

SeismoStruct

Verification Report

For version 2018

Copyright

Copyright © 2002-2018 Seismosoft Ltd. All rights reserved.

SeismoStruct® is a registered trademark of Seismosoft Ltd. Copyright law protects the software and all associated documentation.

No part of this report may be reproduced or distributed in any form or by any means, without the prior explicit written authorization from Seismosoft Ltd.:

Seismosoft Ltd.
Piazza Castello 19
27100 Pavia (PV) - Italy
e-mail: info@seismosoft.com
website: www.seismosoft.com

Every effort has been made to ensure that the information contained in this report is accurate. Seismosoft is not responsible for printing or clerical errors.

Mention of third-party products is for informational purposes only and constitutes neither an engagement nor a recommendation.

Finally, it is noted that this report has been prepared in close collaboration with the European Centre for Training and Research in Earthquake Engineering (EUCENTRE, Pavia, Italy), which provided access to technical expertise, experimental results and independent hand-calculation examples.

Table of Contents

SeismoStruct	1
Verification Report	1
Copyright	3
Chapter 1 INTRODUCTION	7
Presentation of the analysis program.....	7
Structure of the report.....	7
Program features covered by the program	7
Chapter 2 COMPARISON WITH INDEPENDENT HAND-CALCS	11
<i>EXAMPLE 1 – General loading</i>	<i>11</i>
<i>EXAMPLE 2 – Rotated local axes</i>	<i>15</i>
<i>EXAMPLE 4 – Moment/Force releases</i>	<i>20</i>
<i>EXAMPLE 5 – Partial fixity end releases.....</i>	<i>23</i>
<i>EXAMPLE 6 – End offset lengths</i>	<i>25</i>
<i>EXAMPLE 7 – No tension, no compression frame object</i>	<i>27</i>
<i>EXAMPLE 8 – Eigenvalue problem.....</i>	<i>30</i>
<i>EXAMPLE 9 – Bending and axial deformations in a rigid frame</i>	<i>33</i>
<i>EXAMPLE 10 – Bathe and Wilson eigenvalue problem.....</i>	<i>35</i>
<i>EXAMPLE 11 – Two-dimensional frame with static and dynamic loads</i>	<i>37</i>
<i>EXAMPLE 12 – Eigenvalue analysis of a three-dimensional frame with rigid floor diaphragm.....</i>	<i>41</i>
<i>EXAMPLE 13 – Large axial displacements.....</i>	<i>43</i>
<i>EXAMPLE 14 – Large bending displacements</i>	<i>45</i>
<i>EXAMPLE 15 – Linear link with ramp loading.....</i>	<i>47</i>
<i>EXAMPLE 16 – Damper element under harmonic loading.....</i>	<i>50</i>
<i>EXAMPLE 17 – Plastic link (bilinear symmetric)</i>	<i>52</i>
<i>EXAMPLE 18 – Plastic link (trilinear asymmetric).....</i>	<i>55</i>
Chapter 3 COMPARISONS WITH LITERATURE CASE-STUDIES	58
<i>EXAMPLE 1 – Von Mises truss.....</i>	<i>58</i>
<i>EXAMPLE 2 – Cantilever beam under a moment at the tip.....</i>	<i>60</i>
<i>EXAMPLE 3 – Lee's frame</i>	<i>63</i>
<i>EXAMPLE 4 – Williams' toggle frame.....</i>	<i>65</i>
<i>EXAMPLE 5 – Clamped right-angle frame under tip load</i>	<i>67</i>
<i>EXAMPLE 6 – Cantilever beam with vertical tip load.....</i>	<i>69</i>
Chapter 4 COMPARISONS WITH EXPERIMENTAL RESULTS.....	71
Typical existing non-seismically-designed RC european building	71
<i>EXAMPLE 1 – Multi-storey, 2D frame (ICONS frame - bare)</i>	<i>71</i>
<i>EXAMPLE 2 – Seven storey, full-scale, RC shear wall building.....</i>	<i>77</i>
<i>EXAMPLE 3 – Full-scale, three storey, three-dimensional RC moment frame (SPEAR building)</i>	<i>83</i>
Steel frames	88
<i>EXAMPLE 1 – Three storey, three-dimensional steel moment frame.....</i>	<i>88</i>
<i>EXAMPLE 2 – Full-scale, four-storey 3D steel frame.....</i>	<i>94</i>
Infilled frames (masonry infills)	101
<i>EXAMPLE 1 – One storey, single bay infilled frame (Crisafulli).....</i>	<i>101</i>
<i>EXAMPLE 2 – One storey, single bay infilled frame (Colangelo et al.)</i>	<i>105</i>
<i>EXAMPLE 3 – Multi-storey 3D frame (Negro)</i>	<i>109</i>
<i>EXAMPLE 4 – Multi-storey, 2D infilled frame (ICONS frame - infilled)</i>	<i>114</i>

<i>EXAMPLE 5 – Half scale, three-storey, three-dimensional infilled RC frame</i>	<i>119</i>
Beam-Column joints	123
<i>EXAMPLE 1 – RC joints</i>	<i>123</i>
<i>EXAMPLE 2 – Steel joints</i>	<i>130</i>
RC bridges	135
<i>EXAMPLE 1 – Multi-span continuous deck bridge</i>	<i>135</i>
<i>EXAMPLE 2 – Scaled bridge pier</i>	<i>139</i>
Blind Prediction Contest successes	143
<i>EXAMPLE 1 – Full-scale bridge column</i>	<i>143</i>
<i>EXAMPLE 2 – Two simple one storey, three-dimensional RC frame structures</i>	<i>151</i>
<i>EXAMPLE 3 – Half-scale three-storey RC frame</i>	<i>157</i>

Chapter 1 INTRODUCTION

PRESENTATION OF THE ANALYSIS PROGRAM

SeismoStruct is a Finite Element package capable of predicting the large displacement behaviour of space frames under static or dynamic loadings, taking into account both geometric nonlinearities and material inelasticity. Concrete, steel, frp and sma material models are available, together with a large library of 3D elements that may be used with a wide variety of pre-defined steel, concrete and composite section configurations. The spread of inelasticity along the member length and across the section depth is explicitly modelled, allowing for accurate estimation of damage distribution. Coupled with the program's numerical stability and accuracy at high strain levels, it enables the precise determination of the inelastic response and the collapse load of any frame-type structural configuration. SeismoStruct accepts static (forces and displacements) as well as dynamic (accelerations) actions and has the ability to perform eigenvalue, nonlinear static pushover (conventional and adaptive), nonlinear static time-history analysis, nonlinear dynamic analysis, incremental dynamic analysis and response spectrum analysis.

The accuracy of this software in nonlinear analysis of framed structures is demonstrated in this report, and has also been laid evident by the successes in recent Blind Test Prediction Exercises, such as 'Concrete Column Blind Prediction Contest 2010' (UCSD, San Diego, USA), '15WCEE Blind Test Challenge' (LNEC, Lisbon, Portugal) and the Blind Prediction Contest organized in 2011 by the "Earthquake Resistance and Disaster Prevention Branch of the Architectural Society of China", where SeismoStruct ranked first amongst tens of entries from around the world. Such case-studies are included at the end of the current Verification Report.

The SeismoStruct results presented in this document were obtained using **version 2018** of the program, running on an Intel® Core™ i7-6700HQ CPU @ 2.60GHz machine with Windows 10 64-bit. All model files are included in SeismoStruct's installation folder.

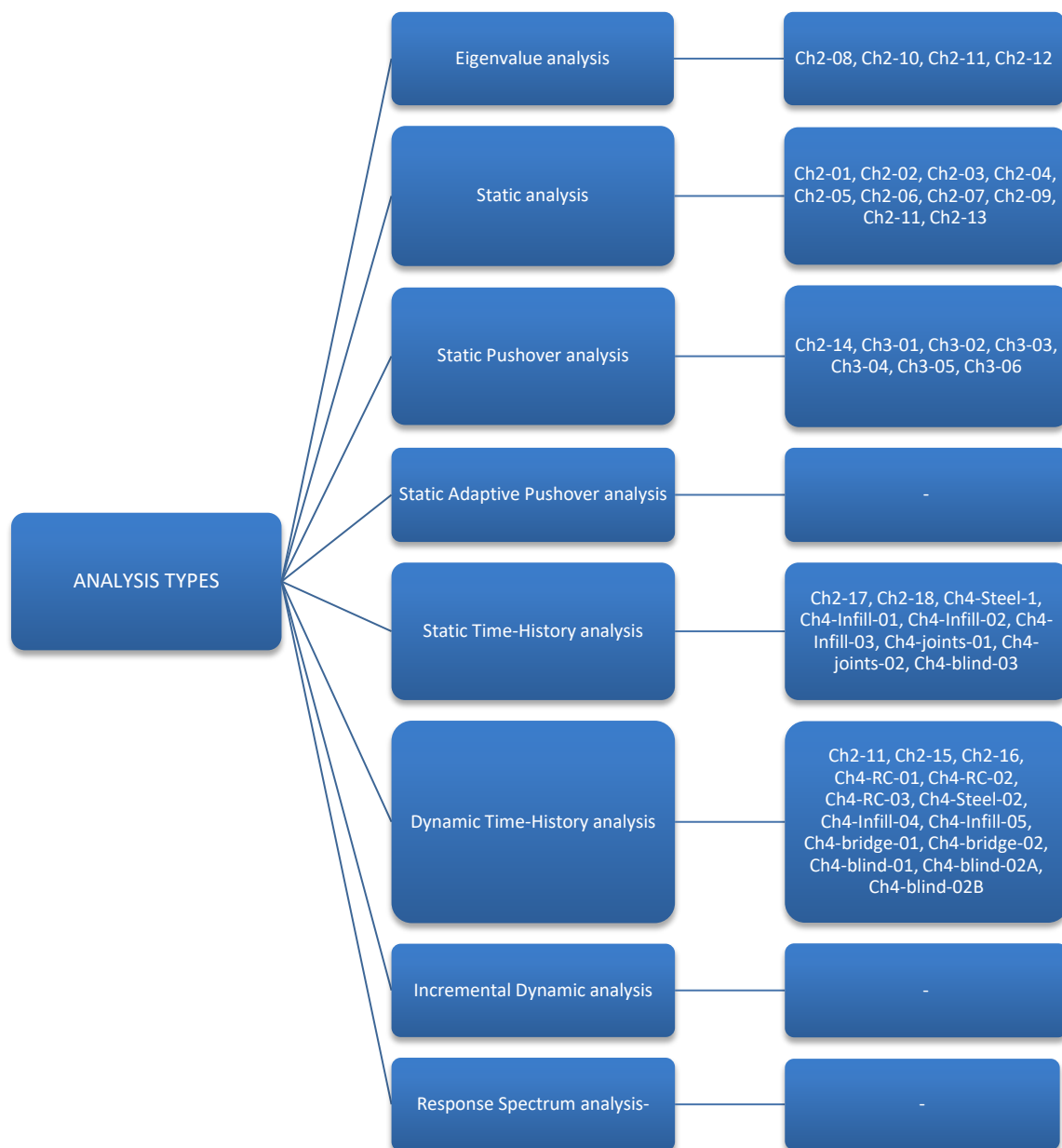
STRUCTURE OF THE REPORT

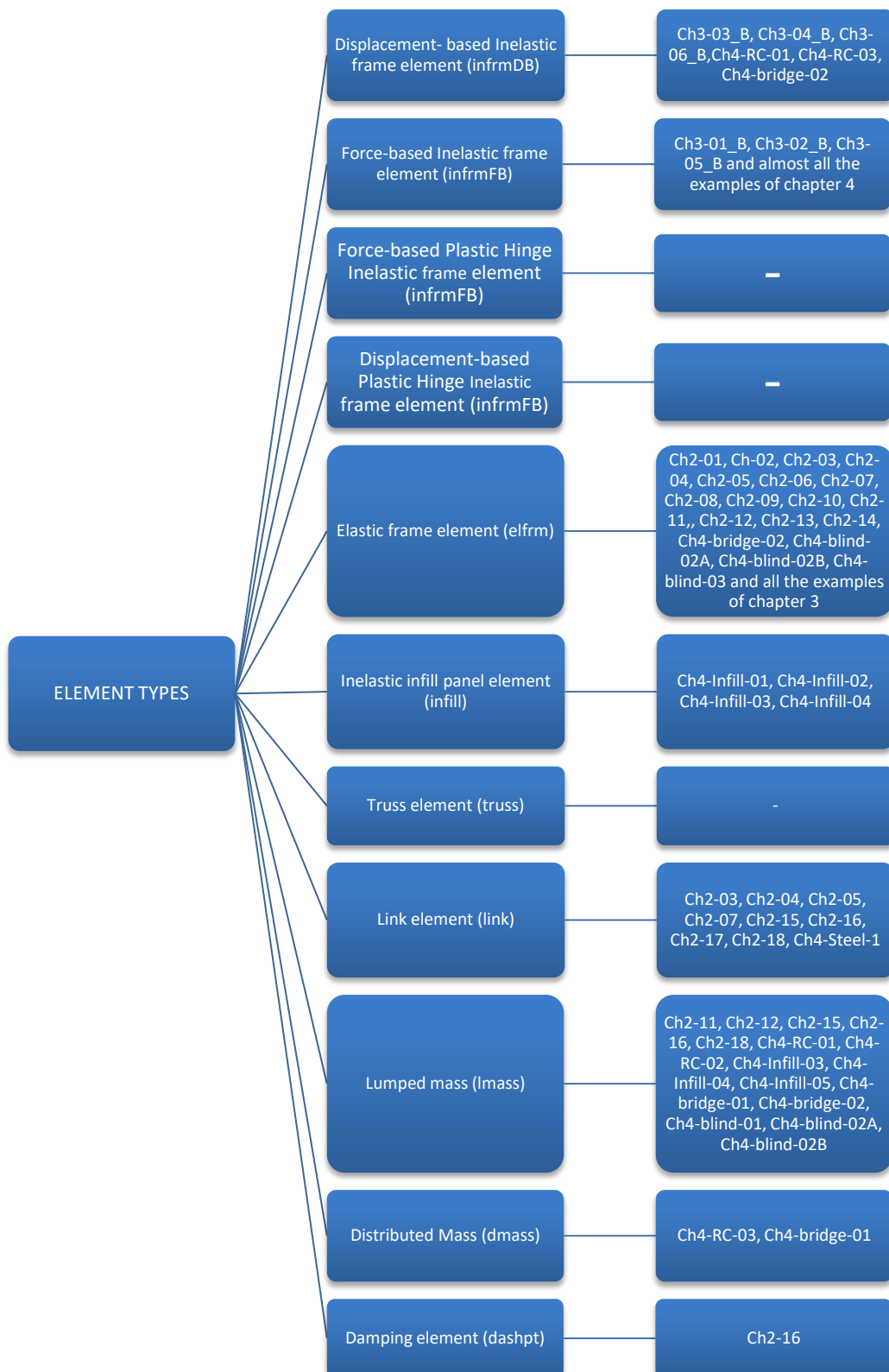
The present report consists of a comprehensive collection of examples, which have been selected to test the various program features. It is structured in three main sections, which are briefly described below:

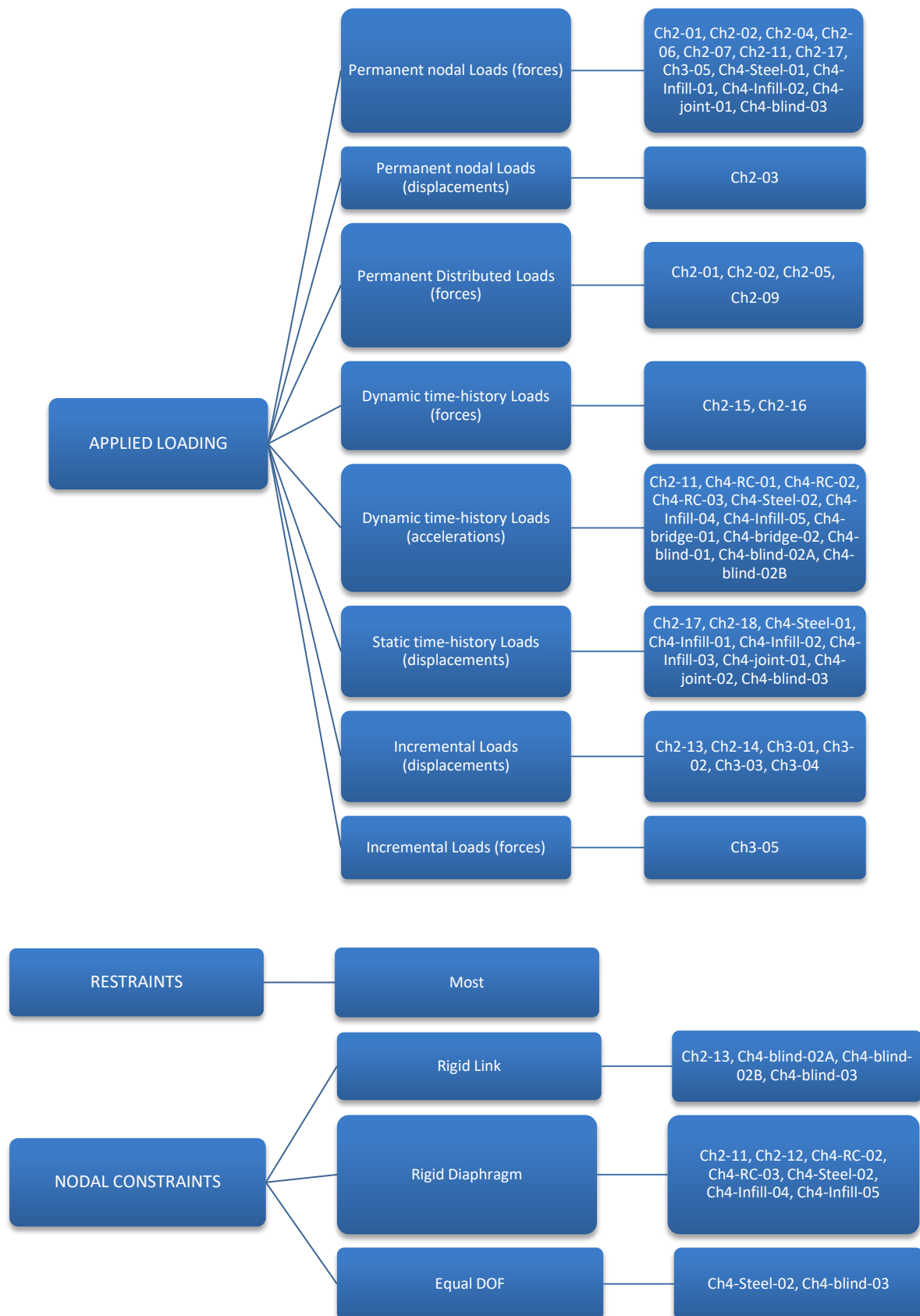
- In the first section (Chapter 2), the results produced by SeismoStruct are compared with the independent hand-calculations proposed in SAP2000's Software Verification report (SAP2000, Integrated Software for Structural Analysis and Design, Analysis Verification Examples", Computers and Structures, Inc., 1992). The results are provided in tabular form;
- In the second section (Chapter 3), the results produced by SeismoStruct are compared with well-known literature case-studies. The results are provided in graphical form;
- Finally, the third section (Chapter 4), deals with the comparison between SeismoStruct results and experimental results, obtained from various laboratory tests. In particular, the last three examples of this section describe the numerical models used in recent Blind Prediction contests (i.e. the "Concrete Column Blind Prediction Contest 2010", the "15WCEE Blind Test Challenge" and the Blind Prediction Contest organized by the "Earthquake Resistance and Disaster Prevention Branch of the Architectural Society of China"), where SeismoStruct came out as the winning software. Again the results are provided in graphical form.

PROGRAM FEATURES COVERED BY THE PROGRAM

The aim of this section is to illustrate, through the charts provided below, which program features (i.e. types of analyses, elements, restraints, constraints and applied loads) are addressed in each example of the present report.







Chapter 2 COMPARISON WITH INDEPENDENT HAND-CALCS

As noted above, this chapter makes use of examples, and their corresponding independent hand-calculations, described in SAP2000's Software Verification report (Computers and Structures, Inc.). On occasions, a direct correspondence between the models used in different programs was not possible (e.g. an oblique load cannot be explicitly defined on some of the programs), but solutions were always found to attain equivalent models

EXAMPLE 1 – General loading

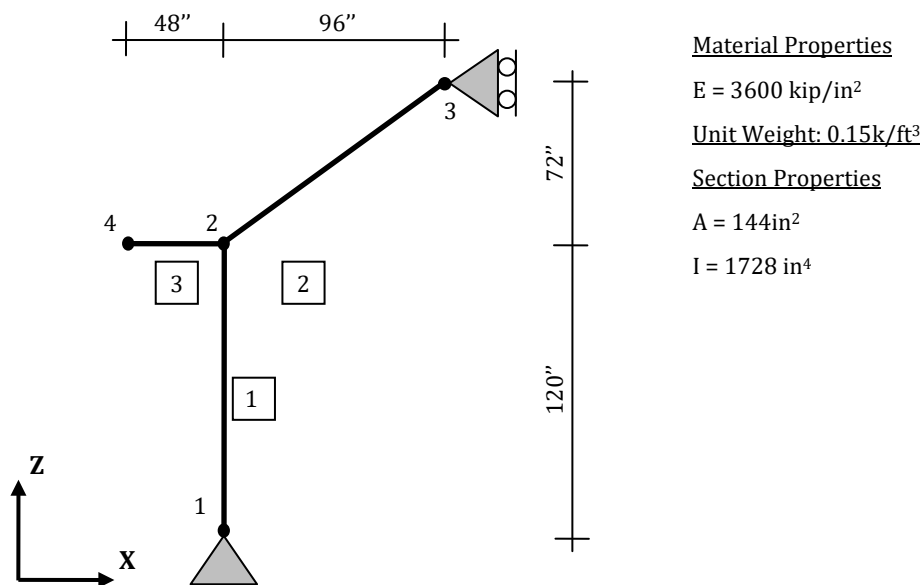
DESCRIPTION

A three-element frame is subjected to six load cases with various types of load (distributed and concentrated). Six different models have been created, one for each load case.

The resulting displacements at specified joints obtained with the FE analysis program SeismoStruct are compared with hand calculations. The FE models lie in the X-Z plane.

NOTE: Only bending deformations are considered in the analysis. In order to ignore the axial deformations, the element's axial stiffness is multiplied by a 1000. Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES



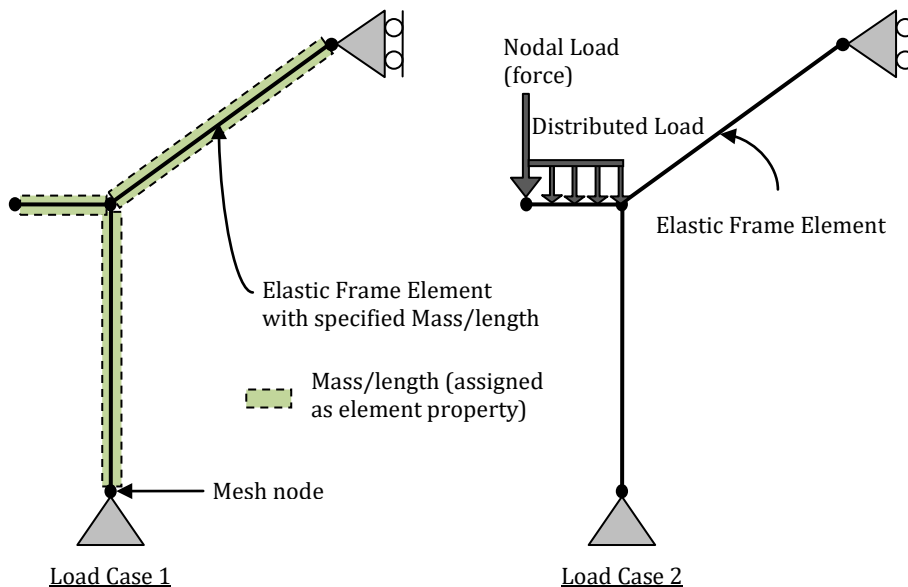
MODELLING AND LOADING

The frame members are modelled through an elastic frame element (elfrm) with the following properties: $EA = 518400000 \text{ [kip]}$; $EI \text{ (axis2)} = 6220800 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 6220800 \text{ [kip-in}^2\text{]}$; $GJ = 4380845 \text{ [kip-in}^2\text{]}$. Four different load cases are considered:

- **Load Case 1:** Self-weights (the weights are inserted in the program as distributed masses, and in the "Gravity and Mass" page of the "Project Settings" panel, it is selected to derive loads from masses, based on the g value in the gravity direction);

- Load Case 2: Global uniform distributed load on frame element 3, plus concentrated load on joint 4 (the uniform distributed load is inserted as element permanent load in the Z direction ($F_z = -1.8$ k/ft); the concentrated load is applied as permanent load in terms of forces in the Z direction ($F_z = -10$ k);
- Load Case 3: Global joint force and joint moment at joint 2 (force and moment are applied as permanent loads, in terms of forces ($F_z = -17.2$ k) and moments ($M_y = -652.8$ kip-in) in the Z and RY directions respectively);
- Load Case 4: Concentrated load on frame element 2 (it has been decomposed in a vertical and a horizontal component, so they are applied as permanent loads in terms of forces in the X and Z directions respectively, $F_x = 9$ k and $F_z = -12$ k);
- Load Case 5: Uniformly distributed projected load on frame element 2 (it has been applied on the full element length as element permanent load in the Z direction, $F_z = -1.6$ k/ft);
- Load Case 6: Uniformly distributed load on frame elements 1 and 2 in frame local direction (in element 1 the frame local direction coincide with the global X axis, so it is applied as element load in terms of forces in the X direction ($F_x = 2$ k/ft); the load on frame element 2 has been decomposed in a vertical and a horizontal component, so they are applied as element loads in terms of forces in the X and Z directions respectively, $F_x = 1.2$ k/ft and $F_z = -1.6$ k/ft).

The FE models are shown below:



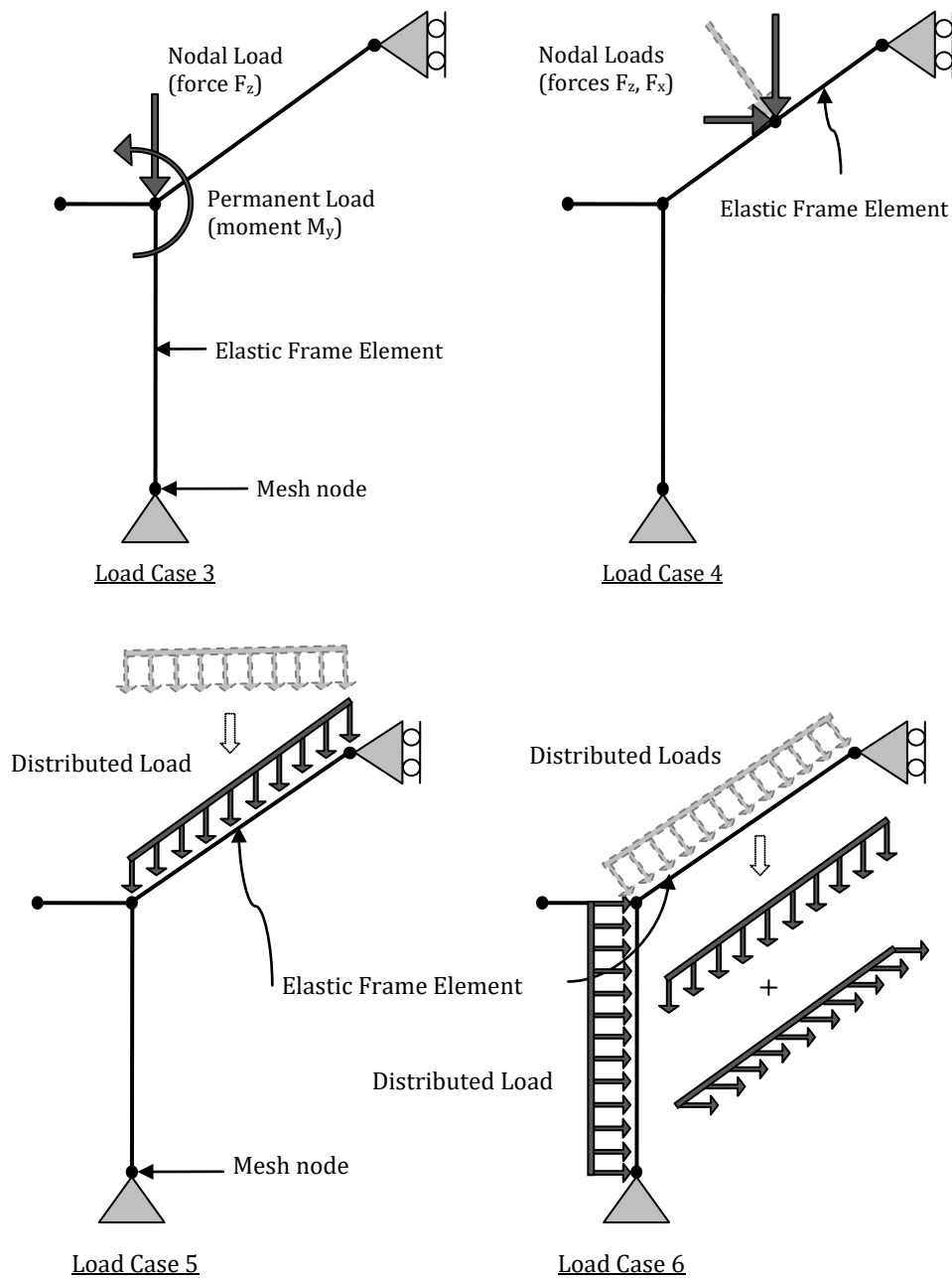


Figure 2.1. Load cases of EXAMPLE 1

Node 1 is fully restrained and node 3 is restrained in X, Y, RX, RZ.

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.1. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 1

LOAD CASE	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
CASE 1	$U_z(jt. 3)$ [in]	-0.02639	-0.02639
CASE 2	$U_z(jt. 3)$ [in]	0.06296	0.06296
CASE 3	$U_z(jt. 3)$ [in]	0.06296	0.06296
CASE 4	$U_x(jt. 2)$ [in]	0.00651	0.00651
CASE 5	$U_z(jt. 3)$ [in]	-0.2963	-0.2963
CASE 6	$U_x(jt. 2)$ [in]	0.3125	0.3125

COMPUTER FILES

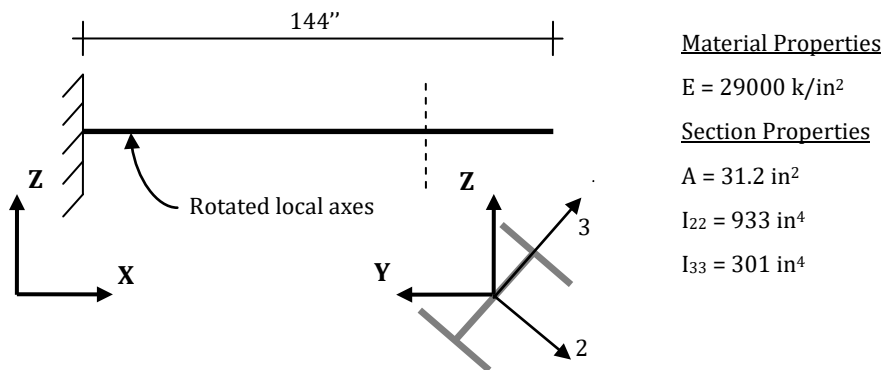
- Ch2-01_1.spf
- Ch2-01_2.spf
- Ch2-01_3.spf
- Ch2-01_4.spf
- Ch2-01_5.spf
- Ch2-01_6.spf

EXAMPLE 2 – Rotated local axes**DESCRIPTION**

A cantilever beam, with its local axes (2 and 3) not parallel to the global ones, is subjected to three load cases with various types of load. An I-section cantilever beam is used with the local 3 axis rotated 30 degrees from the global Z. Three different models have been created, one for each load case.

The resulting displacements at specified joints obtained with the FE analysis program SeismoStruct are compared with hand calculations. The FE models lie in the X-Z plane.

NOTE: Only bending deformations are considered in the analysis. Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The cantilever beam is modelled through an elastic frame element (elfrm) with the following properties: $EA = 904800 \text{ [kip]}$; $EI \text{ (axis2)} = 2.7057\text{E}+007 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 8729000 \text{ [kip-in}^2\text{]}$; $GJ = 101835 \text{ [kip-in}^2\text{]}$.

Three different load cases are considered:

- Load Case 1: Global uniform distributed load on frame element (the uniform distributed load is assigned as element permanent load in terms of forces in the Z direction, $F_z = -0.01 \text{ k/in}$);
- Load Case 2: Concentrated load at free end in global Z direction (the concentrated load is applied as permanent load in terms of forces in the Z direction, $F = -1 \text{ k}$);
- Load Case 3: Concentrated moment at free end about global Z axis (the concentrated load is applied as permanent load in terms of moments in RY direction, $M = 240 \text{ kip-in}$).

The FE models are shown below:

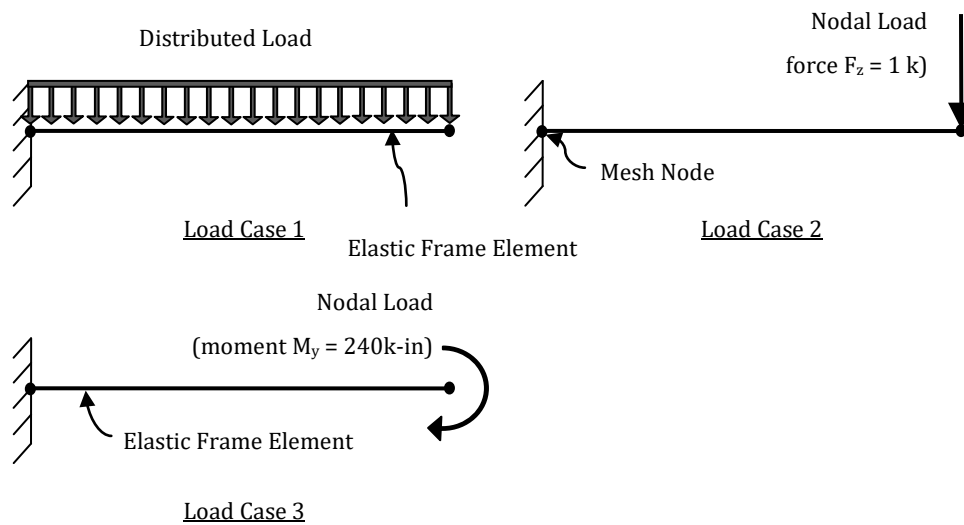


Figure 2.2. Load cases of EXAMPLE 2

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.2. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 2

LOAD CASE	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
CASE 1	$U_y(jt. 2)$ [in]	-0.01806	-0.01806
	$U_z(jt. 2)$ [in]	-0.03029	-0.03029
CASE 2	$U_y(jt. 2)$ [in]	-0.03345	-0.03345
	$U_z(jt. 2)$ [in]	-0.05610	-0.05610
CASE 3	$U_y(jt. 2)$ [in]	-0.08361	-0.08361
	$U_z(jt. 2)$ [in]	-0.14024	-0.14024

COMPUTER FILES

- Ch2-02_1.spf
- Ch2-02_2.spf
- Ch2-02_3.spf

EXAMPLE 3 – Displacement loading

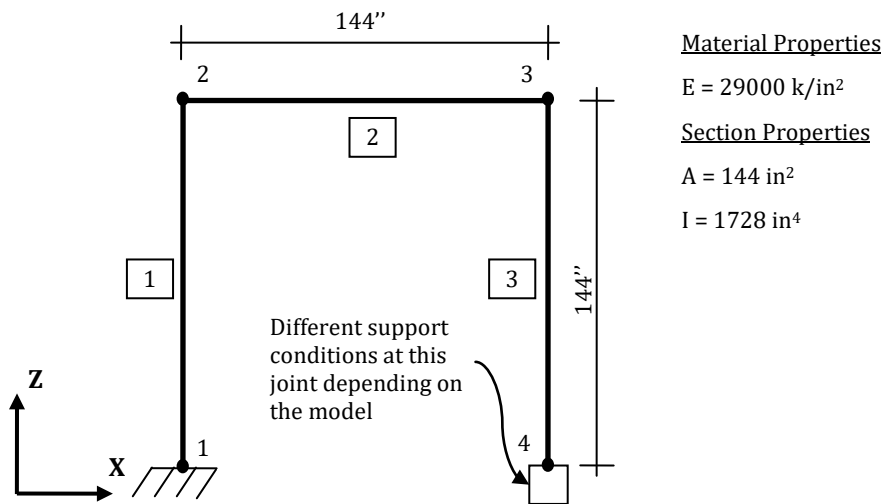
DESCRIPTION

This example tests SeismoStruct for settlement and rotation of normal supports and spring supports on a portal frame. Four different models have been created. The models are identical, except for the loading and the support condition at joint 4.

The results obtained with the FE analysis program SeismoStruct at specified joints and in each model are compared with hand calculations. The FE models lie in the X-Z plane.

NOTE: Only bending deformations are considered in the analysis. Shear and axial deformations are ignored. In order to ignore the axial deformations the value of axial stiffness of the element is multiplied by a factor of 10^5 . The program does not consider the shear deformations by default.

GEOMETRY AND PROPERTIES



MODELLING AND LOADING

The frame members are modelled through an elastic frame element (elfrm) with the following properties: $EA = 4.1760\text{E}+011 \text{ [kip]}$; $EI \text{ (axis2)} = 5.0112\text{E}+007 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 5.0112\text{E}+007 \text{ [kip-in}^2\text{]}$; $GJ = 3.2576\text{E}+007 \text{ [kip-in}^2\text{]}$.

Node 1 is fully restrained.

Four different models have been considered:

- Model A: imposed displacement in the Z direction at joint 4 (it is applied as permanent load in terms of displacements in the Z direction, $d = -0.5 \text{ in}$);
- Model B: spring with displacement in the Z direction at joint 4 (the displacement is applied as permanent load in terms of displacements in the Z direction, $d = -0.2 \text{ in}$);
- Model C: rotation around global Y axis at joint 1 (it is applied as permanent load in terms of rotations in RY direction, $r_y = 0.01 \text{ rad}$);
- Model D: spring with rotation around global Y axis at joint 4 (the rotation is applied as permanent load in terms of rotations in RY direction, $r_y = 0.01 \text{ rad}$)).

In those models in which it is necessary to introduce a spring (translational or rotational), this is modelled by using a link element. This element connects two initially coincident structural nodes and

requires the definition of an independent force-displacement response curve for each of its local six degrees-of-freedom (F1, F2, F3, M1, M2, M3) (see Figure 2.3).

In particular, a translational spring is introduced in Model B by using a link element characterized by a stiffness k equal to 10 kip/in in the Z direction (F1) with a linear symmetric response curve (the other five degrees-of-freedom have the stiffness value set to zero). Then a rotational spring, characterized by a rotational stiffness k equal to 80000 kip-in/rad about Y axis (M2), is introduced in Model D.

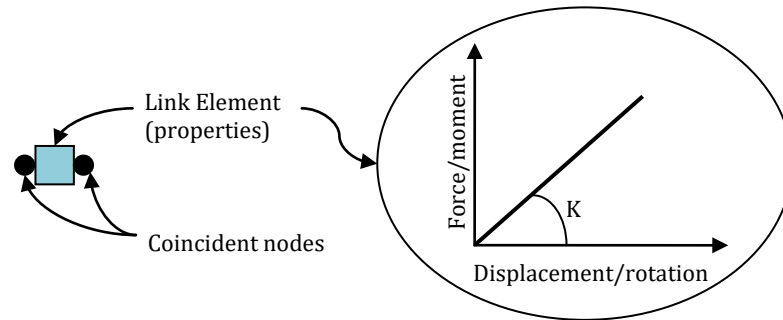


Figure 2.3. Modelling of a link element

The FE models are shown below:

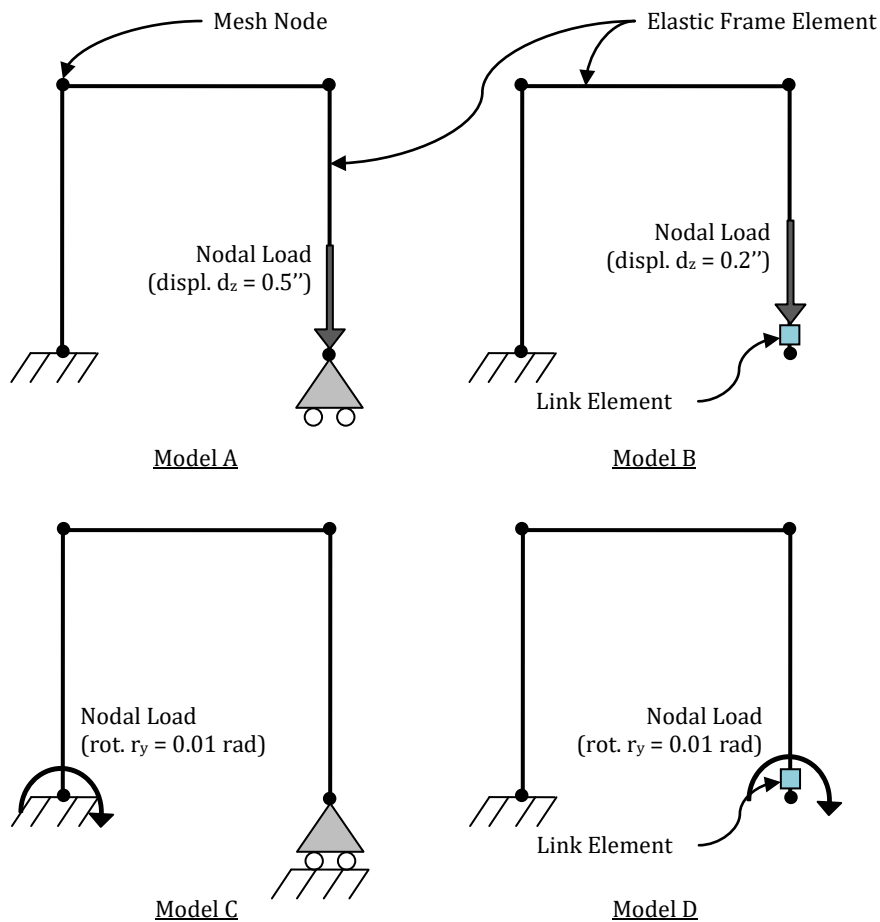


Figure 2.4. Models for EXAMPLE 3

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.3. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 3

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	$F_z(\text{jt. 1})$ [k]	6.293	6.293
	$M_y(\text{jt. 1})$ [k-in]	-906.25	-906.25
B	$F_z(\text{jt. 1})$ [k]	1.115	1.115
	$M_y(\text{jt. 1})$ [k-in]	-160.492	-160.492
C	$F_z(\text{jt. 1})$ [k]	-18.125	-18.125
	$M_y(\text{jt. 1})$ [k-in]	2610	2610
D	$M_y(\text{jt. 1})$ [k-in]	-473.469	-473.469
	$R_y(\text{jt. 4})$ [rad]	0.00408	0.00408

COMPUTER FILES

- Ch2-03_A.spf
- Ch2-03_B.spf
- Ch2-03_C.spf
- Ch2-03_D.spf

EXAMPLE 4 – Moment/Force releases

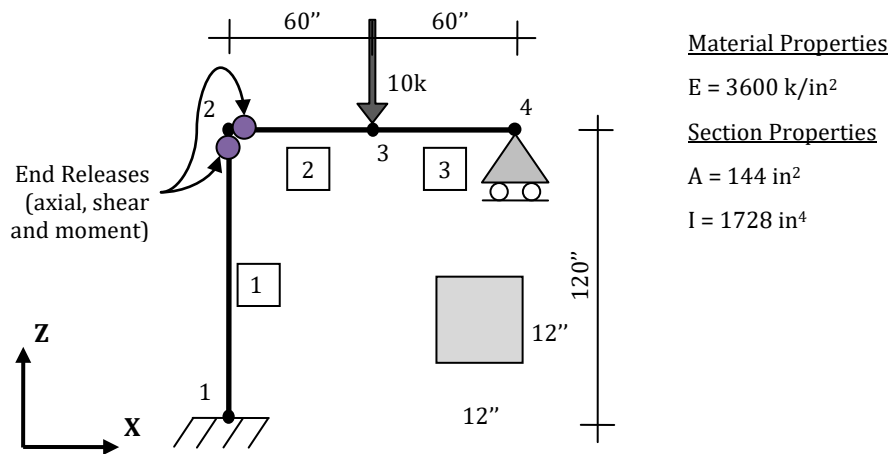
DESCRIPTION

This example describes how to model the end releases in SeismoStruct. Three identical models have been created: they only differ each another for the types of end releases (i.e. Model A has a shear release, Model B an axial release and Model C a moment release).

The results obtained with the FE analysis program SeismoStruct at specified joints are compared with hand calculations. The FE models lie in the X-Z plane.

NOTE: Shear deformations are ignored: the program does not consider the shear deformations by default.

GEOMETRY AND PROPERTIES



MODELLING AND LOADING

The members have a rectangular solid section (rss) that employs the elastic material type. The section's dimensions are shown in the figure above, and its elastic properties are: $EA = 518400 \text{ [kip]}$; $EI \text{ (axis2)} = 6220800 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 6220800 \text{ [kip-in}^2\text{]}$; $GJ = 4380845 \text{ [kip-in}^2\text{]}$. The frame members are modelled through an elastic frame element (elfrm).

Three different models have been considered:

- Model A: Shear release
- Model B: Axial release
- Model C: Moment release

The end releases defined in model A, are modelled by using a link element (see Figure 2.3). The link is characterized by a linear symmetric response curve with a stiffness $k = 0$ in those degrees-of-freedom which need to be release (F3 in Model A). The remaining degrees-of-freedom are characterized again by a linear symmetric response curve with a stiffness $k = 1.00\text{E}+012 \text{ kip/in}$. Axial and moment releases are modelled through the '**Moment/Force releases**' facility, which is available in the element dialog window. For the axial release in Model B the F check-box is selected in elements 1 and 2; for the moment release in Model C the M2b check-box is selected in element 1 and the M2a check-box in element 2.

Node 1 is fully restrained, whereas node 4 is restrained in Y, Z, RX and RZ. The FE models are shown below:

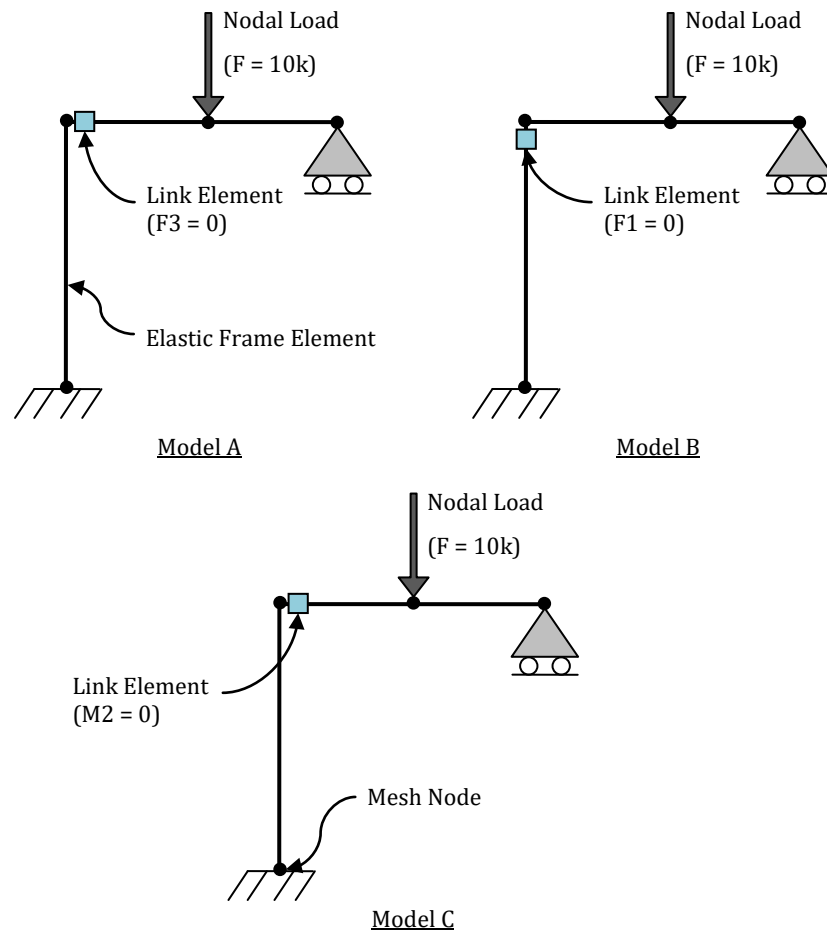


Figure 2.5. Models for EXAMPLE 4

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the tables below:

Table 2.4. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 4

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	$F_z(jt. 1)$ [k]	0	0
	$M_y(jt. 1)$ [k-in]	600	600
B	$F_z(jt. 1)$ [k]	0	0
	$M_y(jt. 1)$ [k-in]	600	600
C	$F_z(jt. 1)$ [k]	5	5
	$M_y(jt. 1)$ [k-in]	0	0

COMPUTER FILES

- Ch2-04_A.spf
- Ch2-04_B.spf
- Ch2-04_C.spf

EXAMPLE 5 – Partial fixity end releases

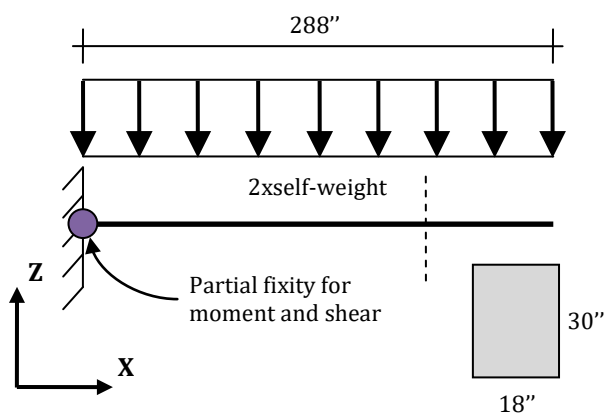
DESCRIPTION

This example describes how to model the partial fixity end releases in SeismoStruct. The model consists of a cantilever beam subjected to a uniform load equal to twice its self-weight. At the fixed end of the frame element the partial fixity moment (M_y) and shear (V_z) springs have been assigned.

