest OpenGL® (SGI) standards to provide fast rendering for fringe plots and animation results.

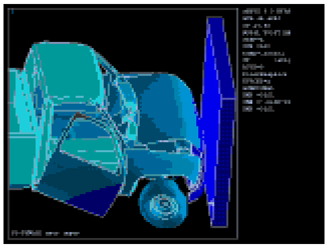
Site URL: www.lstc.com

Engineering Technology Associates

FEMB

ETA's Finite Element Model Builder is a finite element pre- and post-processor for use with all major analysis codes and CAD software. Developed in the mid 80's by Engineering Technology Associates, Inc., FEMB has been consistently updated and improved to become the high-power pre- and post- processor represented in Version 27.

Site URL: www.eta.com

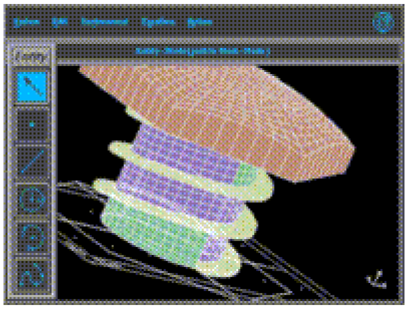


ANSYS Inc.

ANSYS/LS-DYNA

Performs difficult dynamic nonlinear calculations such as deformations, loads, and collision-related stresses. These capabilities make it ideal for performing simulations involving metal forming, crash dynamics, drop tests, and rapid manufacturing processes.

Site URL: www.ansys.com

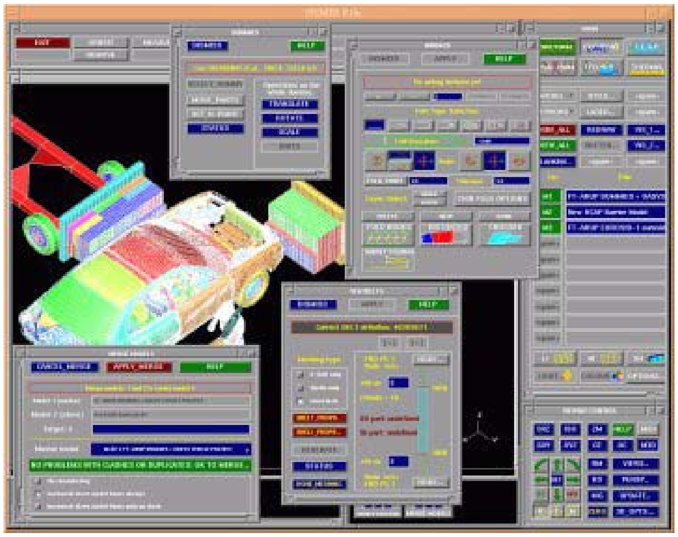


Japanese Research Institute, Ltd.

JVISION

A general purpose pre-post processor for FEM software. Designed to prepare data for, as well as support, various types of analyses, and to facilitate the display of the subsequent results.

Site URL: www.jri.co.jp



Oasys Ltd.

Oasys Primer

A model editor for preparation of LS-DYNA input decks.

Oasys D3Plot

A 3D visualization package for post-processing LS-DYNA analyses using OpenGL® (SGI) graphics.

Site URL: www.arup.com/dyna

MSC Software

MSC.Patran

Provides a complete software environment for companies performing simulation of mechanical products.

A finite element modeler, MSC.Patran enables the user to conceptualize, develop and test a product using computer-based simulation prior to making manufacturing and material commitments.

Site URL: www.mscsoftware.com

Special Announcements
Highlights of News Pages

Posted on FEA Information archived on month on News Page

April 7	**LSTC**	Version 970 now available
	ANSYS	Ansys FEMXplorer™
April 14	**ETA**	ea/FEMB27-PC
	HP	HP Superdome
	Altair Italy	Distributor
April 21	**MSC. Software**	Patran
	JRI	JMAG
	Cril – France	Distributor

Events & Courses from the Events page on www.feainformation.com

May 19	**BETECH 2003**	**USA**
May 22 – 23	**4th European LS-DYNA Conference**	**Germany**
June 3-5	**Testing Expo**	**Germany**
June 4-5	**MSC.Software Virtual Product Development Conference**	**Germany**
June 9	**Dr. Paul Dubois Course held by Numerica, SRL**	**Italy**
June 12-13	**SGI – 2003 User's Conference**	**USA**
June 17-20	**2nd M.I.T Conf. on Computational Fluid & Solid Mechanics**	**USA**
June 24-26	**MSC.Software Virtual Product Development Conference**	**Germany**
Oct 02-05	**Int'l Conference on CAE**	**Italy**
Oct 29-31	**Testing Expo North America**	**USA**
Nov 12-14	**CAD-FEM User Conference**	**Germany**
Nov 18-19	**MSC.Software Virtual Product Development Conference**	**UK**