The vertical tip deflection of the cantilever obtained with the FE analysis program SeismoStruct is compared with hand calculations. The FE models lie in the X-Z plane.

Important Note: Only bending deformations are considered in the analysis for this example. Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES



Material Properties

$$E = 4320 \text{ k/in}^2$$

$$\text{Unit Weight: } 0.15 \text{ k/ft}^3$$

$$G = 1800 \text{ k/in}^2$$

Section Properties

$$A = 540 \text{ in}^2$$

$$I = 40500 \text{ in}^4$$

Note

An element distributed load equal to 0.09375 k/in is applied on the members

MODELLING AND LOADING

The members have a rectangular solid section (rss) that employs the elastic material type. The section dimensions are shown in the figure above, and its elastic properties are: $EA = 2.3328\text{E}+006$ [kip]; EI (axis2) = $1.7496\text{E}+008$ [kip-in²]; EI (axis3) = $6.2986\text{E}+007$ [kip-in²]; $GJ = 6.5724\text{E}+007$ [kip-in²]; Mass/length = 0.00024288 [kip×sec²/in²].

Instead of the uniform load equal to 2xbeam self-weight, a distributed mass is directly associated to the elastic frame element and then automatically converted to gravity load during the analysis. A uniformly distributed permanent force load is applied on the element in the Z direction, $F_z = -0.09375$ k/in).

The partial fixity moment and shear springs are modelled by using a link element, which is characterized by a stiffness value $k = 540$ k/in in F3 and $k = 3888000$ k-in/rad in M2 respectively, both with a linear symmetric response curve. The other degrees-of-freedom are considered infinitely stiff.

The FE model is shown below:

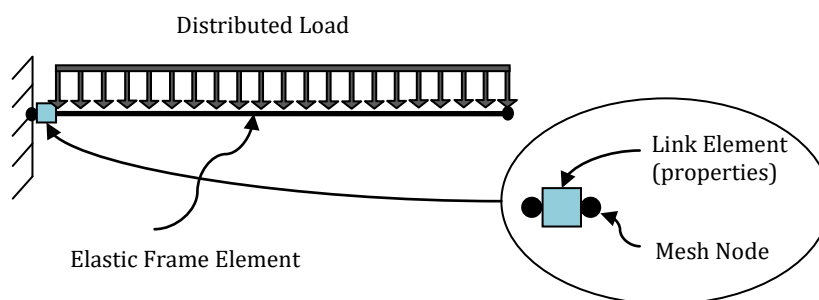


Figure 2.6. Model for EXAMPLE 5

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the tables below:

Table 2.5. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 5

OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
$U_z(jt. 2)$ [in]	-0.7988	-0.7988

COMPUTER FILES

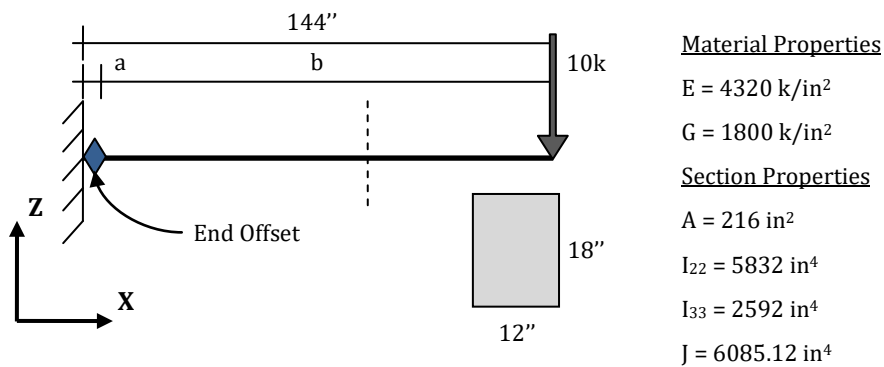
- Ch2-05.spf

EXAMPLE 6 – End offset lengths**DESCRIPTION**

This example tests the SeismoStruct end offsets. Three models have been built for this purpose. The first one has the end offset length set equal to zero, the second and third are characterized by 3'' and 6'' long end offsets respectively.

The vertical displacement at the free end of the cantilever obtained with the FE analysis program SeismoStruct is compared with hand calculations. The FE models lie in the X-Z plane.

Important Note: Only bending deformations are considered in the analyses for this example. Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The members have a rectangular solid section (rss) that employs the elastic material type. The section dimensions are shown in the figure above, and its elastic properties are: $EA = 933120 \text{ [kip]}$; $EI \text{ (axis2)} = 25194240 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 11197440 \text{ [kip-in}^2\text{]}$; $GJ = 10953216 \text{ [kip-in}^2\text{]}$. In every model, the cantilever beam is modelled through an elastic frame element (elfrm).

The “rigid zone”(said “a” in the figure above) is defined through the ‘End offset lengths’ option, which is available in the element dialogue window. In particular:

- **Model A:** no offset length;
- **Model B:** the end offset length at node 1 is equal to 3'' (-> 50% rigidity);
- **Model C:** the end offset length at node 1 is equal to 6'' (-> 100% rigidity).

A concentrated load is applied as permanent load in terms of forces in the Z direction ($F = -10 \text{ k}$). The FE models are presented below:

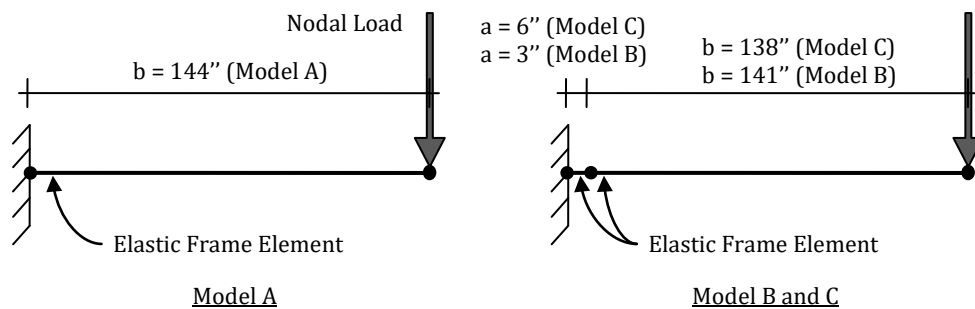


Figure 2.7. Models for EXAMPLE 6

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the tables below:

Table 2.6. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 6

LOAD CASE	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	$U_z(jt. 2)$ [in]	-0.39506	-0.39506
B	$U_z(jt. 2)$ [in]	-0.37088	-0.37088
C	$U_z(jt. 2)$ [in]	-0.34771	-0.34771

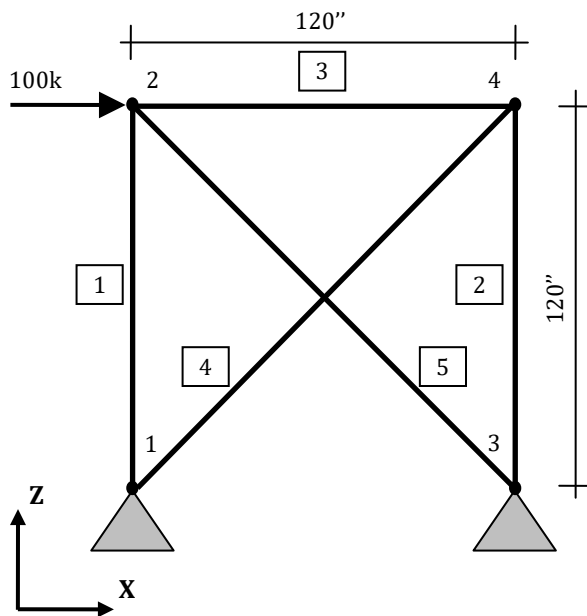
COMPUTER FILES

- Ch2-06_A.spf
- Ch2-06_B.spf
- Ch2-06_C.spf

EXAMPLE 7 – No tension, no compression frame object**DESCRIPTION**

This example uses a one-bay, one-storey braced frame subjected to a horizontal load applied at the top of the frame to test the tension and compression limits for frame objects. Three models have been created for this example with different element properties.

The horizontal displacements at the top of the frame and the support reactions obtained with the FE analysis program SeismoStruct are compared with hand calculations. The FE models lie in the X-Z plane.

GEOMETRY AND PROPERTIESMaterial Properties

$$E = 30000 \text{ k/in}^2$$

Section Properties

$$A = 8 \text{ in}^2$$

$$I_{22} = 1 \text{ in}^4$$

$$I_{33} = 1 \text{ in}^4$$

Notes

- Frame objects 4 and 5 have pinned ends (see par. *Modelling and Loading*)
- Braces are not connected at their intersection

MODELLING AND LOADING

The frame objects are modelled through an elastic frame element (elfrm) with: $EA = 240000$ [kip]; EI (axis2) = 30000 [kip-in²]; EI (axis3) = 30000 [kip-in²]; $GJ = 11538$ [kip-in²].

Three models with the following tension/compression limits have been considered:

- Model A: No tension or compression limits
- Model B: No compression in frame object 5
- Model C: No tension in frame object 4

In order to simulate pinned-ends, element releases are defined in the “Element Connectivity” module As follows:

- Model A: M2a and M2b releases in elements 1, 2 and 3
- Model B: M2a and M2b releases in elements 1, 2 and 3, and F release in element 5
- Model C: M2a and M2b releases in elements 1, 2, 3, and F release in element 4

A concentrated force load in the X direction ($F = 100 \text{ k}$) is applied in node 2. The FE models are shown below:

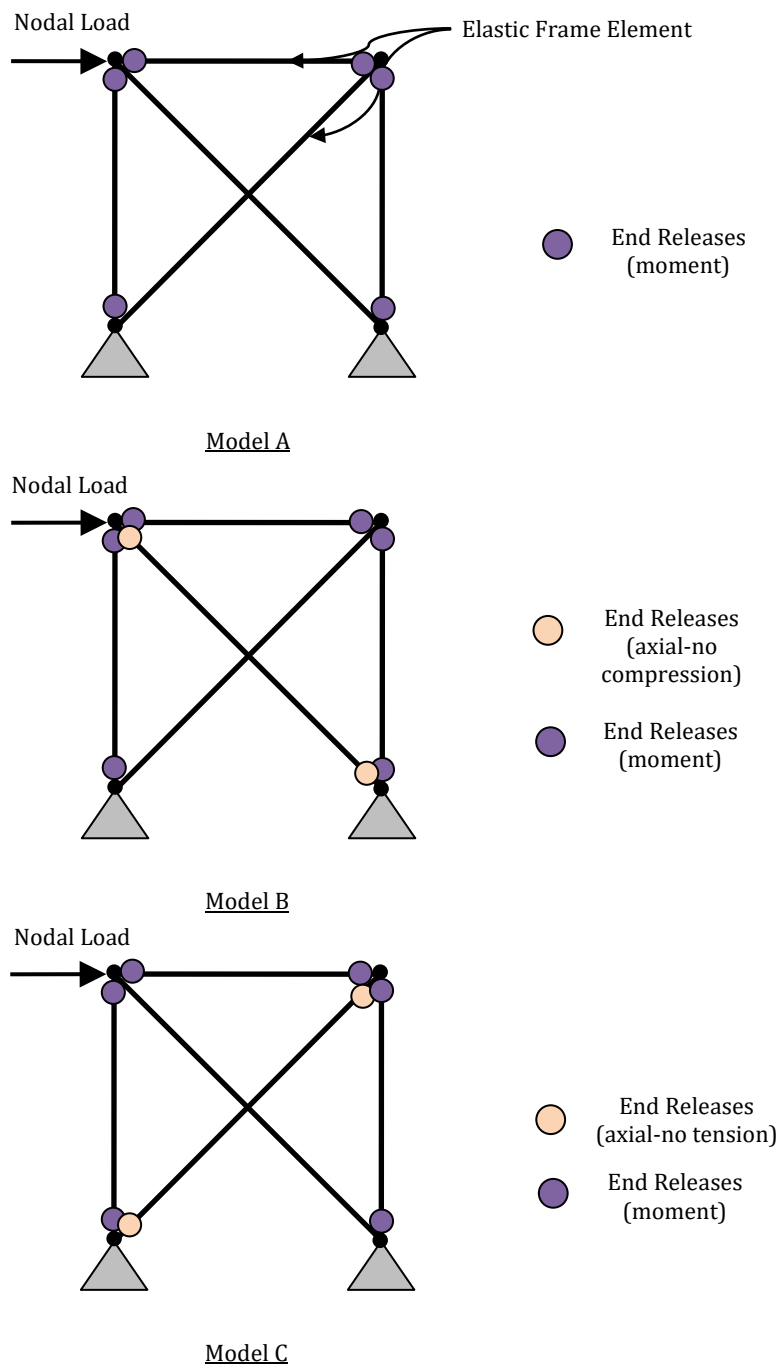


Figure 2.8. Models for EXAMPLE 7

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the tables below:

Table 2.7. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 7

LOAD CASE	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	U _x (jt. 2) [in]	0.10677	0.10677
	F _x (jt. 1) [kip]	-44.224	-44.224
	F _z (jt. 1) [kip]	-100	-100
	F _x (jt. 3) [kip]	-55.776	-55.776
	F _z (jt. 3) [kip]	100	100
B	U _x (jt. 2) [in]	0.24142	0.24142
	F _x (jt. 1) [kip]	-100	-100
	F _z (jt. 1) [kip]	-100	-100
	F _x (jt. 3) [kip]	0	0
	F _z (jt. 3) [kip]	100	100
C	U _x (jt. 2) [in]	0.19142	0.19142
	F _x (jt. 1) [kip]	0	0
	F _z (jt. 1) [kip]	-100	-100
	F _x (jt. 3) [kip]	-100	-100
	F _z (jt. 3) [kip]	100	100

COMPUTER FILES

- Ch2-07_A.spf
- Ch2-07_B.spf
- Ch2-07_C.spf

EXAMPLE 8 – Eigenvalue problem

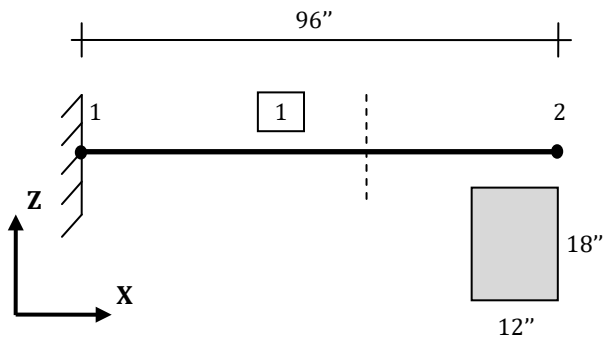
DESCRIPTION

In this example a ninety-six inch long cantilever concrete beam with $I_{22} \neq I_{33}$ is used to test SeismoStruct eigenvalue analysis. Seven models have been created: they differ one another in the discretization of the frame object.

The periods of vibration of the first three modes obtained for each model with the FE analysis program SeismoStruct are compared with hand calculations. The models are created in the X-Z plane.

Important Note: Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES



Material Properties

$$E = 3600 \text{ k/in}^2$$

$$\text{Unit Weight: } 8.878\text{E-}05 \text{ k/in}^3$$

Section Properties

$$A = 216 \text{ in}^2$$

$$I_{22} = 5832 \text{ in}^4$$

$$I_{33} = 2592 \text{ in}^4$$

MODELLING AND DISCRETIZATION

The members have a rectangular solid section (rss) that employs the elastic material type. The section's dimensions are shown in the figure above, and its elastic properties are: $EA = 777600$ [kip]; EI (axis2) = 20995200 [kip-in²]; EI (axis3) = 9331200 [kip-in²]; $GJ = 9127680$ [kip-in²]; $\text{Mass/Length} = 4.968\text{E-}05$ [kip-sec²/in²]. The cantilever beam is modeled through an elastic frame element (elfrm).

Seven different models have been created depending on the beam discretization:

- Model A: 1 element 96 inches long
- Model B: 2 elements each 48 inches long
- Model C: 4 elements each 24 inches long
- Model D: 6 elements each 16 inches long
- Model E: 8 elements each 12 inches long
- Model F: 10 elements each 9.6 inches long
- Model G: 96 elements each 1 inch long

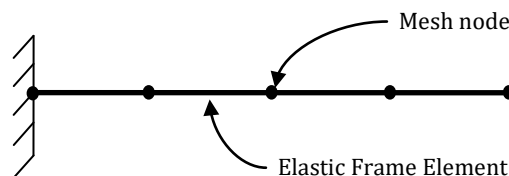


Figure 2.9. Model C of EXAMPLE 8

ANALYSIS TYPE

Eigenvalue analysis (Lanczos algorithm).

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.8. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 8

OUTPUT PARAMETER	MODEL	SeismoStruct 2018	Hand calculations
Period [sec] (1 st mode for bending about Z axis)	A	0.0545	0.038005
	B	0.0423	
	C	0.0391	
	D	0.0385	
	E	0.0383	
	F	0.0382	
	G	0.0380	
Period [sec] (1 st mode for bending about Y axis)	A	0.0364	0.025337
	B	0.0282	
	C	0.0261	
	D	0.0257	
	E	0.0255	
	F	0.0255	
	G	0.0253	
Period [sec] (2 nd mode for bending about Z axis)	A	N.A.	0.006064
	B	0.0082	
	C	0.0067	
	D	0.0063	
	E	0.0062	
	F	0.0062	
	G	0.0061	
Period [sec] (2 nd mode for bending about Y axis)	A	N.A.	0.004043
	B	0.0055	
	C	0.0044	
	D	0.0042	
	E	0.0041	
	F	0.0041	
	G	0.0040	
Period [sec] (3 rd mode for bending about Z axis)	A	N.A.	0.002165
	B	N.A.	
	C	0.0025	
	D	0.0023	
	E	0.0023	
	F	0.0022	
	G	0.0022	

COMPUTER FILES

- Ch2-08_A.spf
- Ch2-08_B.spf
- Ch2-08_C.spf
- Ch2-08_D.spf
- Ch2-08_E.spf
- Ch2-08_F.spf
- Ch2-08_G.spf

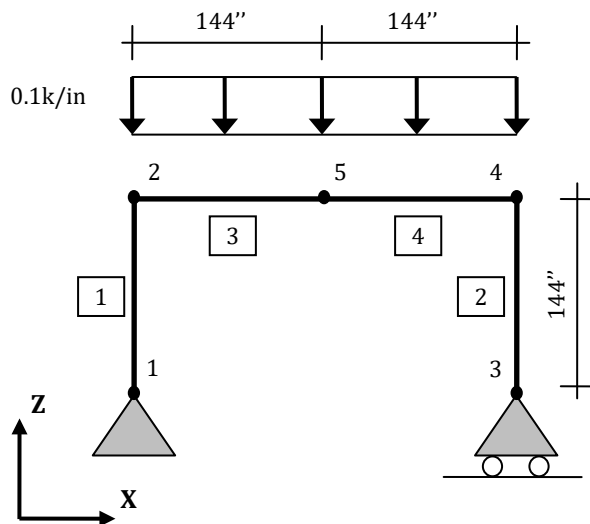
EXAMPLE 9 – Bending and axial deformations in a rigid frame**DESCRIPTION**

A one-storey, one-bay rigid frame is subjected to a uniform vertical load across the horizontal beam. Three different models with combined bending and axial deformation have been created for this example.

The resulting vertical displacement measured at the center of the horizontal member obtained with the FE analysis program SeismoStruct is compared with hand calculations. The FE models lie in the X-Z plane.

Important Note: SeismoStruct ignores shear deformation by default. For models with ignored bending deformations, the values of flexional stiffness are multiplied by 10^7 . For models with ignored axial deformations, the value of axial stiffness is multiplied by 10^5 .

NOTE: SeismoStruct ignores shear deformation by default. For models with ignored bending deformations, the values of flexional stiffness are multiplied by 10^7 . For models with ignored axial deformations, the value of axial stiffness is multiplied by 10^5 .

GEOMETRY AND PROPERTIES**Material Properties**

$$E = 29900 \text{ k/in}^2$$

$$G = 11500 \text{ k/in}^2$$

Section Properties

$$A = 9.12 \text{ in}^2$$

$$I_{22} = 110 \text{ in}^4$$

$$I_{33} = 37.1 \text{ in}^4$$

$$J = 0.536 \text{ in}^4$$

Note

An element distributed load equal to 0.1 k/in is applied on frame objects 3 and 4

MODELLING AND LOADING

A uniformly distributed load is applied on frame elements 3 and 4 as element permanent forces in the Z direction ($F_z = -0.1 \text{ k/in}$).

Three different models have been considered:

- Model A: bending and axial deformations
- Model B: bending deformations only
- Model C: axial deformations only

The frame members are modelled through an elastic frame element (elfrm) with the following properties:

- Model A: $EA = 272688 \text{ [kip]}$; $EI \text{ (axis2)} = 3289000 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 1109290 \text{ [kip-in}^2\text{]}$; $GJ = 6164 \text{ [kip-in}^2\text{]}$

- Model B: $EA = 27268800000$ [kip]; EI (axis2) = 3289000 [kip-in²]; EI (axis3) = 1109290 [kip-in²]; $GJ = 6164$ [kip-in²]
- Model C: $EA = 272688$ [kip]; EI (axis2) = 3289000000000000 [kip-in²]; EI (axis3) = 1109290000000000 [kip-in²]; $GJ = 6164$ [kip-in²]

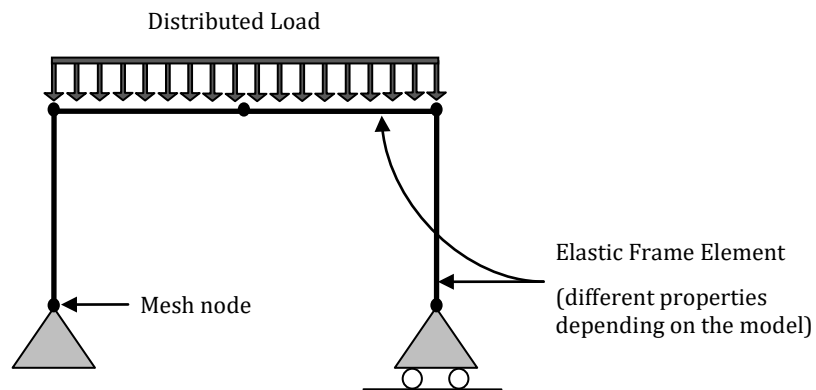


Figure 2.10. Model of EXAMPLE 10

ANALYSIS TYPE

Static analysis (with geometric nonlinearities disabled)

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.9. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 9

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	$U_z(jt. 5)$ [in]	-2.73121	-2.73121
B	$U_z(jt. 5)$ [in]	-2.72361	-2.72361
C	$U_z(jt. 5)$ [in]	-0.00760	-0.00760

COMPUTER FILES

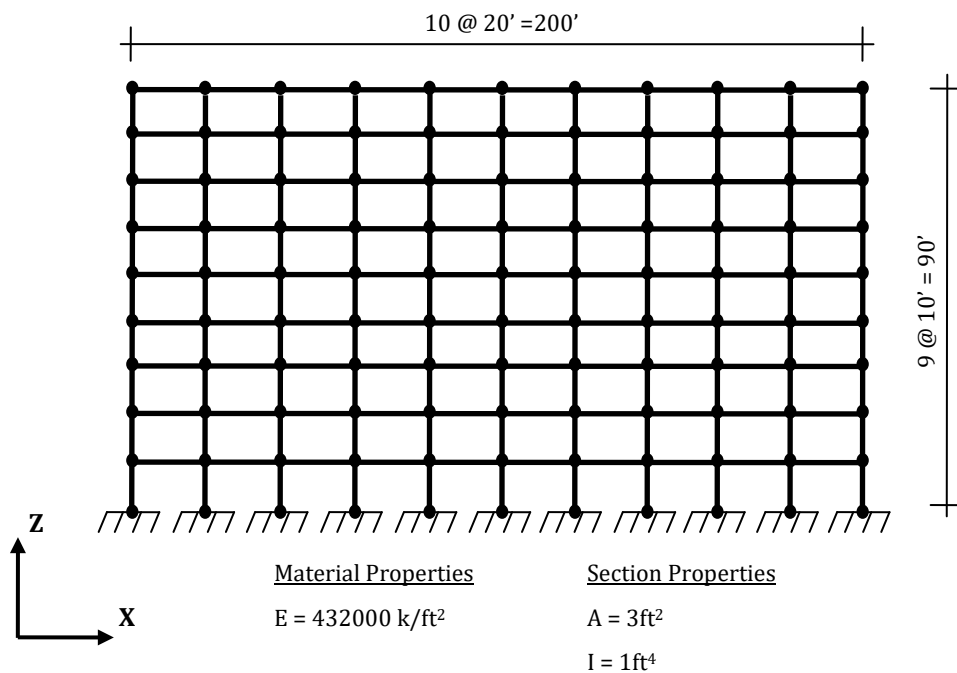
- Ch2-09_A.spf
- Ch2-09_B.spf
- Ch2-09_C.spf

EXAMPLE 10 – Bathe and Wilson eigenvalue problem**DESCRIPTION**

A ten bay, nine storey, two dimensional frame structure solved in Bathe and Wilson (1972) is analyzed for the first three eigenvalues. The material and section properties, and the mass per unit length used for all members, as shown below, are consistent with those used in the above-mentioned reference.

The results, in terms of ω^2 , obtained with the FE analysis program SeismoStruct are compared with independent results. The FE model is in the X-Z plane.

Important Note: Only bending and axial deformations are considered in the analysis. Shear deformations are ignored by default.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The frame objects are modelled through elastic frame elements (elfrm) with the following properties: $EA = 1296000 \text{ [kip]}$; $EI \text{ (axis2)} = 432000 \text{ [kip-ft}^2\text{]}$; $EI \text{ (axis3)} = 432000 \text{ [kip-ft}^2\text{]}$; $GJ = 166154 \text{ [kip-ft}^2\text{]}$; $\text{Mass/Length} = 3 \text{ [kipxsec}^2\text{/ft}^2\text{]}$.

All the base nodes are fully restrained.

The FE model is shown below:

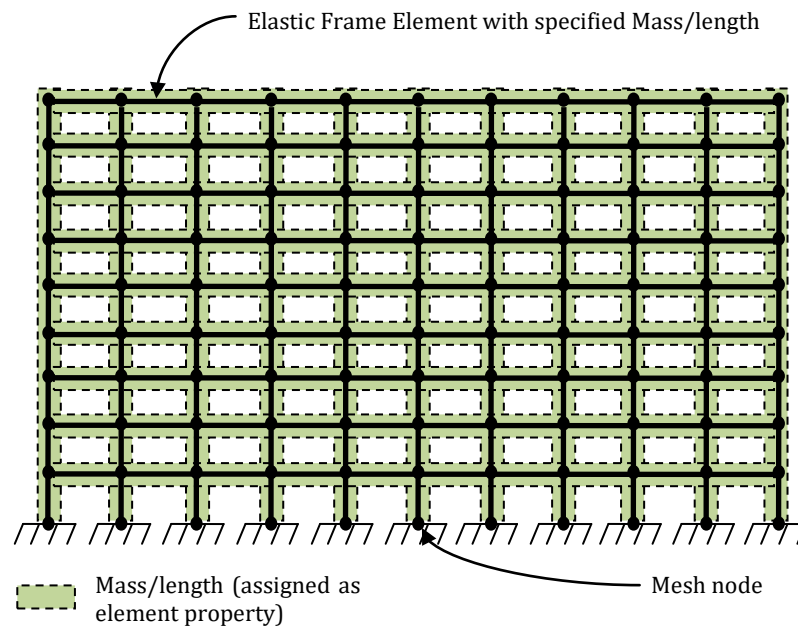


Figure 2.11. Model of EXAMPLE 10

ANALYSIS TYPE

Eigenvalue analysis (Lanczos algorithm).

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.10. Comparison between SeismoStruct and independent results (Bathe & Wilson) for EXAMPLE 10

OUTPUT PARAMETER	SeismoStruct 2018	INDEPENDENT RESULTS
Eigenvalue (1 st mode)	0.58954	0.58954
Eigenvalue (2 nd mode)	5.52696	5.52695
Eigenvalue (3 rd mode)	16.5879	16.5878

COMPUTER FILE

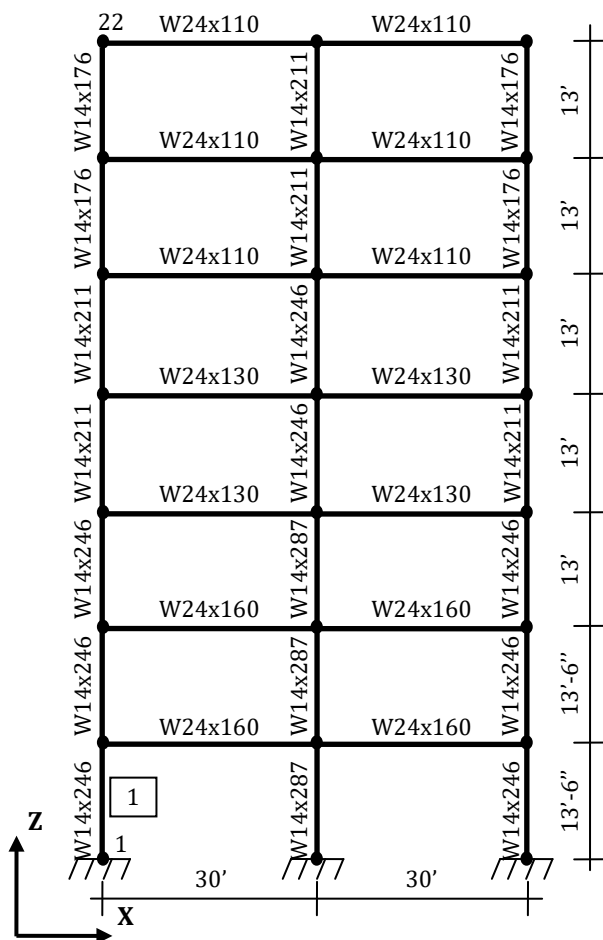
- Ch2-10.spf

EXAMPLE 11 – Two-dimensional frame with static and dynamic loads**DESCRIPTION**

A seven-storey, two-dimensional frame is subjected to earthquake loads. After the eigenvalue analysis, a static analysis (with static lateral loads) and a dynamic time-history analysis (with the N-S component of “El Centro”) have been performed.

The SeismoStruct results are compared with hand calculations. The FE model lies in the X-Z plane.

Important Note: Only bending and axial deformations are considered in the analysis.

GEOMETRY AND PROPERTIESMaterial Properties

$E = 29500 \text{ k/in}^2$

Section Properties

W14X176

$A = 51.7 \text{ in}^2$

$I = 2150 \text{ in}^4$

W14X211

$A = 62.1 \text{ in}^2$

$I = 2670 \text{ in}^4$

W14X246

$A = 72.3 \text{ in}^2$

$I = 3230 \text{ in}^4$

W14X287

$A = 84.4 \text{ in}^2$

$I = 3910 \text{ in}^4$

W24X110

$A = 32.5 \text{ in}^2$

$I = 3330 \text{ in}^4$

W24X130

$A = 38.3 \text{ in}^2$

$I = 4020 \text{ in}^4$

W24X160

$A = 47.1 \text{ in}^2$

$I = 5120 \text{ in}^4$

MODELLING AND LOADING

Three different models are considered (one for each analysis): (i) Model A for eigenvalue analysis, (ii) Model B for static analysis and (iii) Model C for dynamic time-history analysis.

In each model, columns and beams are modelled with elastic frame elements (elfrm) with the following properties:

- W14X176: $EA = 1525150 \text{ [kip]}; EI (\text{axis2}) = 6.3425\text{E}+007 \text{ [kip-in}^2\text{]}; EI (\text{axis3}) = 6.3425\text{E}+007 \text{ [kip-in}^2\text{]}; GJ = 1.0000\text{E}+008 \text{ [kip-in}^2\text{]};$

- W14X211: EA = 1831950 [kip]; EI (axis2) = 7.8765E+007 [kip-in²]; EI (axis3) = 7.8765E+007 [kip-in²]; GJ = 1.0000E+008 [kip-in²];
- W14X246: EA = 2132850 [kip]; EI (axis2) = 9.5285E+007 [kip-in²]; EI (axis3) = 9.5285E+007 [kip-in²]; GJ = 1.0000E+008 [kip-in²];
- W14X287: EA = 2489800 [kip]; EI (axis2) = 1.1535E+008 [kip-in²]; EI (axis3) = 1.1535E+008 [kip-in²]; GJ = 1.0000E+009 [kip-in²];
- W24X110: EA = 958750 [kip]; EI (axis2) = 9.8235E+007 [kip-in²]; EI (axis3) = 9.8235E+007 [kip-in²]; GJ = 1.0000E+008 [kip-in²];
- W24X130: EA = 1129850 [kip]; EI (axis2) = 1.1859E+008 [kip-in²]; EI (axis3) = 1.1859E+008 [kip-in²]; GJ = 1.0000E+009 [kip-in²];
- W24X160: EA = 1389450 [kip]; EI (axis2) = 1.5104E+008 [kip-in²]; EI (axis3) = 1.5104E+008 [kip-in²]; GJ = 1.0000E+009 [kip-in²].

A lumped mass of 0.49 kip-sec²/in is applied in the X direction at joints 5, 8, 11, 14, 17, 20 and 23.

A rigid diaphragm is introduced at each floor, in order to constrain the lateral displacements of the columns.

All the base nodes are fully restrained.

In model B the static lateral loads are applied as permanent loads in terms of forces in the X direction (F1 = 2.5 k, F2 = 5 k, F3 = 7.5 k, F4 = 10 k, F5 = 12.5 k, F6 = 20 k).

In model C a time-history curve (El Centro accelerogram) is loaded in the “Time-history Curves” dialog box, as defined in Figure 2.12 (accelerations are in g). The used output sampling time interval is 0.001 seconds and the response is calculated for the first 8 seconds of the record. Then, the dynamic loads are applied as dynamic time-history loads in terms of accelerations in the X direction.

A mass proportional damping coefficient of 0.3686 and a stiffness proportional coefficient of 0.005127 are defined in the “Project settings” panel. These coefficients are calculated by assuming 5% damping for the first two modes.

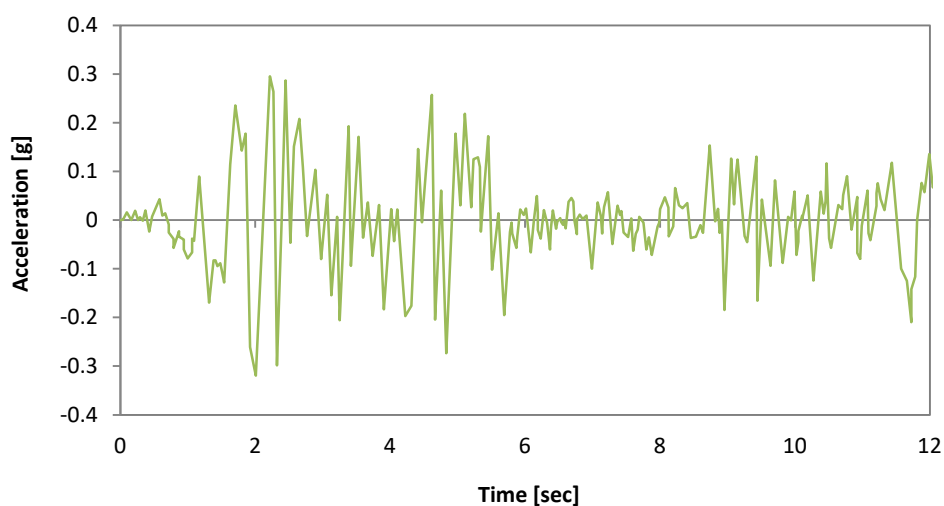


Figure 2.12. El Centro Accelerogram

The FE models are shown below:

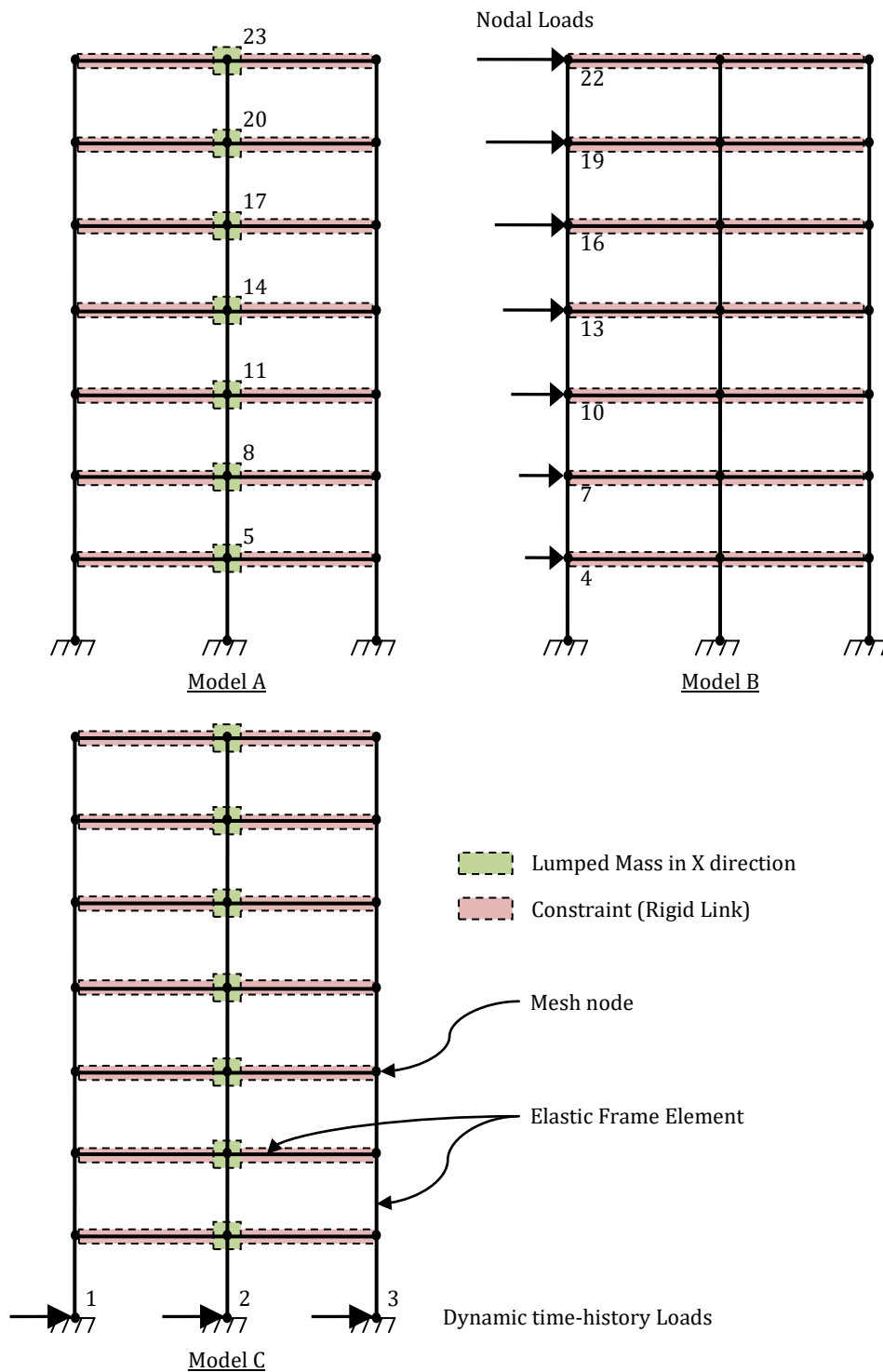


Figure 2.13. Models of EXAMPLE 11

ANALYSIS TYPE

Eigenvalue analysis, static analysis and dynamic time-history analysis (direct integration), using the Hilber-Hughes-Taylor method for time integration (with the alpha factor equal to zero).

RESULTS COMPARISON

The most significant results are compared in the tables below:

Table 2.11. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 11 (Eigenvalue analysis)

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A	Period [sec] (1 st mode)	1.2732	1.2732
	Period [sec] (2 nd mode)	0.4313	0.4313
	Period [sec] (3 rd mode)	0.2420	0.2420
	Period [sec] (4 th mode)	0.1602	0.1602
	Period [sec] (5 th mode)	0.1190	0.1190
	Period [sec] (6 th mode)	0.0951	0.0951
	Period [sec] (7 th mode)	0.0795	0.0795

Table 2.12. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 11 (Static analysis)

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
B	Ux (jt. 22) [in]	1.451	1.451
	Axial force (fr. 1) [kip]	70.01	69.99
	Moment (fr. 1, jt. 1) [k-in]	2324.7	2324.68

Table 2.13. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 11 (Dynamic time-history analysis with dt = 0.001 s)

MOD.	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
C	Ux (jt. 22) [in]	5.44	5.46
	Axial force (fr. 1) [kip]	261	258
	Moment (fr. 1, jt. 1) [k-in]	9077	8740

* Considering dt equal to 0.02 seconds.

COMPUTER FILES

- Ch2-11_A.spf
- Ch2-11_B.spf
- Ch2-11_C.spf

NOTE

If the output sampling time interval is sufficiently small (0.001 seconds), the results are more accurate (than those obtained with hand calculations, for which the assumed output sampling is 0.01 seconds).

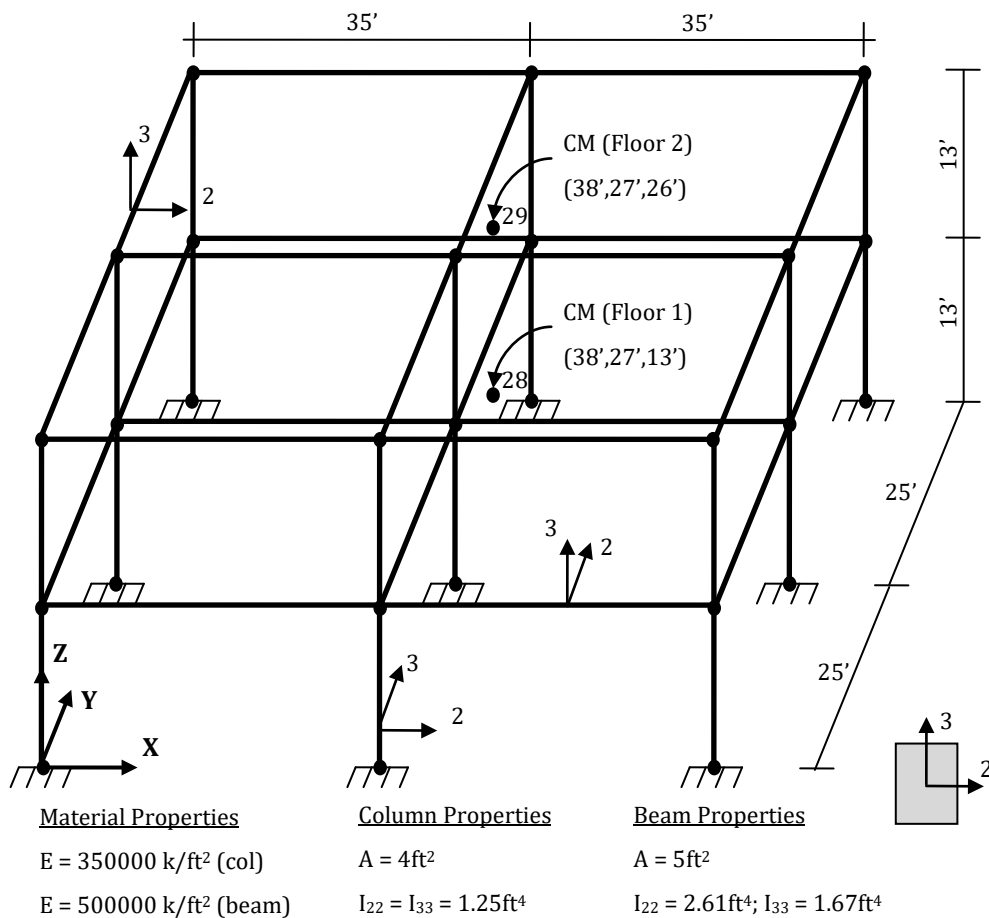
EXAMPLE 12 – Eigenvalue analysis of a three-dimensional frame with rigid floor diaphragm

DESCRIPTION

A two-storey, two-bay, three-dimensional frame structure is analyzed for its four natural periods. The structure is doubly symmetric in plan, except that the center of mass at each storey level is eccentric, as shown in the figure below. The entire storey mass is applied at these joints in the X and Y directions only.

The SeismoStruct results are compared with independent results. The FE model lies in the X-Y-Z space.

GEOMETRY AND PROPERTIES



MODELLING AND LOADING

The frame members are modelled through elastic frame elements (elfrm) with the following properties:

- Beam = 2500000 [kip]; EI (axis2) = 1305000 [kip-ft²]; EI (axis3) = 835000 [kip-ft²]; GJ = 1.0000E+010 [kip-ft²];
- Column = 1400000 [kip]; EI (axis2) = 437500 [kip-ft²]; EI (axis3) = 437500 [kip-ft²]; GJ = 1.0000E+010 [kip-ft²];

A lumped mass is applied to joints 28 and 29 with a value of 6.2112 k-sec²/ft in the X and Y directions. Two rigid diaphragm constraints are introduced (one at each floor level). All the base nodes are fully restrained.

The FE model is presented below:

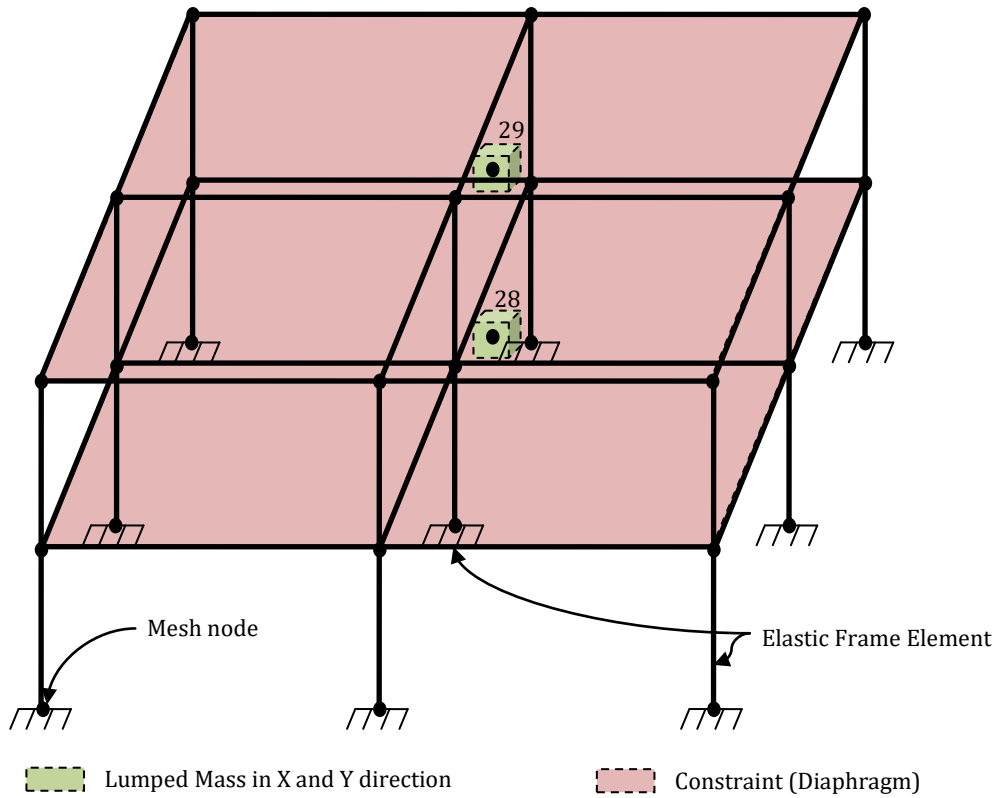


Figure 2.14. Model of EXAMPLE 12

ANALYSIS TYPE

Eigenvalue analysis (Lanczos algorithm).

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.14. Comparison between SeismoStruct and independent results for EXAMPLE 12

OUTPUT PARAMETER	SeismoStruct 2018	INDEPENDENT RESULTS
Period [sec] (1 st mode)	0.2271	0.2271
Period [sec] (2 nd mode)	0.2156	0.2156
Period [sec] (3 rd mode)	0.0733	0.0733
Period [sec] (4 th mode)	0.0720	0.0720

COMPUTER FILE

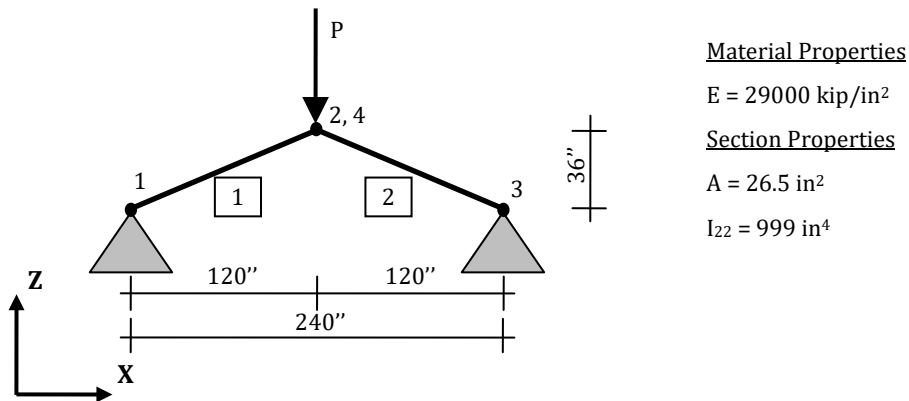
- Ch2-12.spf

EXAMPLE 13 – Large axial displacements**DESCRIPTION**

A three-hinged arch is subjected to a pseudo-static vertical load P (displacement) applied to the central joint in order to test static nonlinear analysis with large axial displacements.

The resulting vertical support reaction at joint 1 obtained with the FE analysis program SeismoStruct is compared with independent results. The model lies in the X-Z plane.

Important Note: Only bending and axial deformations are considered in the analysis. Shear deformations are ignored by SeismoStruct by default.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The frame objects are modelled through an elastic frame element (elfrm) with the following properties: $EA = 768500 \text{ [kip]}$; $EI \text{ (axis2)} = 2.8971\text{E}+007 \text{ [kip-in}^2\text{]}$; $EI \text{ (axis3)} = 2.8971\text{E}+007 \text{ [kip-in}^2\text{]}$; $GJ = 1.0000\text{E}+012 \text{ [kip-in}^2\text{]}$.

A rigid link is introduced between the two coincident nodes 2 (master) and 4 (slave), restraining the X, Y and Z degrees of freedom. The base nodes are restrained in X, Y, Z, RX and RZ.

The load P is defined as a permanent displacement load, applied at joint 2 in the Z direction with a value of 12 inches.

The FE model is presented below:

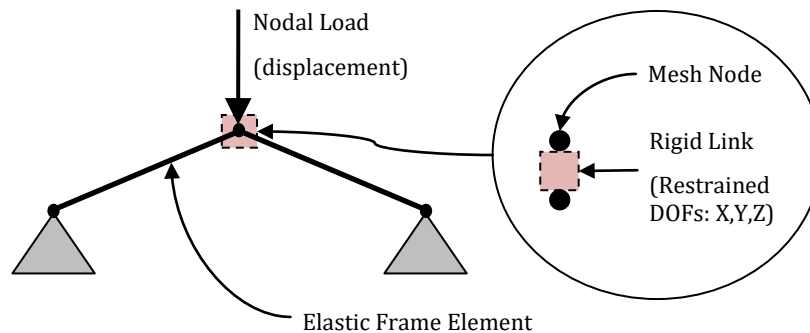


Figure 2.15. Model of EXAMPLE 13

ANALYSIS TYPE

Static analysis (non-variable load).

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.15. Comparison between SeismoStruct and independent results for EXAMPLE 13

OUTPUT PARAMETER	SeismoStruct 2018	INDEPENDENT RESULTS
$F_z(jt. 1)$ [k]	3497.35	3497.35

COMPUTER FILE

- Ch2-13.spf

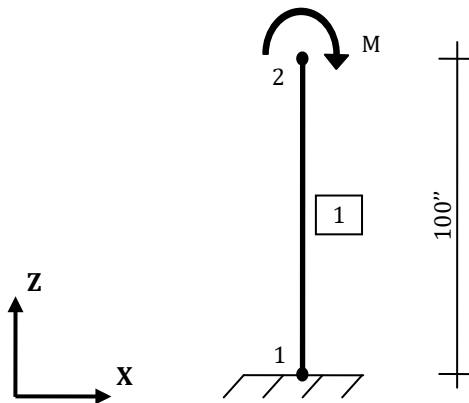
EXAMPLE 14 – Large bending displacements**DESCRIPTION**

A fixed base cantilever column is subjected to a concentrated moment M applied at the top of the element in order to test static nonlinear analysis with large bending displacements. The moment M is increased until the rotation at the top of the column is equal to π radians (180 degrees).

Three different models have been built for this example. The models are identical except for the column discretization, which is subdivided into 4, 16 and 64 elements respectively.

The resulting vertical and horizontal displacements of the top of the column and the moment required to have the desired deflected shape obtained with the FE analysis program SeismoStruct are compared with hand calculations. The model is in the X-Z plane.

Important Note: Only bending and axial deformations are considered in the analysis. Shear deformations are ignored by SeismoStruct by default.

GEOMETRY AND PROPERTIESMaterial Properties

$$E = 29000 \text{ kip/in}^2$$

Section Properties

$$A = 3.83 \text{ in}^2$$

$$I_{22} = 11.3 \text{ in}^4$$

$$I_{33} = 3.86 \text{ in}^4$$

$$J = 0.151 \text{ in}^4$$

MODELLING AND LOADING

The cantilever beam is modelled through an elastic frame element (elfrm) with the following properties: $EA = 111070$ [kip]; EI (axis2) = 327700 [kip-in²]; EI (axis3) = 111940 [kip-in²]; $GJ = 1684$ [kip-in²].

Three different models with different frame discretization are considered:

- Model A: 4 elements
- Model B: 16 elements
- Model C: 64 elements

Moment M is defined as an incremental load in terms of rotations at joint 2 in RY direction with a value of 3.14 (π).

The base node is fully restrained.

The FE model is presented below:

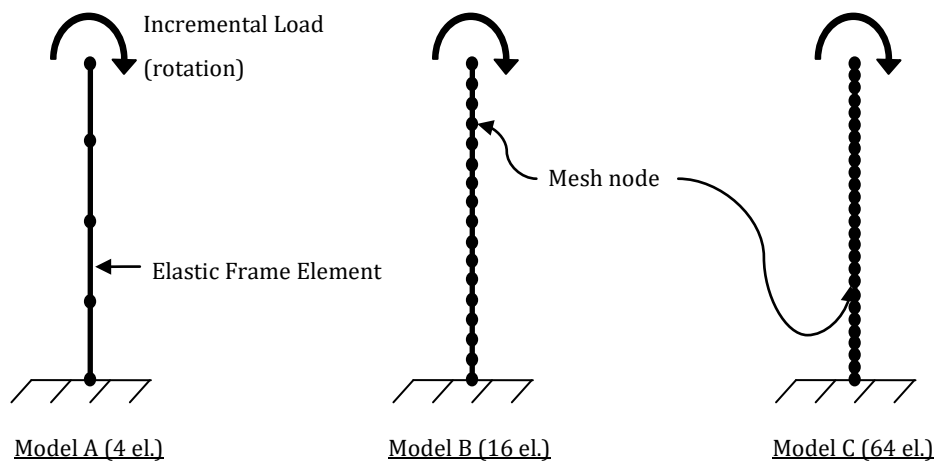


Figure 2.16. Models of EXAMPLE 14

ANALYSIS TYPE

Static pushover analysis.

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.16. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 14

MODEL	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A (4 el.)	U_x (jt. 2) [in]	65.328	63.662
	U_z (jt. 2) [in]	-100	-100
	Moment [k-in]	-10295.023	10295
B (16 el.)	U_x (jt. 2) [in]	63.764	63.662
	U_z (jt. 2) [in]	-100	-100
	Moment [k-in]	-10295.023	10295
C (64 el.)	U_x (jt. 2) [in]	63.668	63.662
	U_z (jt. 2) [in]	-100	-100
	Moment [k-in]	-10295.023	10295

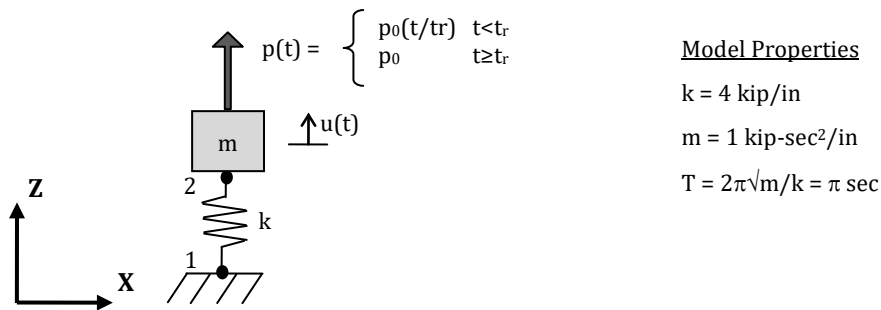
COMPUTER FILES

- Ch2-14_A.spf
- Ch2-14_B.spf
- Ch2-14_C.spf

EXAMPLE 15 – Linear link with ramp loading**DESCRIPTION**

An undamped single degree-of-freedom structure, consisting of two coincident joints (one fixed and the other free) connected by a link element and with a lumped mass applied at the free joint, is subjected to a ramp load. The ramp loading has a finite rise time t_r and it is constant thereafter (see description below). Two different load cases are defined: one (called Load Case A) considering t_r equal to the period of the structure ($t_r = T = \pi$), the other (called Load Case B) considering t_r equal to one-half the period of the structure ($t_r = T/2 = \pi/2$).

The vertical displacement of the joint at two selected time steps obtained with the FE analysis program SeismoStruct are compared with hand calculations. The FE model lies in the X-Z plane.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The model consists of two coincident joints connected by a linear link element fixed at the first joint and free at the other one. As shown in the figure above, the link is modelled in the X-Z plane and is oriented such that its local 1 axis is parallel to the global Z axis. The only active degree-of-freedom in the model is U_z . Similarly, the only degree-of-freedom in the linear link element with nonzero properties is the F1 (axial) degree-of-freedom.

A lumped mass is applied at joint 2 with a value of 1 kip-sec²/in in the Z direction only.

Four different models have been created:

- Model A1: t_r is equal to the period of the structure (RAMP1) and the time step is set to 0.25 sec;
- Model A2: t_r is equal to the period of the structure (RAMP1) and the time step is set to 0.0025 sec;
- Model B1: t_r is equal to one-half the period of the structure (RAMP2) and the time step is set to 0.25 sec;
- Model B2: t_r is equal to one-half the period of the structure (RAMP2) and the time step is set to 0.0025 sec.

Both loads, RAMP1 and RAMP2, are loaded in the “Time-history Curves” dialog box and are defined as follows:

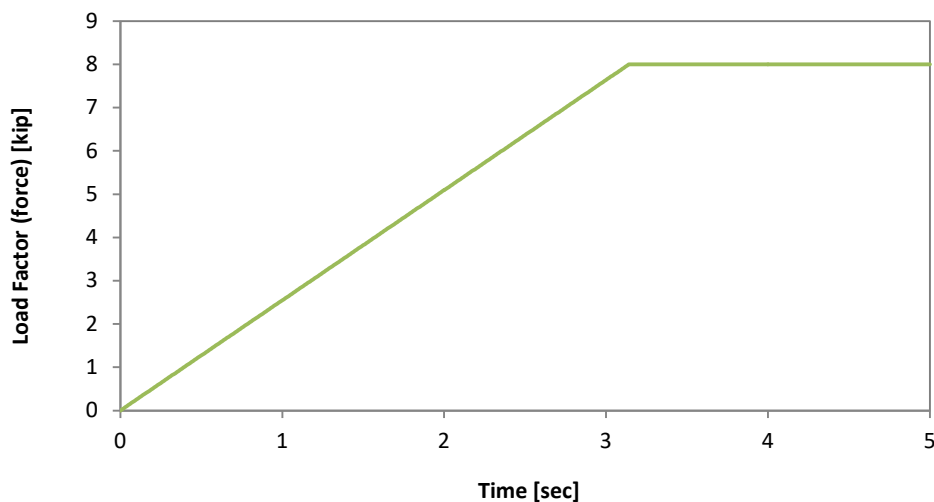


Figure 2.17. Time-history curve (RAMP1)

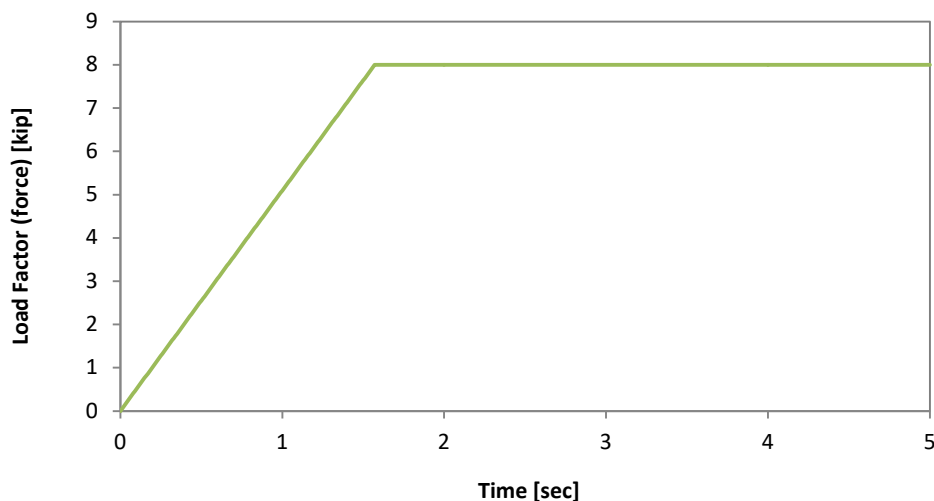


Figure 2.18. Time-history curve (RAMP2)

In each model, the ramp loading is applied at the free end of the link (joint 2) as dynamic time-history load in terms of forces in the Z direction.

The FE model is presented below:

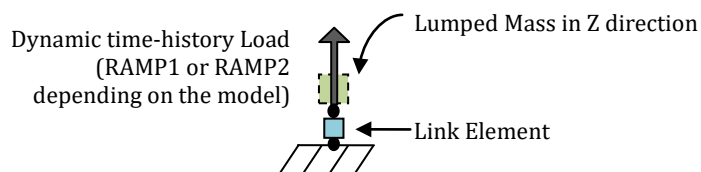


Figure 2.19. Model of EXAMPLE 15

ANALYSIS TYPE

Dynamic time-history analysis (direct integration), using the Hilber-Hughes-Taylor method for time integration with an alpha factor of 0, a beta factor of 0.25 and a gamma factor of 0.5

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.17. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 15

MODEL	OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
A1	U_z (jt. 2) [in] at $t = 1$ sec	0.34210	0.34718
	U_z (jt. 2) [in] at $t = 4$ sec	1.9888	2
A2	U_z (jt. 2) [in] at $t = 1$ sec	0.34718	0.34718
	U_z (jt. 2) [in] at $t = 4$ sec	2	2
B1	U_z (jt. 2) [in] at $t = 1$ sec	0.68419	0.69436
	U_z (jt. 2) [in] at $t = 4$ sec	0.74300	0.74031
B2	U_z (jt. 2) [in] at $t = 1$ sec	0.69436	0.69436
	U_z (jt. 2) [in] at $t = 4$ sec	0.74032	0.74031

COMPUTER FILES

- Ch2-15_A1.spf
- Ch2-15_A2.spf
- Ch2-15_B1.spf
- Ch2-15_B2.spf

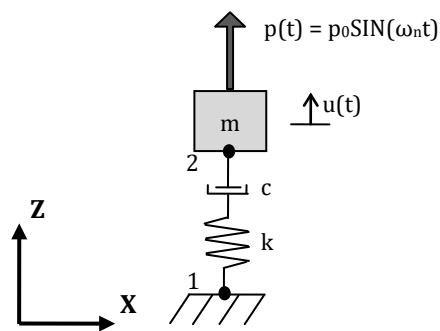
EXAMPLE 16 – Damper element under harmonic loading

DESCRIPTION

A damped single degree-of-freedom structure, which consists of two coincident joints (one fixed and the other free) connected by a link element and with a lumped mass and a damping element applied at the free joint, is subjected to a harmonic load. The frequency of the harmonic load is chosen to be equal to the frequency of the spring-mass-damper system. The damper is assumed to provide 5% of critical damping.

The joint vertical displacement of the spring-mass-damper system, evaluated at various time steps and the steady-state deformation of the system obtained with the FE analysis program SeismoStruct are compared with hand calculations. The FE model is in the X-Z plane.

GEOMETRY AND PROPERTIES



Model Properties

$$k = 100 \text{ kip/in}$$

$$m = 1 \text{ kip-sec}^2/\text{in}$$

$$\omega_n = \sqrt{k/m} = 10 \text{ rad/sec}$$

$$c = 2\zeta\omega_n m = 1 \text{ kip-sec/in}$$

MODELLING AND LOADING

The model consists of two coincident joints connected by a linear link element that is fixed at the first joint and is free at the other one. As shown in the figure above, the link is oriented such that its local 1 axis is parallel to the global Z axis. The only degree-of-freedom with nonzero properties is F1 (axial). The stiffness of the link element is set to 100 k/in.

A lumped mass is applied at joint 2 with a value of 1 kip-sec²/in in the Z direction only. A single-node damping element is applied at the same joint with the damping coefficient set to 1 kip-sec/in.

A harmonic time-history curve is loaded in the “Time-history Curves” dialog box. It is defined as follows:

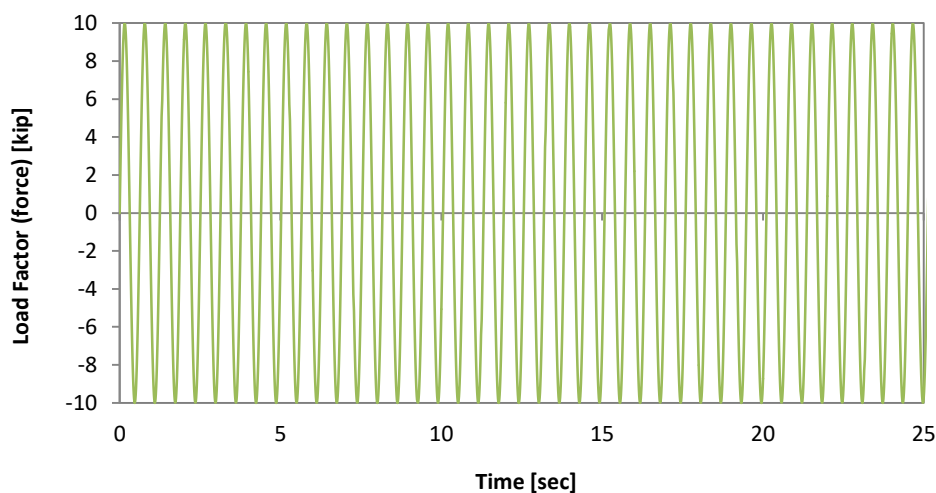


Figure 2.20. Harmonic time-history curve

The load is applied at the free end of the link (joint 2) as dynamic time-history load in terms of forces in the Z direction.

The FE model is presented below:

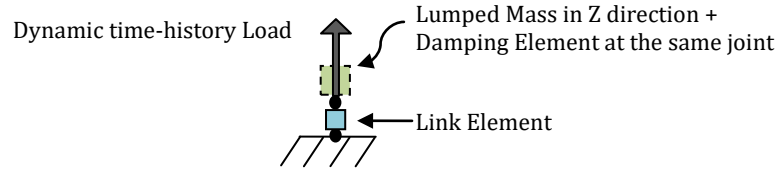


Figure 2.21. Model of EXAMPLE 19

ANALYSIS TYPE

Dynamic time-history analysis (direct integration), using the Hilber-Hughes-Taylor method for time integration with an alpha factor of zero, a beta factor of 0.25 and a gamma factor of 0.5

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.18. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 16

OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
U_z (jt. 2) [in] at $t = 0.5$ sec	-0.10431	-0.10488
U_z (jt. 2) [in] at $t = 5$ sec	-0.88456	-0.88858
U_z (jt. 2) [in] at $t = 11$ sec	0.99463	0.99497
Steady-State deformation [in]	0.99892	1.00000

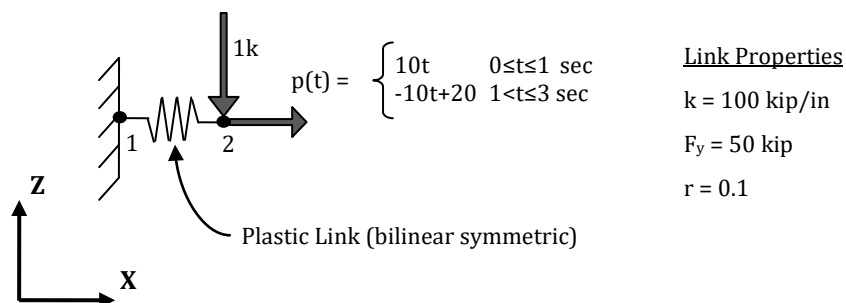
COMPUTER FILES

- Ch2-16_Damp.spf

EXAMPLE 17 – Plastic link (bilinear symmetric)**DESCRIPTION**

This example uses a single degree-of-freedom structure to test the behaviour of a link element with a bilinear symmetric response curve. A static analysis is used to push first the link element to a positive 10 inch displacement (the applied load is a displacement) and then, starting from the final conditions, to a negative 10 inch displacement (20 inch push).

The resulting link force at various deformations obtained with the FE analysis program SeismoStruct is compared with hand calculations. The FE model lies in the X-Z plane.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The model consists of two coincident nodes connected with a link element. The link element is oriented such that its positive local 2 axis is parallel to the positive global Z axis. Only F1 degree-of-freedom properties are defined for the link element. A bilinear symmetric response curve is chosen with the following properties: k_0 (stiffness) = 100 k/in, yield force $F_y = 50$ kips, ratio of initial stiffness to yield stiffness = 0.1. The yielding exponent, controlling the sharpness of the transition from the initial stiffness to the yielded stiffness is not modelled with this response curve.

A time-history curve is loaded in the “Time-history Curves” dialog box and is defined as follows:

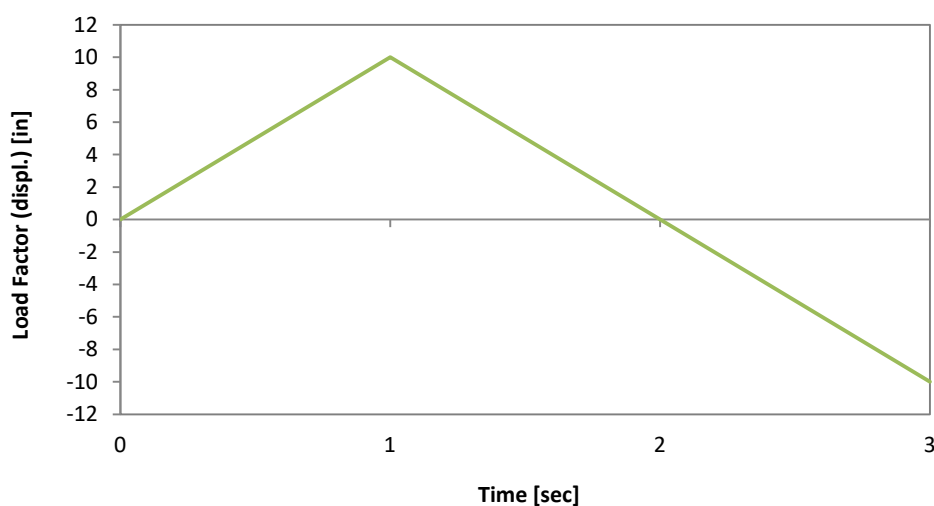


Figure 2.22. Time-history curve

The load is applied at the free end of the link (joint 2) as static time-history load in terms of displacements in the X direction.

The weight of the link is 1 kip. It is applied as permanent load acting in the Z direction.

The FE model is presented below:

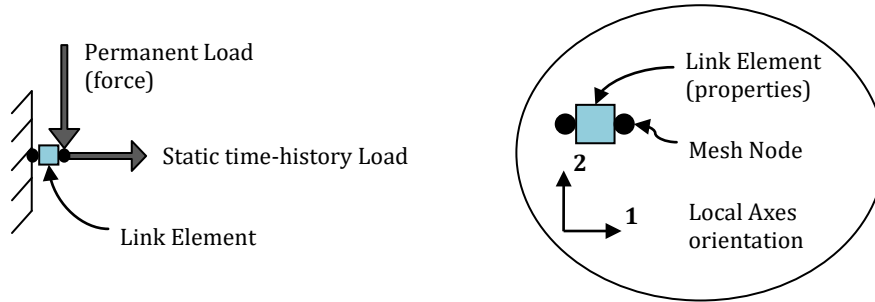


Figure 2.23. Model of EXAMPLE 17

ANALYSIS TYPE

Static time-history analysis.

RESULTS COMPARISON

Because a bilinear symmetric response curve is adopted in order to model the link element in SeismoStruct, only few results can be compared with the independent results.

Table 2.19. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 17

OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
Link force at 9.6 in deformation [kips] (A)	141	141
Link force at 9.6 in deformation [kips] (B)	105	105
Link force at -9.6 in deformation [kips] (C)	-141	-141

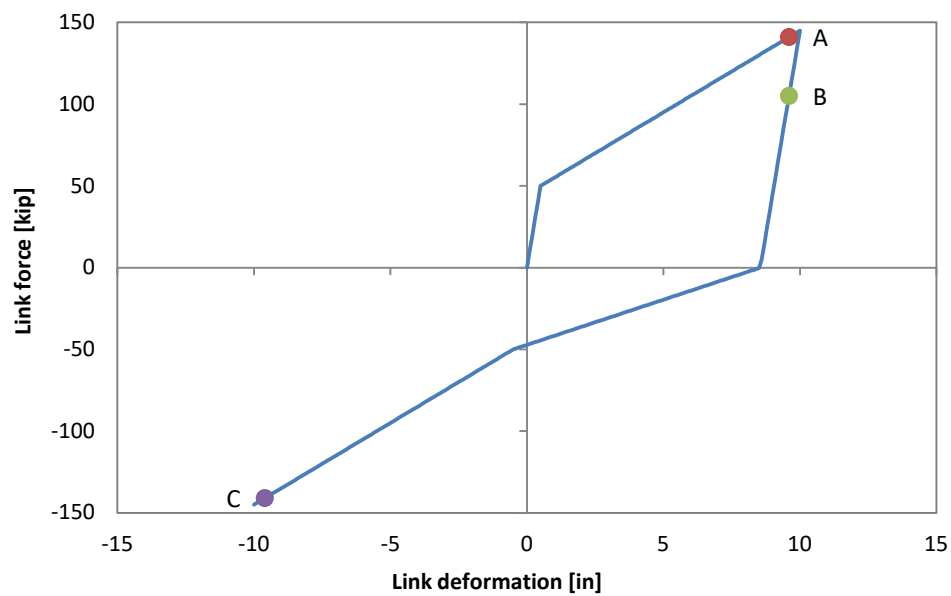


Figure 2.24. Link force vs. Link deformation

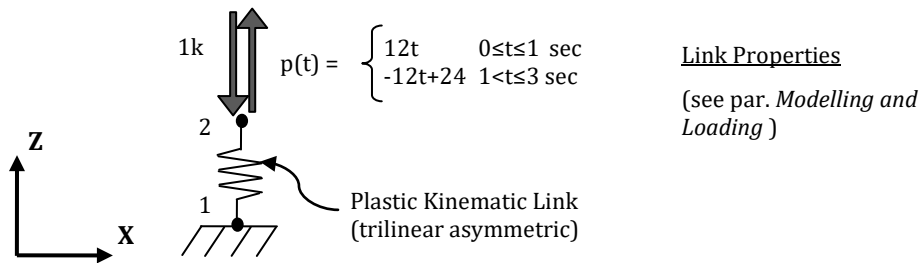
COMPUTER FILES

- Ch2-17.spf

EXAMPLE 18 – Plastic link (trilinear asymmetric)**DESCRIPTION**

This example uses a single degree-of-freedom structure to test the behaviour of a link element with a trilinear asymmetric response curve for which an isotropic hardening rule is adopted. Multi-linear force-deformation characteristics are defined for the link element for both tension and compression behaviour. The force-deformation characteristics modelled for tension are different from those modelled for compression. A nonlinear static analysis is used to push the link element to a positive 12 inch displacement (the applied load is a displacement), then a second load is started from the final conditions of the first analysis cased and used to push the link element to a negative 12 inch displacement (24 inch push).

The resulting link force at various deformations obtained with the FE analysis program SeismoStruct is compared with hand calculations. The FE model is in the X-Z plane.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

The model consists of two coincident nodes connected by a link element. The link element is oriented such that its positive local 1 axis is parallel to the positive global Z axis. Only F1 degree-of-freedom properties are defined for the link element. A trilinear asymmetric response curve is chosen with the following properties: initial stiffness $k_0 (+)$ = 25 k/in, displacement $d_1 (+)$ = 2 in, stiffness of second branch $k_1 (+)$ = 5 k/in, displacement $d_2 (+)$ = 6 in, stiffness of third branch $k_2 (+)$ = 1 k/in, initial stiffness $k_0 (-)$ = 40 k/in, displacement $d_1 (-)$ = -1 in, stiffness of second branch $k_1 (-)$ = 2 k/in, displacement $d_2 (-)$ = -6 in, stiffness of third branch $k_2 (-)$ = 0.5 k/in.

A time-history curve is loaded in the “Time-history Curves” dialog box and is defined as follows:

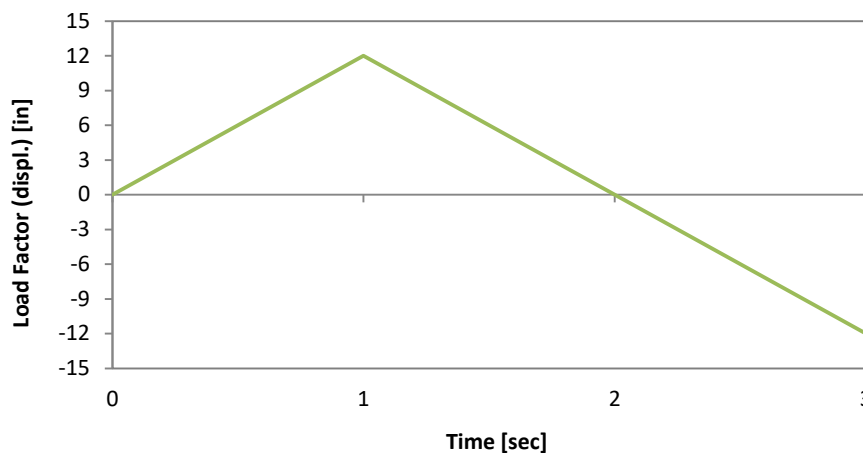


Figure 2.25. Time-history curve

The load is applied at the free end of the link (joint 2) as static time-history load in terms of displacements in the Z direction.

The link's weight is 1 kip, defined as lumped mass in the Z direction only with the value of 0.00259067 [kip-sec²/in] and then automatically converted to gravity load.

The FE model is presented below:

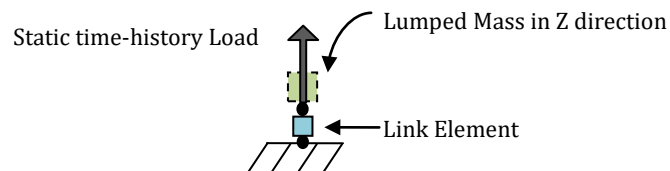


Figure 2.26. Model of EXAMPLE 18

ANALYSIS TYPE

Static time-history analysis.

RESULTS COMPARISON

The most significant results are compared in the table below:

Table 2.20. Comparison between SeismoStruct and hand-calculated results for EXAMPLE 18

OUTPUT PARAMETER	SeismoStruct 2018	Hand calculations
Link force at 1.2 in deformation [kips] (A)	30	30
Link force at 4.2 in deformation [kips] (B)	61	61
Link force at 12 in deformation [kips] (C)	76	76
Link force at 11.52 in deformation [kips] (D)	64	64
Link force at 9.6 in deformation [kips] (E)	16	10
Link force at 2.4 in deformation [kips] (F)	-33.2	-36
Link force at -12 in deformation [kips] (G)	-53	-52.25

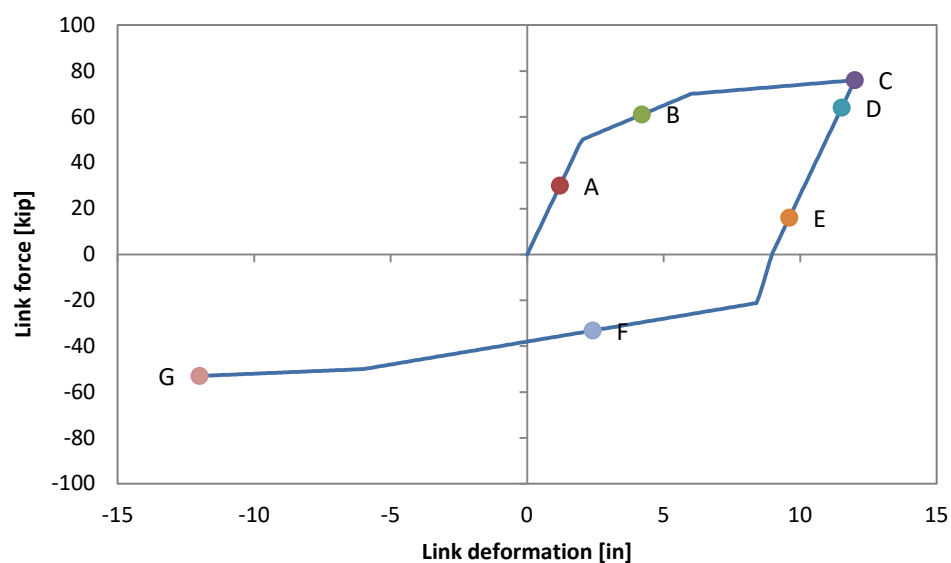


Figure 2.27. Link force vs. Link deformation

COMPUTER FILES

- Ch2-18.spf

NOTE: Comparison of the SeismoStruct results with the independent results is exact only for the ascending branch. The other results differ because in the SeismoStruct's link element the adopted hardening rule is isotropic, rather than kinematic.

Chapter 3 COMPARISONS WITH LITERATURE CASE-STUDIES

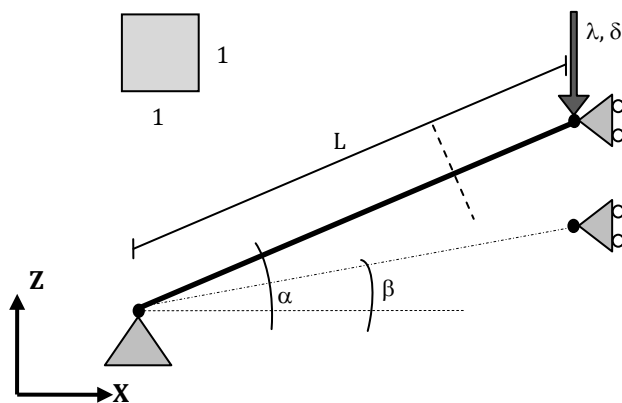
EXAMPLE 1 – Von Mises truss

DESCRIPTION

A single frame element, defined in literature as *Von Mises truss*, is subjected to a vertical load at its final joint. This example tests SeismoStruct's static pushover analysis by applying a pseudo-static load (displacement) at the end of the element and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of force vs. displacement, obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model is in the X-Z plane.

GEOMETRY AND PROPERTIES



Material Properties

$E = 29000 \text{ kip/in}^2$

Geometrical Properties

W14X90

$L \cos(\alpha) = 120''$

$L \sin(\alpha) = 36''$

$\alpha = 0.291 \text{ rad}$

Section Properties

$A = 26.5 \text{ in}^2$

MODELLING AND LOADING

Two different models have been created:

- Model A: the element is modelled with an elastic frame element (elfrm) which has $EA = 768500 \text{ kip}$;
- Model B: an I-shaped section is adopted (W14X90 -> this section has the same elastic section properties of the model above); the element is modelled with a force-based inelastic frame element (infrmFB), with 5 integration sections and 100 section fibres.

A pseudo-static load (displacement) is applied incrementally at the final joint in the negative Z direction with a magnitude of 120 in.

The total response is completely attained with the load control phase, thus the program is never required to change for the automatic response control.

ANALYSIS TYPE

Static pushover analysis (Load Control).

RESULTS COMPARISON

The comparison between the reference solution and the analysis results, in terms of forces vs. displacements at the final joint of the frame element, is shown below:

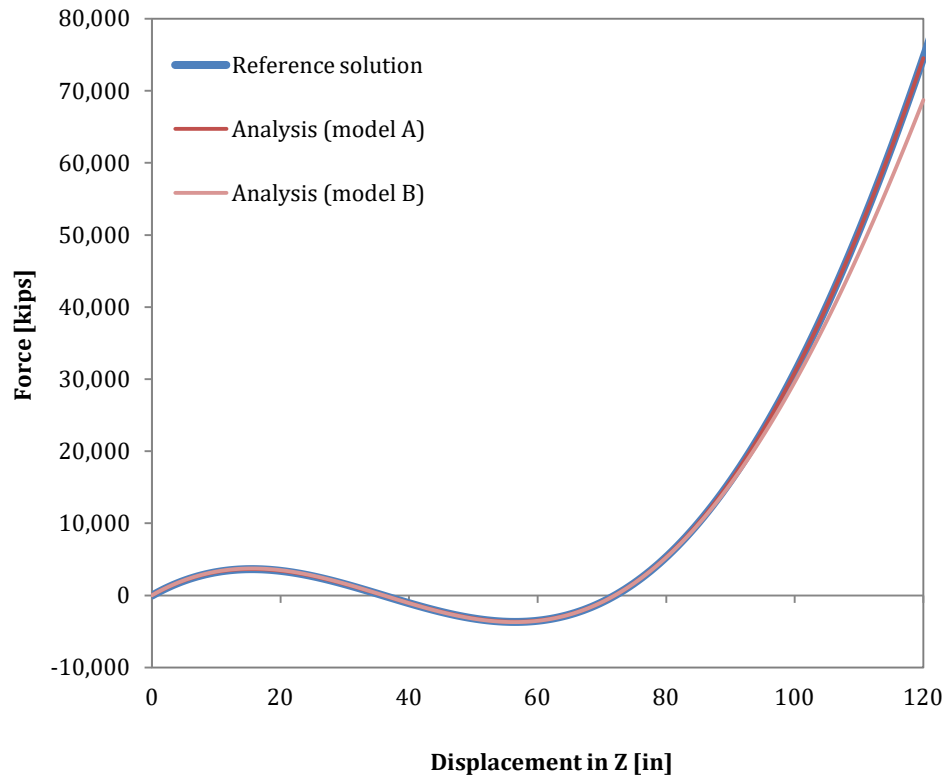


Figure 3.1. Reference solution vs. Analysis results (force-displacement)

COMPUTER FILES

- Ch3-01_A.spf
- Ch3-01_B.spf

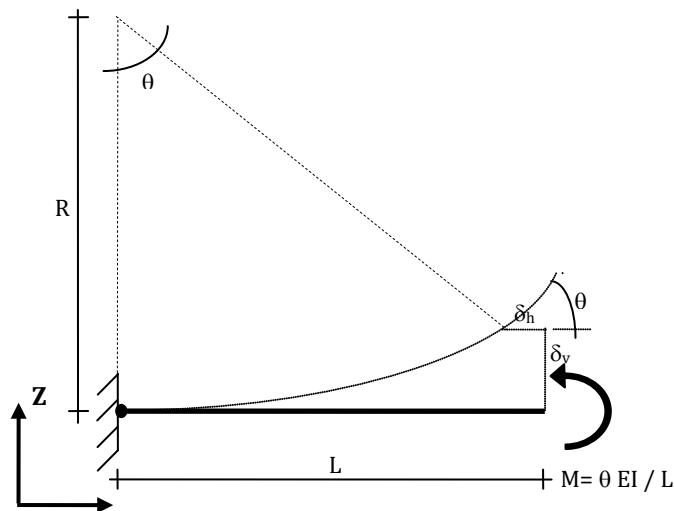
EXAMPLE 2 – Cantilever beam under a moment at the tip

DESCRIPTION

A cantilever beam element is subjected to a moment at the tip. This example tests SeismoStruct's static pushover analysis by applying a pseudo-static load (rotation) at the free end of the element and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of end rotation vs. displacement, obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model is in the X-Z plane.

GEOMETRY AND PROPERTIES



Material Properties

$$E = 29000 \text{ kip/in}^2$$

Geometrical Properties

$$L = 100''$$

Section Properties

W4X13

$$A = 3.83 \text{ in}^2$$

$$I (\text{axis } 2) = 11.3 \text{ in}^4$$

MODELLING AND LOADING

Two different models have been created:

- Model A: the element is modelled with eight elastic frame elements (elfrm) of equal length ($EA = 111070 \text{ kip}$, $EI = 327700 \text{ kip-in}^2$);
- Model B: an I-shaped section is adopted (W4X13); the element is modelled with eight force-based inelastic frame elements (infrmFB) with 3 integration sections and 100 section fibres.

A pseudo-static load (rotation) is incrementally applied at the free end of the cantilever beam final joint in RY direction with a magnitude of 12.5664 rad.

The total response is completely attained with the load control phase.

ANALYSIS TYPE

Static pushover analysis (Load Control).

RESULTS COMPARISON

The comparison between the reference solution (horizontal and vertical components) and the analysis results, in terms of displacement vs. end rotation at the final joint of the frame element, is shown below:

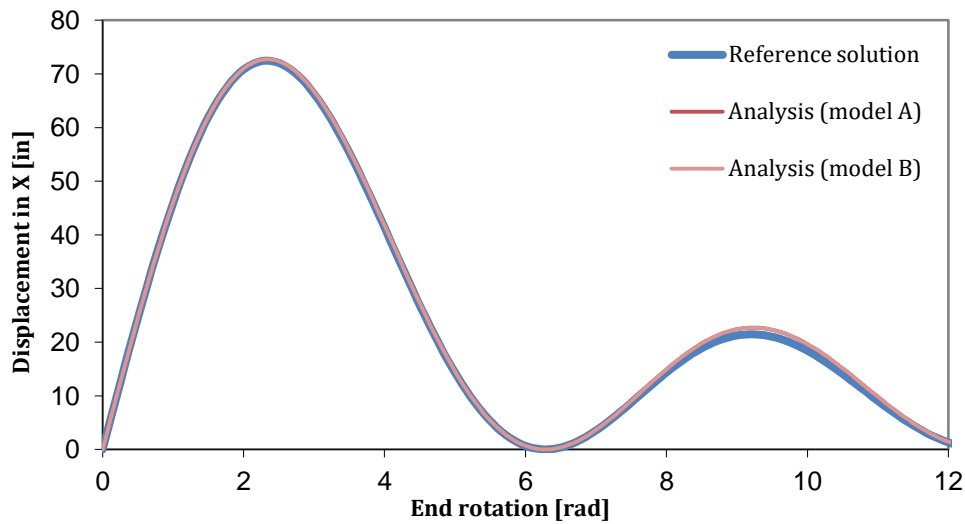


Figure 3.2. Reference solution (horizontal) vs. Analysis results (displacement-end rotation)

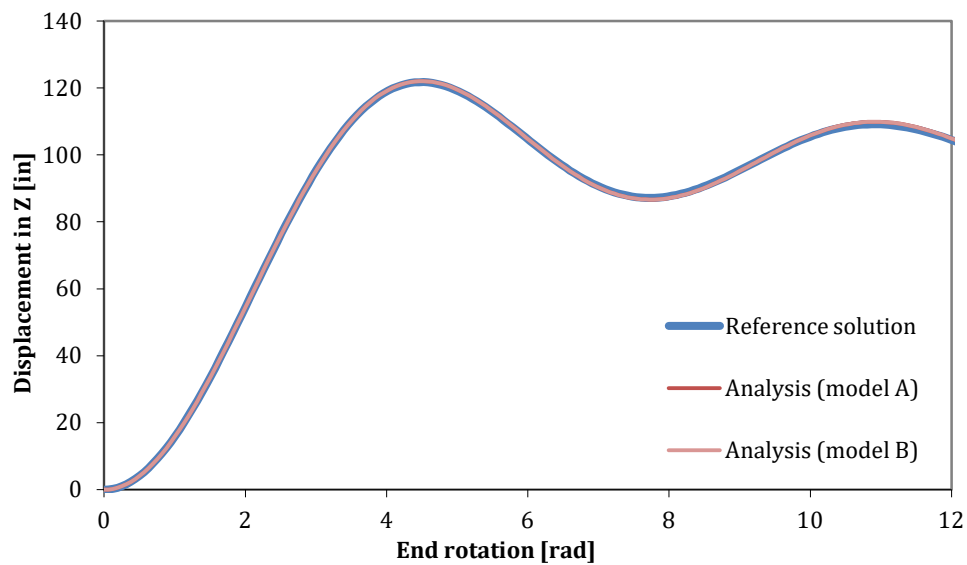


Figure 3.3. Reference solution (vertical) vs. Analysis results (displacement-end rotation)

COMPUTER FILES

- Ch3-02_A.spf
- Ch3-02_B.spf

REFERENCES

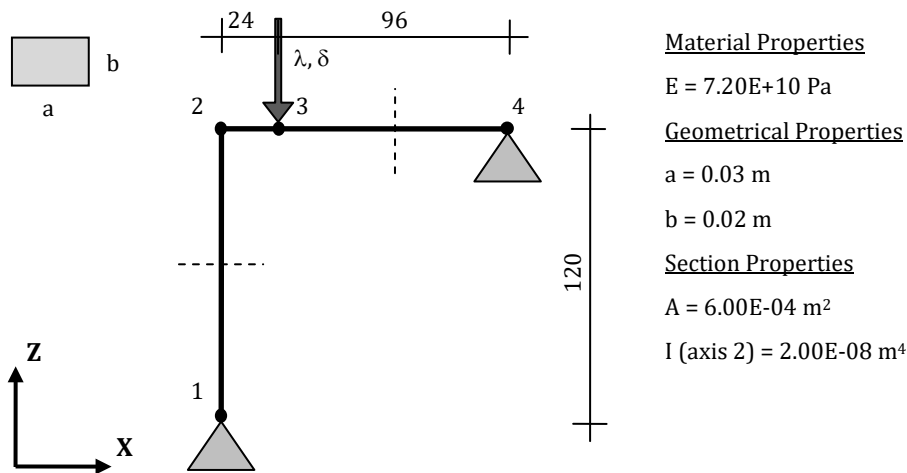
- Bathe K.-J. and Bolourchi S. [1979] "*Large displacement analysis of three-dimensional beam structures*", International Journal of Numerical Methods in Engineering, 14, 961-986

- Simo J. C. and Vu-Quoc L. [1986] "*A three-dimensional finite strain rod model: Part 2: Computational aspects*", Comput. Methods Appl. Mech. Engrg., 58, 79-116
- Crisfield MA [1990] "*A consistent co-rotational formulation for non-linear, three-dimensional, beam-elements*", Comp. Meth. Appl. Mech. Eng. 81, 131-150
- Gummadi L. N. B. and Palazotto A. N. [1998] "*Large strain analysis of beams and arches undergoing large rotations*", Int. J. Non-Linear Mechanics, 33(4), 615-645
- Waszczyszyn Z. and Janus-Michalska M. [1998] "*Numerical approach to the 'exact' finite element analysis of in-plane finite displacements of framed structures*", Comput. Struct., 69, 525-535
- Souza [2000] "*Force-based Finite Element for Large Displacement Inelastic Analysis of Frames*", PhD Thesis, University of California (Berkley)
- Schulz M. and Filippou F. C. [2001] "*Nonlinear Spatial Timoshenko Beam Element with Curvature Interpolation*", International Journal of Numerical Methods in Engineering, 50(4), 761-785

EXAMPLE 3 – Lee’s frame**DESCRIPTION**

A two member frame, defined in literature as *Lee’s frame*, is subjected to a load. This example tests SeismoStruct’s static pushover analysis by applying a pseudo-static load (displacement) at 1/5 of the beam length and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of applied load vs. displacement, obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model is in the X-Z plane.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

Two different models have been created:

- **Model A:** the frame is modelled with five elastic frame elements (elfrm): the column consists of two elements of equal length, whereas the beam consists of three elements of 24, 48 and 48 cm, respectively;
- **Model B:** the frame is modelled with five displacement-based inelastic frame elements (infrmDB) (for the discretization see above); a rectangular section is adopted for both the frame objects and the number of section fibers is set to 100.