FEA Information Participants
Commercial and Educational

Headquarters	Company	
Australia	Leading Engineering Analysis Providers	www.leapaust.com.au
Canada	Metal Forming Analysis Corp.	www.mfac.com
China	Ansys - China	www.ansys.com.cn
France	Cril Technology Simulation	www.criltechnology.com
Germany	DYNAmore	www.dynamore.de
Germany	CAD-FEM	www.cadfem.de
India	GissEta	www.gisseta.com
Italy	Altair Engineering srl	www.altairtorino.it
Italy	Numerica srl	www.numerica-srl.it
Japan	The Japan Research Institute, Ltd	www.jri.co.jp
Japan	Fujitsu Ltd.	www.fujitsu.com
Japan	NEC	www.nec.com
Korea	THEME Engineering	www.lsdyna.co.kr
Korea	Korean Simulation Technologies	www.kostech.co.kr
Russia	State Unitary Enterprise - STRELA	www.ls-dynarussia.com
Sweden	Engineering Research AB	www.erab.se
Taiwan	Flotrend Corporation	www.flotrend.com
UK	OASYS, Ltd	www.arup.com/dyna
USA	INTEL	www.intel.com
USA	Livermore Software Technology	www.lstc.com
USA	Engineering Technology Associates	www.eta.com
USA	ANSYS, Inc	www.ansys.com
USA	Hewlett Packard	www.hp.com
USA	SGI	www.sgi.com
USA	MSC.Software	www.mscsoftware.com
USA	DYNAMAX	www.dynamax-inc.com
USA	AMD	www.amd.com
Educational Participants		
USA	Dr. T. Belytschko	Northwestern University
USA	Dr. D. Benson	Univ. California – San Diego
USA	Dr. Bhavin V. Mehta	Ohio University
USA	Dr. Taylan Altan	The Ohio State U – ERC/NSM
USA	Prof. Ala Tabiei	University of Cincinnati
Italy	Prof. Gennaro Monacelli	Prode – Elasis & Univ. of Napoli, Federico II
Russia	Dr. Alexey I. Borovkov	St. Petersburg State Tech. University

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

By

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

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.

1. INTRODUCTION

1.1 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.

1.2 Berkeley Full-Scale Frame 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 18.

1.3 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.

2. UNBONDED BRACED FRAME SIMULATION

2.1 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.

2.2 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.

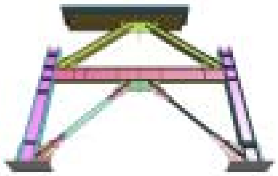
	
Test 1 Specimen	**Finite Element Model**

Figure 1 Test 1 Setup

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.

2.3 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.

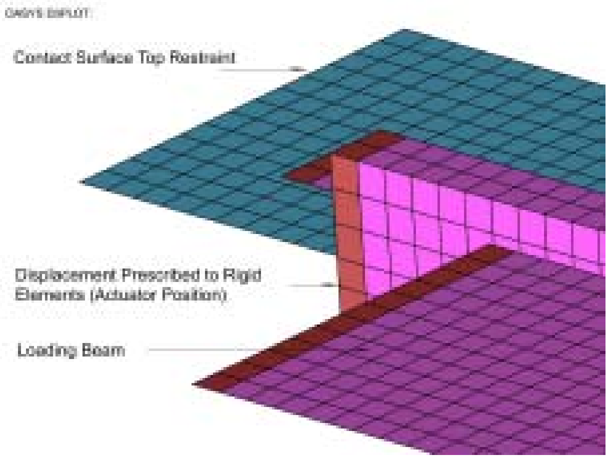


Figure 2 Contact Surface/Loading Beam Detail

2.4 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

2.5 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

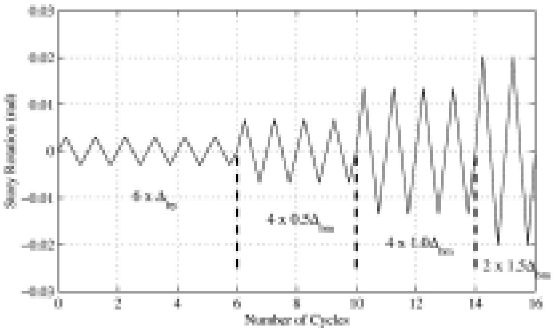


Figure 3 Loading Protocol

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.