A pseudo-static load (displacement) is incrementally applied at 1/5 of the beam length in the negative Z direction with a magnitude of 0.86.

The total response is attained by employing one load control and one automatic load control phase.

ANALYSIS TYPE

Static pushover analysis (Load Control and Automatic Response Control).

RESULTS COMPARISON

The comparison between the reference solution and analysis results, in terms of forces (shear) vs. displacements at the joint of the frame element where the load is applied, is shown below:

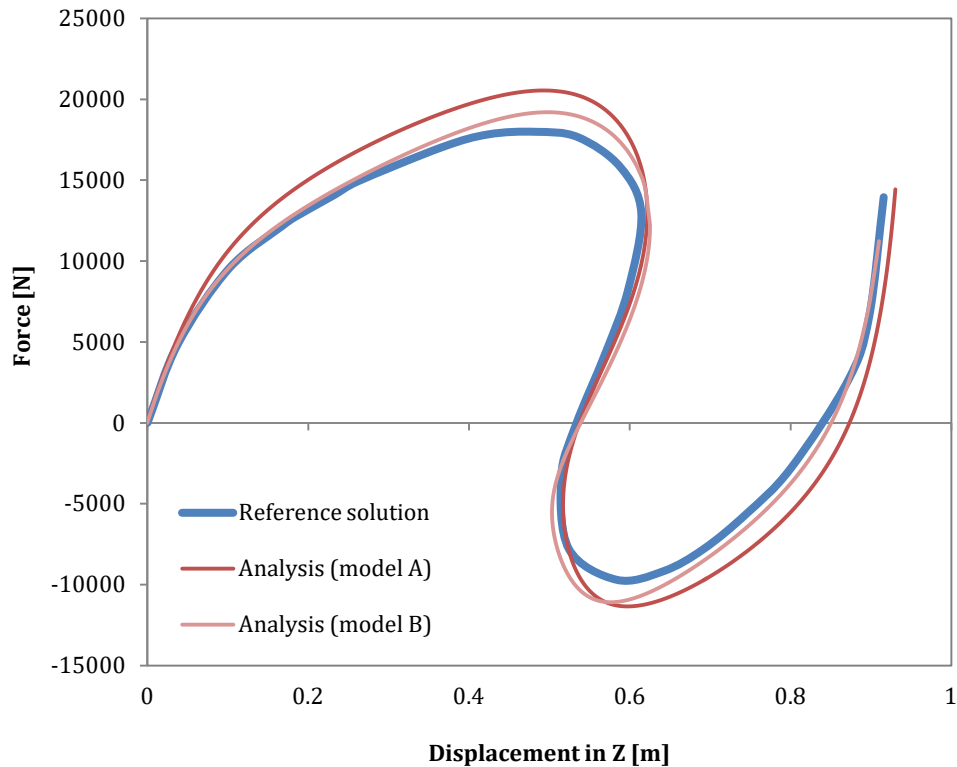


Figure 3.4. Reference solution vs. Analysis results (force-displacement)

COMPUTER FILE

- Ch3-03_A.spf
- Ch3-03_B.spf

NOTE

The reference solution results have been obtained by digitizing the curve plotted in the publication Souza, 2000 (p. 157). Although the global response is well predicted (including the modelling of snap-through and snap-back), the process of finding the best combination of applied loads/loading phases and associated convergence parameters was not straightforward, requiring "a priori" knowledge of the response history.

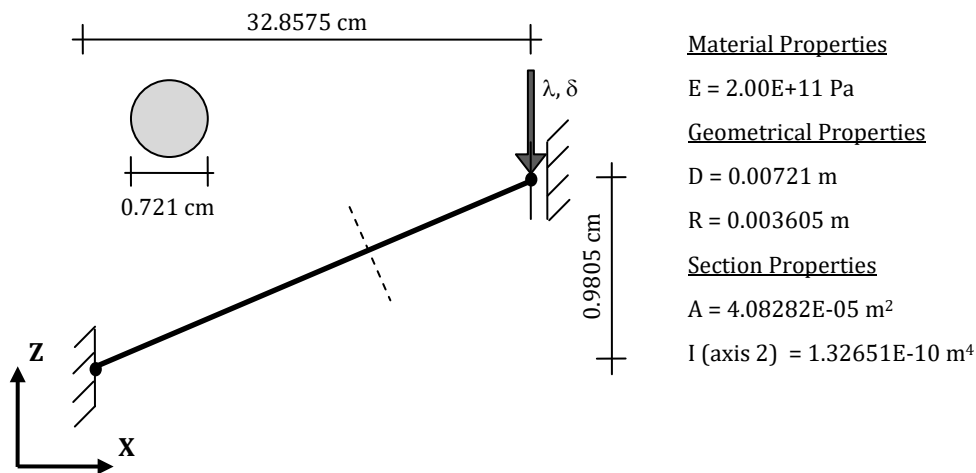
REFERENCES

- Lee S., Manuel F. S., Rossow, E. C. [1968] "*Large deflections and stability of elastic frames*", Journal of the Engineering Mechanics Division, ASCE, EM2, 521-547;
- Cichon C. [1984] "*Large displacements in-plane analysis of elastic plastic frames*", Comput. Struct, 19, 737-745;
- Souza [2000] "*Force-based Finite Element for Large Displacement Inelastic Analysis of Frames*", PhD Thesis, University of California (Berkley)

EXAMPLE 4 – Williams’ toggle frame**DESCRIPTION**

A single element frame, defined in literature as *Williams’ toggle frame*, is subjected to a vertical load at its final joint. This example tests SeismoStruct’s static pushover analysis by applying a pseudo-static load (displacement) at the end of the element and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of shear force (at the fixed end) vs. displacement (at the other end), obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model is in the X-Z plane.

GEOMETRY AND PROPERTIES**MODELLING AND LOADING**

Two different models have been created:

- **Model A:** the element is modelled with eight elastic frame elements (elfrm) of equal length ($EA = 8153803 \text{ N}$, $EI = 26.5 \text{ N-m}^2$);
- **Model B:** a circular section is adopted; the element is modelled with four displacement-based inelastic frame elements (infrmDB) with 120 section fibres.

A pseudo-static load (displacement) is incrementally applied at the right end of the element in the negative Z direction with a magnitude of 0.02.

The total response is completely attained with the load control phase.

ANALYSIS TYPE

Static pushover analysis (Load Control).

RESULTS COMPARISON

The comparison between reference solution and analysis results, in terms of forces vs. displacements at the final joint of the frame element, is shown in the following:

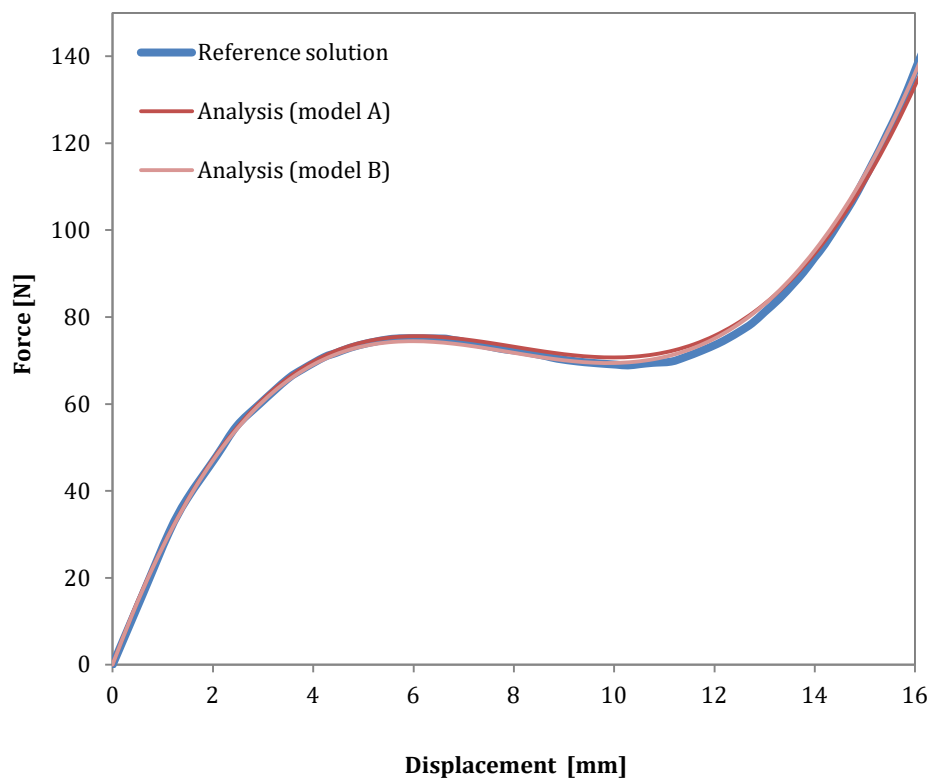


Figure 3.5. Reference solution vs. Analysis results (force-displacement)

COMPUTER FILES

- Ch3-04_A.spf
- Ch3-04_B.spf

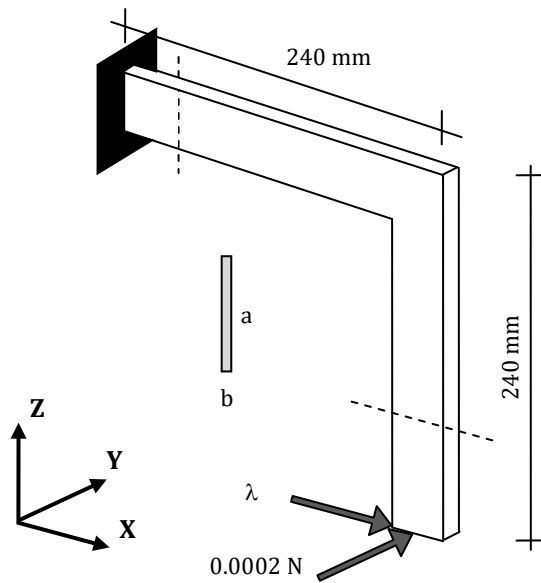
REFERENCES

- Williams F. W. [1964] “An Approach to the non-linear behaviour of the members of a rigid jointed plane framework with finite deflections”, Quarterly Journal of Mechanics and Applied Mathematics, 17(Pt. 4), 451-469
- Chan S. L. [1988] “Geometric and material non-linear analysis of beam-columns and frames using the minimum residual displacement method”, International Journal for Numerical Methods in Engineering, 26, 2657-2669
- Souza [2000] “Force-based Finite Element for Large Displacement Inelastic Analysis of Frames”, PhD Thesis, University of California (Berkley)

EXAMPLE 5 – Clamped right-angle frame under tip load**DESCRIPTION**

A clamped right-angle frame is subjected to a tip load in the X direction. A small permanent load is also applied in the Y direction, in order to artificially induce the buckling instability. This example tests SeismoStruct's static pushover analysis by applying a pseudo-static load (force) at the tip and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of force vs. displacement, obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model lies in the X-Z plane.

GEOMETRY AND PROPERTIESMaterial Properties

$$E = 7.12\text{E}+04 \text{ MPa}$$

$$G = 27190.84 \text{ MPa}$$

Geometrical Properties

$$a = 30 \text{ mm}$$

$$b = 0.6 \text{ mm}$$

$$L = 240 \text{ mm}$$

Section Properties

$$A = 18 \text{ mm}^2$$

$$I (\text{axis } 2) = 1350 \text{ mm}^4$$

$$I (\text{axis } 3) = 0.54 \text{ mm}^4$$

$$J = 2.16 \text{ mm}^4$$

MODELLING AND LOADING

- Each member (horizontal and vertical) is modelled with four elastic frame elements (elfrm) ($EA = 1882320 \text{ N}$, $EI_2 = 96174000 \text{ N-mm}^2$, $EI_3 = 38469.6 \text{ N-mm}^2$, $GJ = 58732.1 \text{ N-mm}^2$);

Two different loads are applied at the tip: a permanent load (force) of 0.0002 N in the Y direction and a pseudo-static load (force) with a value of 1.8 N in the X direction.

The total response is completely attained with the load control phase.

ANALYSIS TYPE

Static pushover analysis.

RESULTS COMPARISON

The comparison between reference solution and analysis results, in terms of force vs. displacement at the tip, is shown in the following:

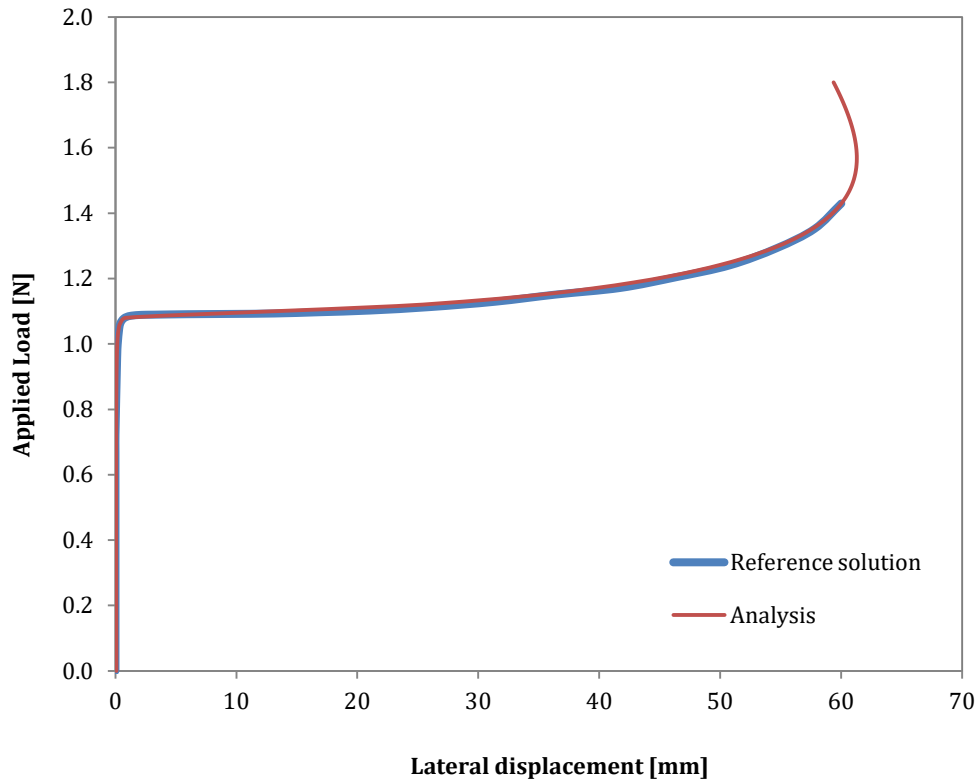


Figure 3.6. Reference solution vs. Analysis results (force-displacement)

COMPUTER FILES

- Ch3-05.spf

NOTE

The reference solution results have been obtained by digitizing the curve plotted in the publication Souza, 2000 (p. 167).

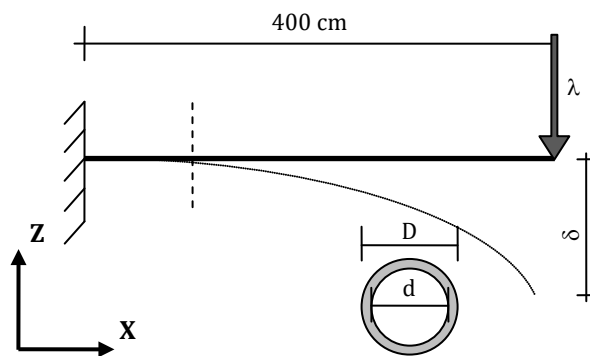
REFERENCES

- Crisfield MA [1990] "A consistent co-rotational formulation for non-linear, three-dimensional, beam-elements", Comp. Meth. Appl. Mech. Eng. 81, 131-150
- Souza [2000] "Force-based Finite Element for Large Displacement Inelastic Analysis of Frames", PhD Thesis, University of California (Berkley)

EXAMPLE 6 – Cantilever beam with vertical tip load**DESCRIPTION**

A cantilever beam element is subjected to a vertical tip load. This example tests SeismoStruct's static pushover analysis by applying a pseudo-static load (displacement) at the end of the element and by using the load control strategy for the incrementation of the loading factor.

The resulting response in terms of applied load vs. lateral displacement, obtained with the FE analysis program SeismoStruct, is compared with the reference solution. The model is in the X-Z plane.

GEOMETRY AND PROPERTIES**Material Properties**

$E = 20 \text{ MPa}$

$G = 27190.84 \text{ MPa}$

Geometrical Properties

$D = 355.46 \text{ mm}$

$d = 347.86 \text{ mm}$

Section Properties

$A = 4198.1 \text{ mm}^2$

$I (\text{axis } 2) = 64902751 \text{ mm}^4$

MODELLING AND LOADING

Two different models have been created:

- Model A: a circular hollow section is adopted; the element is modelled with four elastic frame elements (elfrm);
- Model B: a circular hollow section is adopted; the element is modelled with four displacement-based inelastic frame elements (infrmDB) with 100 section fibres.

In each model, a pseudo-static load (force), that is incrementally varied, is defined at the tip in the negative Z direction with a magnitude of 2000.

The total response is completely attained with the load control phase.

ANALYSIS TYPE

Static pushover analysis (Load Control)

RESULTS COMPARISON

The comparison between the reference solution and analysis results, in terms of force vs. displacement at the tip, is shown in the following figures:

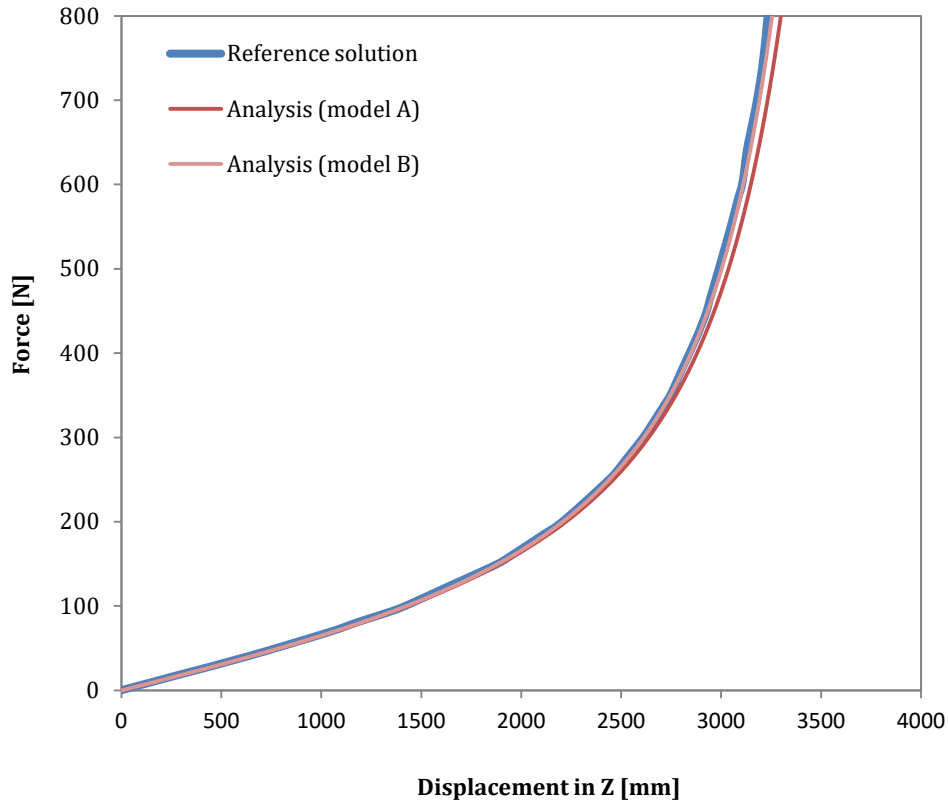


Figure 3.7. Reference solution vs. Analysis results (force-displacement)

COMPUTER FILES

- Ch3-06_A.spf
- Ch3-06_B.spf

REFERENCES

- Frish-Fay R. [1962] "*Flexible bars*", Butterworths, London
- Chan S. L. [1988] "*Geometric and material non-linear analysis of beam-columns and frames using the minimum residual displacement method*", International Journal for Numerical Methods in Engineering, 26, 2657-2669
- Souza [2000] "*Force-based Finite Element for Large Displacement Inelastic Analysis of Frames*", PhD Thesis, University of California (Berkley)

Chapter 4 COMPARISONS WITH EXPERIMENTAL RESULTS

TYPICAL EXISTING NON-SEISMICALLY-DESIGNED RC EUROPEAN BUILDING

EXAMPLE 1 – Multi-storey, 2D frame (ICONS frame - bare)

DESCRIPTION

This example describes the modelling of a full-scale, four-storey, 2D bare frame, which was designed essentially for gravity loads and a nominal lateral load of 8% of its weight. The reinforcement details attempted to reproduce the construction practices used in southern European countries in the 1950's and 1960's. The frame was tested at the ELSA laboratory (Joint Research Centre, Ispra) under two subsequent pseudo-dynamic loadings, first using the Acc-475 input motion and then the Acc-975 input motion.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is defined in the X-Z plane.

NOTE: Further information about the ICONS frame and the tests conducted in ELSA, can be found in Pinto et al. (1999), Carvalho et al. (1999), Pinho and Elnashai (2000) and Varum (2003).



Figure 4.1. ICONS frame tested at the ELSA laboratory of Ispra

GEOMETRY AND PROPERTIES

The model consists of a three-bay RC frame. The dimensions are indicated in the figures below:



[1999])

Structural Geometry

The geometrical details of the structural elements, columns and beams, are given in the following tables:

Table 4.1. Geometrical details of columns

Storey	Col1	Col2	Col3	Col4
3-4	0.4x0.2 (6 ϕ 12)	0.25x0.5 (4 ϕ 16 + 2 ϕ 12)	0.4x0.2 (6 ϕ 12)	0.3x0.2 (6 ϕ 12)
1-2	0.4x0.2 (6 ϕ 12)	0.25x0.6 (8 ϕ 16 + 2 ϕ 12)	0.4x0.2 (8 ϕ 12)	0.3x0.2 (6 ϕ 12)

Table 4.2. Geometrical details of beams

Floor	Beam (1 st and 2 nd bay)	Beam (3 rd bay)
R	0.5x0.25x1.05x0.15*	0.5x0.25x0.65x0.15*
4		
3		
2		

* beam height x beam width x slab effective width x slab thickness

* beam height x beam width x slab effective width x slab thickness

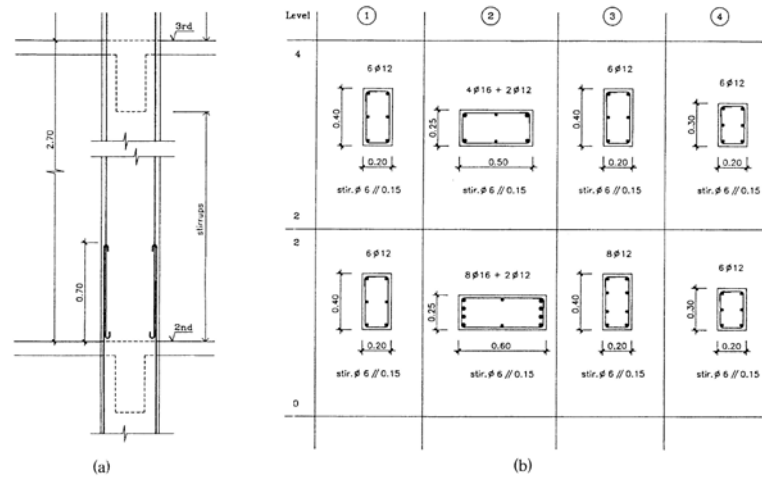


Figure 4.3. Four-storey, three-bay RC frame geometry (reinforcement details of the columns, after Carvalho et al. [1999])

Material Properties of the frame

The Mander et al. concrete model is employed for defining the concrete material. The characteristic parameters are listed below:

- $f_c = 16300 \text{ kPa}$; $f_t = 1900 \text{ kPa}$; $\epsilon_c = 0.002 \text{ m/m}$.

The Menegotto-Pinto steel model is employed for defining the steel material with the following properties:

- $E_s = 2.00\text{E}+008 \text{ kPa}$; $f_y = 343000 \text{ kPa}$; $\mu = 0.0024$

MODELLING AND LOADING

Two different models have been created:

- **Model A:** Both columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200.
- **Model B:** Both columns and beams are modelled through 3D displacement-based inelastic frame elements (infrmDB). The number of fibres used in section equilibrium computations is set to 200.

The masses, proportional to the tributary areas, are applied as (i) lumped (to each beam-column joints) and as (ii) distributed along beams (using the “section additional mass” feature at the section level). The values are summarized in the table below:

Table 4.3. Lumped and distributed masses applied to the frame (in ton and ton/m)

Floor	Col1	Col2	Col3	Col4	Distributed Mass
Roof	4.5	7.8	6.1	2.9	1.295
2-3-4	5.7	9	7.4	4.1	1.539

All the base nodes are fully restrained.

In order to run a dynamic time-history analysis, a time-history curve, constituted by two artificial records (Acc475 and Acc975) in series and separated by 35 s interval with no acceleration, is loaded in the “Time-history Curves” dialog box.

The time step for the dynamic analysis is selected as 0.005 s and 0.01 s, respectively (coincident with the input record sampling time step).

The two input motions are given in the figures below:

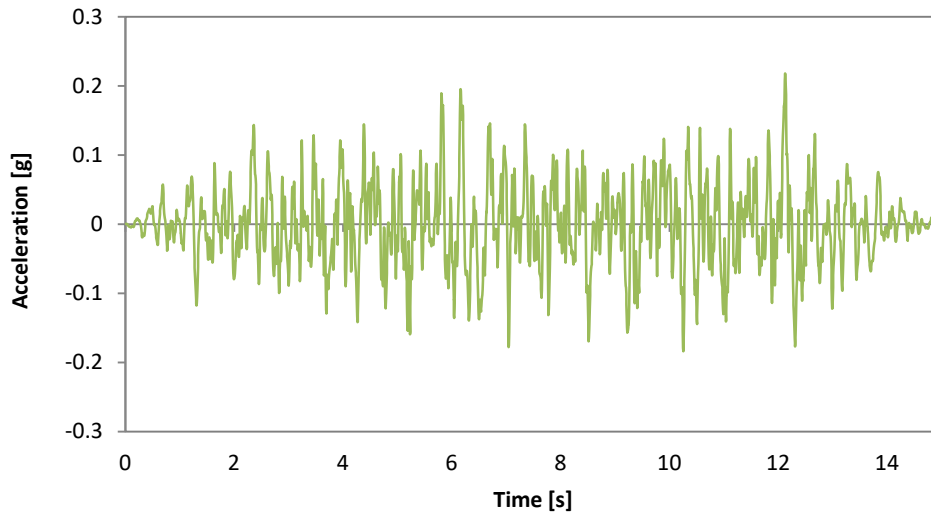


Figure 4.4. Artificial acceleration time history for 475 year return period (Acc-475)

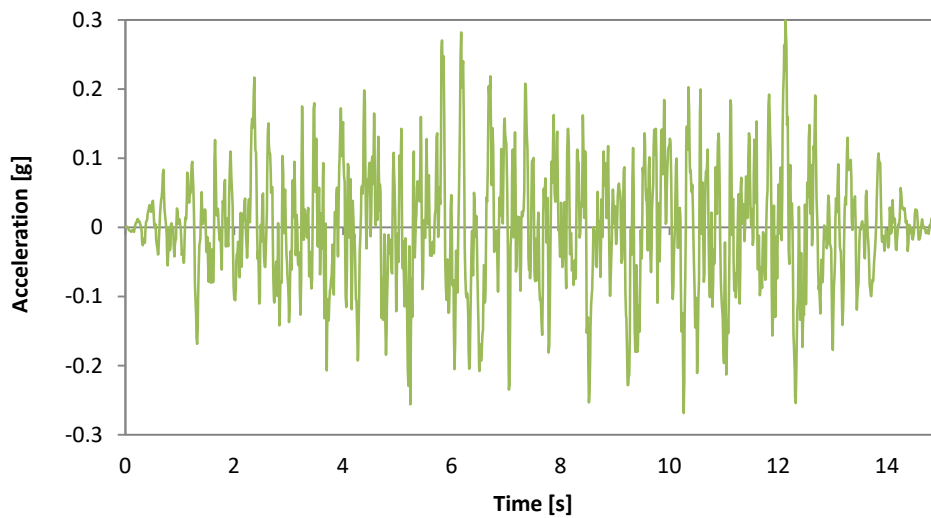


Figure 4.5. Artificial acceleration time history for 975 year return period (Acc-975)

The vertical loads are automatically computed by the program through the option loads are derived from masses, based on the g value, but ONLY in the gravity direction (see Project Settings).

A sketch of the FE model is presented in the following figure:

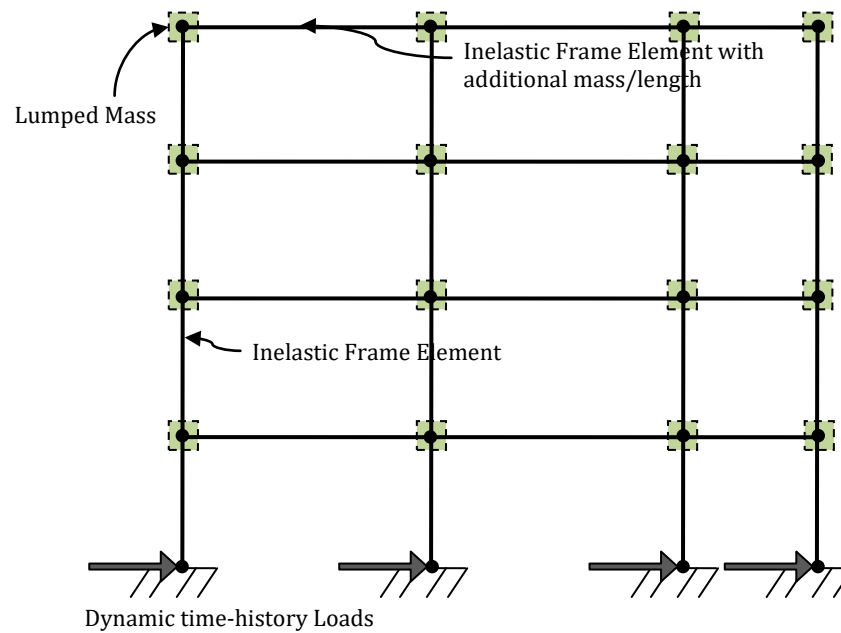


Figure 4.6. FE model sketch of ICONS frame

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

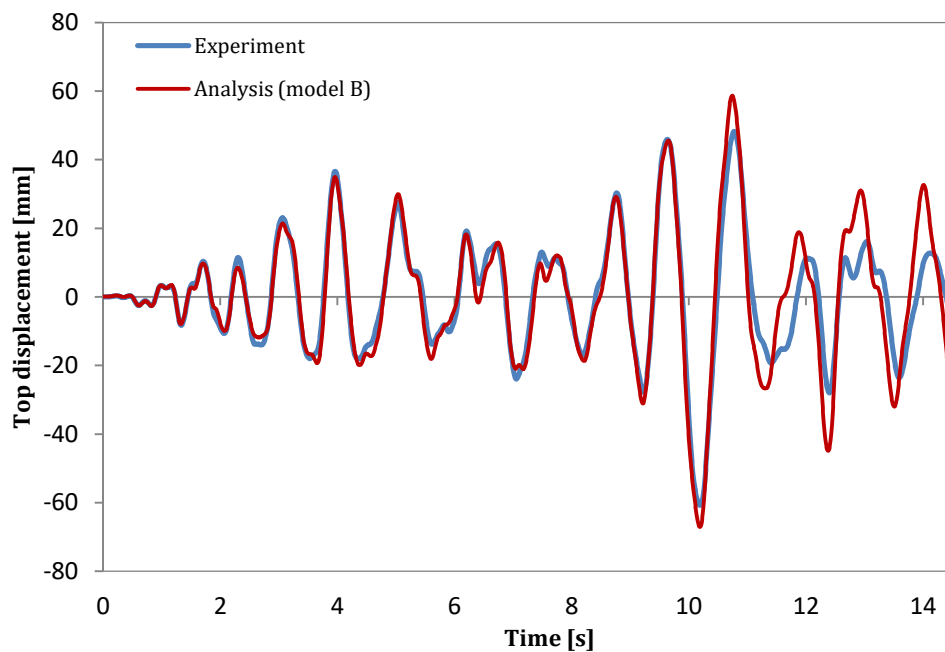


Figure 4.7. Experimental vs. Analytical results – top displacement-time (475yrp)

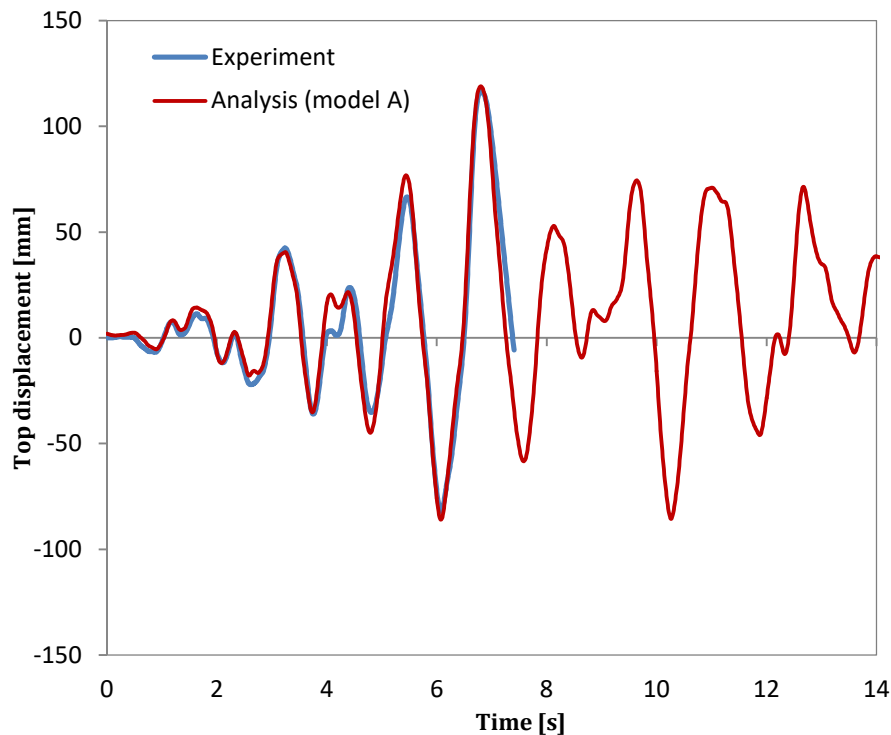


Figure 4.8. Experimental vs. Analytical results – top displacement-time (975yrp)

COMPUTER FILE

- Ch4-RC-01_A.spf
- Ch4-RC-01_B.spf

REFERENCES

- Pinto A., Verzeletti G., Molina F.J., Varum H., Pinho R., Coelho E. [1999]. “*Pseudo-Dynamic Tests on Non-Seismic Resisting RC Frames (Bare and Selective Retrofit Frames)*”. EUR Report, Joint Research Centre, Ispra, Italy.
- Carvalho E.C., Coelho E., Campos-Costa A. [1999] “*Preparation of the Full-Scale Tests on Reinforced Concrete Frames. Characteristics of the Test Specimens, Materials and Testing Conditions*”, ICONS Report, Innovative Seismic Design Concepts for New and Existing Structures, European TMR Network, LNEC.
- Pinho, R. and Elnashai, A.S. [2000]. “*Dynamic collapse testing of a full-scale four storey RC frame*”, ISET Journal of Earthquake Technology, Paper No. 406; 37(4), pp. 143-164.
- Varum, H. [2003]. “*Seismic Assessment, Strengthening and Repair of Existing Buildings*”, PhD Thesis, Department of Civil Engineering, University of Aveiro

EXAMPLE 2 – Seven storey, full-scale, RC shear wall building**DESCRIPTION**

This example describes the modelling of a prototype building (seven-storey, full-scale RC shear wall frame) tested on the NEES Large High-Performance Outdoor Shake Table at UCSD's Englekirk Structural Engineering under dynamic conditions (Panagiotou et al. 2006), by applying four subsequent uniaxial ground motions. The structure was designed with the displacement-based capacity approach for a site in Los Angeles: hence, the design lateral forces are smaller than those currently specified in U.S. building codes for regions of high seismic risk.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is defined in the X-Y-Z space.

NOTE: Further information about the test specimen can be found in Panagiotou et al. (2006).

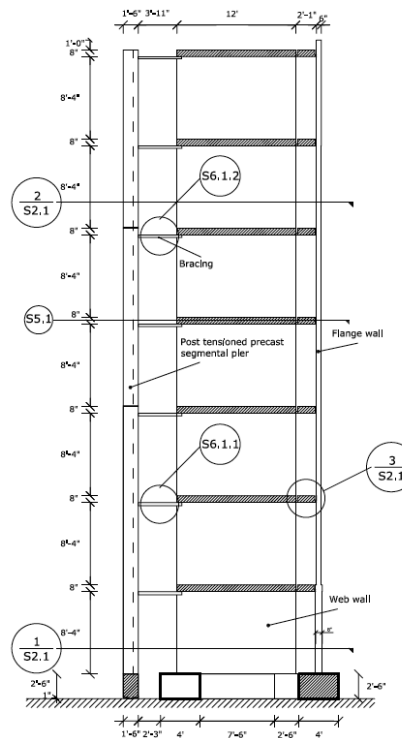


Figure 4.9. Seven-storey, full-scale RC shear wall building tested at the NEES Large High-Performance Outdoor Shake Table at UCSD's Englekirk Structural Engineering Center [Martinelli P. and Filippou F.C., 2009]

GEOMETRY AND PROPERTIES

The frame consists of (i) a cantilever web wall, (ii) a flange wall, (iii) a precast segmental pier and (iv) gravity columns. At each floor, the slab is simply supported by the wall and the columns. The plan dimensions are indicated in the figure below:

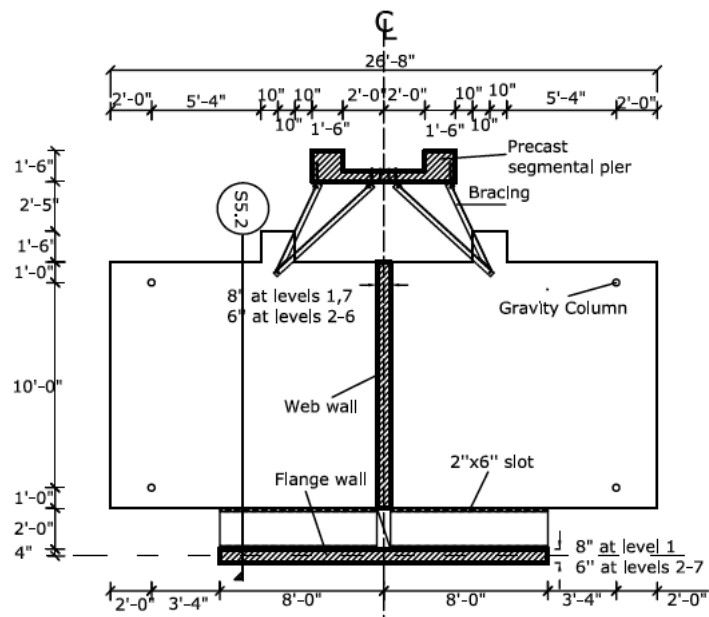


Figure 4.10. Frame geometry (floor plan view) [Martinelli P. and Filippou F.C., 2009]

Structural Geometry

The geometrical details of the structural elements, walls and gravity columns, are given in the following table:

Table 4.4. Geometrical details of walls

Storey	Web Wall	Flange Wall	Gravity Columns Wall
7	144 in x 8 in	192 in x 6 in	d = 4 in; t = 1.3125 in
6	144 in x 6 in	192 in x 6 in	d = 4 in; t = 1.125 in
5	144 in x 6 in	192 in x 6 in	d = 4 in; t = 1.125 in
4	144 in x 6 in	192 in x 6 in	d = 4 in; t = 1.125 in
3	144 in x 6 in	192 in x 6 in	d = 4 in; t = 1.125 in
2	144 in x 6 in	192 in x 6 in	d = 4 in; t = 1.125 in
1	144 in x 8 in	192 in x 6 in	d = 4 in; t = 1.125 in

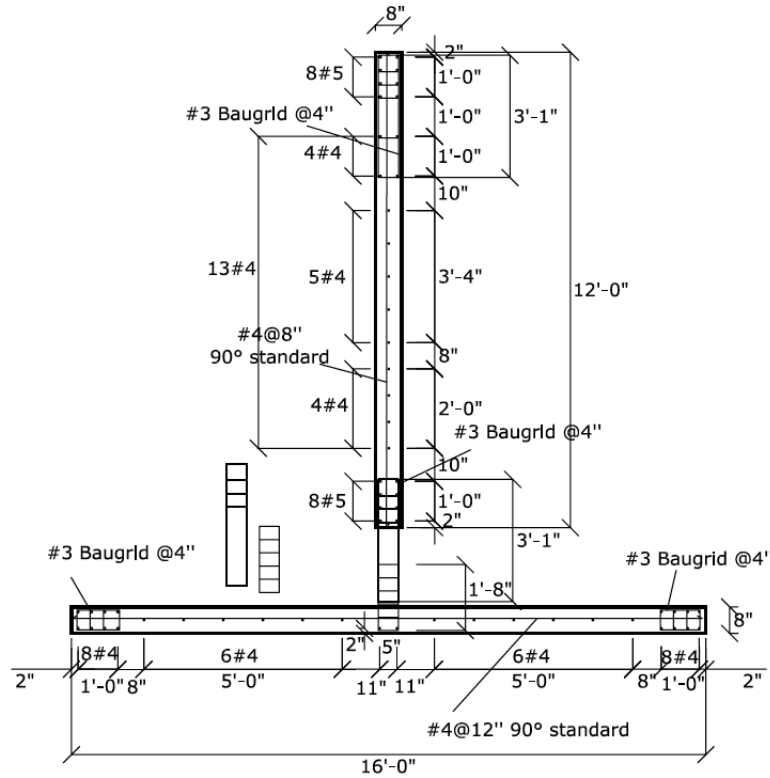


Figure 4.11. Wall reinforcement (floor plan view – level 1) [Martinelli P. and Filippou F.C., 2009]

Material Properties of the frame

The Mander et al. concrete model is employed for defining the concrete material used in the RC rectangular sections for modelling the walls. The characteristic parameters are listed below:

- $f_c = 5426.39$ psi; $f_t = 542.71$ psi; $\epsilon_c = 0.00269$ in/in.

The Menegotto-Pinto steel model is employed for defining the steel material used for the wall reinforcement:

- $E_s = 2.9007E+007$ psi; $f_y = 66497.42$ psi; $\mu = 0.014$.

The uniaxial elastic material model with symmetric behaviour in tension and compression is employed for defining the material used in the RC circular hollow sections for modelling the gravity columns with $E_s = 2.9007E+007$ psi.

MODELLING AND LOADING

Both walls are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200.

The pinned gravity columns are modelled through truss elements (truss) where the number of fibres used in section equilibrium computations is set to 200.

The precast column, since it was designed in order to remain elastic, is modelled through an elastic frame element (elfrm) with the following properties: $EA = 3.9720E+009$ [lb]; EI (axis2) = $3.3035E+012$ [lb-in²]; EI (axis3) = $1.7548E+011$ [lb-in²]; $GJ = 5.7887E+010$ [lb-in²].

The modelling of the slabs is realized through rigid diaphragms (for details, refer to SeismoStruct input file).

The masses are computed starting from the values of weights given by the organizing committee and are assigned in a lumped fashion to each floor node. The applied values are summarized in the table below:

Table 4.5. Lumped masses applied to the frame (in ton)

Floor	Web Wall	Flange Wall	Columns	Precast Pier
7	1.8	2.4	3.7	2.2
1-6	3.6	4.8	3.8	4.3

The base nodes are fully restrained, in order to reproduce the anchorage between the structure and the shaking table.

In order to run a nonlinear dynamic time-history analysis, a time-history curve (the last accelerogram of the experimental series) is loaded in the “Time-history Curves” dialog box. The time step for the dynamic time-history analysis is set to 0.02 s.

The applied ground motions are shown below:

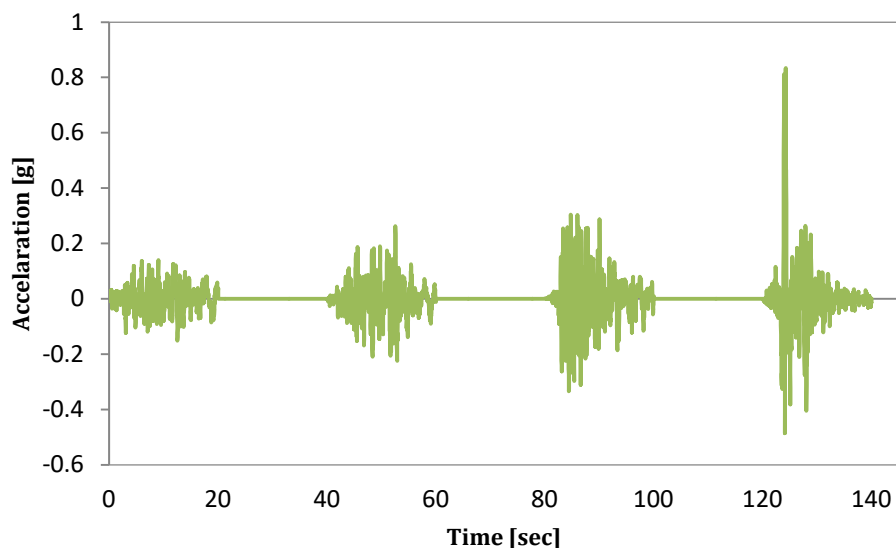


Figure 4.12. Input ground motion

Dynamic time-history loads are applied at the base nodes, in terms of accelerations in the Y direction.

NOTE: A 5% tangent stiffness-proportional damping is applied as global damping in the “Project Settings” menu.

The FE model is presented below:

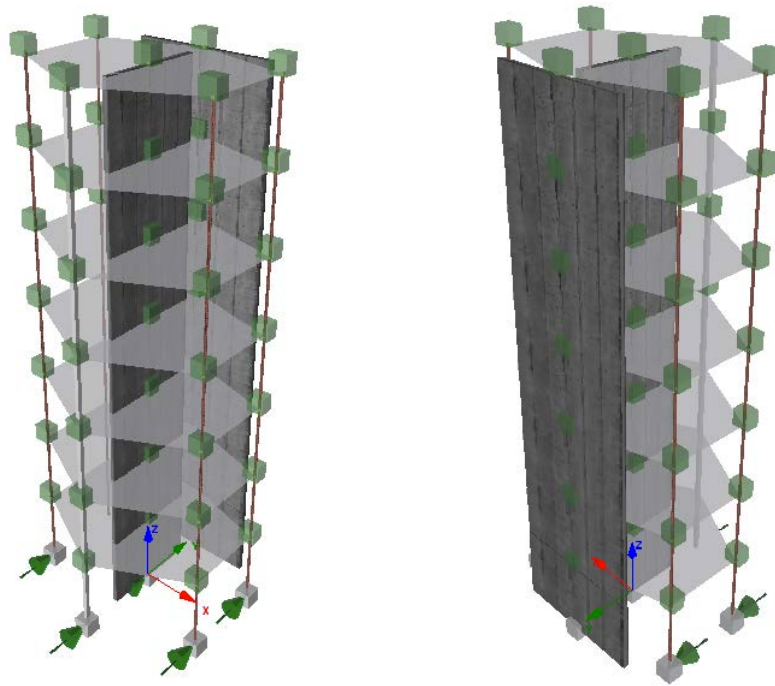


Figure 4.13. FE model

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

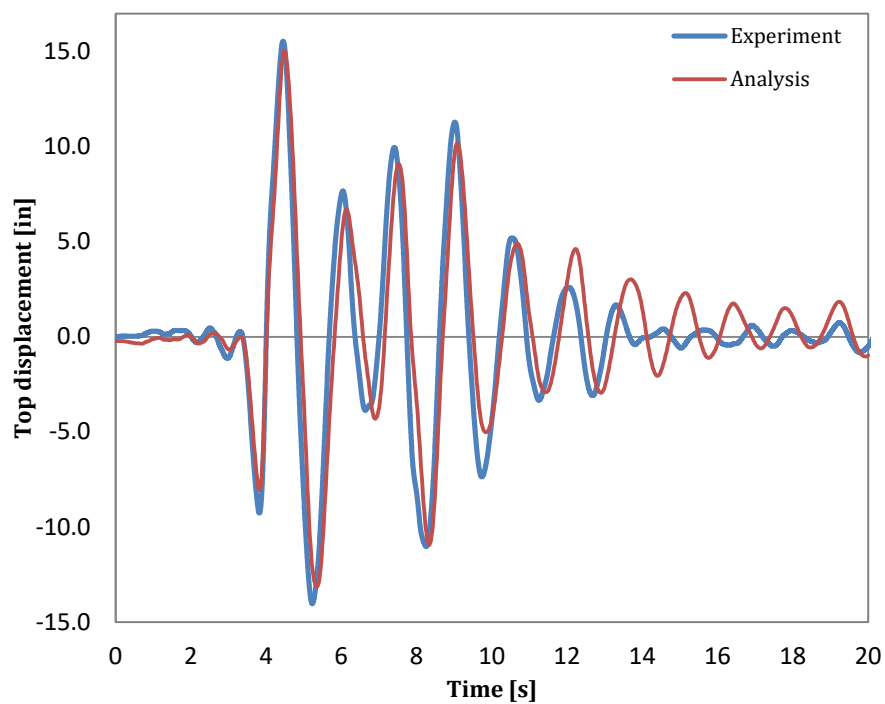


Figure 4.14. Experimental vs. Analytical results – top displacement-time (EQ4)

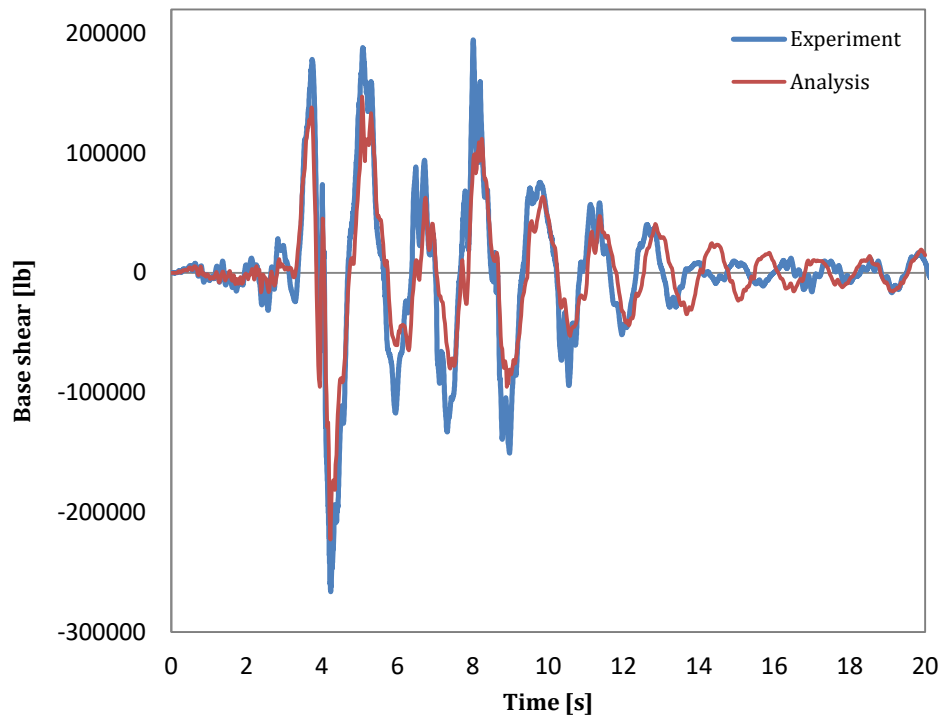


Figure 4.15. Experimental vs. Analytical results – base shear-time (EQ4)

COMPUTER FILE

- Ch4-RC-02.spf

REFERENCES

- Quaglini L. [2006] *“Modellazione di un edificio di sette piani in c.a. sottoposto ad una prova dinamica a scala reale”*, Undergraduate Thesis, Structural Mechanics Department, University of Pavia. (in Italian).
- Panagiotou M., Restrepo J.I. and Englekirk R.E. [2006] *“Experimental seismic response of a full scale reinforced concrete wall building”*, Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 201.
- Martinelli P. and Filippou F.C. [2009] *“Simulation of the Shaking Table Test of a Seven-Storey Shear Wall Building”*, Earthquake Engineering and Structural Dynamics, Vol. 38, No. 5, pp. 587-607.

EXAMPLE 3 – Full-scale, three storey, three-dimensional RC moment frame (SPEAR building)**DESCRIPTION**

This example describes the modelling of a full-scale, three-storey, three-dimensional RC building, which was designed for gravity loads only, according to the 1954-1995 Greek Code. The prototype building was built with the construction practice and materials used in Greece in the early 70's (non-earthquake resistant construction). It is regular in height but highly irregular in plan. The prototype building was tested at the European Laboratory for Structural Assessment (ELSA) of the Joint Research Centre of Ispra (Italy) under pseudo-dynamic conditions using the Herceg-Novi bi-directional accelerogram registered during the Montenegro 1979 earthquake.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is created in the X-Y-Z space.

NOTE: Detailed information about the structural beam member dimensions and reinforcing bars may be found in Fardis and Negro (2006).



Figure 4.16. Full-scale, three-storey prototype building [Fardis & Negro, 2006]

GEOMETRY AND PROPERTIES

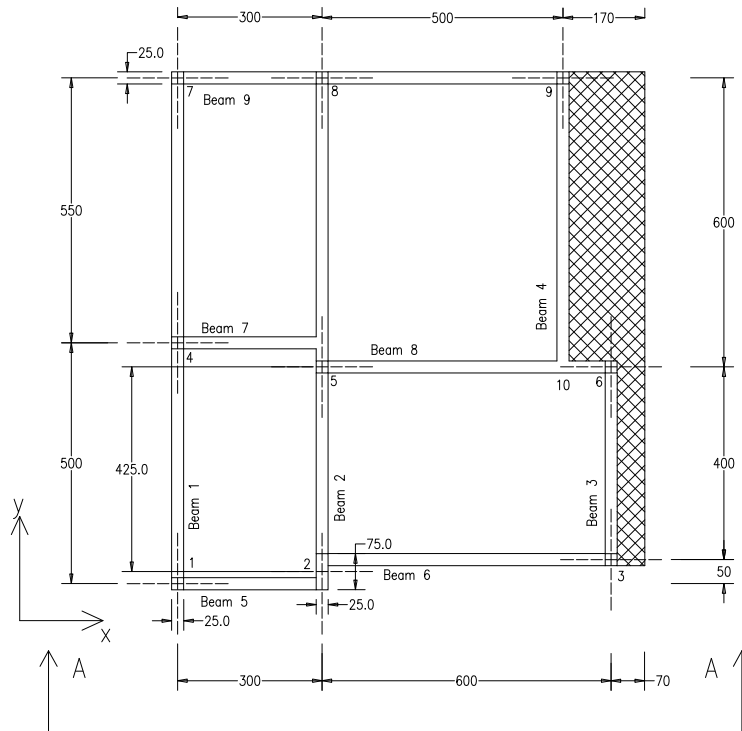


Figure 4.17. Plan view of the full-scale, three-storey prototype building [Lanese et al., 2008]

Structural Geometry

The model consists of seven RC columns (0.25 x 0.25 m) and one RC column (0.25 x 0.75) per floor and RC beams (0.25 x 0.50 m).

Material Properties of the building

The Mander et al. concrete model is employed for defining the concrete material with the following parameters:

- $f_c = 26.5 \text{ MPa}$; $f_t = 0.001 \text{ MPa}$; $\epsilon_c = 0.002 \text{ mm/mm}$

Then, the bilinear model with kinematic strain-hardening is employed for defining the steel material with the following parameters:

- $E_s = 200000 \text{ MPa}$; $f_v = 459 \text{ MPa}$; $\mu = 0.004$

MODELLING AND LOADING

Columns and beams are modelled through 3D displacement-based inelastic frame elements (infrmDB). The number of fibres used in section equilibrium computations is set to 200.

The connection between column C6 and the adjacent beams is modelled as rigid through elastic frame elements (elfrm) with high stiffness values.

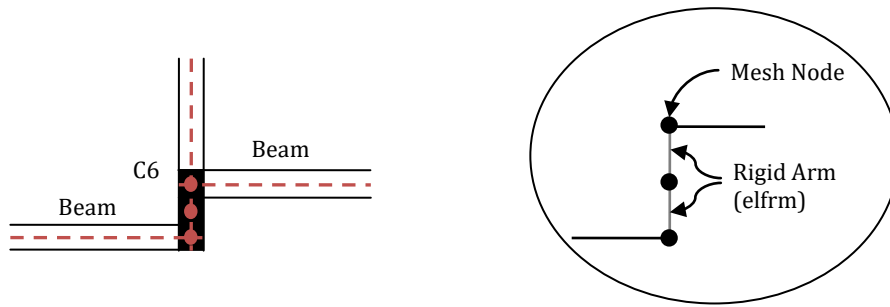


Figure 4.18. Modelling of column C6 and its connection to the adjacent beams

Regarding the applied masses, they are distributed along columns and beams, in order to represent (i) the self-weight of the frame, (ii) a permanent load of 0.5 kN/m^2 and (iii) a variable load of 2 kN/m^2 . For the exact values refer to the SeismoStruct frame model.

All foundation nodes are considered as fully restrained against rotations and translations.

The slabs are modelled by introducing a rigid diaphragm in the X-Y plane for each floor level (see SeismoStruct model).

In order to run a nonlinear dynamic time-history analysis, two time-history curves are loaded in the “Time-history Curves” dialog box (one for X direction, another for Y direction, respectively). The two curves are defined as follows:

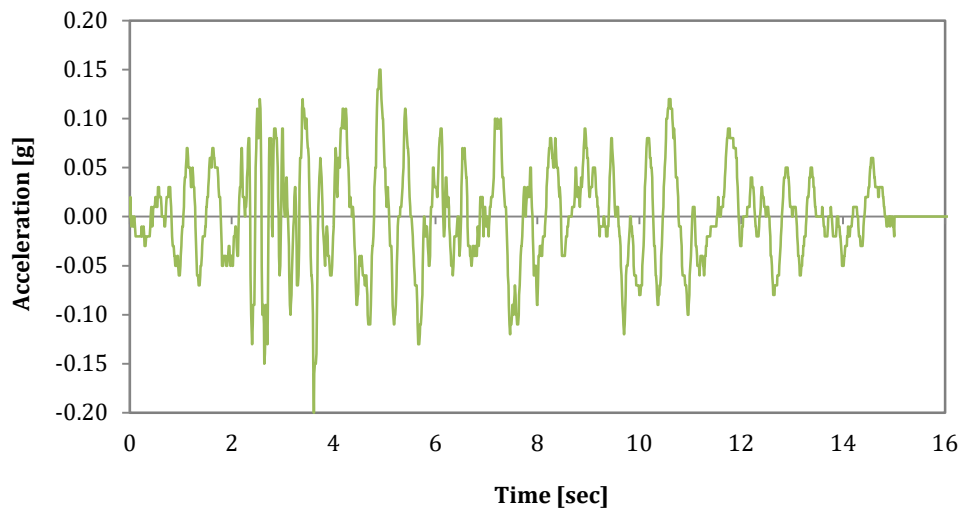


Figure 4.19. H-BCR140 accelerogram in the X direction

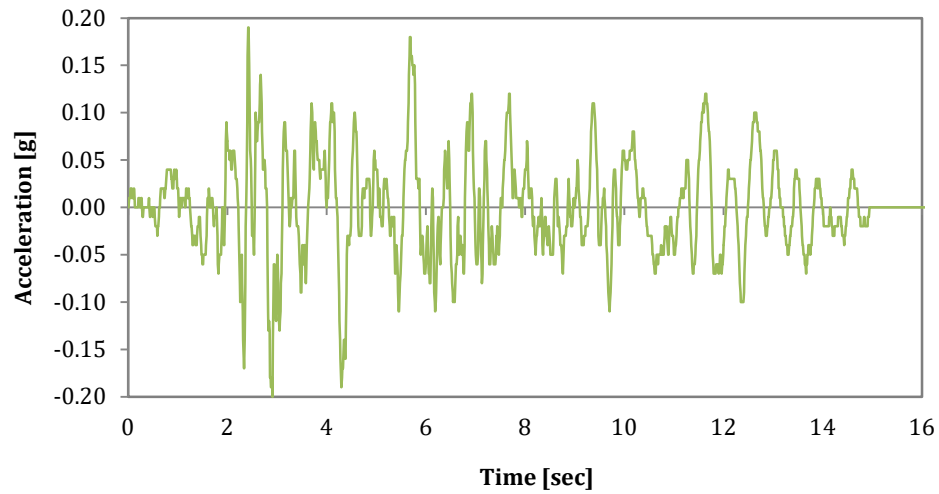


Figure 4.20. H-BCR230 accelerogram in the Y direction (b)

The time step for the dynamic time-history analysis is set to 0.01 s.

Dynamic time-history loads are applied at the base nodes, in terms of accelerations in the X and Y directions.

The FE model of the building is presented below:

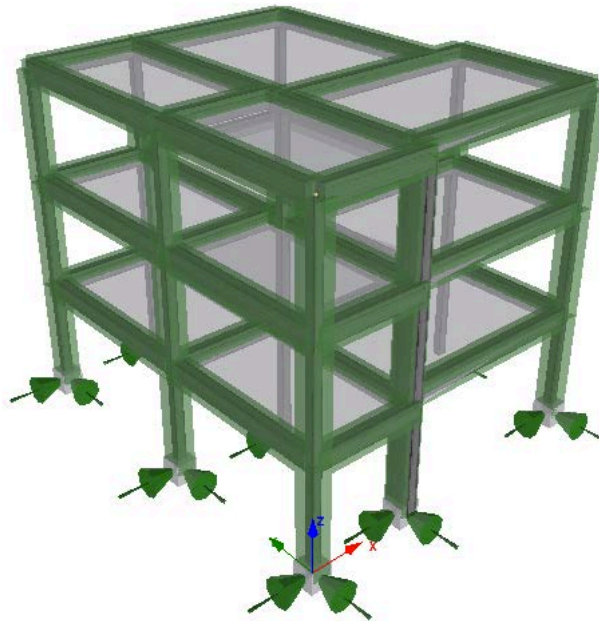


Figure 4.21. FE model of the building

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

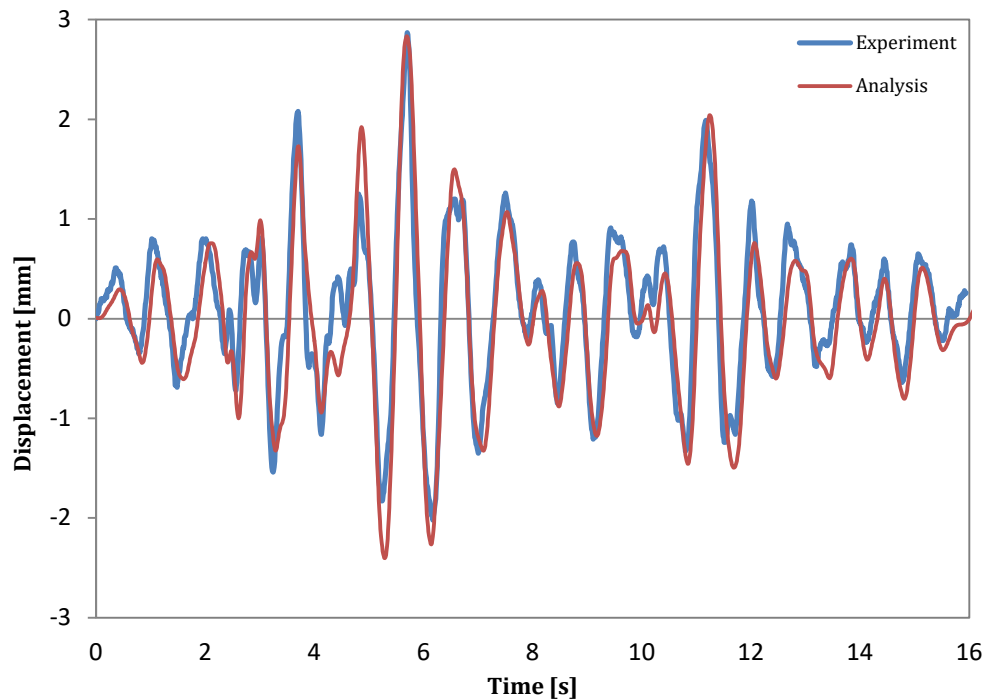


Figure 4.22. Experimental vs. Analytical results – displacement vs. time

COMPUTER FILE

- Ch4-RC-03.spf

REFERENCES

- Fardis, M.N. [2002] *“Design of an Irregular Building for the SPEAR Project – Description of the 3-Storey Structure”*, Research Report, University of Patras, Greece.
- Fardis, M.N. and Negro P. [2006] *“SPEAR – Seismic performance assessment and rehabilitation of existing buildings”*, Proceedings of the International Workshop on the SPEAR Project, Ispra, Italy.
- Lanese et al. (2008)

STEEL FRAMES

EXAMPLE 1 – Three storey, three-dimensional steel moment frame

DESCRIPTION

This example describes the modelling of a three-storey, steel moment frame tested at the University of Kyoto by prof. Nakashima and his team of researchers under a quasi-static cyclic loading. The building was designed following the most common design considerations exercised in Japan for post-Kobe steel moment frames. It is noted that the columns have been extended to the approximate mid-height in the third storey, at which level steel braces were connected horizontally to the columns by high strength bolts through gusset plates to allow for the rotation at the column top.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is created in the X-Z plane.

NOTE: Further details can be found in Matsumiya et al. (2004) and Nakashima et al. (2006).



Figure 4.23. Three-storey, three-dimensional steel moment frame [Nakashima et al., 2006]

GEOMETRY AND PROPERTIES

The test structure plan dimensions are 12 m in the longitudinal direction and 8.5 m in the transversal direction. The total height is 8.5 m (without considering the “steel blocks”).

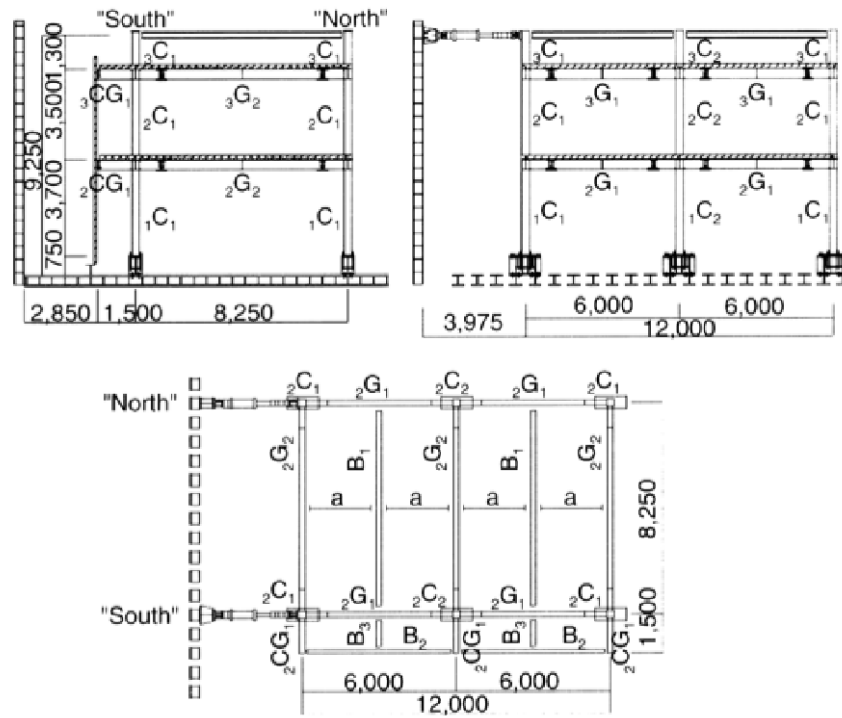


Figure 4.24. Elevation and plan of the test structure (unit: mm) [Nakashima et al., 2006]

Structural Geometry

The test structure consists of two lateral resisting steel frames. They lie along the longitudinal direction and, since they work almost independently, only one of them has been modelled. In particular, the geometrical details of the principal elements (columns and beams) are listed below:

- Column 1, 2 and 3: rectangular hollow section of 0.3 m x 0.3 m, with section thickness of 9 mm, 12 mm and 16 mm, respectively;
- Beam: symmetric I-section with a bottom and top flange width of 0.2 m, a bottom and top flange thickness of 16 mm, a web height of 0.368 m and a web thickness of 9 mm.

Material Properties of the structure

A bilinear steel model is employed for defining the steel materials:

- Steel1: $E_s = 2.0000E+008$ kPa; $f_y = 512000$ kPa; $\mu = 0.01$;
- Steel2: $E_s = 2.0000E+008$ kPa; $f_y = 522000$ kPa; $\mu = 0.01$;
- Steel3: $E_s = 2.0000E+008$ kPa; $f_y = 537000$ kPa; $\mu = 0.01$;
- Steel4: $E_s = 2.0000E+008$ kPa; $f_y = 375000$ kPa; $\mu = 0.01$.

MODELLING AND LOADING

The lateral resisting frame is modelled through steel beam and column elements where a bilinear hysteretic model according with the measured material properties after testing (Nakashima et al., 2006) is adopted.

Beams and columns are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 100.

Panel zones are modelled considering a panel size of 400 mm in depth and 300 mm in width (beam length and column height are shortened by the panel width and depth, respectively).

At the top of the frame, the link beams connecting horizontally the columns are modelled with an elastic frame element (elfrm) with the following properties: $EA = 1.0000E+010$ [kN]; EI (axis2) = 1 [kN-m²]; EI (axis3) = 1 [kN-m²]; $GJ = 1$ [kN-m²].

Rotational springs are inserted at the bottom of the first storey columns to allow for the rotational flexibility of the column base. They are modelled with link elements, in which for F_1 , F_2 , F_3 , M_1 , and M_3 degrees of freedom a linear symmetric response curve is chosen with a value of stiffness k_0 equal to $1.00E+09$, whereas for M_2 degree-of-freedom a trilinear symmetric response curve is chosen.

All the base nodes are fully restrained.

The vertical loads assigned for the frame are applied to each beam-column joint as permanent loads in terms of forces in the Z direction (for details refer to the computer file of the model).

A quasi-static cyclic loading with increasing displacement amplitude is loaded in the “Time-history Curves” dialog box. The used output sampling time interval is set to 0.1 seconds. The cyclic loading is applied at the top lateral node as a static time-history load in terms of displacements in the X direction.

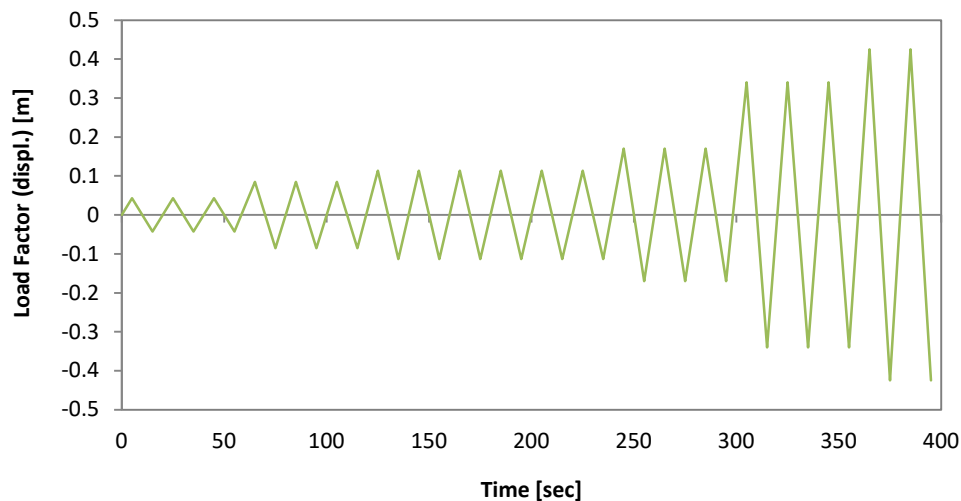


Figure 4.25. Cyclic loading

The FE model of the steel frame is presented below:

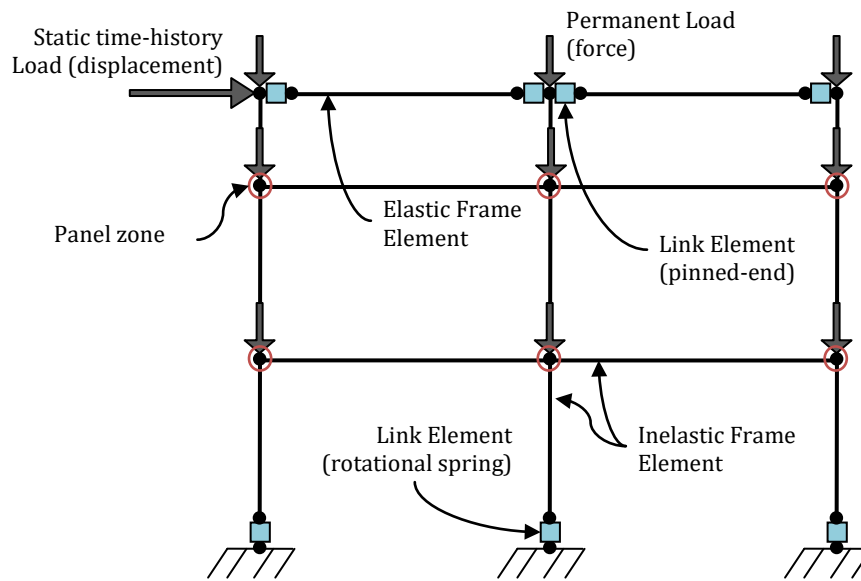


Figure 4.26. FE model

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results, is shown below:

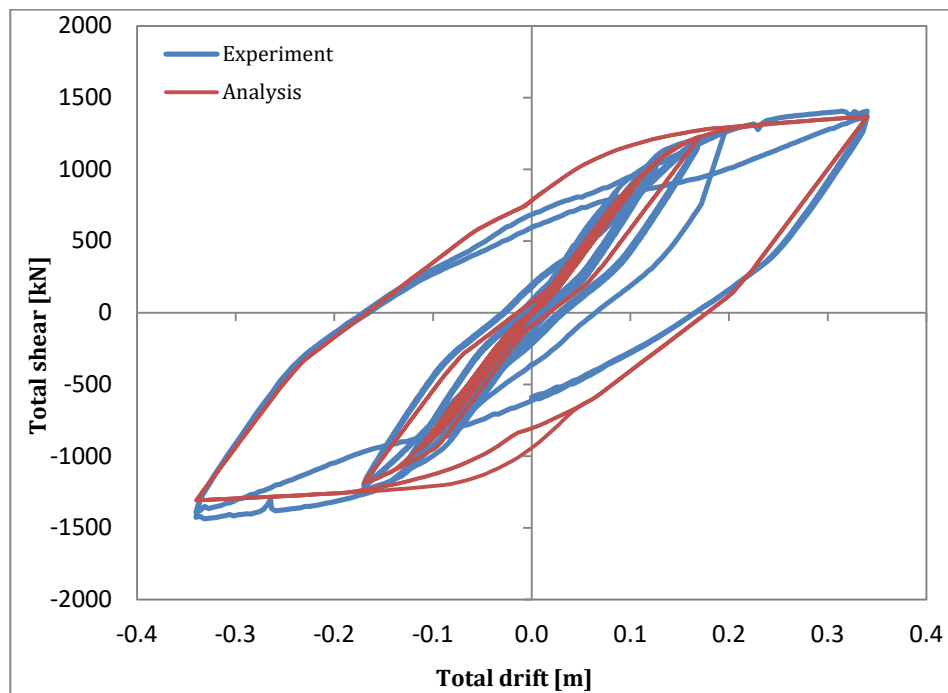


Figure 4.27. Total shear vs. Total drift

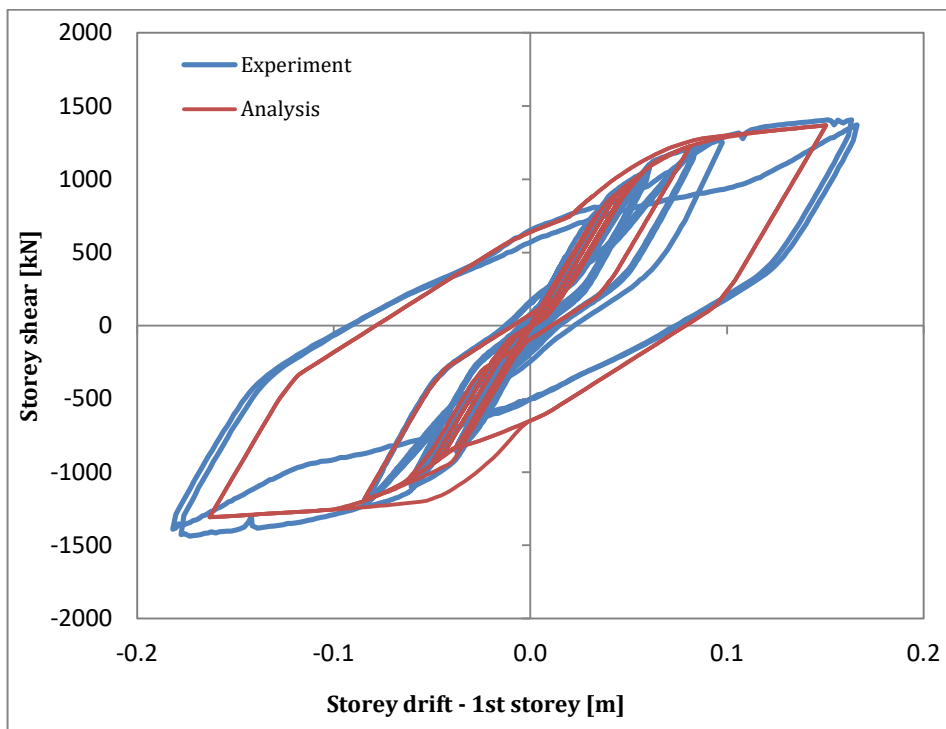


Figure 4.28. Storey shear vs. Storey drift – 1st storey

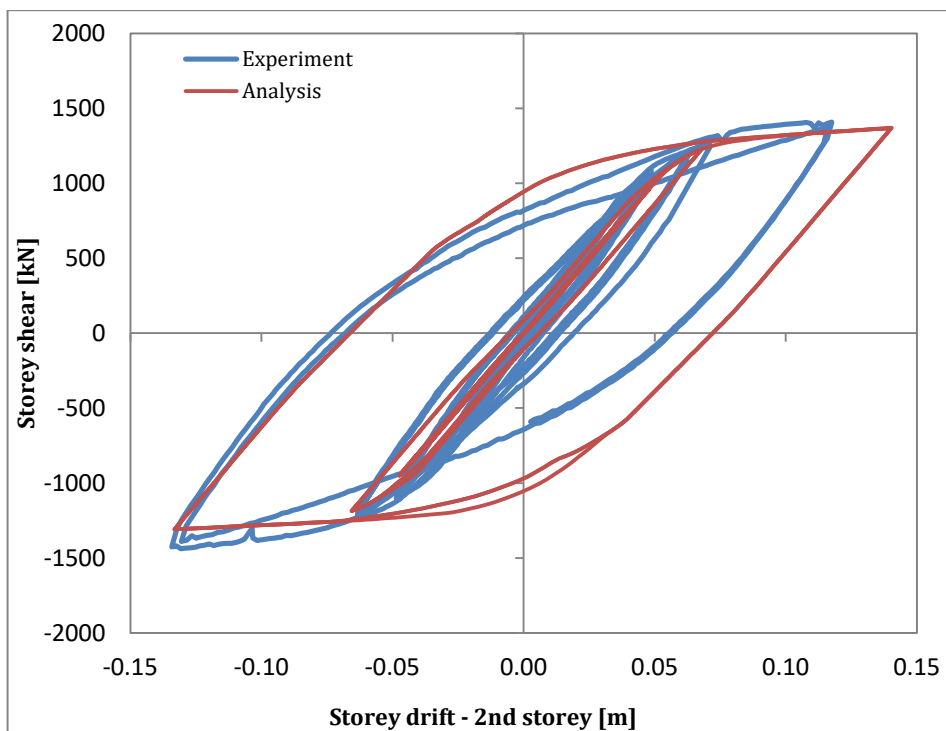


Figure 4.29. Storey shear vs. Storey drift – 2nd storey

COMPUTER FILE

- Ch4-steel-01.spf

REFERENCES

- A.V. Pinto, G. Verzeletti, P. Pegon, G. Magonette, P. Negro, J. and Guedes [1996] *"Pseudo-Dynamic Testing of Large-Scale R/C Bridges"*, ELSA Lab, Report EUR 16378 EN.
- J. Guedes [1997] *"Seismic behaviour of reinforced concrete bridges. Modelling, numerical analysis and experimental assessment"*, PhD Thesis, Univ. of Porto, Porto, Portugal.
- Nakashima M., Matsumiya T, Suita K, Liu D. [2006] *"Test on full-scale three-storey steel moment frame and assessment of ability of numerical simulation to trace cyclic inelastic behaviour"*, Earthquake Engineering and Structural Dynamics, No. 35, pp. 3-19.
- Matsumiya T, Suita K, Chusilp P, Nakashima M. [2004] *"Full-scale test of three-storey steel moment frames for examination of extremely large deformation and collapse behaviour"*, Proceedings of the 13th World Conference on Earthquake Engineering, Vancouver, Paper No. 3471, 1–6 August 2004.

EXAMPLE 2 – Full-scale, four-storey 3D steel frame

DESCRIPTION

This example describes the modelling of a prototype building (full-scale, 3D steel moment resisting frame) tested on the world's largest three-dimensional shaking table located at Miki City, Hyogo Prefecture (Japan) under dynamic conditions, by applying a scaled version of the near-fault motion recorded in Takatori during the 1995 Kobe earthquake. More 3D shaking table tests have been performed consecutively with increasing levels of seismic motion to evaluate the effect of plastic deformation: Takatori scaled to 40% (elastic level), Takatori scaled to 60% (incipient collapse level) and Takatori in full scale (collapse level).

The analytical results, obtained with the FE analysis program SeismoStruct after the test (post-test results), are compared with experimental results. The model is created in the X-Y-Z space.

NOTE: Full details on the structure's geometrical and material characteristics may be found in Pavan (2008).

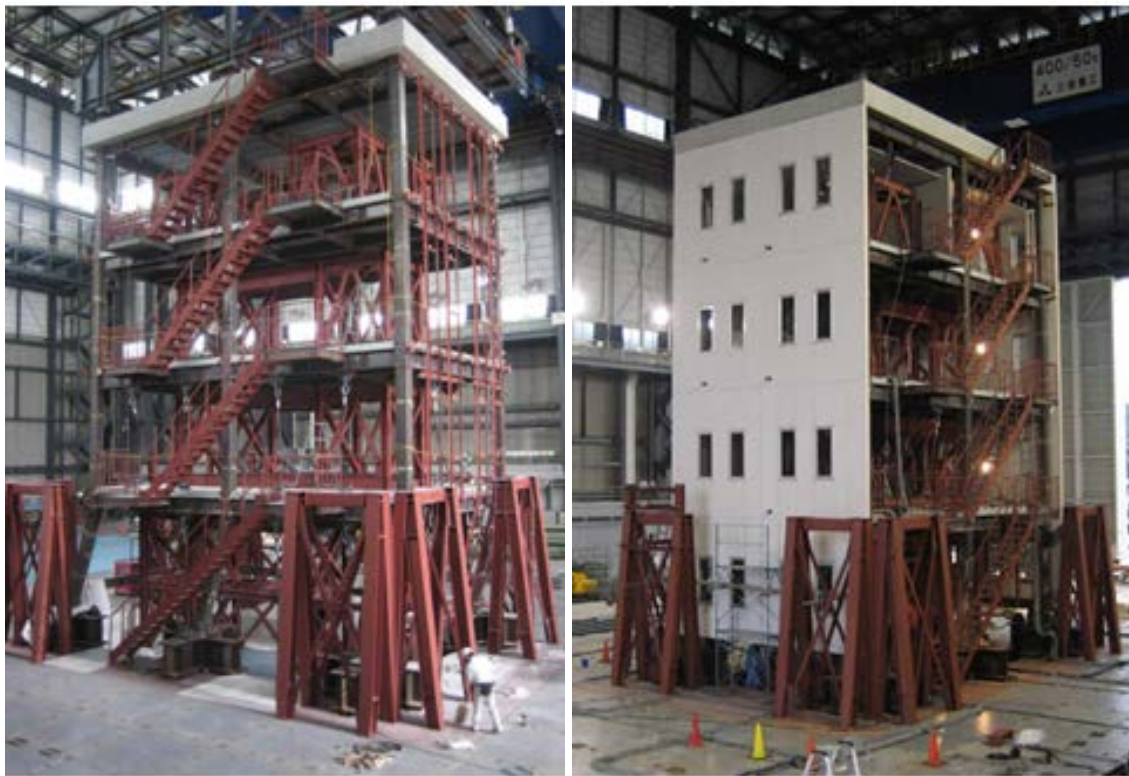


Figure 4.30. Four-storey 3D steel moment resisting frame [NRIESDP, 2007]

GEOMETRY AND PROPERTIES

The model consists of two steel frames composed by two bays 5 m long, along NS-direction, and three steel frames (with one bay 6 m long) in EW-direction. The interstorey height is equal to 3.5 m.

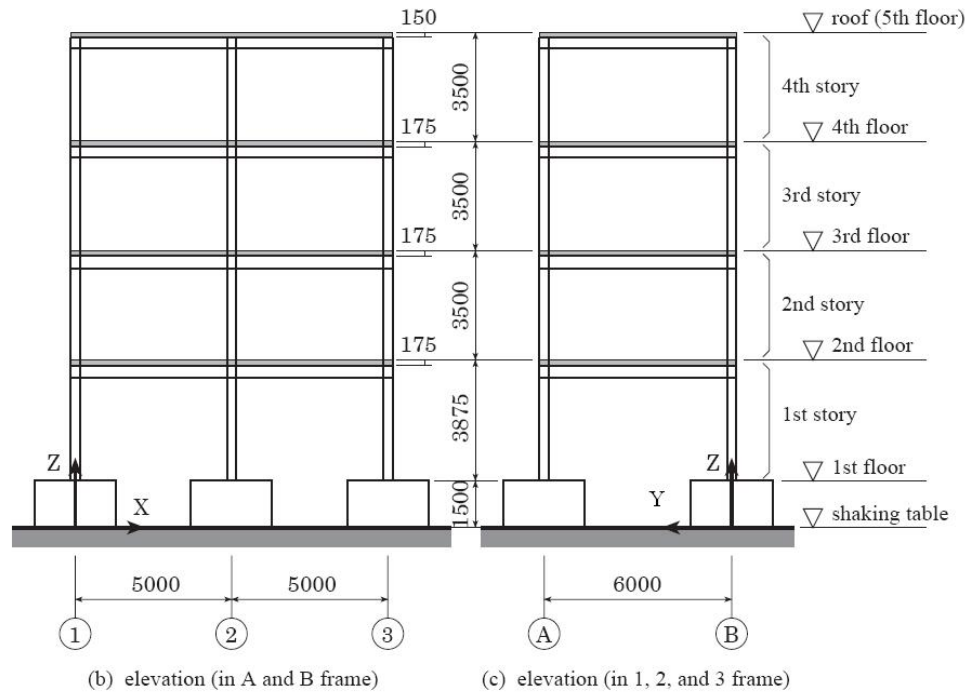


Figure 4.31. Four-storey 3D steel frame geometry (frontal and lateral views) [Pavan, 2008]

Structural Geometry

The geometrical details of the principal structural elements (columns and beams) are given in the table below:

Table 4.6. Geometrical details of beams and columns

Beam				Column	
Floor	G1	G11	G12	Storey	-
R	H- 346x174x6x9	H- 346x174x6x9	H- 346x174x6x9	4	RHS- 300x300x9
4	H- 350x175x7x11	H- 350x175x7x11	H- 340x175x9x14	3	RHS- 300x300x9
3	H- 396x199x7x11	H- 400x200x8x13	H- 400x200x8x13	2	RHS- 300x300x9
2	H- 400x200x8x13	H- 400x200x8x13	H- 390x200x10x16	1	RHS- 300x300x9

H- height x width x web thickness x flange thickness, RHS- height x width x thickness

In order to consider that the slabs consist in composite deck floor (at the second, third and forth level) and reinforced concrete (at the roof), each beam section is modelled as a composite I-section.

Material Properties of the structure

The Mander et al. concrete model is employed for defining the concrete material used in the composite I-sections for modelling the slabs. The characteristic parameters are listed below:

Table 4.7. Characteristic parameters of concrete

PARAMETERS	2 nd Level	3 rd Level	4 th Level	Roof Level
f_c (kPa)	33530	28800	27540	35900
f_t (kPa)	3353	2880	2754	3590
ϵ_c (m/m)	0.00154	0.0012	0.00127	0.002467
γ (kN/m ³)	24	24	24	24

Then, the bilinear model is employed for defining the steel materials. In particular, eight different kind of steel materials are defined:

Table 4.8. Steel bi-linear constitutive model characteristic parameters (1)

PARAMETERS	Column 1 (steel_column1)	Column 2 (steel_column2)	2G1, 2G11, 3G11, 3G12 (steel_beam1)	4G12 (steel_beam2)
E (kPa)	200000000	200000000	203750000	212830000
f_y (kPa)	380000	380000	326000	308600
μ	0.05	0.05	0.12	0.10
γ (kN/m ³)	78	78	78	78

Table 4.9. Steel bi-linear constitutive model characteristic parameters (2)

PARAMETERS	RG1, RG11, RG12 (steel_beam3)	4G1, 4G11 (steel_beam4)	2G12 (steel_beam5)	3G1 (steel_beam6)
E (kPa)	201820000	223480000	174770000	199550000
f_y (kPa)	333000	301700	279100	311100
μ	0.15	0.13	0.20	0.18
γ (kN/m ³)	78	78	78	78

MODELLING AND LOADING

Two different models have been created:

- Model A: Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 100;
- Model B: Columns and beams are modelled through 3D displacement-based inelastic frame elements (infrmDB). The number of fibres used in section equilibrium computations is set to 100.

The masses are computed starting from the values of weights given by the organizing committee and are assigned to each beam section as additional mass. The applied values are summarized in the table below:

Table 4.10. Additional mass distributed on girders

Level	Element	Additional Mass (ton/m)
Roof	RG1	1.064
	RG11	1.203
	RG12	1.954
4	4G1	0.668
	4G11	0.779
	4G12	1.559
3	3G1	0.632
	3G11	0.737
	3G12	1.385
2	2G1	0.594
	2G11	0.692
	2G12	1.385

The base nodes are fully restrained, in order to reproduce the anchorage between the structure and the shaking table.

The slabs are modelled by introducing a rigid diaphragm in the X-Y plane for each floor level (see SeismoStruct model).

In order to run a nonlinear dynamic time-history analysis, three time-history curves are loaded in the “Time-history Curves” dialog box, one for each direction (NS component, EW component and vertical component). Each curve includes 10 s interval with no acceleration (needed to damp out the structure motion after each earthquake run). The three curves are defined as follows:

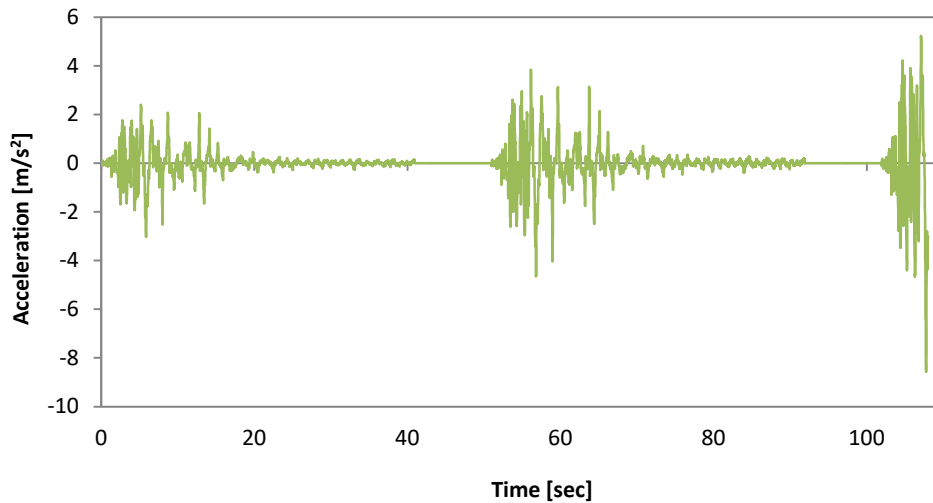


Figure 4.32. Post-test shaking table acceleration time-history (NS component)

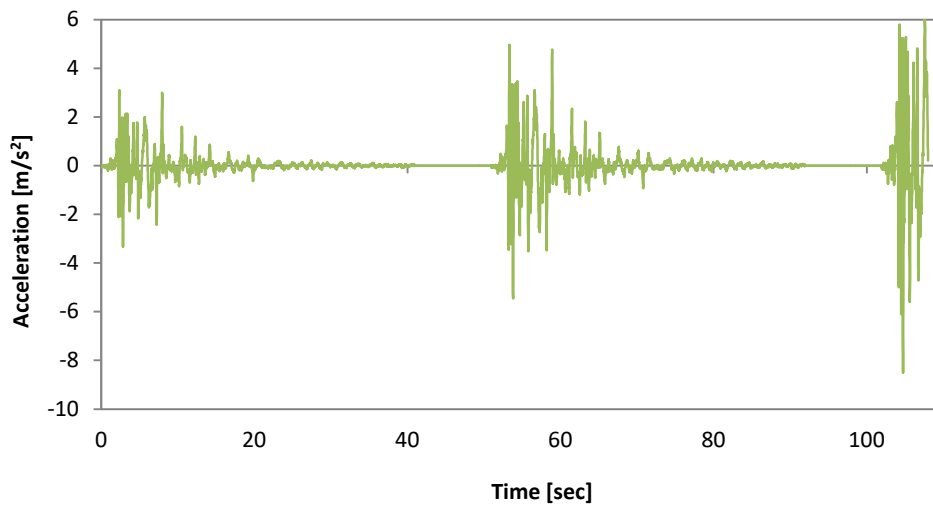


Figure 4.33. Post-test shaking table acceleration time-history (EW component)

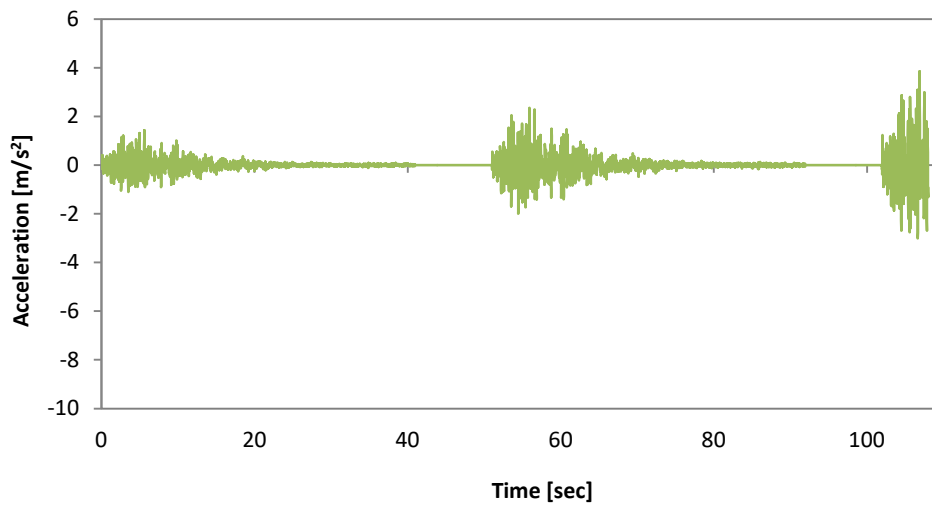


Figure 4.34. Post-test shaking table acceleration time-histories (vertical component)

The time step for the dynamic time-history analysis is set to 0.01 s, with the exception of the last and computationally more demanding 8 seconds, where it was set to 0.001s.

Dynamic time-history loads are applied at the base nodes, in terms of accelerations in the X, Y and Z directions, respectively.

In the “Project Settings” panel 0.5% tangent-stiffness proportional damping is selected.