3. TEST 1 RESULTS

3.1 Observation Comparison

3.1.1 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.

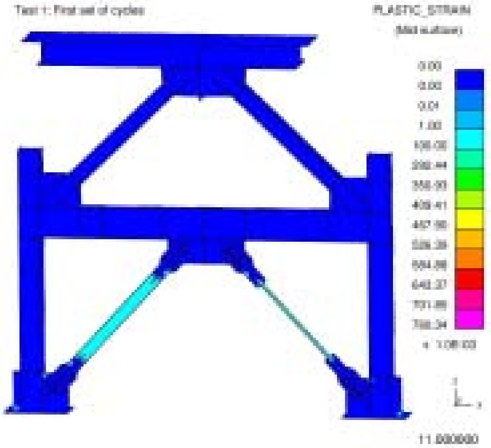


Figure 4 Plastic Strain at end of first set of cycles

3.1.2 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.

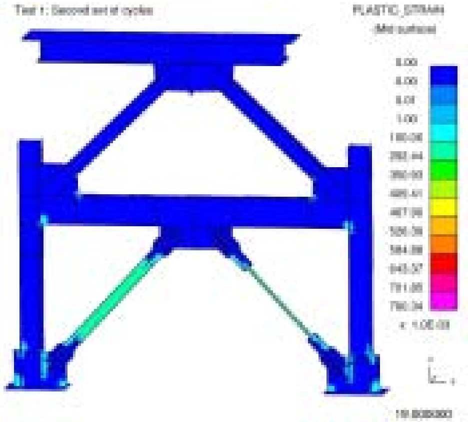


Figure 5 Plastic Strain at end of second set of cycles

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

3.1.3 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.

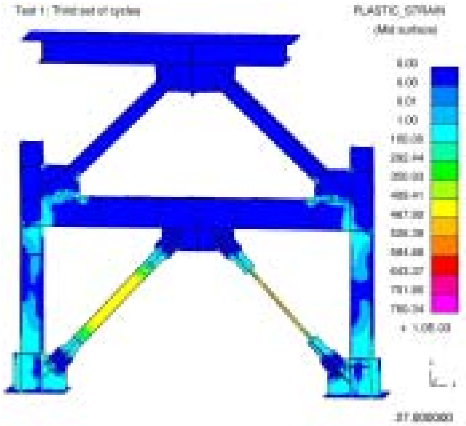


Figure 7 Plastic Strain at end of third set of cycles

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).

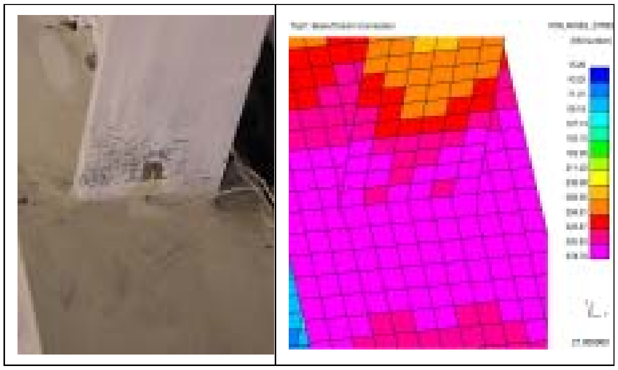
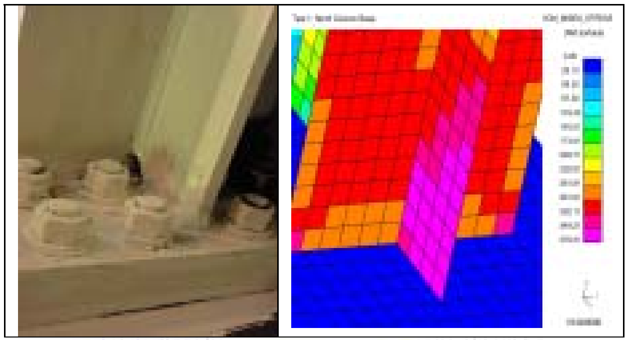


Figure 8 Beam-Column Connection

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.