The FE analysis model of the steel building is presented below:

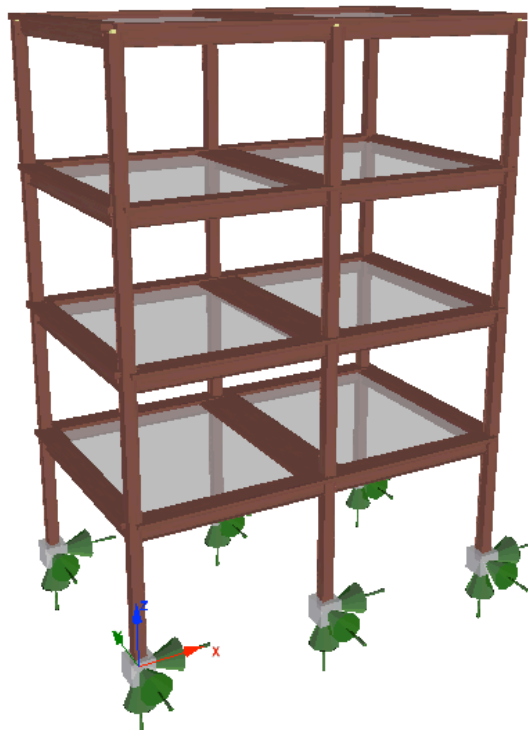


Figure 4.35. Four-storey 3D steel frame geometry (3D model)

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results (post-test results), in terms of maximum relative displacements and maximum storey shear for the second time-history curve, is shown below:

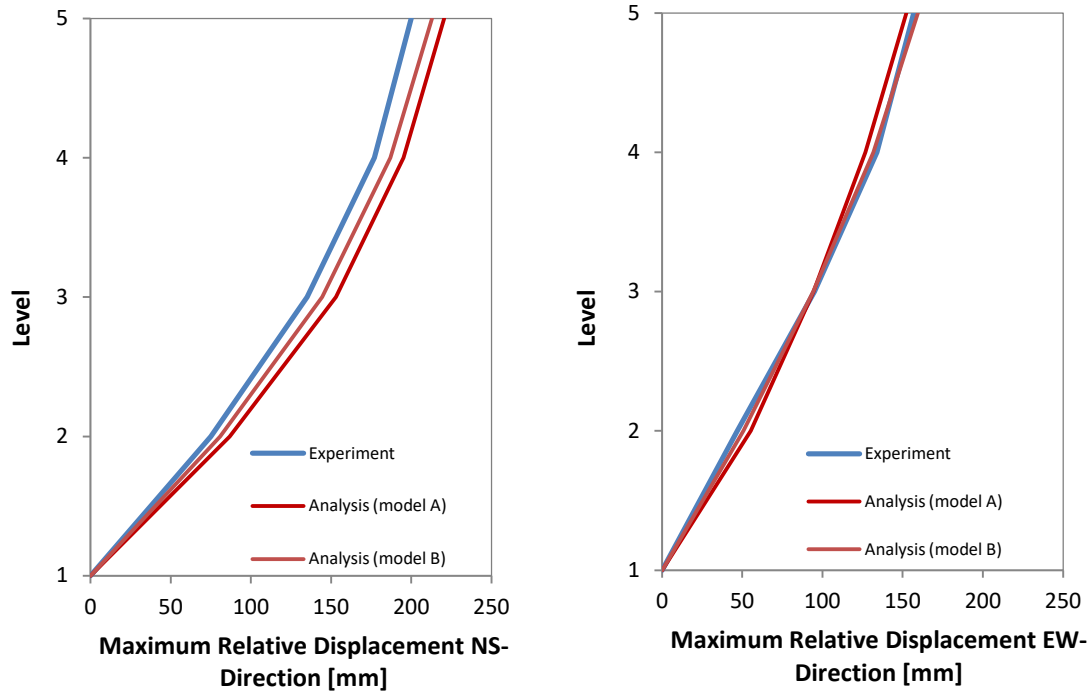


Figure 4.36. Experimental vs. Analytical results – maximum relative displacement-floor level

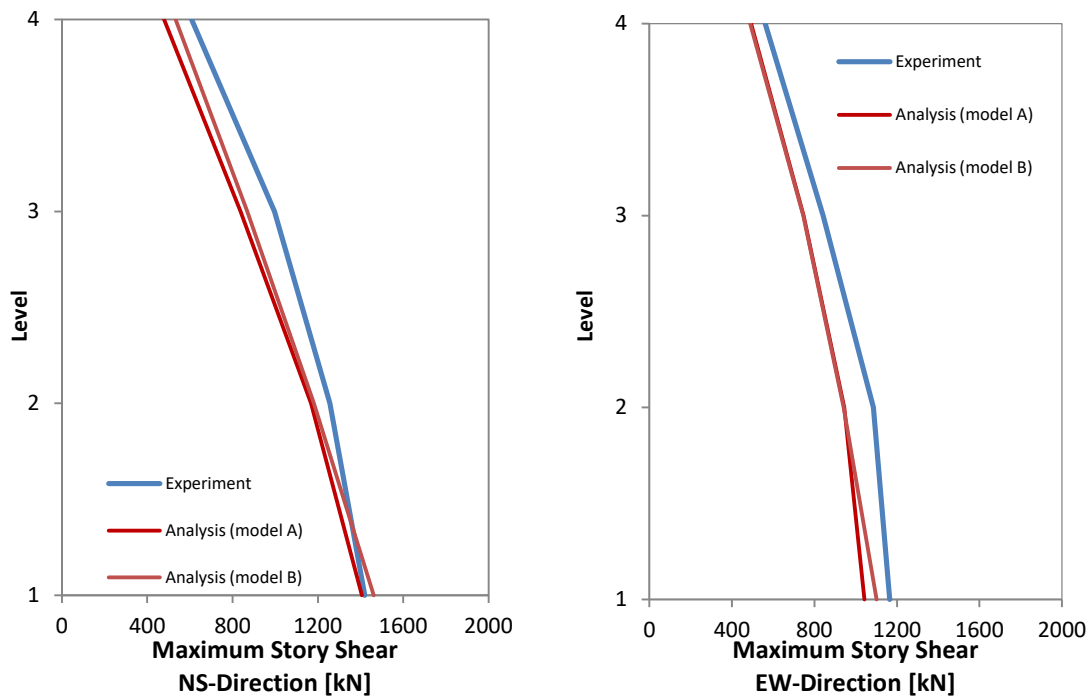


Figure 4.37. Experimental vs. Analytical results – maximum storey shear-floor level

COMPUTER FILE

- Ch4-steel-02_A.spf
- Ch4-steel-02_B.spf

REFERENCES

- Pavan A. [2008] *"Blind Prediction of a Full-Scale 3D Steel Frame Tested under Dynamic Conditions"*, MSc Dissertation, ROSE School, Pavia, Italy.

INFILLED FRAMES (MASONRY INFILLS)

EXAMPLE 1 – One storey, single bay infilled frame (Crisafulli)

DESCRIPTION

This example describes the modelling of one storey, single bay infilled frame constructed to a reduced scale of $\frac{3}{4}$ and tested by Crisafulli (1997) under pseudo-static cyclic loadings. Actuators have been used to apply lateral and vertical forces, simulating thus the gravity loads and overturning moment corresponding to a typical two-storey building with infill panels.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with experimental results. The model is defined in the X-Z plane.

NOTE: Detailed information about the material properties and the test arrangements can be found in Crisafulli (1997).

GEOMETRY AND PROPERTIES

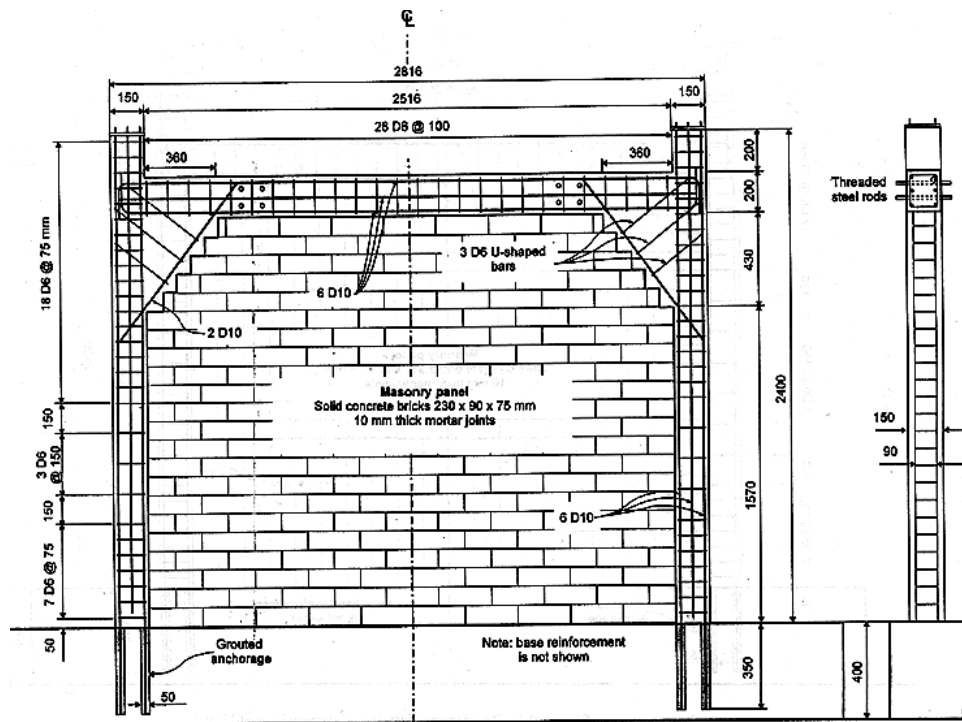


Figure 4.38. Infilled frame tested by Crisafulli (1997)

Geometry

The model consists of two RC columns (0.15×0.15 m), a RC beam (0.2×0.15 m) and an infill panel with the properties listed in the paragraph “Modelling and Loading”.

Material Properties of the frame

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters are listed below:

- $f_c = 31200$ kPa; $f_t = 0$ kPa; $\epsilon_c = 0.002$ m/m

The Menegotto-Pinto steel model is employed for defining the steel material with the following properties:

- $E_s = 2.0700\text{E}+008$ kPa; $f_y = 323000$ kPa; $\mu = 0.004$

Properties of the infill panel

See paragraph *Modelling and Loading*.

MODELLING AND LOADING

Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200.

The infill panel is modelled through a four-node masonry panel element (inelastic infill panel element). This element is characterized by the following parameters:

- Strut curve parameters (inf strut response curve): initial Young modulus $E_m = 1100000$, compressive strength $f_m = 950$, tensile strength $f_t = 0.01$, strain at maximum stress $\varepsilon_m = 0.001$, ultimate strain $\varepsilon_u = 0.02$, closing strain $\varepsilon_{cl} = 0.002$, strut area reduction strain $\varepsilon_1 = 0.0004$, residual strut area strain $\varepsilon_2 = 0.0008$, starting unloading stiffness factor $\gamma_u = 2.0$, strain reloading factor $\alpha_r = 1.5$, strain inflection factor $\alpha_{ch} = 0.6$, complete unloading strain factor $\beta_a = 2.0$, stress inflection factor $\beta_{ch} = 0.7$, zero stress stiffness factor $\gamma_{pu} = 1.0$, reloading stiffness factor $\gamma_{pr} = 1.1$, plastic unloading stiffness factor $e_{x1} = 1.5$, repeated cycle strain factor $e_{x2} = 1.0$
- Shear curve parameters (inf shear response curve): shear bond strength = 50, friction coefficient = 0.2, maximum shear resistance = 200, reduction shear factor = 1.6
- Panel thickness (t): 0.09 m
- Out-of-plane failure drift: 5% of ver. panel side
- Strut Area 1 (A1): 0.1 m²
- Strut Area 2 (A2): 70% of A1
- Equivalent contact length (hz): 7% of vert. panel side
- Horizontal and vertical offsets (xo and yo): 5% of horiz. panel side and 5% of vert. panel side
- Proportion of stiffness assigned to shear: 50%
- Specific weight: 10 kN/m³

All the base nodes are fully restrained.

In order to run a nonlinear static time-history analysis, a time-history curve is loaded in the “Time-history Curves” dialog box. It is defined as follows:

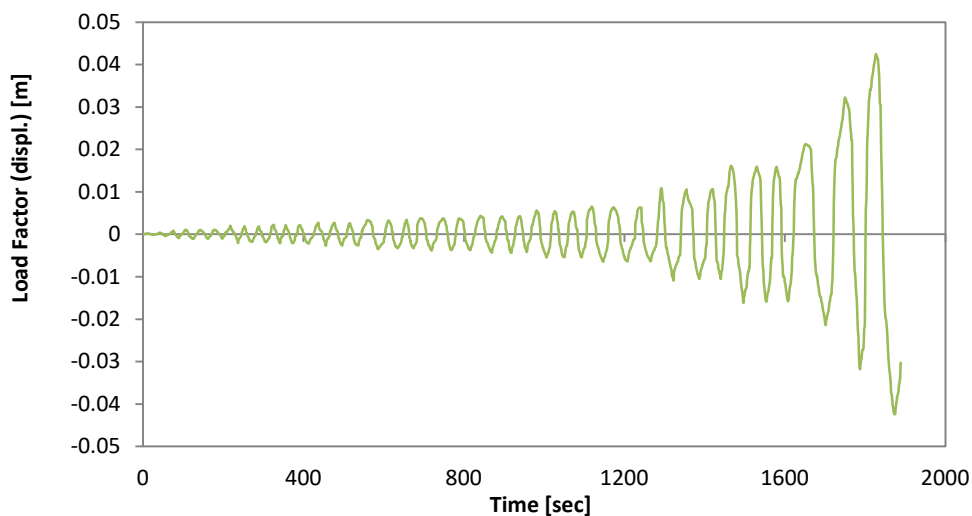


Figure 4.39. Cyclic loading

The time step for the static analysis is selected as 1 s (coincident with the input record sampling time step).

Two permanent loads of 20 kN each are applied at the top of the columns in the negative Z direction in terms of forces, in order to simulate the gravity loads, whereas a static time-history load, in terms of displacements, is imposed at the node nb121 in the X direction.

The FE model of the infilled frame is shown below:

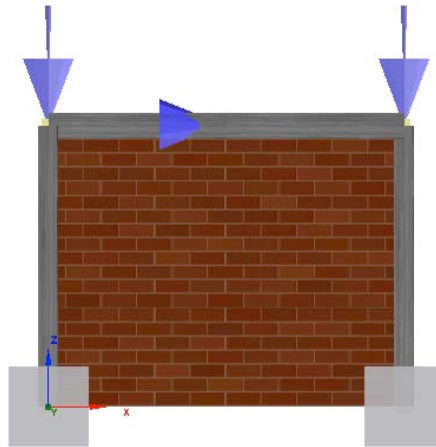


Figure 4.40. FE model (screenshot)

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

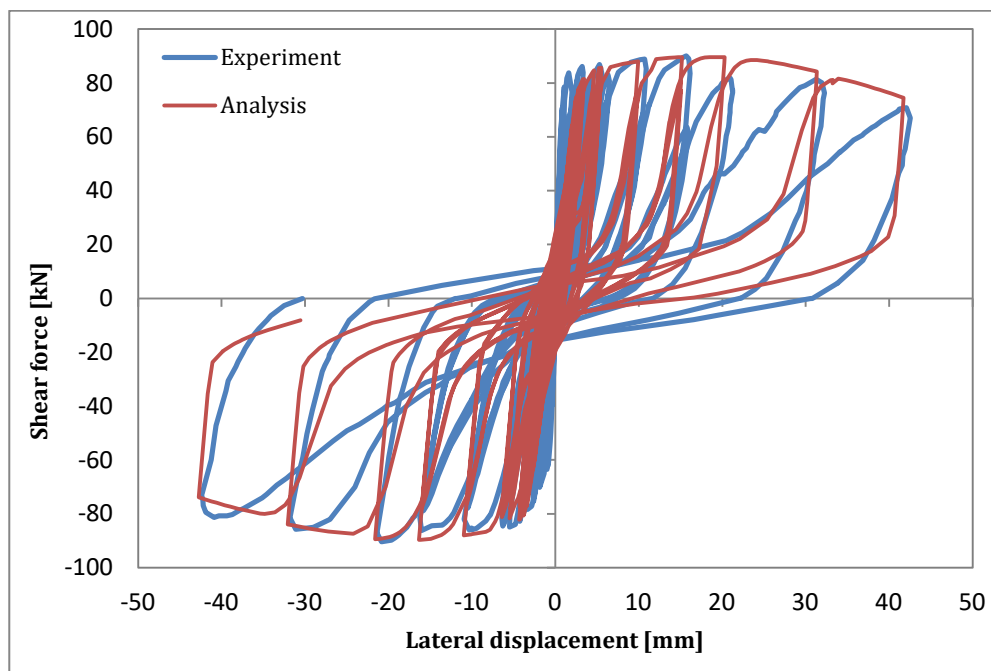


Figure 4.41. Experimental vs. Analytical results – shear force-lateral displacement

COMPUTER FILE

- Ch4-infill-01.spf

REFERENCES

- Crisafulli F.J. [1997] *"Seismic Behaviour of Reinforced Concrete Structures With Masonry Infills"*, PhD Thesis, University of Canterbury, New Zealand.
- Smyrou E., Blandon-Urbe C., Antoniou S. and Pinho R. [2006] *"Implementation and verification of a masonry panel model for nonlinear dynamic analysis of for infilled RC frames"*, Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 355.

EXAMPLE 2 – One storey, single bay infilled frame (Colangelo et al.)**DESCRIPTION**

This example describes the modelling of one storey, single bay infilled frame (in reduced scale) tested by Colangelo et al. (1999) and built to comply with the requirements of an earlier version of Eurocode 8 (CEN, 1995). The specimen was tested in the laboratory of materials and structures at the University of L'Aquila (Italy) under pseudo-dynamic loading using the E-W component of Tolmezzo earthquake (Friuli 1976).

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is defined in the X-Z plane.

NOTE: Detailed information about the material properties and the test arrangements can be found in Biondi et al. (2000).

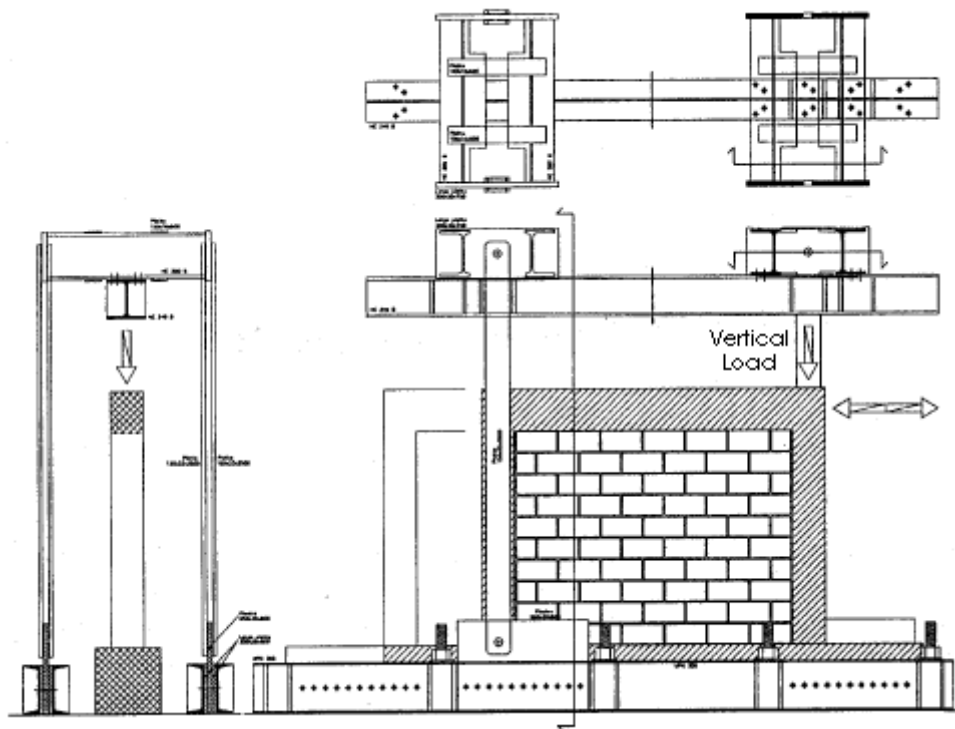
GEOMETRY AND PROPERTIES

Figure 4.42. Infilled frame tested by Colangelo (1999)

Geometry

The model consists of two RC columns (0.2 x 0.2 m), a RC beam (0.25 x 0.2 m) and an infill panel, the properties of which are listed in the paragraph “Modelling and Loading”.

Material Properties of the frame

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters are listed below:

- $f_c = 48900 \text{ kPa}$; $f_t = 0 \text{ kPa}$; $\epsilon_c = 0.002 \text{ m/m}$.

The bilinear steel model with kinematic strain hardening is employed for defining the steel material with the properties listed below:

- $E_s = 2.0000E+008$ kPa; $f_y = 550000$ kPa; $\mu = 0.003$

Properties of the infill panel

See paragraph *Modelling and Loading*.

MODELLING AND LOADING

Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200.

The infill panel is modelled through a four-node masonry panel element (inelastic infill panel element). It is characterized by the following parameters:

- Strut curve parameters (inf strut response curve): initial Young modulus $E_m = 4088000$, compressive strength $f_m = 800$, tensile strength $f_t = 0.001$, strain at maximum stress $\varepsilon_m = 0.002$, ultimate strain $\varepsilon_u = 0.005$, closing strain $\varepsilon_{cl} = 0.0000001$, strut area reduction strain $\varepsilon_1 = 0.001$, residual strut area strain $\varepsilon_2 = 0.002$, starting unloading stiffness factor $\gamma_u = 2.0$, strain reloading factor $\alpha_r = 1.5$, strain inflection factor $\alpha_{ch} = 0.6$, complete unloading strain factor $\beta_a = 2.0$, stress inflection factor $\beta_{ch} = 0.6$, zero stress stiffness factor $\gamma_{pu} = 1.0$, reloading stiffness factor $\gamma_{pr} = 1.1$, plastic unloading stiffness factor $e_{x1} = 1.5$, repeated cycle strain factor $e_{x2} = 1.0$
- Shear curve parameters (inf shear response curve): shear bond strength = 150, friction coefficient = 0.4, maximum shear resistance = 800, reduction shear factor = 1.46
- Panel thickness (t): 0.12 m
- Out-of-plane failure drift: 5% of ver. panel side
- Strut Area 1 (A1): 0.11 m²
- Strut Area 2 (A2): 60% of A1
- Equivalent contact length (hz): 15% of vert. panel side
- Horizontal and vertical offsets (xo and yo): 4% of horiz. panel side and 4% of vert. panel side
- Proportion of stiffness assigned to shear: 50%
- Specific weight: 10 kN/m³

All the base nodes are fully restrained.

In order to run a nonlinear static time-history analysis, a time-history curve is loaded in the “Time-history Curves” dialog box. It is defined as follows:

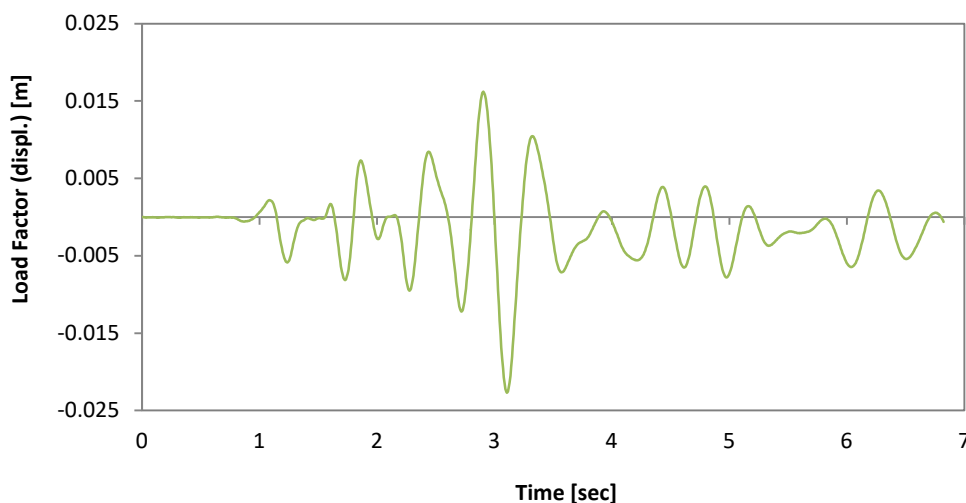


Figure 4.43. E-W component of Tolmezzo earthquake (Friuli, 1976) – displacement history

The time step for the static analysis is selected as 0.007 s (coincident with the input record sampling time step).

Two permanent loads of 250 kN each are applied at the top of the columns in the negative Z direction in terms of forces, in order to represent a ground floor infilled panel in a four-storey structure. A static time-history load, in terms of displacements, is imposed at the node nb121 in the X direction.

The FE model of the infilled frame is shown below:

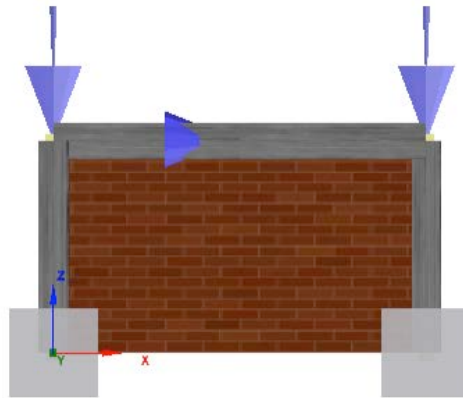


Figure 4.44. FE model (screenshot)

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

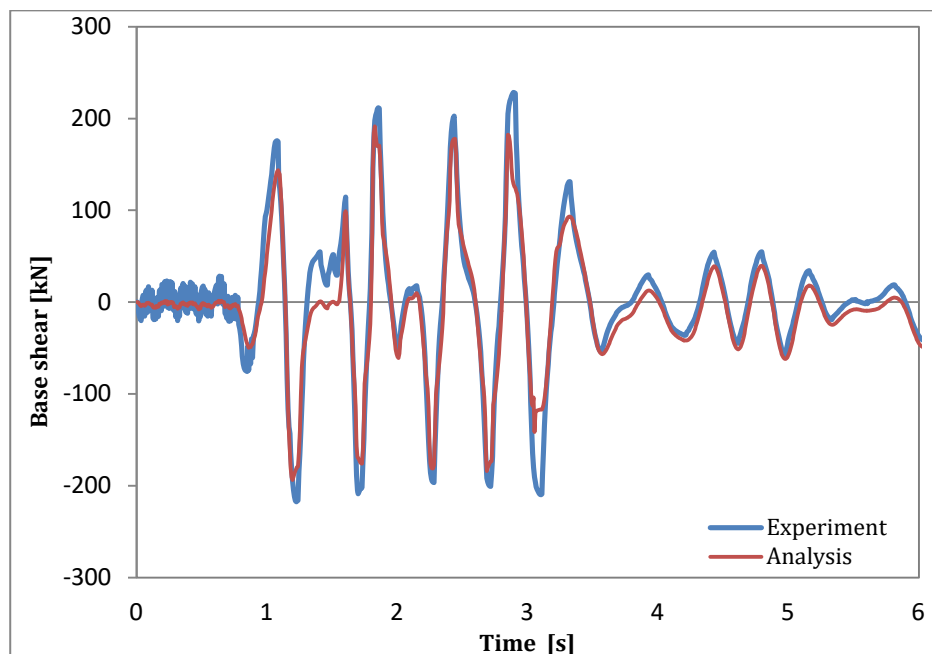


Figure 4.45. Experimental vs. Analytical results – base shear-time

COMPUTER FILE

- Ch4-infill-02.spf

REFERENCES

- Colangelo F. [1999] *“Qualificazione, risposta sismica pseudodinamica e modelli fenomenologici di portali di c.a. tamponati con laterizio”*, Research Report, DISAT, University of L'Aquila, Italy (in Italian).
- Biondi S., Colangelo F., Nuti C. [2000] *“La Risposta Sismica dei Telai con Tamponature Murarie”*, Research Report, GNDT, Rome, Italy. (in Italian).
- Smyrou E., Blandon-Uribe C., Antoniou S. and Pinho R. [2006] *“Implementation and verification of a masonry panel model for nonlinear dynamic analysis of for infilled RC frames”*, Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 355.

EXAMPLE 3 – Multi-storey 3D frame (Negro)**DESCRIPTION**

This example describes the modelling of a full-scale, four-storey building, which was designed according to initial versions of Eurocode 8 (CEN, 1995) and Eurocode 2 (CEN, 1991). The structure was tested at the ELSA laboratory (Joint Research Centre, Ispra) under pseudo-dynamic loading using an artificial accelerogram derived from the 1976 Friuli earthquake.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is defined in the X-Y-Z space.

NOTE: Full details on the structure's geometrical and material characteristics may be found in Negro et al. (1996).



Figure 4.46. Four-storey 3D infilled frame tested by Negro et al. (1996)

GEOMETRY AND PROPERTIES

The model consists of three RC frames, two infilled (exterior) and one bare (interior), composed by two bays (6 m and 4 m long, respectively), along the NS-direction and three bare RC frames composed by two bays (5 m long), in the EW-direction. The interstorey height is 3.5 m for the first floor and 3 m for the others.

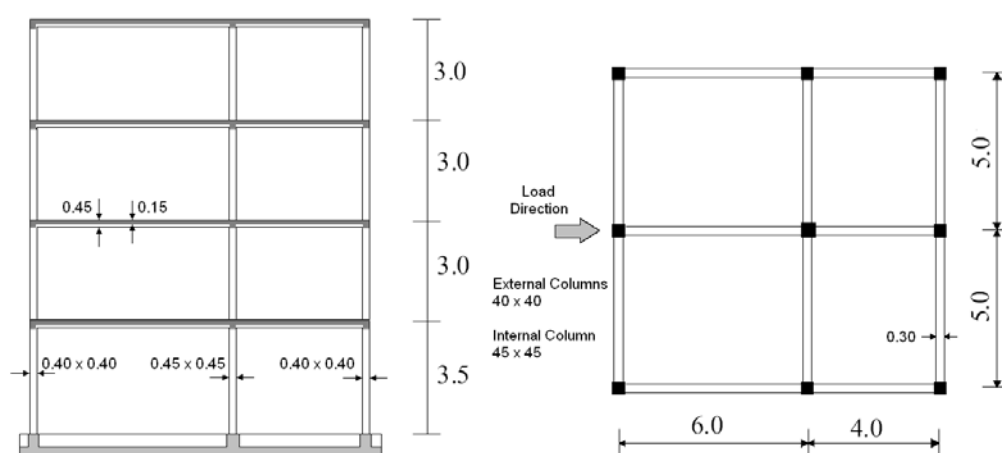


Figure 4.47. Four-storey 3D infilled frame geometry (frontal and plan views) [Negro et al., 1996]

Structural Geometry

The model consists of eight (exterior) RC columns (0.4 x 0.4 m), one (interior) RC column (0.45 x 0.45), T-section RC beams with different dimensions and infill panels.

Material Properties of the frames

The Mander et al. concrete model is employed for defining the concrete material used for columns and beams. In particular, eight different types of concrete are defined:

Table 4.11. Characteristic parameters of concrete (columns)

PARAMETERS	1 st Storey	2 nd Storey	3 rd Storey	4 th Storey
f_c (kPa)	44800	42600	27000	41300
f_t (kPa)	0.001	0.001	0.001	0.001
ϵ_c (m/m)	0.002	0.002	0.002	0.002

Table 4.12. Characteristic parameters of concrete (beams)

PARAMETERS	1 st Storey	2 nd Storey	3 rd Storey	4 th Storey
f_c (kPa)	51400	48200	42200	37100
f_t (kPa)	0.001	0.001	0.001	0.001
ϵ_c (m/m)	0.002	0.002	0.002	0.002

Then, the Menegotto-Pinto steel model is employed for defining the steel material with the following properties:

- $E_s = 200000000$ kPa; $f_y = 555000$ kPa, $\mu = 0.02$

Properties of the infill panels

See SeismoStruct input file (--> Ch4-infill-03.spf).

MODELLING AND LOADING

The RC columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200. It is noted that *multiple sections* are assigned to some element classes.

The infill panels are modelled through a four-node masonry panel element (inelastic infill panel element). For the characteristic parameters refer to the input file of the model.

Regarding the masses, they are applied to the structure in lumped fashion (for the three translational inertia values of each lumped mass refer to the SeismoStruct model) and then automatically converted to gravity loads.

The slabs are modelled by introducing a rigid diaphragm in the X-Y plane for each floor level (see SeismoStruct model).

All the base nodes are considered fully restrained against rotations and translations.

In order to run a nonlinear static time-history analysis, four time-history curves (one for each floor) are loaded in the “Time-history Curves” dialog box.

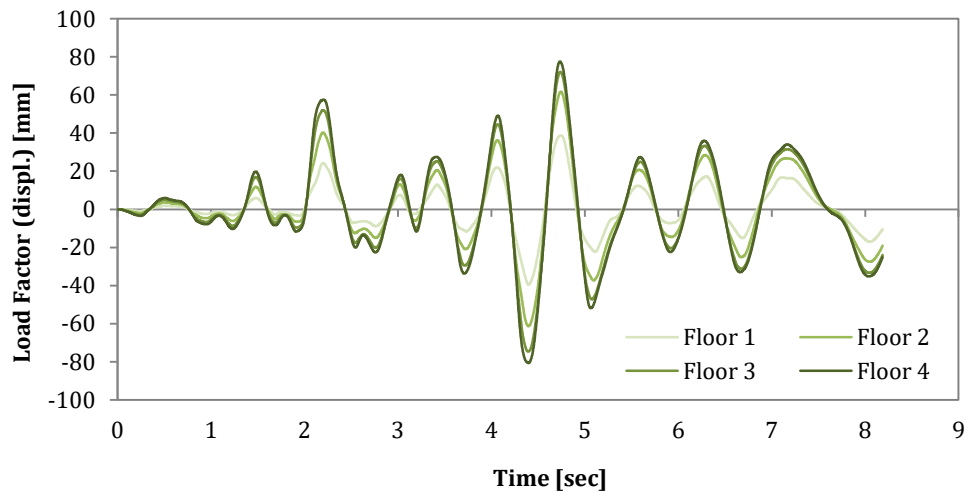


Figure 4.48. Artificial record generated from Friuli accelerogram – displacement histories

The time step for the static time-history analysis is set to 0.002 s.

Only static time-history loads, in terms of displacements, are applied to the structure in the X direction (two loads per floor).

The FE model of the building is presented below:

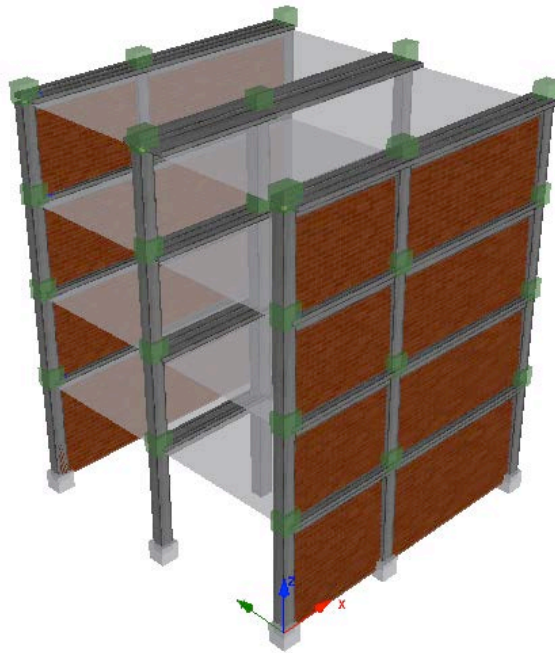


Figure 4.49. FE model (screenshot)

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

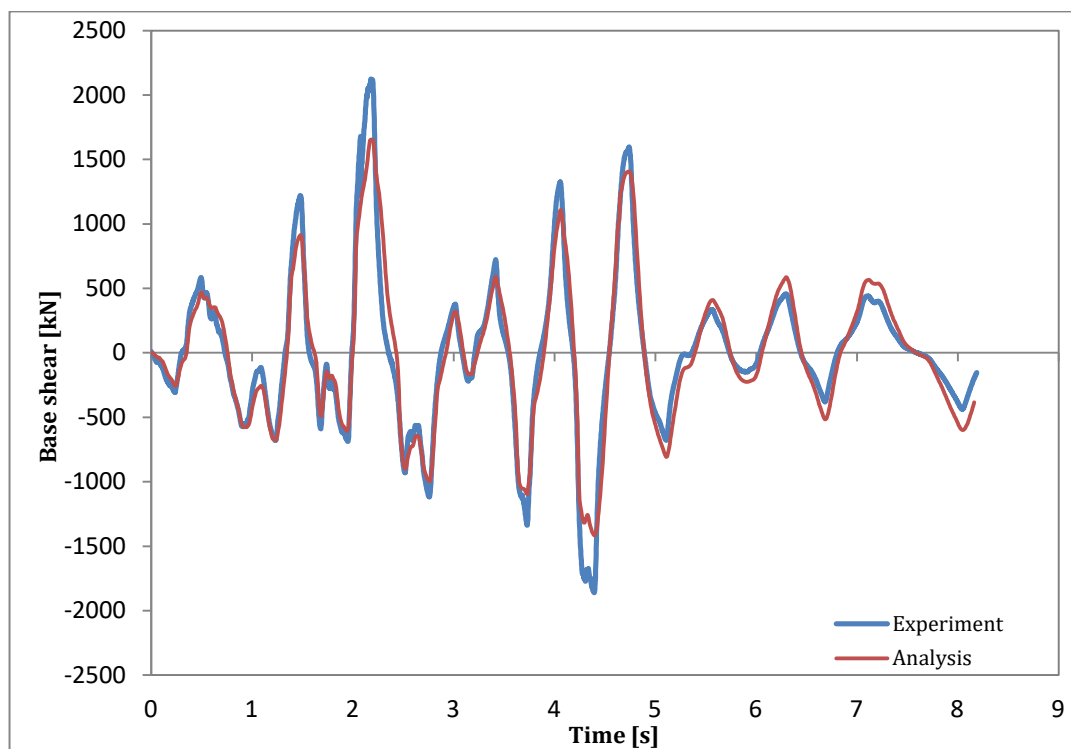


Figure 4.50. Experimental vs. Analytical results – base shear-time

COMPUTER FILE

- Ch4-infill-03.spf

REFERENCES

- Negro, P. *et al* [1995] "*Test on the Four-Storey Full-Scale RC frame with masonry Infills*", Special Publication No I.95.54, European Laboratory for Structural Assessment, Joint Research Centre, Ispra, Italy.
- Negro P., Pinto A.V., Verzeletti G., Magonette G.E. [1996] "*PsD Test on a Four-Storey R/C Building Designed According to Eurocodes*", Journal of Structural Engineering - ASCE, Vol. 122, No. 11, 1409-1417.
- Negro P., Verzeletti G. [1996] "*Effect of Infills on the Global Behaviour of R/C Frames: Energy Considerations from Pseudodynamic Tests*", Earthquake Engineering and Structural Dynamics, Vol. 25, 753-773.
- Negro P. [1996] "*Experimental Assessment of the Global Cyclic Damage of Framed R/C Structures*", Journal of Earthquake Engineering, Vol. 1, No. 3, 543-562, 1997.
- Smyrou E., Blandon-Urbe C., Antoniou S. and Pinho R. [2006] "*Implementation and verification of a masonry panel model for nonlinear dynamic analysis of for infilled RC frames*", Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 355.

EXAMPLE 4 – Multi-storey, 2D infilled frame (ICONS frame - infilled)

DESCRIPTION

This example describes the modelling of a full-scale, four-storey, 2D infilled frame, which is representative of the design and construction practice of the 50s-60s in Southern Mediterranean countries, hence not meeting modern seismic design requirements. The frame includes infill panels with openings of different dimensions. The specimen was tested under two subsequent pseudo-dynamic loading (Acc-475 and Acc-975) at the ELSA laboratory (Joint Research Centre, Ispra) within the framework of the ICONS research program (Carvalho and Coelho, 2001).

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results for the 975 years return period input motion. The model is defined in the X-Z plane.

NOTE: Further information about the ICONS frame as well as the tests conducted in ELSA can be found in Pinto et al. (1999), Carvalho et al. (1999), Pinho and Elnashai (2000) and Varum (2003).

GEOMETRY AND PROPERTIES

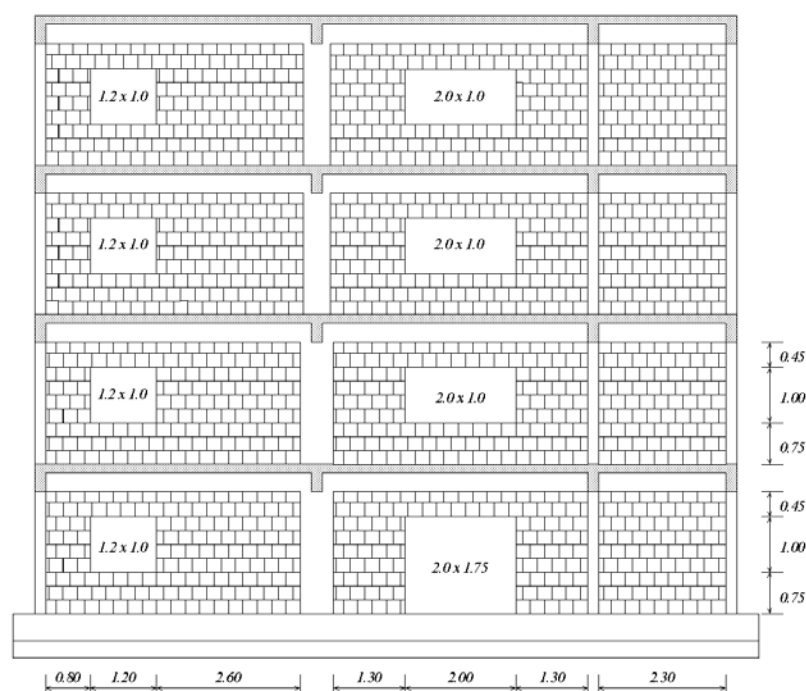


Figure 4.51. Four-storey 2D infilled frame (ICONS frame) [Carvalho et al., 1999]

Structural Geometry

The model consists of RC columns and beams and three kinds of infill panels (with and without openings).

Material Properties of the frame

The material properties are the same of those presented for the bare frame (see *Chapter 4 > Typical existing non-seismically-designed RC European frames > EXAMPLE 1*)

Properties of the infill panel

See SeismoStruct input file.

MODELLING AND LOADING

Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200. It is noted that *multiple sections* are assigned to some element classes.

The infill panels are modelled through a four-node masonry panel element (inelastic infill panel element). For the characteristic parameters refer to the input file of the model.

Regarding the masses, which have been computed as proportional to the tributary areas, they are applied (i) in a lumped fashion (at each beam-column joint) and (ii) distributed along the beams (using the “section additional mass” feature at the section level) – for their values refer to the bare frame (*Chapter 4 > Typical existing non-seismically-designed RC European frames > EXAMPLE 1*).

All the base nodes are fully restrained.

In order to run a dynamic time-history analysis, the same time-history curve employed in the bare frame modelling and constituted by two artificial records (Acc475 and Acc975) in series separated by 35 s interval with no acceleration, is loaded in the “Time-history Curves” dialog box.

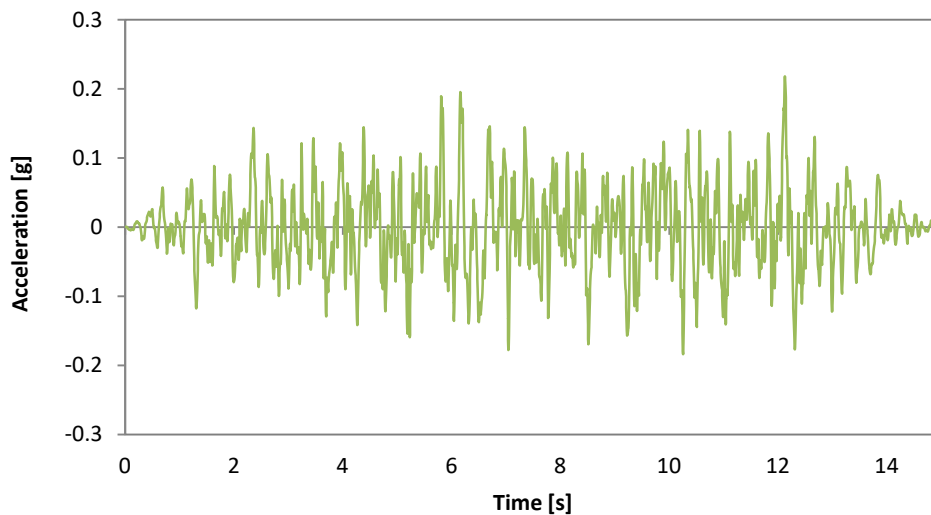


Figure 4.52. Artificial acceleration time histories for 475 year (Acc-475)

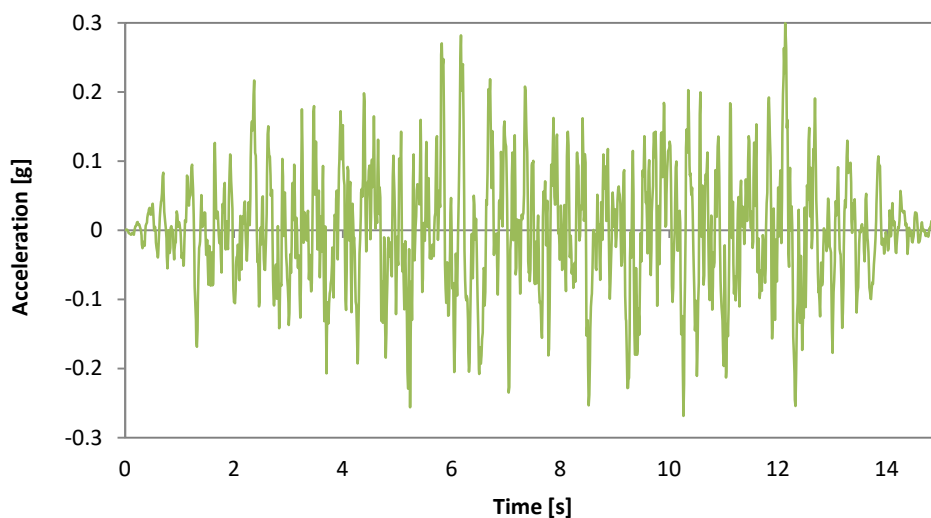


Figure 4.53. Artificial acceleration time histories for 975 year (Acc-975)

The vertical loads are automatically computed by the program through the option to derive loads from masses, based on the g value (see Gravity and mass page of the Project Settings).

The FE model is presented hereafter:

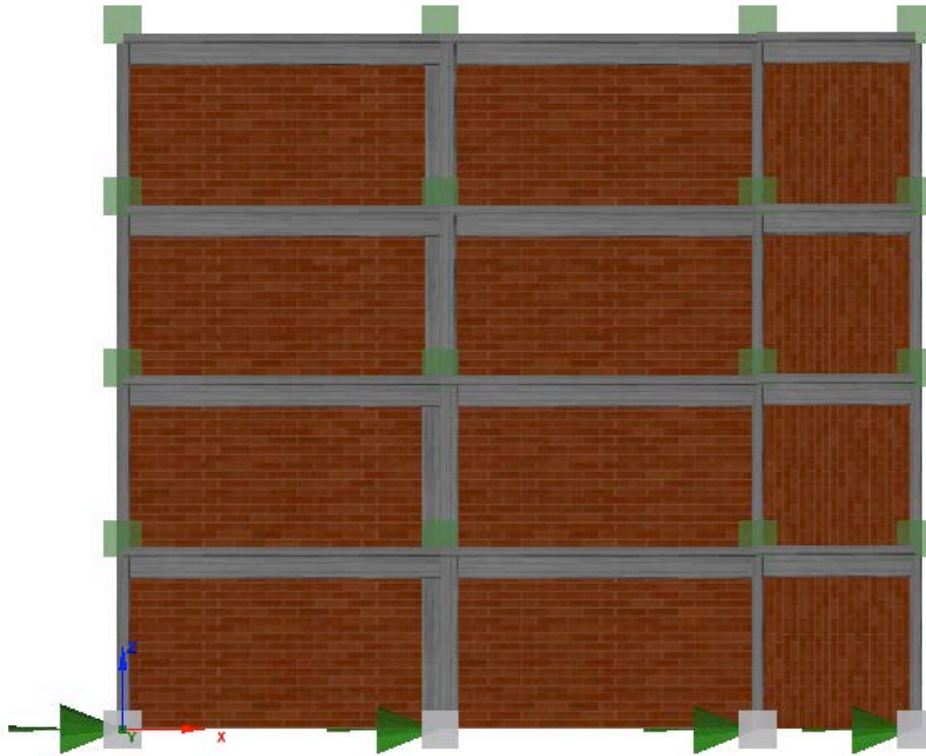


Figure 4.54. FE model (screenshot)

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

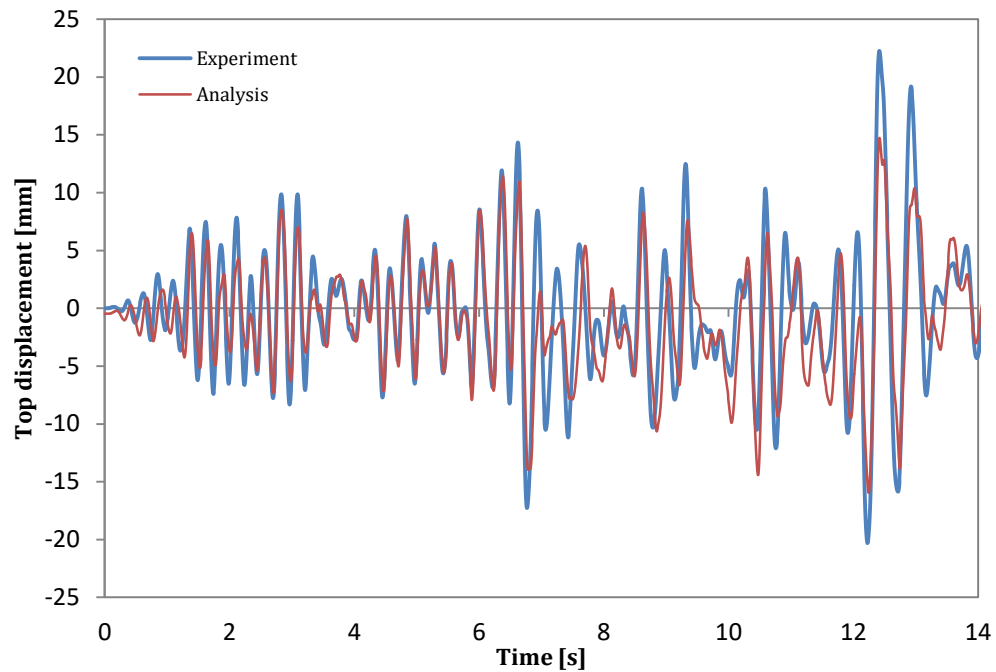


Figure 4.55. Experimental vs. Analytical results – top displacement-time (975yrp)

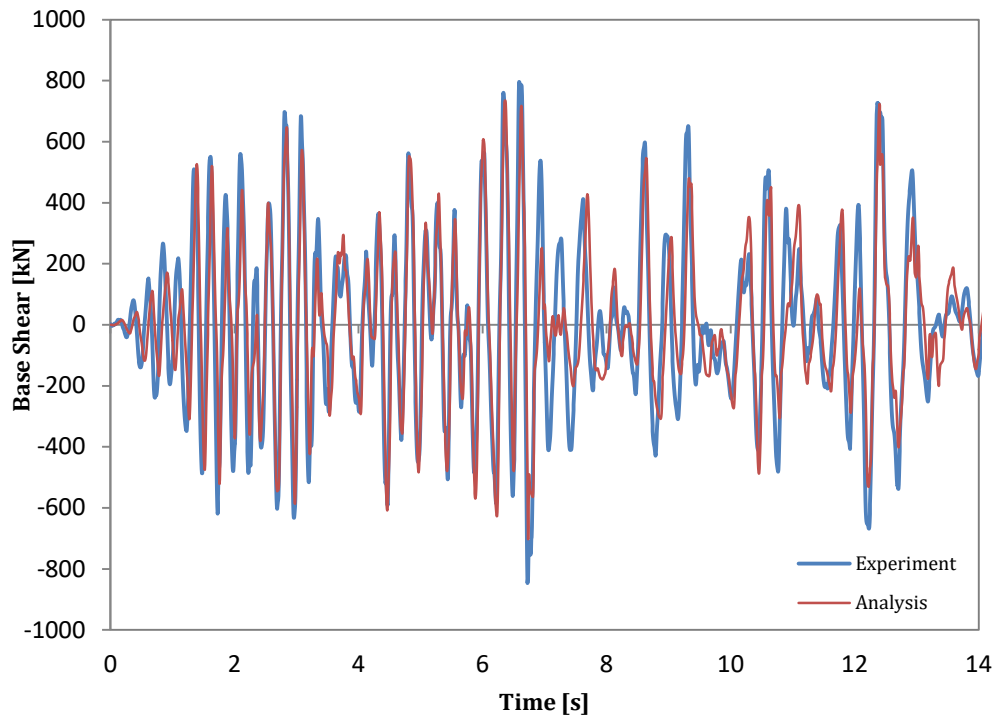


Figure 4.56. Experimental vs. Analytical results – base shear-time (975yrp)

COMPUTER FILE

- Ch4-infill-04.spf

REFERENCES

- Carvalho E.C., Coelho E., Campos-Costa A. [1999] *“Preparation of the Full-Scale Tests on Reinforced Concrete Frames. Characteristics of the Test Specimens, Materials and Testing Conditions”*, ICONS Report, Innovative Seismic Design Concepts for New and Existing Structures, European TMR Network, LNEC.
- Smyrou E., Blandon-Urbe C., Antoniou S. and Pinho R. [2006] *“Implementation and verification of a masonry panel model for nonlinear dynamic analysis of for infilled RC frames”*, Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 355.

EXAMPLE 5 – Half scale, three-storey, three-dimensional infilled RC frame**DESCRIPTION**

This example describes the modelling of a half scale, three-storey RC building, similar to SPEAR structure, filled by nonstructural masonry infilled walls (made on cellular concrete Gasbeton RDB). The prototype building was built with the construction practice and materials used in Greece in the early 70's (non-earthquake resistant construction). As the SPEAR building, it is regular in height but highly irregular in plan. It was tested at the EUCENTRE of Pavia (Italy) under pseudo-dynamic conditions using the Herceg-Novi accelerogram registered during the Montenegro 1979 earthquake.

The analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. The model is defined in the X-Y-Z space.

NOTE: Detailed information about structural beam member dimensions and reinforcing bars may be found in Fardis and Negro (2006).



Figure 4.57. Half scale, three-storey infilled RC building

GEOMETRY AND PROPERTIESStructural Geometry

The model consists of seven RC columns (0.125×0.125 m) and one RC column (0.125×0.375) per floor and RC beams (0.125×0.25 m).

Material Properties of the building

The Mander et al. concrete model is employed for defining the concrete materials with the following parameters:

- (beams 1 and 2): $f_c = 43$ MPa; $f_t = 0$ MPa; $\epsilon_c = 0.002$ mm/mm
- (beam 3): $f_c = 36$ MPa; $f_t = 0$ MPa; $\epsilon_c = 0.002$ mm/mm
- (columns 1 and 2): $f_c = 33$ MPa; $f_t = 0$ MPa; $\epsilon_c = 0.002$ mm/mm

- (column 3): $f_c = 27 \text{ MPa}$; $f_t = 0 \text{ MPa}$; $\epsilon_c = 0.002 \text{ mm/mm}$

The bilinear model with kinematic strain-hardening is employed for defining two types of steel materials with the following parameters:

- (steel for 6mm bars): $E_s = 200000 \text{ MPa}$; $f_y = 364 \text{ MPa}$; $\mu = 0.004$
- (steel for 10mm bars): $E_s = 200000 \text{ MPa}$; $f_y = 293 \text{ MPa}$; $\mu = 0.004$

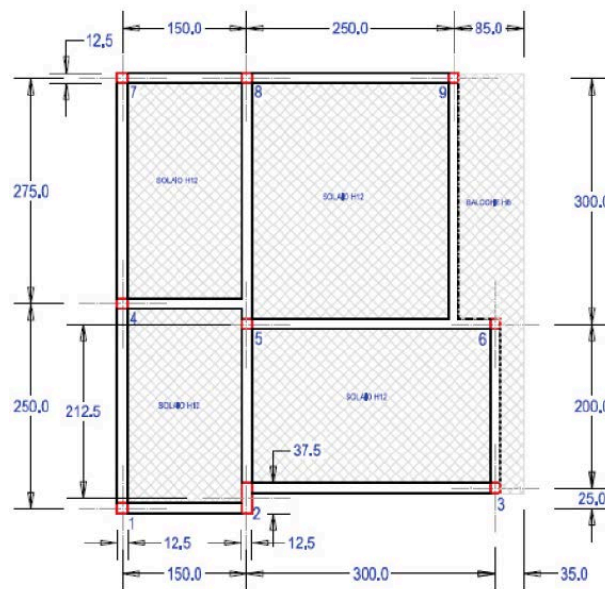


Figure 4.58. Plan view of the half-scale, three-storey prototype building [Ali H., 2009]

MODELLING AND LOADING

Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 5 integration sections. The number of fibres used in section equilibrium computations is set to 200.

The infill panels are modelled through a four-node masonry panel element (inelastic infill panel element). For the characteristic parameters of each panel refer to the input file of the model.

Regarding the applied masses, they are distributed along columns and beams (using the “section additional mass” feature at the section level), in order to represent (i) the self-weight of the frame, (ii) permanent loads and (iii) variable loads. For the values refer to the SeismoStruct frame model.

All foundation nodes are considered as fully restrained against rotations and translations.

The connection between column C6 and the adjacent beams is modelled as rigid through rigid links.

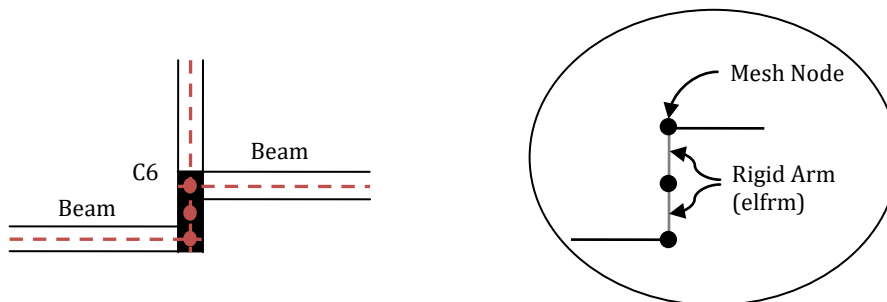


Figure 4.59. Modelling of column C6 and its connection to the adjacent beams

The slabs are modelled by introducing a rigid diaphragm in the X-Y plane at each floor level (see SeismoStruct model).

In order to run a nonlinear dynamic time-history analysis, the scaled Herceg-Novı record with moderate intensity ($\text{PGA} = 0.3\text{g}$) is loaded in the “Time-history Curves” dialog box. The applied curve is presented below:

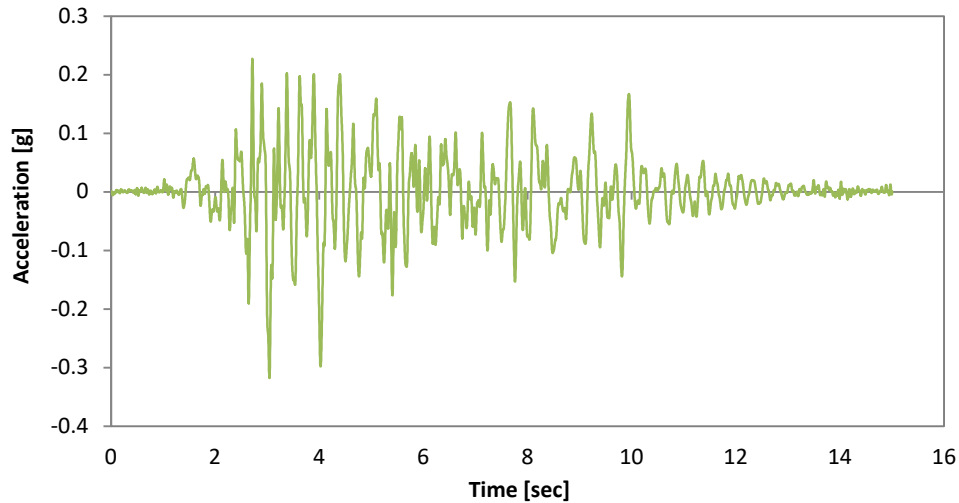


Figure 4.60. Scaled Herceg-Novı record with moderate intensity ($\text{PGA} = 0.3\text{g}$) in the Y direction

The time step for the dynamic time-history analysis is set to 0.0021 s.

Dynamic time-history loads are applied at the base nodes, in terms of accelerations in the Y direction.

The FE model of the building is shown below:

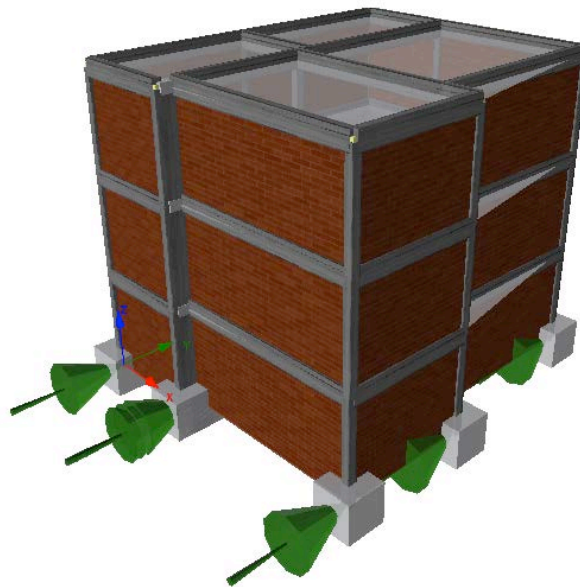


Figure 4.61. FE model of the building

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

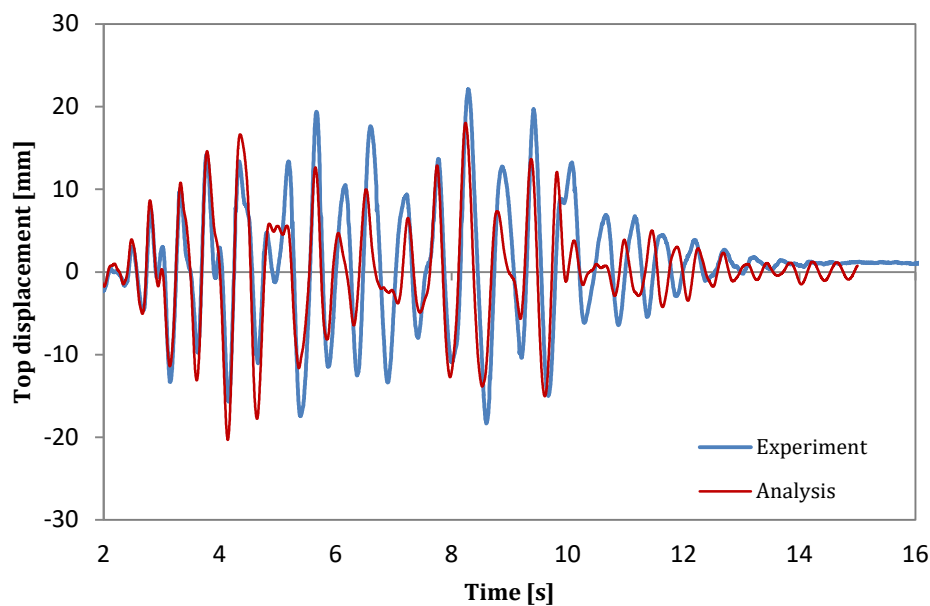


Figure 4.62. Experimental vs. Analytical results – top displacement-time

COMPUTER FILE

- Ch4-infill-05.spf

REFERENCES

- Fardis, M.N. [2002] “*Design of an Irregular Building for the SPEAR Project – Description of the 3-Storey Structure*”, Research Report, University of Patras, Greece.
- Fardis, M.N. and Negro P. [2006] “*SPEAR – Seismic performance assessment and rehabilitation of existing buildings*”, Proceedings of the International Workshop on the SPEAR Project, Ispra, Italy.
- Penna A. [2006] “*Campagna sperimentale su telai in calcestruzzo armato con tamponamenti in calcestruzzo cellulare e diverse soluzioni di rinforzo*”, Technical Report, EUCENTRE – Centro Europeo di Formazione e Ricerca in Ingegneria Sismica.
- Ali H. [2009] “*Half Scale Three-Storey nfilled RC Building; A Comparison of Experimental and Numerical Models*”, MSc Dissertation, ROSE School, Pavia, Italy.

BEAM-COLUMN JOINTS

EXAMPLE 1 – RC joints

DESCRIPTION

This example describes the modelling of three half-scale RC beam-column joints sub-assemblages. The specimens, subdivided into two series (Series A and B), have been tested at the University of Tokyo under quasi-statically reversed cyclic loading with increasing amplitudes simulating seismic loading condition.

For specimens A2, B1 and B2 the analytical results, obtained with the FE analysis program SeismoStruct, are compared with the experimental results. A FE model is created for each joint and is defined in the X-Z plane.

NOTE: Detailed information about geometry and material properties may be found in Yu Wei (2006).



Figure 4.63. RC joint [Shiohara & Kusuhara, 2006]

GEOMETRY AND PROPERTIES

Each model consists of a RC joint of crucial form; two columns and beams are framing to one joint without transverse beam nor slab.

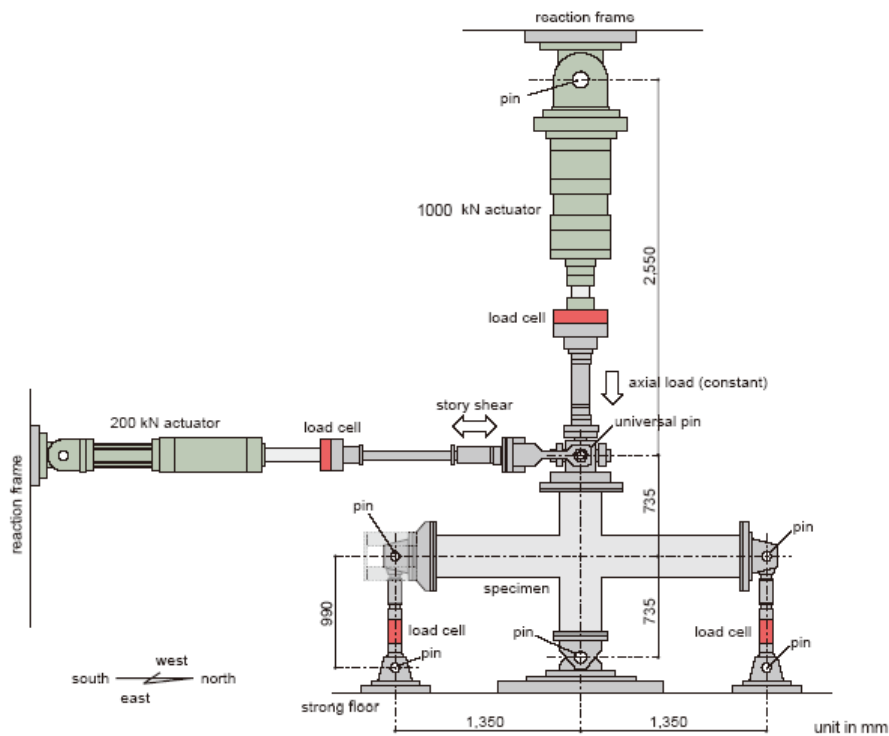


Figure 4.64. RC joint scheme [Shiohara & Kusahara, 2006]

Structural Geometry of specimens A2, B1 and B2

The geometrical details are shown in the figures below:

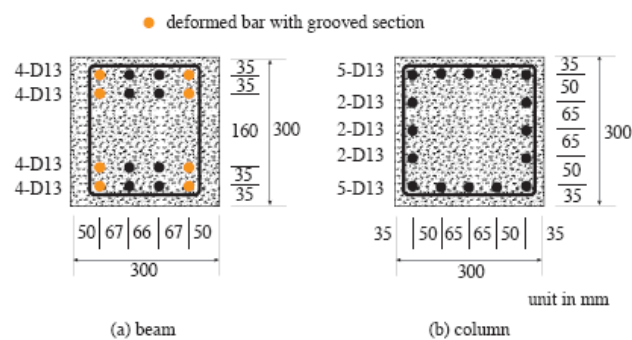


Figure 4.65. Section details of specimen A2 [Yu, 2006]

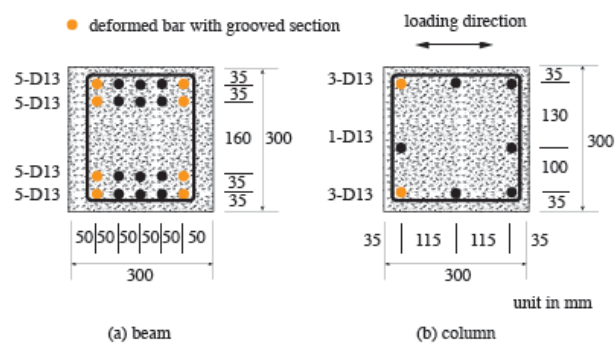


Figure 4.66. Section details of specimens B1 and B2 [Yu, 2006]

Material Properties of the specimens

The material properties are the same for each specimen.

The Mander et al. concrete model is employed for defining the concrete material used for columns and beams. The characteristic parameters are listed below:

- $f_c = 28.3 \text{ MPa}$; $f_t = 2.67 \text{ MPa}$; $\epsilon_c = 0.002 \text{ m/m}$.

The Menegotto-Pinto steel model is employed for defining the steel material with the following properties:

- (steel for beam section): $E_s = 200000 \text{ MPa}$; $f_y = 456.4 \text{ MPa}$; $\mu = 0.004$
- (steel for column section): $E_s = 200000 \text{ MPa}$; $f_y = 356.9 \text{ MPa}$; $\mu = 0.004$

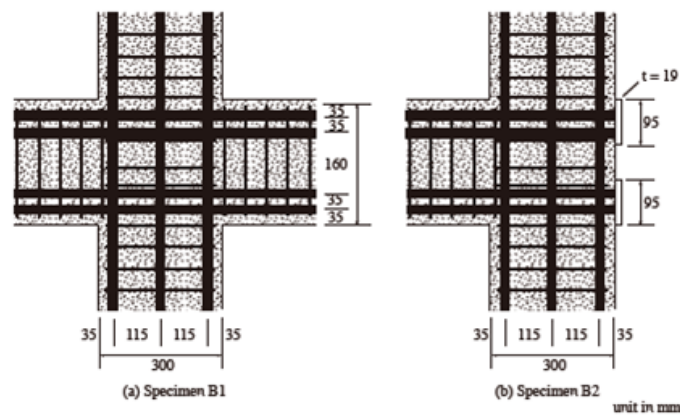


Figure 4.67. Reinforcing details of specimens B1 and B2 [Yu, 2006]

MODELLING AND LOADING

Columns and beams are modelled through 3D force-based inelastic frame elements (infrmFB) with 5 integration sections. The number of fibres used in section equilibrium computations is set to 200.

The intersection between column and beam, defined as “panel zone” is modelled with a 3D elastic nonlinear beam-column element (elfrm) with the following properties: $EA = 1.0000E+019 \text{ [N]}$; $EI \text{ (axis2)} = 1.0000E+019 \text{ [N-mm}^2\text{]}$; $EI \text{ (axis3)} = 1.0000E+019 \text{ [N-mm}^2\text{]}$; $GJ = 1.0000E+019 \text{ [N-mm}^2\text{]}$.

Three link elements are defined, in order to connect the inelastic frame elements to the elastic panel zone. The properties of the link elements employed in the FE model of specimen A2 are listed below. For the properties of the other specimens refer to the input files of the models.

Properties of Link1 (JointA bm)

All degrees of freedom properties are defined for the link element. In particular, for F_1 , F_2 , M_1 and M_3 degrees of freedom a linear symmetric response curve is chosen with a value of stiffness k_0 equal to $1.0000E+016 \text{ N/mm}$, whereas for F_3 and M_2 degrees of freedom a simplified bilinear Takeda response curve is chosen with the following properties: yielding strength $F_y = 405000 \text{ MPa}$, initial stiffness $S_y = 4800000 \text{ N/mm}$, post-yielding to initial stiffness ratio = 0.000001, outer loop stiffness degradation factor = 0.00001, inner loop stiffness degradation factor = 1.0 (for F_3 dof), yielding strength $F_y = 7.7265E+007 \text{ MPa}$, initial stiffness $S_y = 1.0657E+011$, post-yielding to initial stiffness ratio = 0.017058, outer loop stiffness degradation factor = 0.1, inner loop stiffness degradation factor = 0.9 (for M_2 dof)

Properties of Link2 (JointA col)

All degrees of freedom properties are defined for the link element. In particular, for F_2 , F_3 , M_1 and M_3 degrees of freedom a linear symmetric response curve is chosen with a value of stiffness k_0 equal to $1.0000E+016 \text{ N/mm}$, whereas for F_1 and M_2 degrees of freedom a simplified bilinear Takeda response curve is chosen with the following properties: yielding strength $F_y = 405000 \text{ MPa}$, initial stiffness $S_y = 4800000 \text{ N/mm}$, post-yielding to initial stiffness ratio = 0.000001, outer loop stiffness degradation

factor = 0.00001, inner loop stiffness degradation factor = 1.0 (for F_1 dof), yielding strength $F_y = 8.3004\text{E}+007$, initial stiffness $S_y = 1.8653\text{E}+011$ N/mm, post-yielding to initial stiffness ratio = 0.0128875, outer loop stiffness degradation factor = 0.1, inner loop stiffness degradation factor = 0.9 (for M_2 dof).

Properties of Link3 (JointA Centroid)

All degrees of freedom properties are defined for the link element. In particular, for F_1 , F_2 , F_3 , M_1 and M_3 degrees of freedom a linear symmetric response curve is chosen with a value of stiffness k_0 equal to $1.0000\text{E}+016$, whereas for M_2 degree-of-freedom a simplified bilinear Takeda response curve is chosen with the following properties: yielding strength $F_y = 2.3581\text{E}+008$ MPa, initial stiffness $S_y = 2.2735\text{E}+010$ N/mm, post-yielding to initial stiffness ratio = 0.16219, outer loop stiffness degradation factor = 0.00001, inner loop stiffness degradation factor = 1.0

The column base node is fully restrained.

In order to run a nonlinear static time-history analysis, a time-history curve is loaded in the “Time-history Curves” dialog box. It is defined as follows:

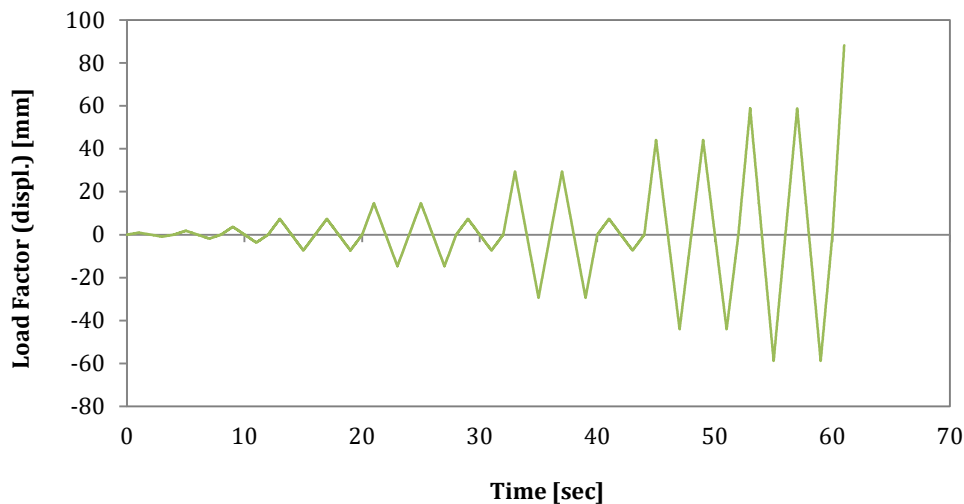


Figure 4.68. Cyclic loading

The time step for the static analysis is selected as 0.01 s, coincident with the input record sampling time step.

Two permanent loads of 3037.5 N and 216000 N respectively are applied at the nodes 5 and 7 in the negative Z direction in terms of forces, whereas a static time-history load, in terms of displacements, is imposed at the node 7 in the X direction.

The FE model of specimens A2, B1 and B2 are shown below:

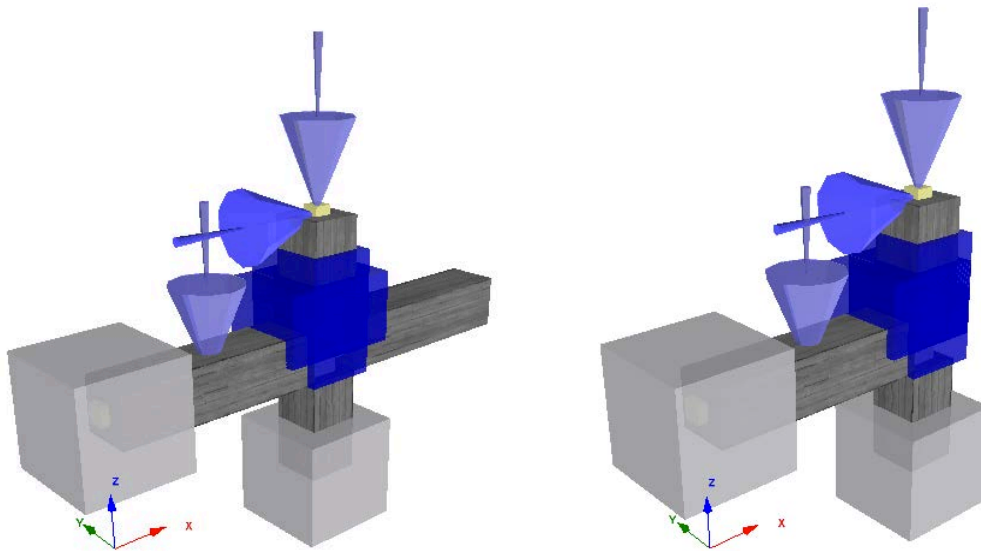


Figure 4.69. FE models of specimen A2, B1 (on the left) and B2 (on the right)

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

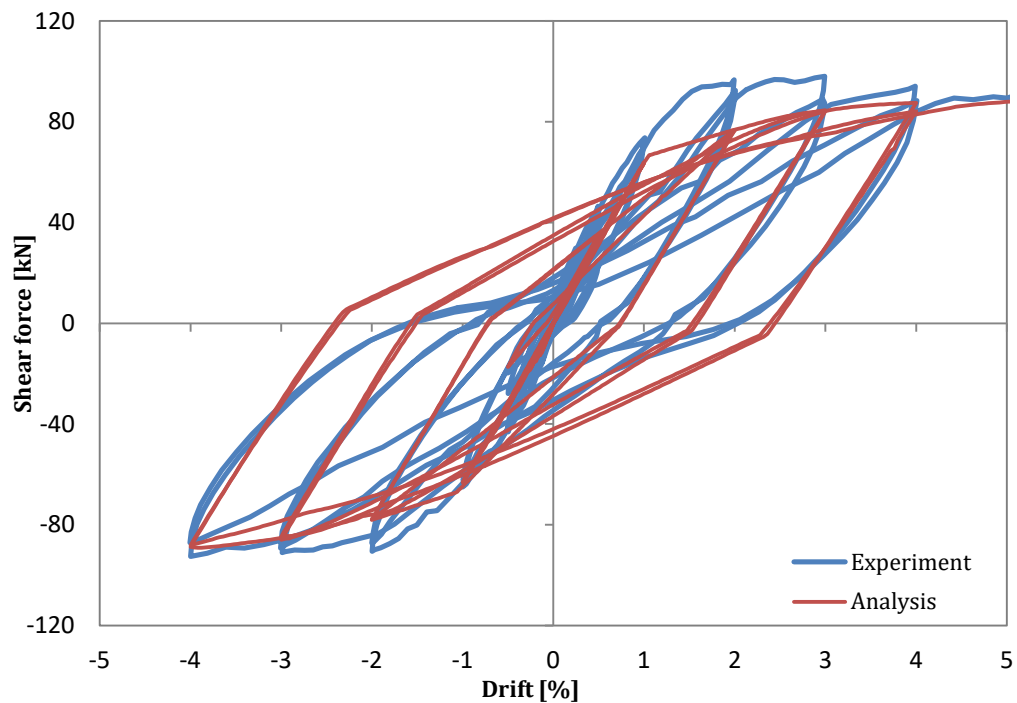


Figure 4.70. Experimental vs. Analytical results – shear force-drift (top node) (specimen B1)

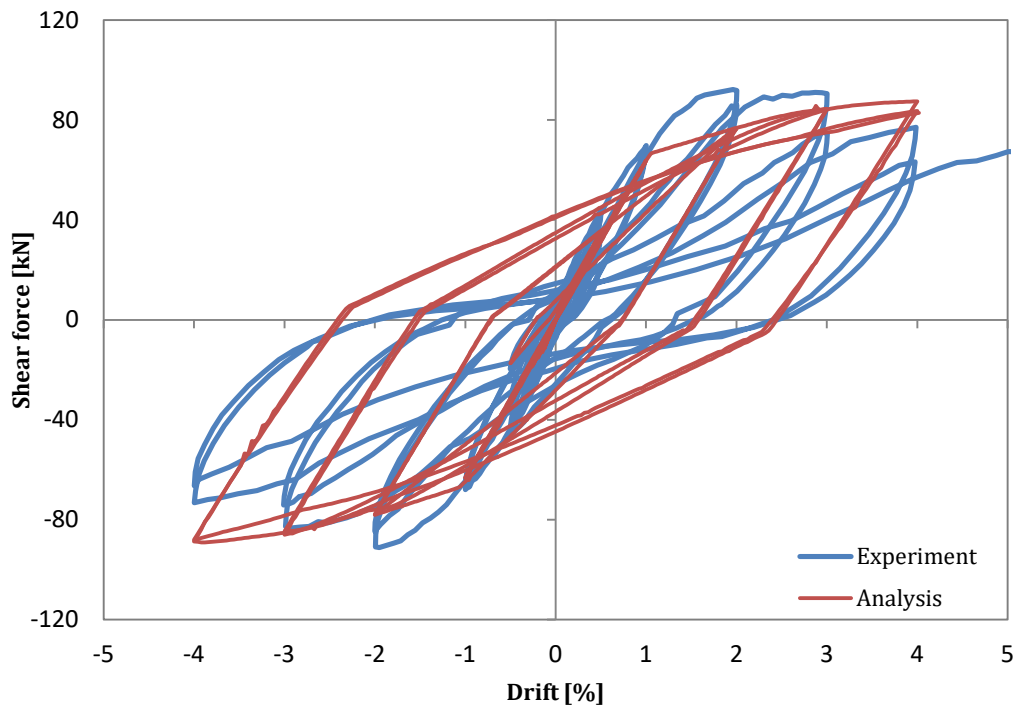


Figure 4.71. Experimental vs. Analytical results – shear force-drift (top node) (specimen B2)

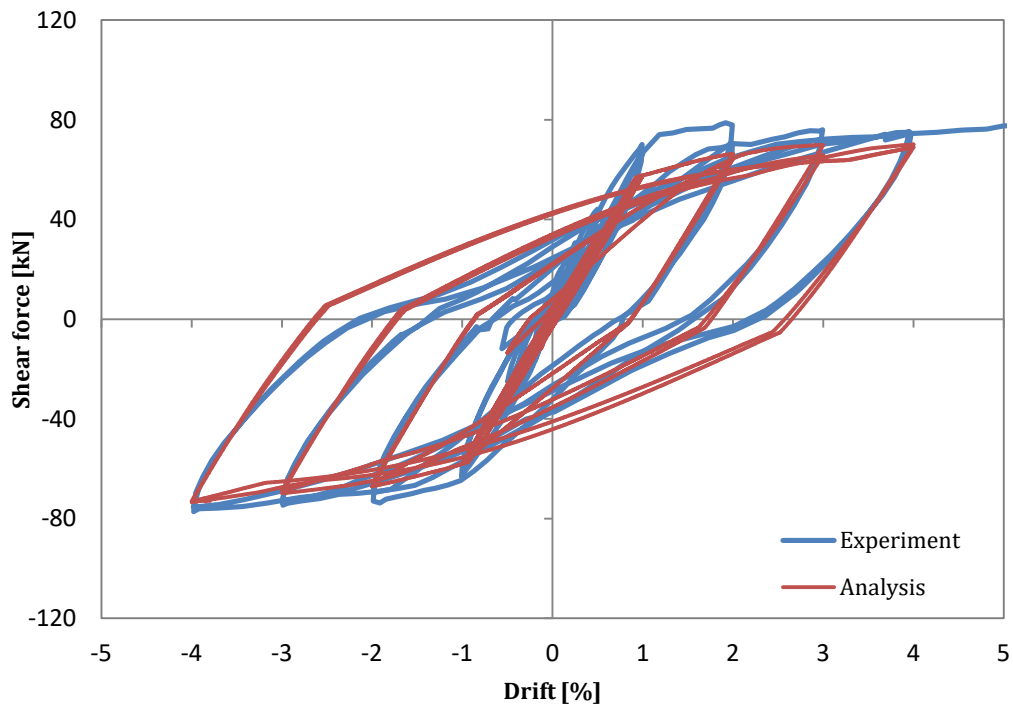


Figure 4.72. Experimental vs. Analytical results – shear force-drift (top node) (specimen A2)

COMPUTER FILES

- Ch4-joint-01.spf (-> specimen B1)
- Ch4-joint-02.spf (-> specimen B2)
- Ch4-joint-03.spf (-> specimen A2)

REFERENCES

- Yu W. [2006] *“Inelastic modeling of reinforcing bars and blind analysis of the benchmark tests on beam-column joints under cyclic loading”*, MSc Dissertation, ROSE School, Pavia, Italy.

EXAMPLE 2 – Steel joints

DESCRIPTION

This example describes the modelling of three steel joints tested by Bursi et al. (2002), Korol et al. (1990) and Broderick & Thomson (2002), respectively, under different cyclic loadings. The first joint presented in this example is an extended end-plate connection with 18 mm plate thickness (see Figure 4.73 (a)), the second one is an extended end-plate connection with stiffeners in the column web as well as in the end-plate (see Figure 4.73 (b)) and the third one is a flush end-plate joint (see Figure 4.73 (c)).

The analytical results, obtained with the FE analysis program SeismoStruct (in which is employed the modified Richard-Abbot hysteretic model able to simulate the cyclic behaviour of steel and composite joints), are compared with the experimental results. For each connection a FE model has been created and is defined in the X-Z plane.

NOTE: Detailed information about the geometry and material properties may be found in Nogueiro et al. (2007).

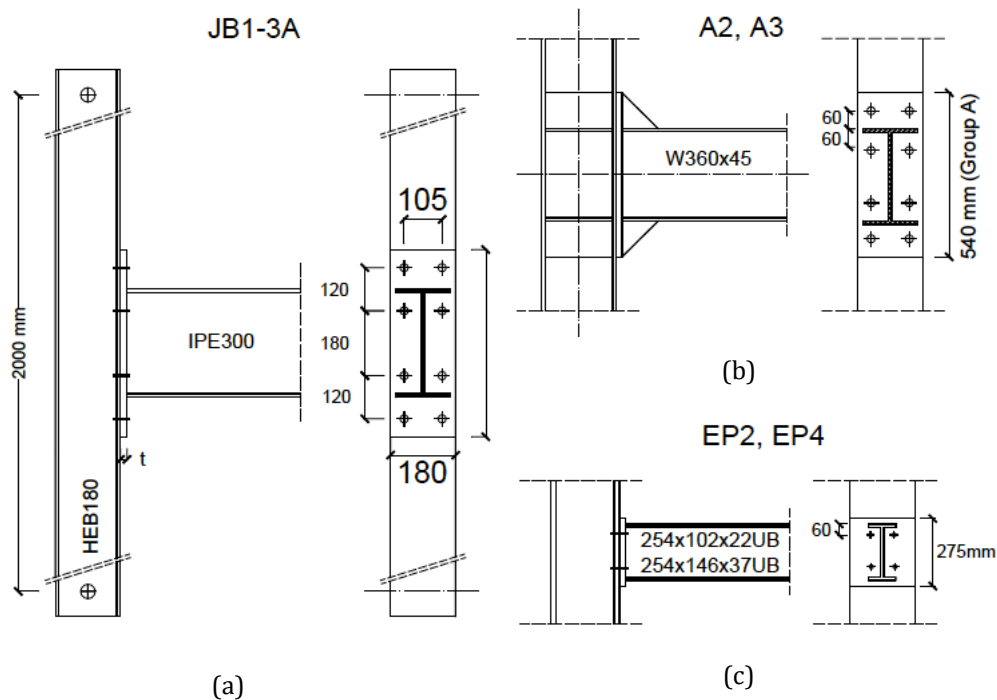
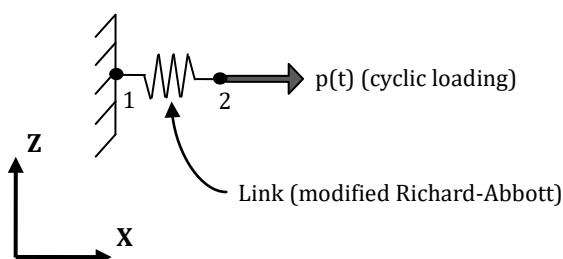


Figure 4.73. Steel joints (a) Bursi et al. (2002), (b) Korol et al. (1990), (c) Broderick et al. (2002)

GEOMETRY AND PROPERTIES



Link Properties

(see paragraph *Modelling and Loading*)

MODELLING AND LOADING

Each model consists of two coincident joints connected by a link element. The link element is defined with the following properties: for F_1 , F_2 , F_3 , M_1 and M_2 degrees of freedom a linear symmetric response curve is chosen with a value of stiffness k_0 equal to $1.00E+012$ kN/m, whereas for M_3 degree-of-freedom a modified Richard-Abbott response curve is chosen with the properties summarized in the tables below:

Table 4.13. Modified Richard-Abbott response curve parameters (ascending branches)

PARAMETERS	TEST N.		
	2 (Bursi)	3 (Korol)	7 (Broderick)
K_a [kN-m/rad]	35000	63751	3550
M_a [kN-m]	130	400	50
K_{pa} [kN-m/rad]	1000	1	150
n_a	1	1	1
K_{ap} [kN-m/rad]	35000	0	3550
M_{ap} [kN-m]	80	0	5
K_{pap} [kN-m/rad]	1000	0	100
n_{ap}	1	0	1
t_{1a}	1	0	20
t_{2a}	0.3	0	0.3
C_a	1	0	1
i_{Ka}	10	30	0
i_{Ma}	0	0.03	0
H_a	0.2	0	0
E_{maxa} [rad]	0.1	0.1	0.1

Table 4.14. Modified Richard-Abbott response curve parameters (descending branches)

PARAMETERS	TEST N.		
	2 (Bursi)	3 (Korol)	7 (Broderick)
K_d [kN-m/rad]	35000	63751	4550
M_d [kN-m]	120	400	60
K_{pd} [kN-m/rad]	1000	1	100
N_d	1	1	1
K_{dp} [kN-m/rad]	35000	0	4550
M_{dp} [kN-m]	80	0	5
K_{pdp} [kN-m/rad]	1000	0	100
n_{dp}	1	0	1
t_{1d}	1	0	20
t_{2d}	0.3	0	0.3
C_d	1	0	1
i_{Kd}	10	30	0
i_{Md}	0	0.03	0
H_d	0.2	0	0
E_{maxd} [rad]	0.1	0.1	0.1

In order to run a nonlinear static time-history analysis, a time-history curve is loaded in the “Time-history Curves” dialog box. It is defined as follows:

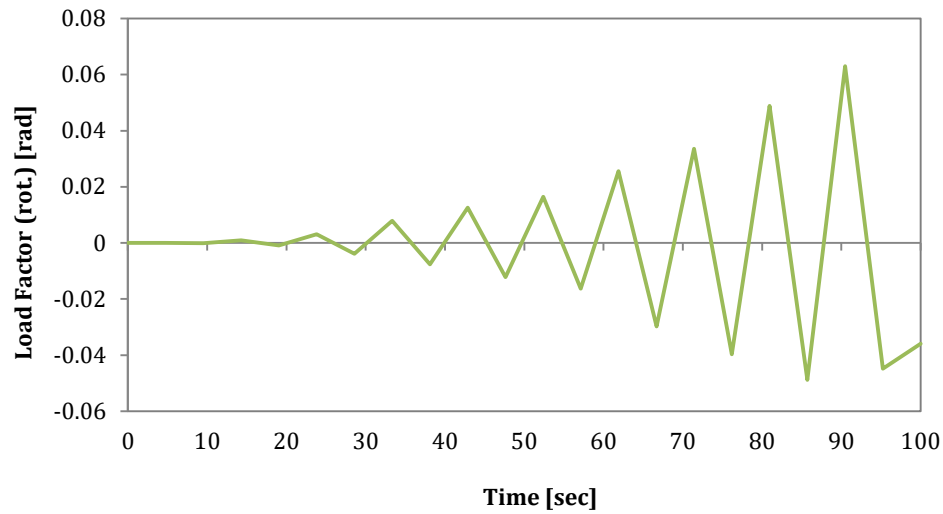


Figure 4.74. Cyclic loading (Ch4-joint-05.spf)

The time step for the static analysis is selected as 0.01 s, coincident with the input record sampling time step.

A static time-history load, in terms of displacements, is imposed at the node 2 in the RY direction.

The FE model is shown below:

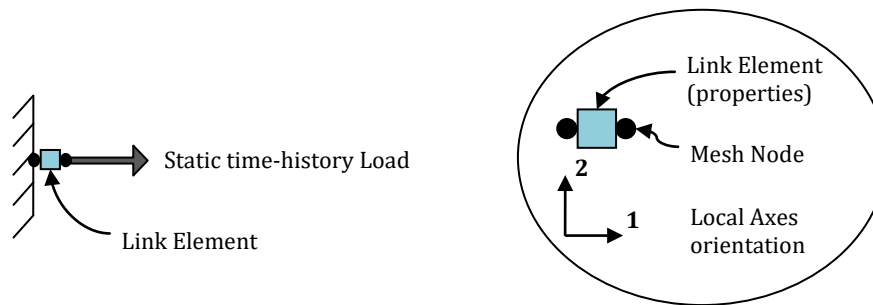


Figure 4.75. FE model of steel joints

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparisons between numerical and experimental results is shown hereafter:

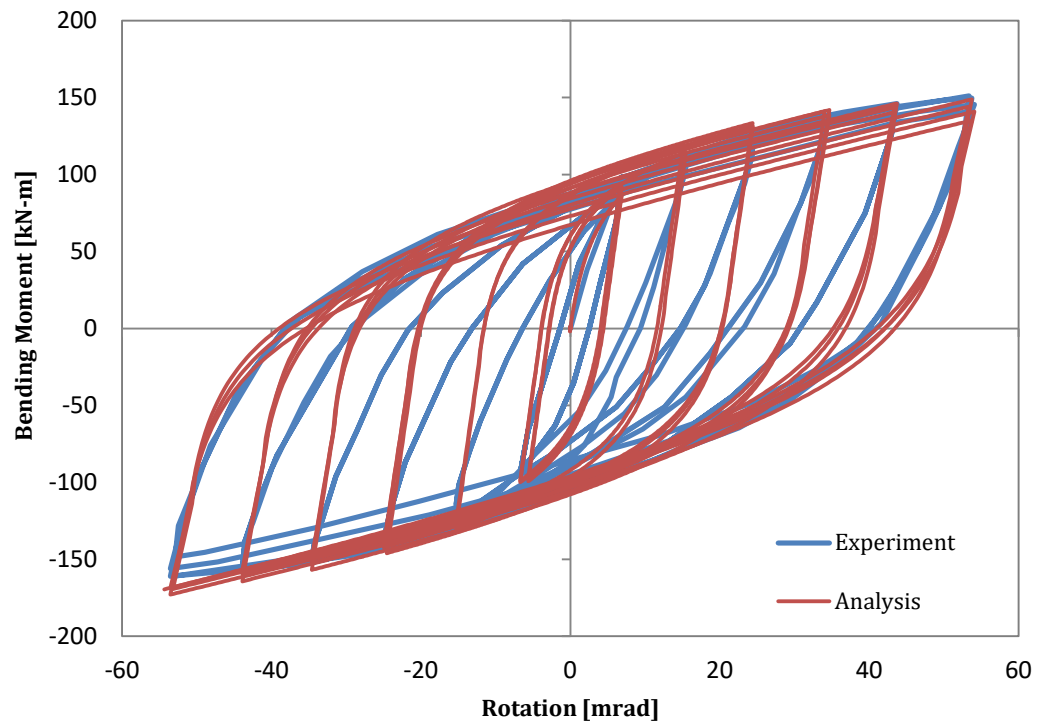


Figure 4.76. Experimental vs. Analytical results – moment-rotation (Bursi et al.)

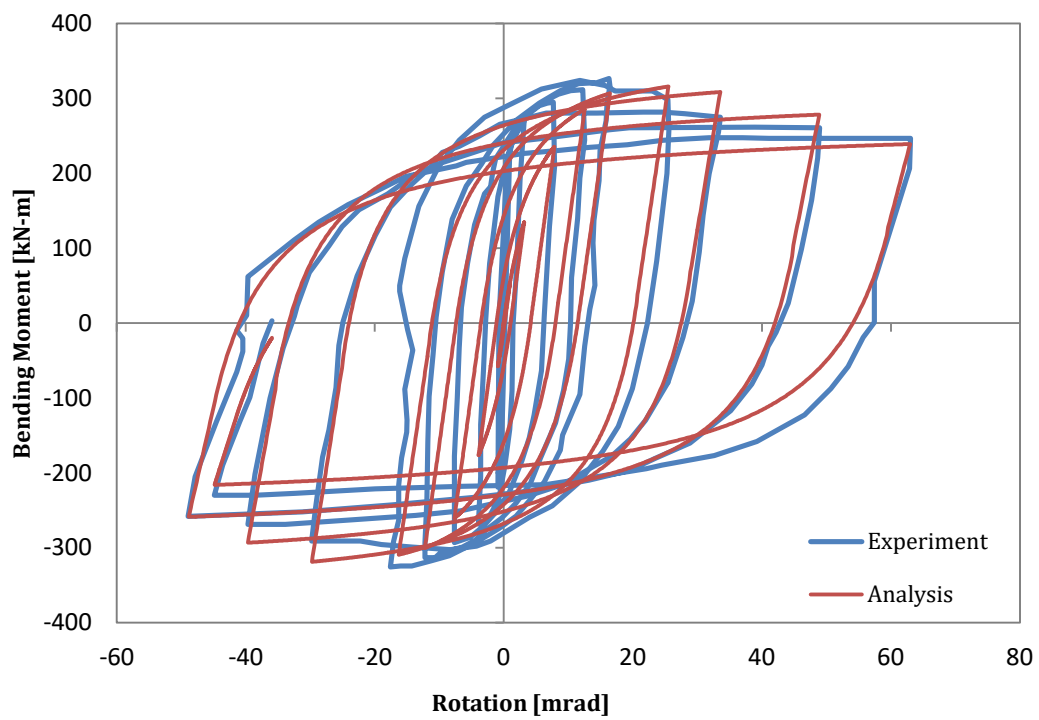


Figure 4.77. Experimental vs. Analytical results – moment-rotation (Korol et al.)