3.1.4 $\Delta_b = 1.5\Delta_{bm}$

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



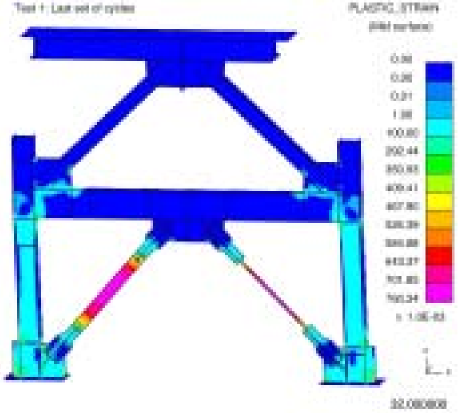


Figure 9 Plastic Strain at end of fourth set of cycles

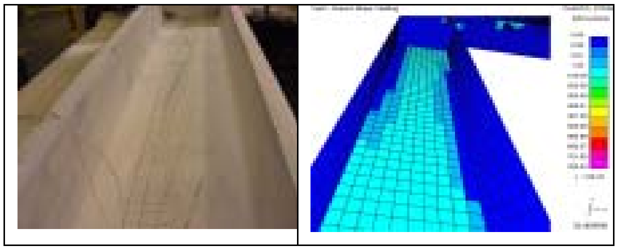


Figure 10 Column Shear Yielding

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

3.2 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.

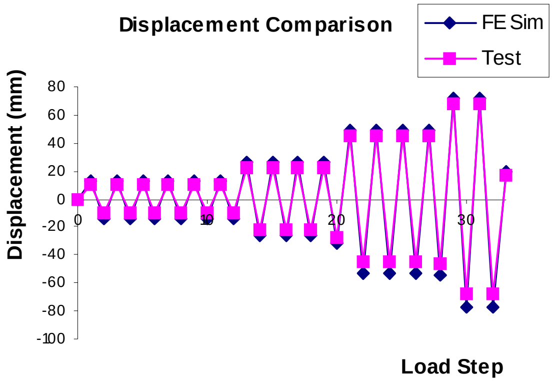


Figure 13 Displacement Comparison

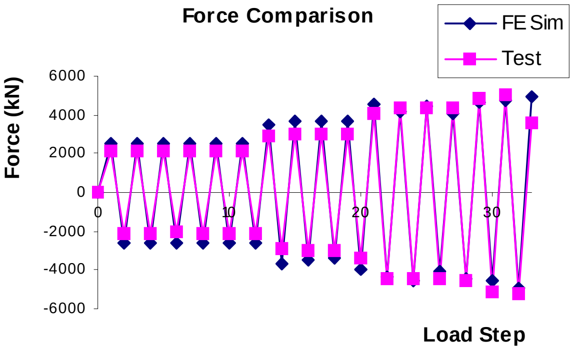


Figure 14 Force/Reaction Comparison

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. This difference could indicate that the stiffness of the upper section of the frame (loading braces and beam) is slightly too stiff in the finite element model.

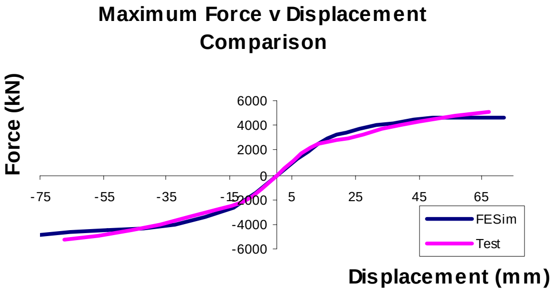


Figure 15 Force/Displacement Comparison

The plot in Figure 15 does not show hysteresis loops as only the maximum and minimum values have been plotted, but shows a good comparison between test and simulation.

4. 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 16 below.

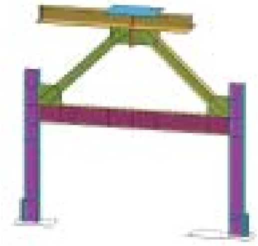


Figure 16 Intermediate Stage: Frame with braces removed

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 17.

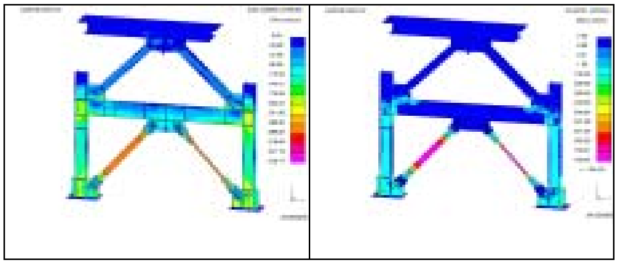


Figure 17 Stresses and Plastic Strains at end of Test 1

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 17 using:

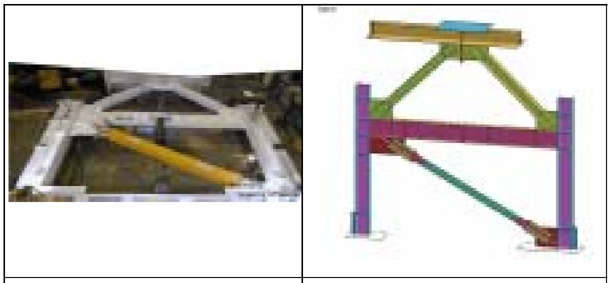
```
*INITIAL_STRESS_SHELL.
```

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.

5. TEST 2 SIMULATION

5.1 Test 2 Setup

The second test specimen had just a single diagonal unbonded brace rather than a chevron arrangement, as shown in Figure 18.



Test 1 Specimen | **Finite Element Model**

Figure 18 Test 2 Setup

The initial stresses and plastic strains for test 2 (output from the intermediate stage) are shown in Figure 19.

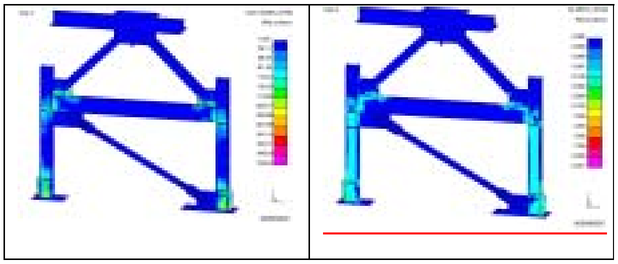


Figure 19 Initial Stresses and Plastic Strains for Test 2

5.2 Test 2 Results

5.2.1 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 20.

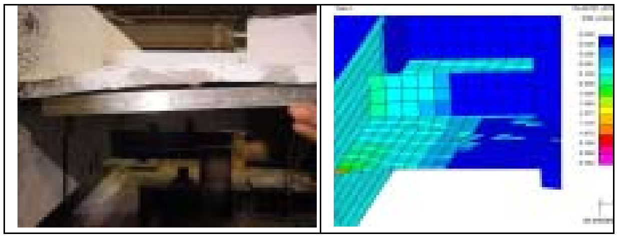


Figure 20 Buckling of gusset plate

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

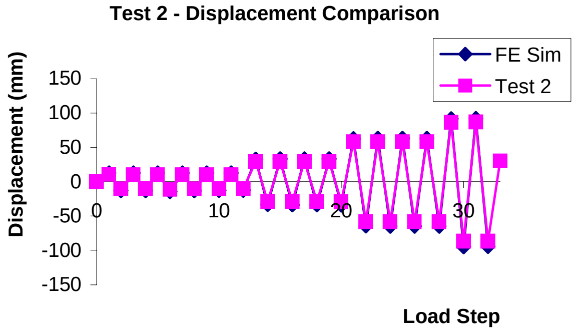


Figure 21 Displacement Comparison

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.

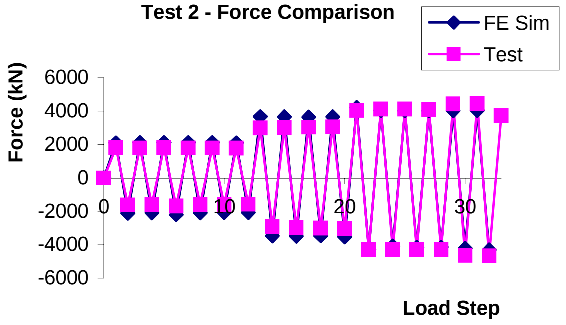


Figure 22 Applied Force/Reaction Comparison

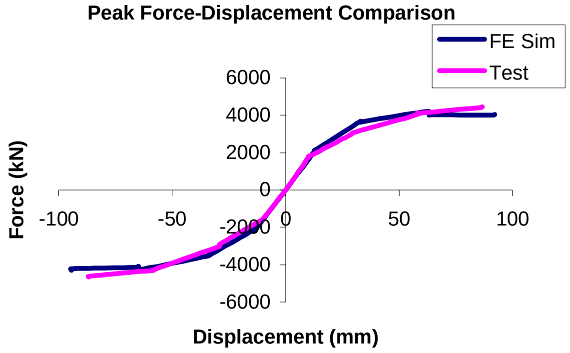


Figure 23 Force/Displacement Comparison

The simulation agrees very closely with the test, as shown in Figures 21-23. Again, the displacements and reactions from the simulation are slightly higher than the test control node displacement and applied (actuator) force, respectively. However, the force-displacement plot in Figure 23 shows a good stiffness match with the test.

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. S.Mahin & P.Uriz (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