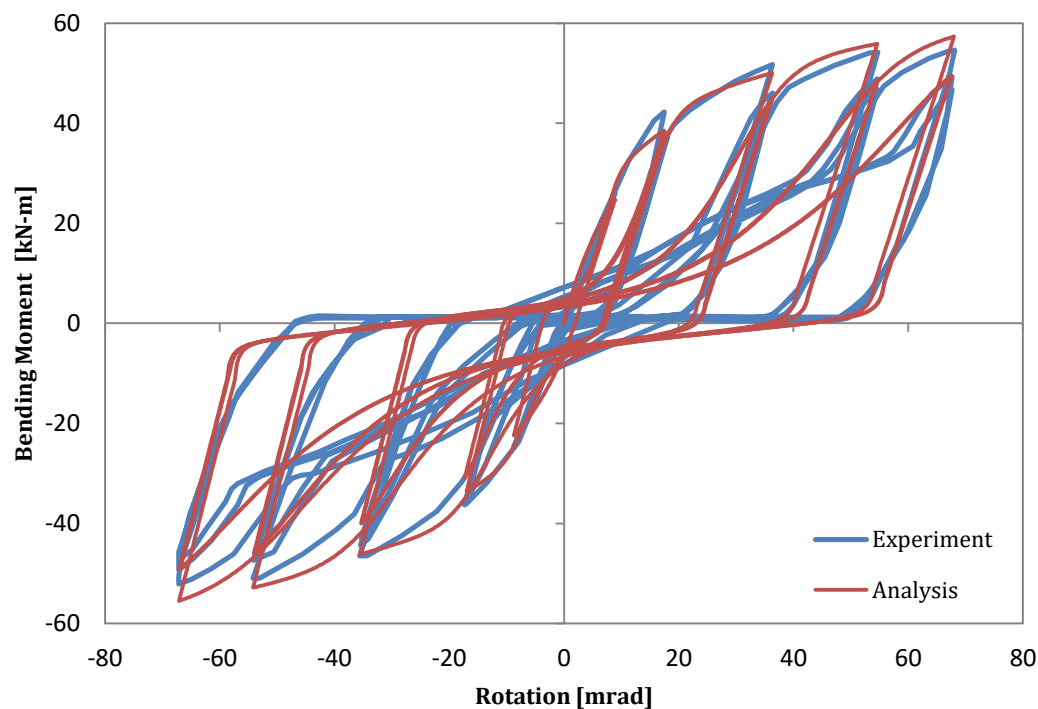


Figure 4.78. Experimental vs. Analytical results – moment-rotation (Broderick et al.)

COMPUTER FILES

- Ch4-joint-04.spf (→ Bursi et al. (2002))
- Ch4-joint-05.spf (→ Korol et al. (1990))
- Ch4-joint-06.spf (→ Broderick et al. (2002))

REFERENCES

- Bursi O.S., Ferrario F., Fontanari V. [2002] “Non-linear analysis of the low-cycle fracture behaviour of isolated Tee stub connections”, Computers & Structures, Vol. 80, pp. 2333-2360.
- Broderick B.M., Thomson A.W. [2002] “The response of flush joints under earthquake loading”, Journal of Constructional Steel Research, Vol. 8, pp. 1161-1175.
- Korol R.M., Ghobarah A., Osman A. [1990] “Extended End-Plate Connections Under Cyclic Loading: Behaviour and Design”, Journal of Constructional Steel Research, Vol. 16, pp. 253-280.
- Nogueiro P., Simões da Silva L., Bento R., Simões R. [2007] “Numerical implementation and calibration of a hysteretic model with pinching for the cyclic response of steel joints”, International Journal of Advanced Steel Construction, Vol. 3, No. 1, pp. 459-484.

RC BRIDGES

EXAMPLE 1 – Multi-span continuous deck bridge

DESCRIPTION

This example describes the modelling of a bridge, composed of three piers of different height and a continuous deck with four identical spans. The input ground motion is represented by an adequately scaled accelerogram. Two pseudo-dynamic tests have been performed on the structure at the European Joint Research Centre of Ispra (Italy): one with the input motion corresponding to the design earthquake and another defined on the basis of the estimated ultimate capacity of the bridge (1.2 times the design earthquake).

The analytical results observed when the bridge is subjected to the second and stronger earthquake input motion, in terms of displacements and forces at the top of pier 1, are compared with the experimental results. The FE model is defined in the X-Z plane.

NOTE: Detailed information about the geometry and material properties may be found in Casarotti and Pinho (2006).

GEOMETRY AND PROPERTIES

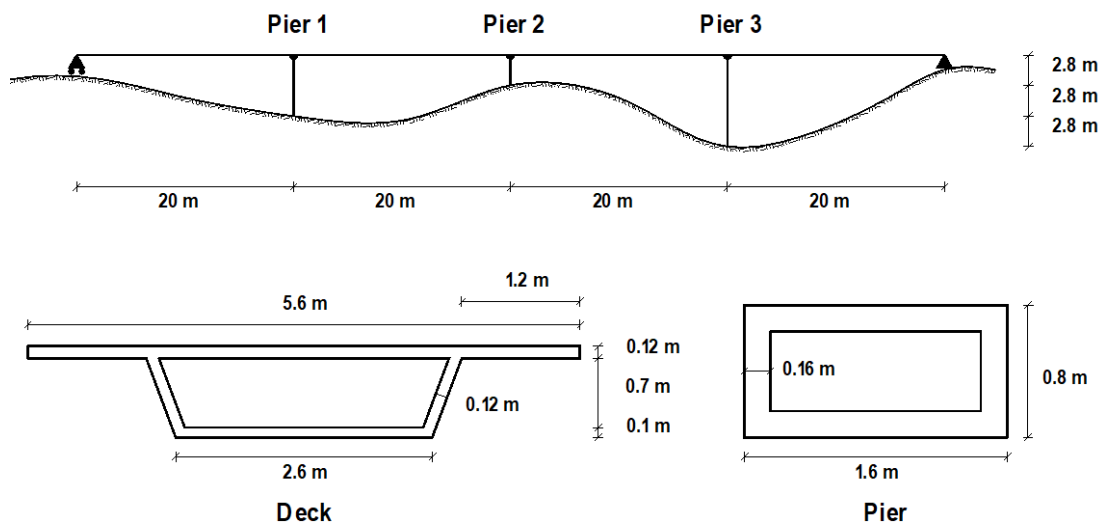


Figure 4.79. Bridge configuration and member cross sections [Casarotti and Pinho, 2006]

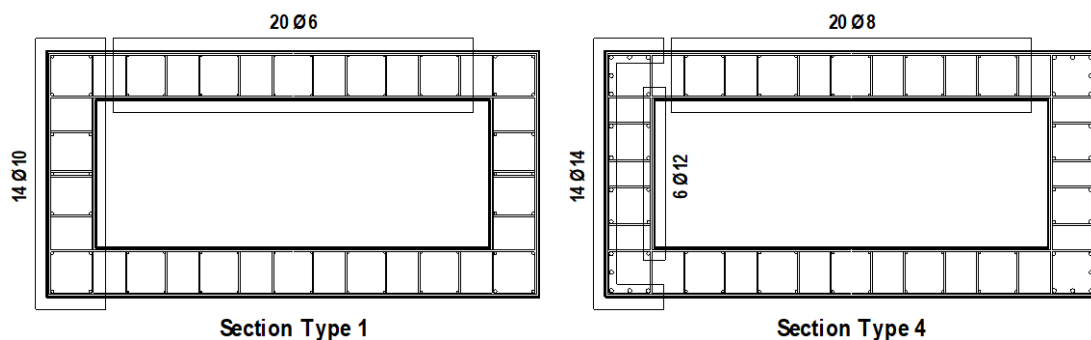


Figure 4.80. Reinforcement layout [Casarotti and Pinho, 2006]

Structural Geometry

The model consists of three RC piers (rectangular hollow section of 0.8 m x 1.6 m, with a wall width of 0.16 m), and a deck (hollow-core pre-stressed concrete girder 5.6 m wide) with the properties described hereafter.

Material Properties of the piers

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters are listed below:

- (concrete Pier 1) $f_c = 31450$ kPa; $f_t = 0$ kPa; $\epsilon_c = 0.002$ m/m
- (concrete Pier 2) $f_c = 35000$ kPa; $f_t = 0$ kPa; $\epsilon_c = 0.002$ m/m
- (concrete Pier 3) $f_c = 42920$ kPa; $f_t = 0$ kPa; $\epsilon_c = 0.002$ m/m

The Menegotto-Pinto steel model is employed for defining the steel materials with the following properties:

- (steel Pier 1) $E_s = 2.0600E+008$ kPa; $f_y = 496000$ kPa; $\mu = 0.0036$
- (steel Pier 2) $E_s = 2.0600E+008$ kPa; $f_y = 468000$ kPa; $\mu = 0.0027$
- (steel Pier 3) $E_s = 2.0600E+008$ kPa; $f_y = 496000$ kPa; $\mu = 0.0036$

MODELLING AND LOADING

The piers are modelled through 3D force-based inelastic frame elements (infrmFB) with 5 integration sections. The number of fibres used in section equilibrium computations is set to 200.

As the deck is assumed to behave elastically, it is modelled with a 3D elastic nonlinear beam-column element (elfrm) with the following properties: $EA = 2.7837E+007$ [kN]; EI (axis 2) = 2590000 [kN-m²]; EI (axis 3) = 4.2400E+007 [kN-m²]; $GJ = 2.3989E+007$ [kN-m²]. Each span of the deck is discretized with four elements, of length equal to 10%, 40%, 40% and 10% of the span.

Rigid connections are also modelled. The rigid arm, connecting the top of the pier and the centre of gravity of the deck, is constituted by an elastic element (elfrm) with the following properties: $EA = 1.0000E+008$ [kN]; EI (axis2) = 1.0000E+010 [kN-m²]; EI (axis3) = 1.0000E+010 [kN-m²]; $GJ = 1.0000E+010$ [kN-m²], whilst the connection between the base of the rigid arm and the top of the pier is modelled as a hinge by introducing a link element, which is set as a spring with infinite stiffness in the vertical and transversal direction, and fully flexible in all the other four degrees of freedom.

The pier masses are defined as lumped and are concentrated at the top of each pier, whilst the deck mass is considered both distributed at each deck span and lumped at the top of each pier; the employed values are summarized in the table below.

Table 4.15. Lumped and distributed masses applied to the bridge

Lumped masses (ton)				Distributed Mass (ton/m)
Pier 1	Pier 2	Pier 3	Deck (per pier)	Deck
9.3184	4.6592	13.9776	56	3.875 (abutments) 1.091 (others)

The base nodes are fully restrained.

In order to run a nonlinear dynamic time-history analysis, a time-history curve, including the 10 s intervals with no acceleration (needed to damp out the structure motion after the first earthquake run), is loaded in the “Time-history Curves” dialog box. It is defined as follows:

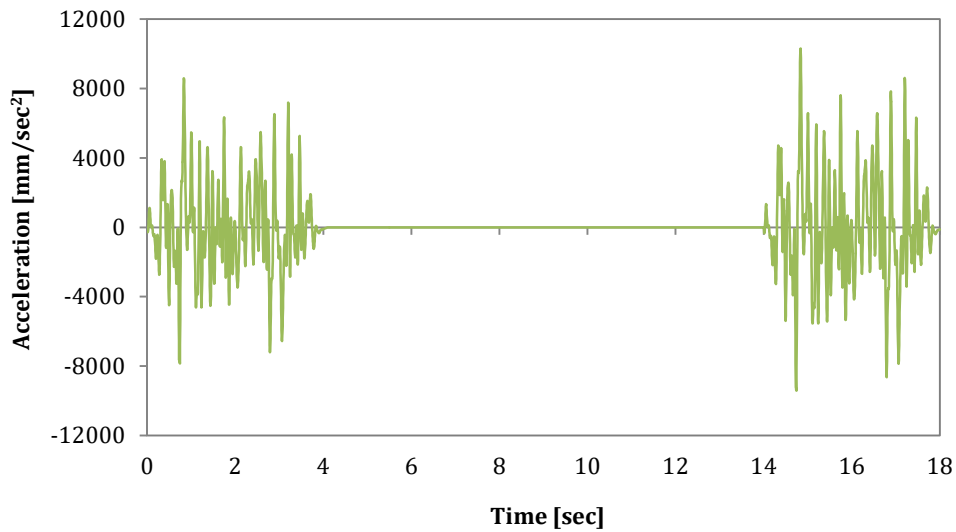


Figure 4.81. Input ground motion, design earthquake (Guedes, 1997)

The time step for the dynamic analysis is selected as 0.004 s (coincident with the input record sampling time step).

Permanent loads of 2450 kN are applied at the top of each pier in the negative Z direction, whilst dynamic time-history loads are imposed at the pier bases and at the abutments in the Y direction.

A sketch of the FE model is presented in the following:

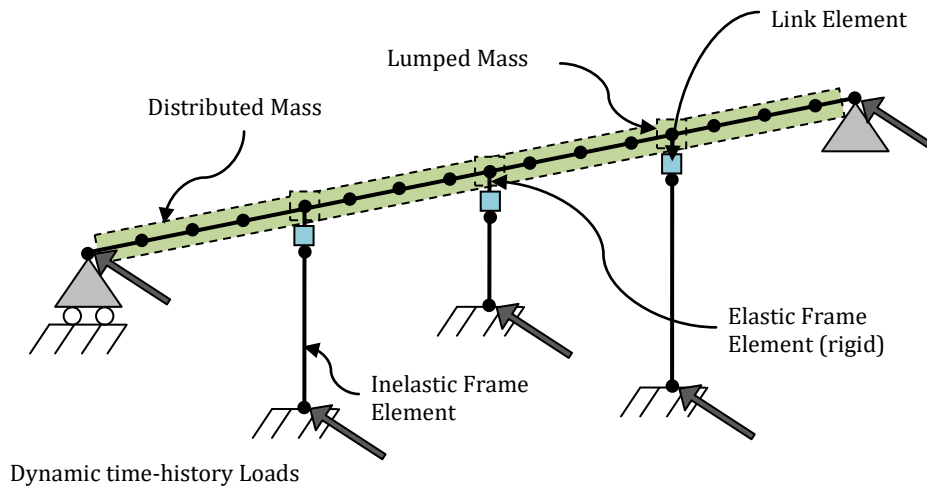


Figure 4.82. Details of the FE model of the large-scale RC viaduct

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results, in terms of displacements and forces at the top of the medium pier (pier 1) observed when the bridge was subjected to the second and stronger earthquake input motion, is shown hereafter:

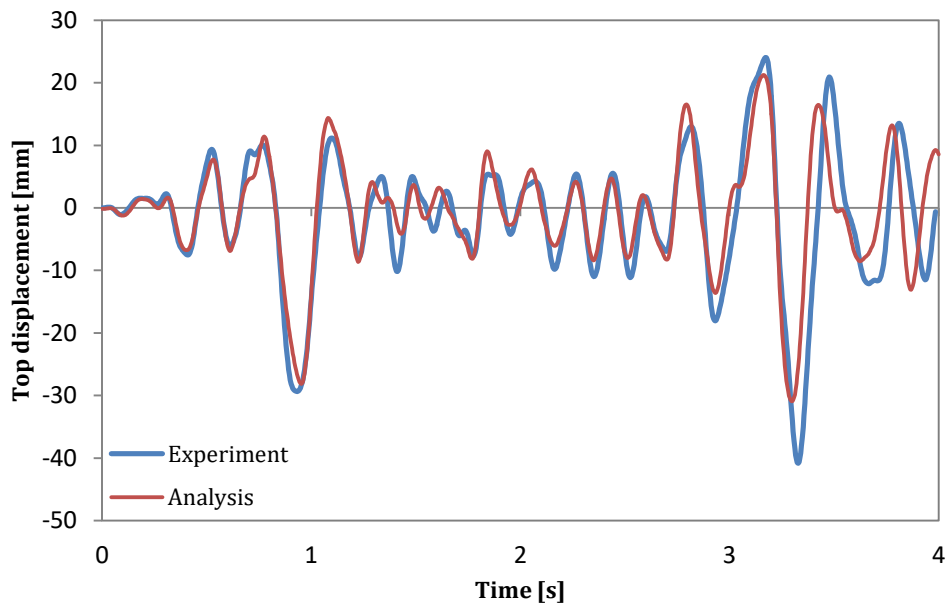


Figure 4.83. Experimental vs. Analytical results – pier 1 top displacement-time

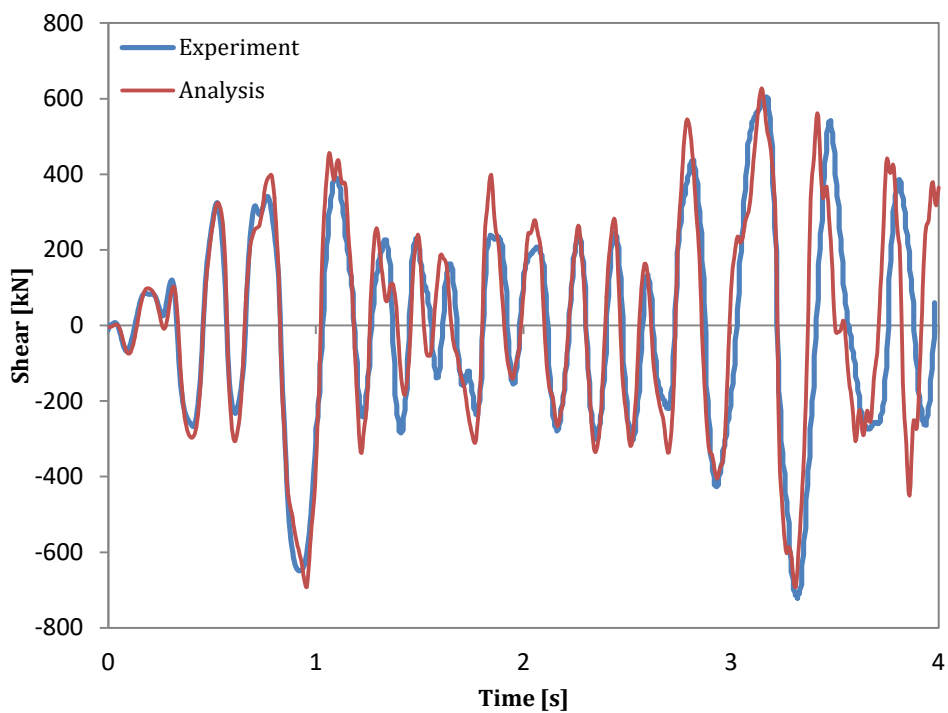


Figure 4.84. Experimental vs. Analytical results – pier 1 shear-time

COMPUTER FILE

- Ch4-bridge-01.spf

REFERENCES

- Casarotti C., Pinho R. [2006] “*Seismic response of continuous span bridges through fibre-based finite element analysis*”, Journal of Earthquake Engineering and Engineering Vibration, Vol. 5, No. 1, pp. 119-131.

EXAMPLE 2 – Scaled bridge pier**DESCRIPTION**

This example describes the modelling of a scaled standard RC hollow-section cantilever bridge pier (scaling factor equal to 4). The specimen was tested under dynamic loading at the EUCENTRE Foundation in Pavia (Italy). The input ground motion is represented by an adequately scaled accelerogram with duration of 40 seconds.

The analytical results, obtained with the FE analysis program SeismoStruct in terms of displacements at the top of the pier, are compared with the experimental results. The FE model is defined in the X-Z plane.

NOTE: Detailed information about the geometry and material properties may be found in Petrini et al. (2008).



Figure 4.85. Bridge pier tested at the EUCENTRE

GEOMETRY AND PROPERTIESStructural Geometry

The model consists of a RC bridge pier (circular hollow section with an outer diameter of 0.45 m and an inner stirrup diameter of 0.3775 m) with the properties described hereafter.

Material Properties of the pier

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters are listed below:

- $f_c = 39440 \text{ kPa}$; $f_t = 0.001 \text{ kPa}$; $\epsilon_c = 0.0022 \text{ m/m}$

The Menegotto-Pinto steel model is employed for defining the steel material as follows:

- $E_s = 2.0000E+008$ kPa; $f_y = 514000$ kPa; $\mu = 0.004$

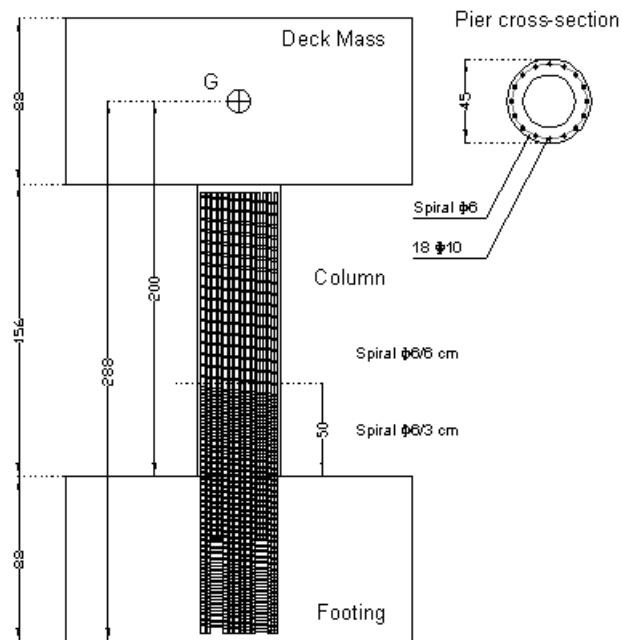


Figure 4.86. Bridge pier specimen configuration and pier cross section [Petrini et al., 2008]

MODELLING AND LOADING

The whole pier is modelled through three elements. In particular, the yielding penetration in the footing is modelled by using an elastic beam-column element (elfrm), 11 mm long, at the bottom with the following properties: $EA = 4294164$ [kN]; EI (axis2) = 13770 [kN-m²]; EI (axis3) = 13770 [kN-m²]; $GJ = 99999999$ [kN-m²].

Then, starting from the base, the column is modelled through four 3D displacement-based inelastic frame elements (infrmDB) of different length (300, 200, 500 and 560 mm long, respectively). The number of fibres used in section equilibrium computations is set to 200. An element specific damping is also defined in the same element's dialog box by using a stiffness proportional damping.

Regarding the applied masses, the effective mass of the pier is neglected, whereas a lumped mass of 7.8 ton in the X direction only is concentrated at the top of the column.

In order to run a nonlinear dynamic time-history analysis, the amplified Morgan Hill accelerogram is loaded in the "Time-history Curves" dialog box. It is defined as follows:

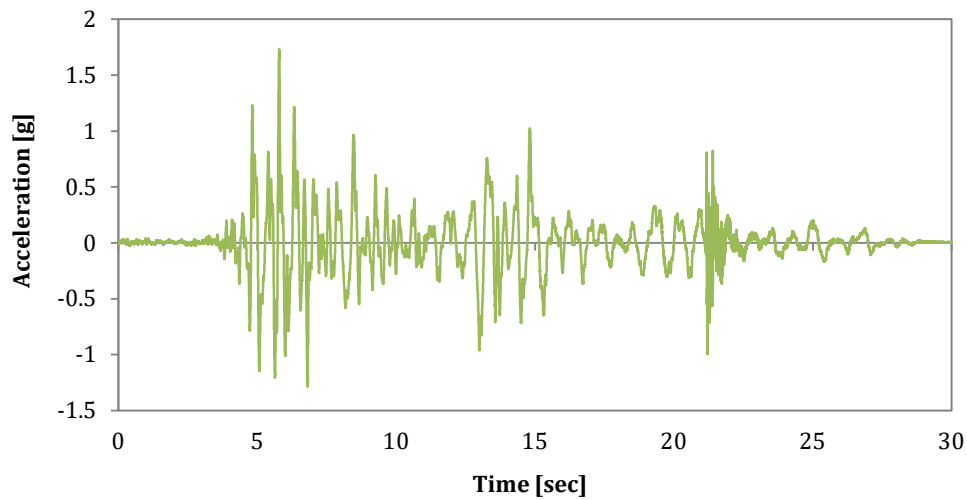


Figure 4.87. Input ground motion (amplified Morgan Hill accelerogram)

The time step for the dynamic analysis is selected as 0.0019501 s.

A permanent load of 76.5 kN is applied at the top of the pier in the negative Z direction, whilst a dynamic time-history load is imposed at the pier base in the X direction.

A sketch of the FE model is presented in the following:

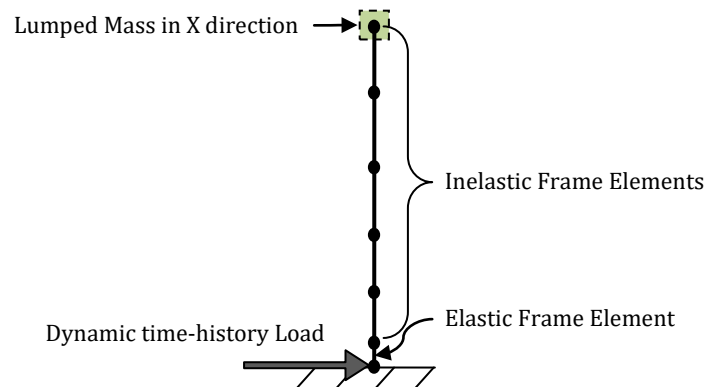


Figure 4.88. FE model of the bridge pier

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

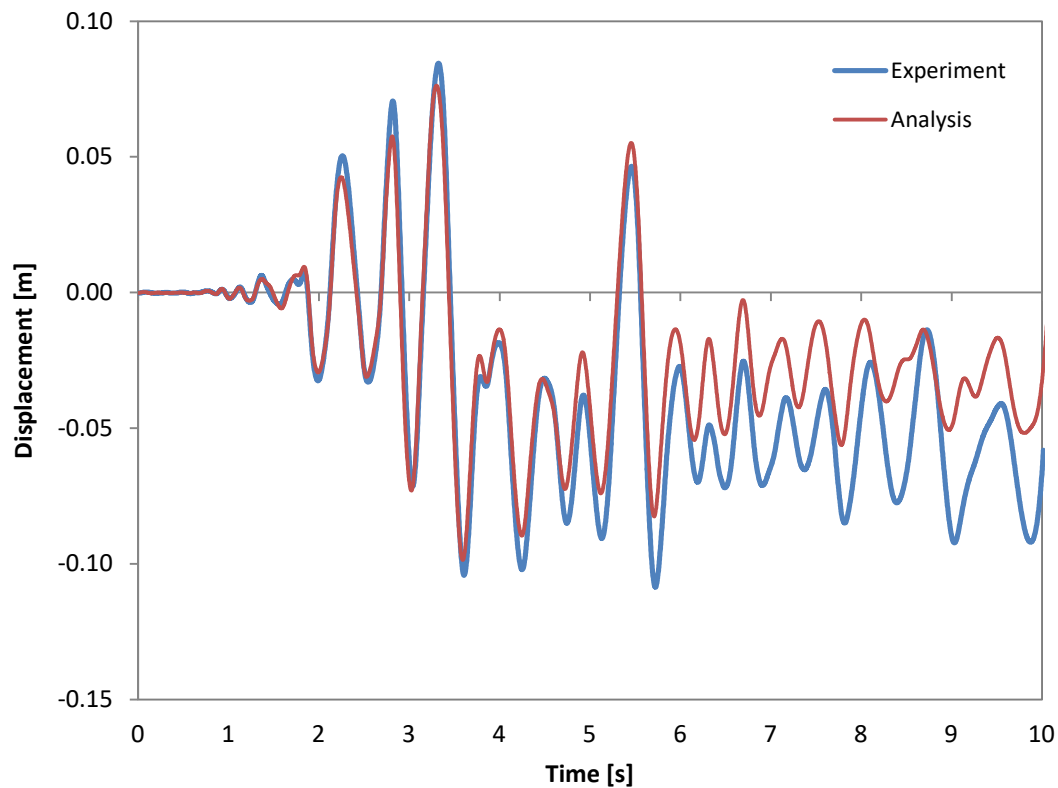


Figure 4.89. Experimental vs. Analytical results – displacement-time

COMPUTER FILE

- Ch4-bridge-02.spf

NOTE

SeismoStruct's prediction of the experimental response is successful up until 5.5 s, at which point the most tensioned longitudinal rebars fractured, leading to a residual displacement level that is underestimated by the software.

REFERENCES

- Petrini L., Maggi C., Priestley M.J.N., Calvi G.M. [2008] "*Experimental verification of viscous damping modelling for inelastic time-history analysis*", Journal of Earthquake Engineering, Vol. 12, Special Issue 1.

BLIND PREDICTION CONTEST SUCCESSES

EXAMPLE 1 – Full-scale bridge column

DESCRIPTION

This example describes the modelling of a full-scale reinforced concrete bridge column. The specimen was tested on the NEES Large High-Performance Outdoor Shake Table at UCSD's Englekirk Structural Engineering Center under dynamic conditions, as part of a blind prediction contest (http://nisee2.berkeley.edu/peer/prediction_contest). Six uniaxial earthquake ground motions, starting with low-intensity shaking, were increased so as to bring the pier progressively to near-collapse conditions.

SeismoStruct was employed by the winner in the Practitioners category, and was also used by two other teams that received an 'Award of Excellence', amongst a total of 41 entries.

The analytical results obtained with SeismoStruct in terms of displacements at the top of the column and base shear are herein compared with the experimental observations. The FE model is defined in the X-Z plane.

NOTE: Detailed information about geometry and material properties may be found on the competition's website (http://nisee2.berkeley.edu/peer/prediction_contest/) and in Bianchi et al. (2011).

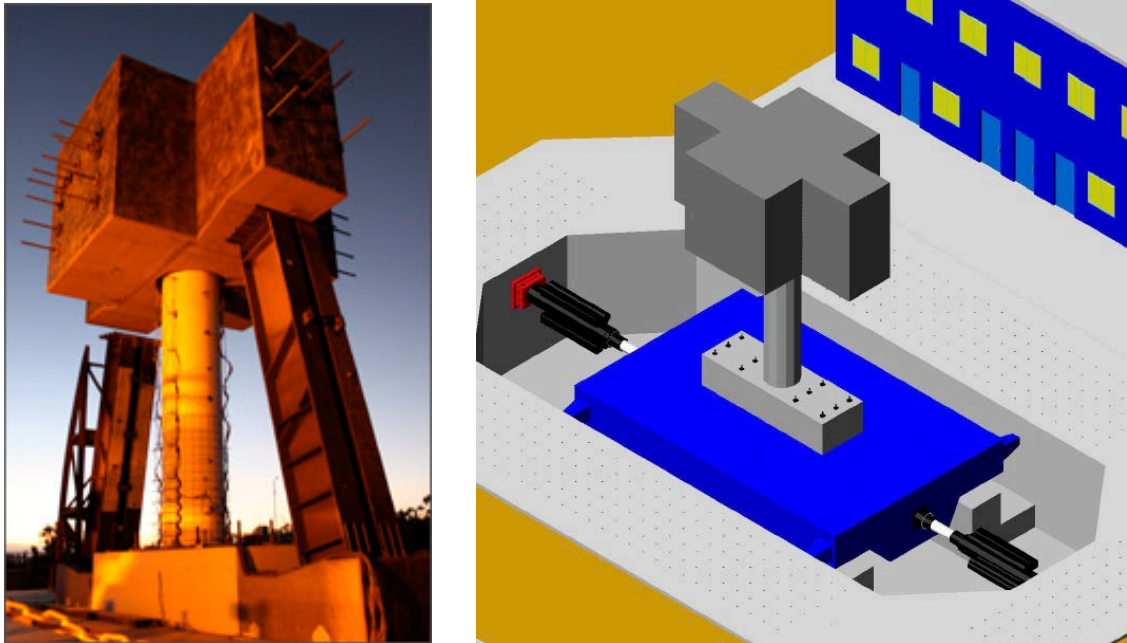


Figure 4.90. Full-scale reinforced concrete bridge column tested on the NEES Large High-Performance Outdoor Shake Table at UCSD's Englekirk Structural Engineering Center

GEOMETRY AND PROPERTIES

Structural Geometry

The model consists of a 1.22 m diameter cantilevered RC circular column with the properties described hereafter.

Material Properties of the column

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters are listed below:

- $f_c = 41500 \text{ kPa}$; $f_t = 0$; $\varepsilon_c = 0.0028 \text{ m/m}$; $\gamma = 23.6 \text{ kN/m}^3$

The Menegotto-Pinto steel model is employed for defining the steel material as follows:

- $E_s = 2.0000\text{E}+008 \text{ kPa}$; $f_y = 518500 \text{ kPa}$; $\mu = 0.008$; $\gamma = 77 \text{ kN/m}^3$

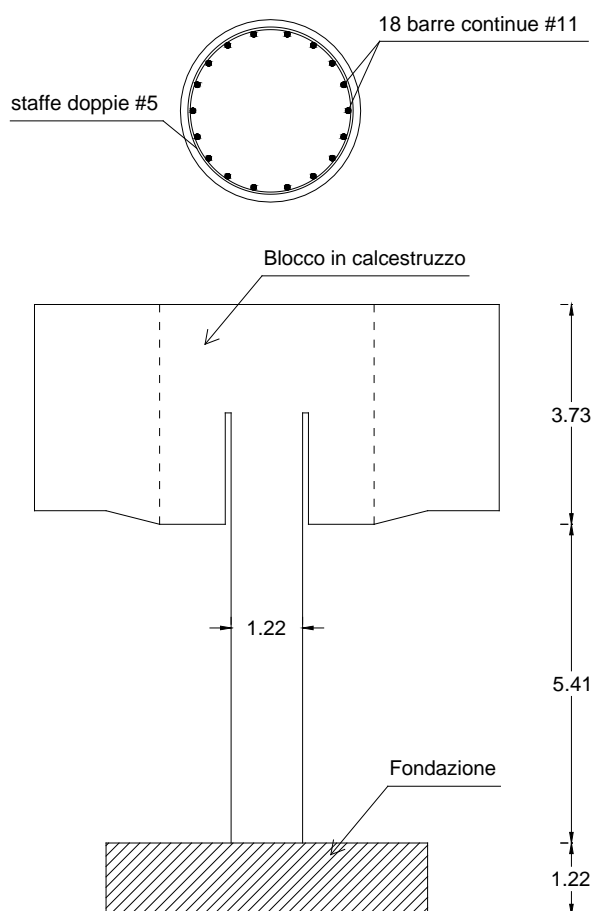


Figure 4.91. Pier cross section and bridge pier specimen configuration [Bianchi et al., 2011]

MODELLING AND LOADING

The column is modelled through a 3D force-based inelastic frame element (infrmFB), where the number of fibres used in section equilibrium computations is set to 300.

Regarding the applied masses, the effective mass of the pier is considered by assigning the specific weight of the materials in the “Materials” module, whereas a lumped mass of 228 ton is concentrated at the top of the column.

In order to run a nonlinear dynamic analysis, a time-history curve, constituted by six records in series and separated by 10 s intervals with no acceleration, is loaded in the “Time-history Curves” dialog box.

The applied ground motions are shown below:

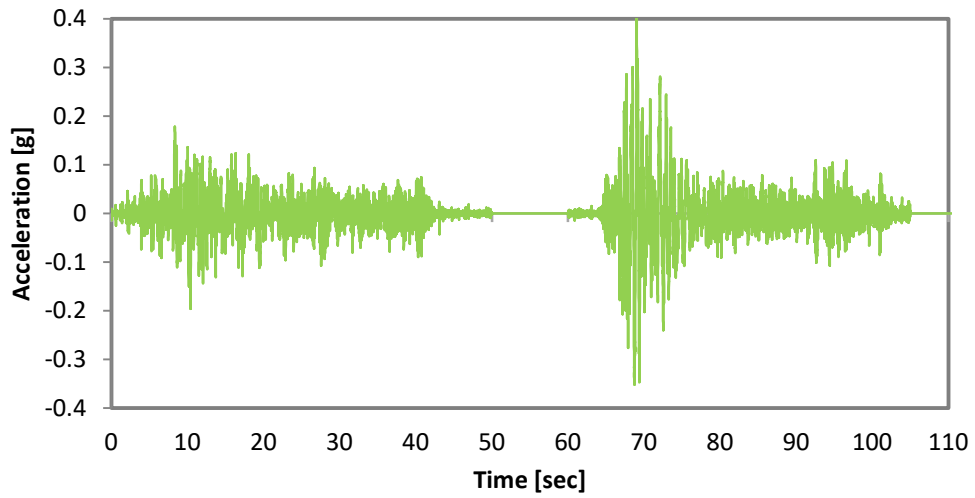


Figure 4.92. Input ground motion (EQ1 and EQ2)

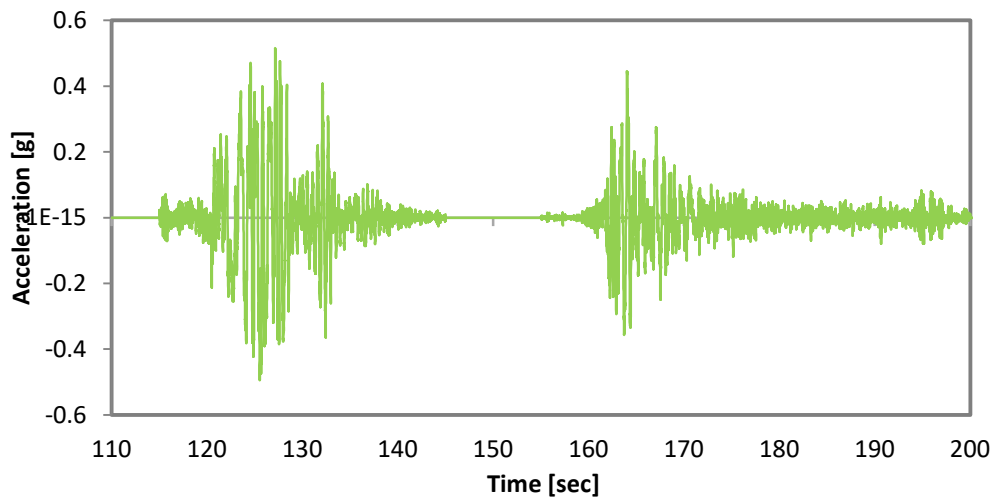


Figure 4.93. Input ground motion (EQ3 and EQ4)

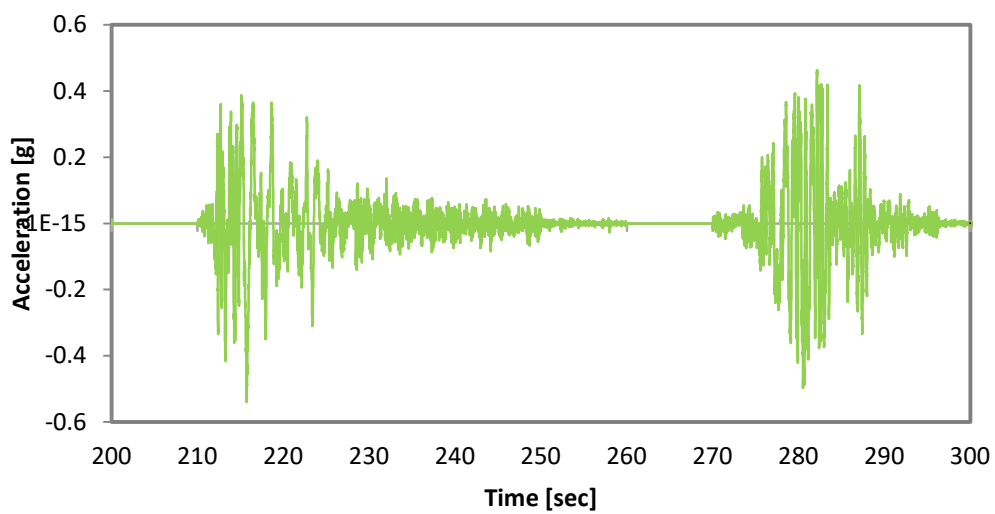


Figure 4.94. Input ground motion (EQ5 and EQ6)

The time step for the dynamic analysis is set as 0.00390625 s.

The dynamic time-history load is imposed at the pier base in the X direction.

NOTE: A 1% tangent stiffness-proportional damping is applied as global damping in the “Project Settings” menu.

A sketch of the FE model is presented in the following:

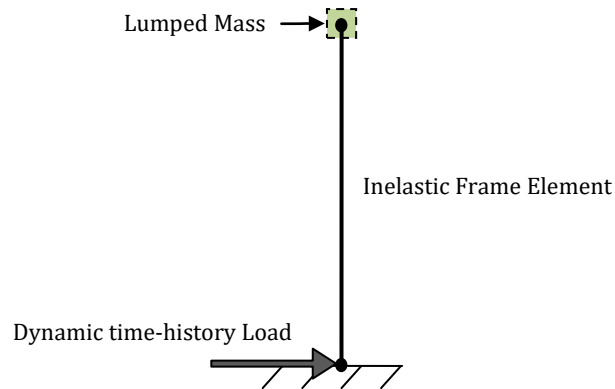


Figure 4.95. FE model of the bridge column

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

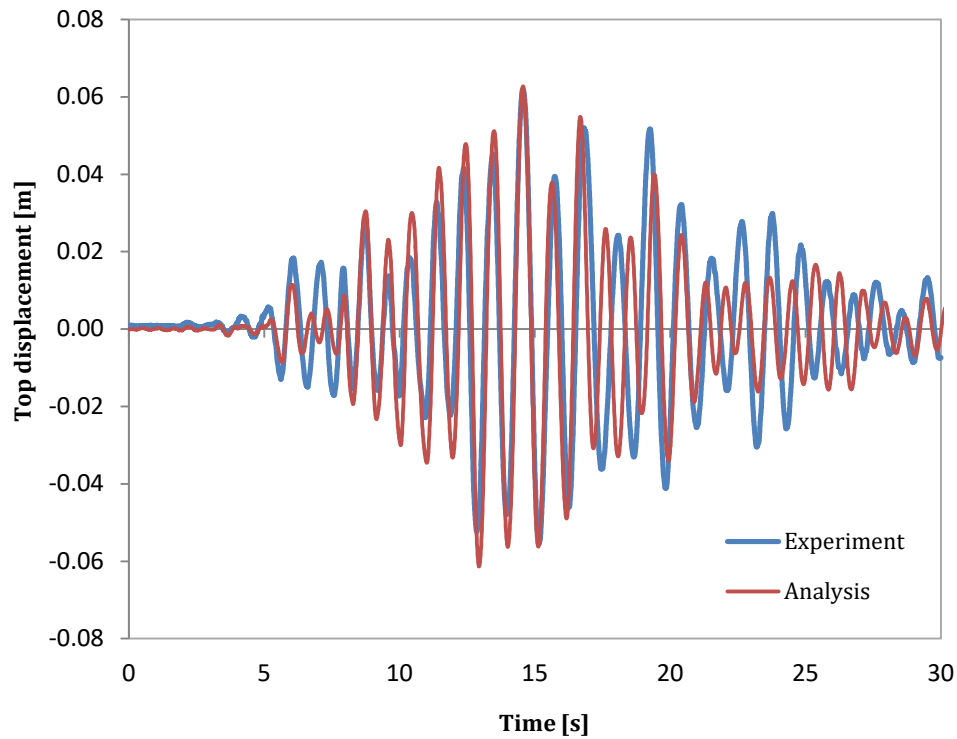


Figure 4.96. Experimental vs. Analytical results – top displacement-time (EQ1)

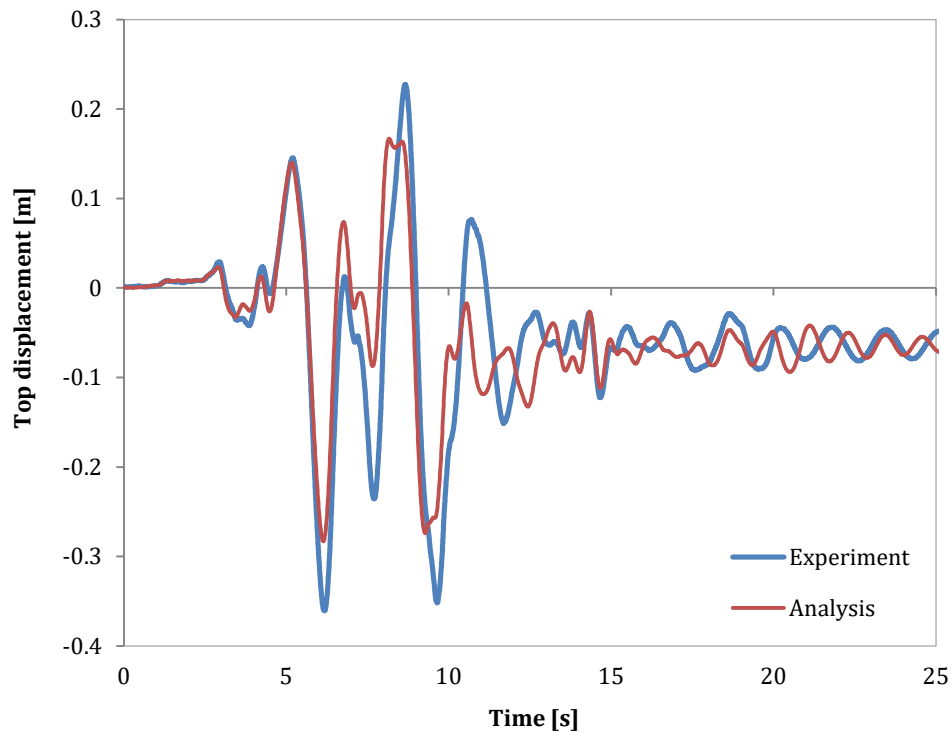


Figure 4.97. Experimental vs. Analytical results – top displacement-time (EQ3)

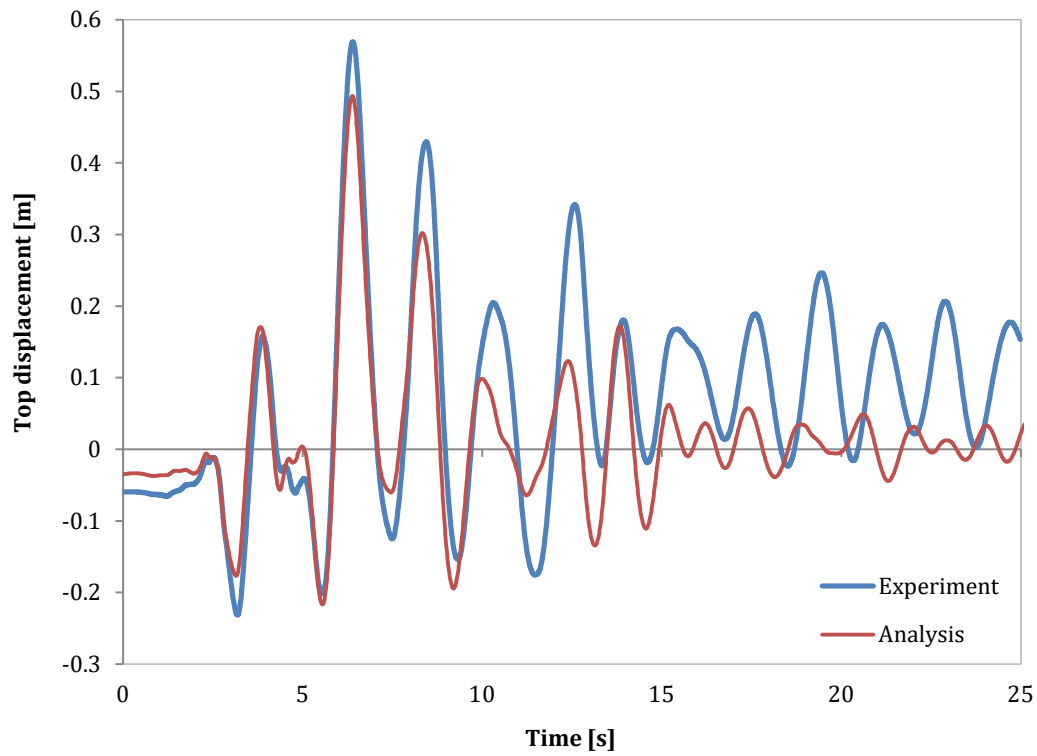


Figure 4.98. Experimental vs. Analytical results - top displacement-time (EQ5)

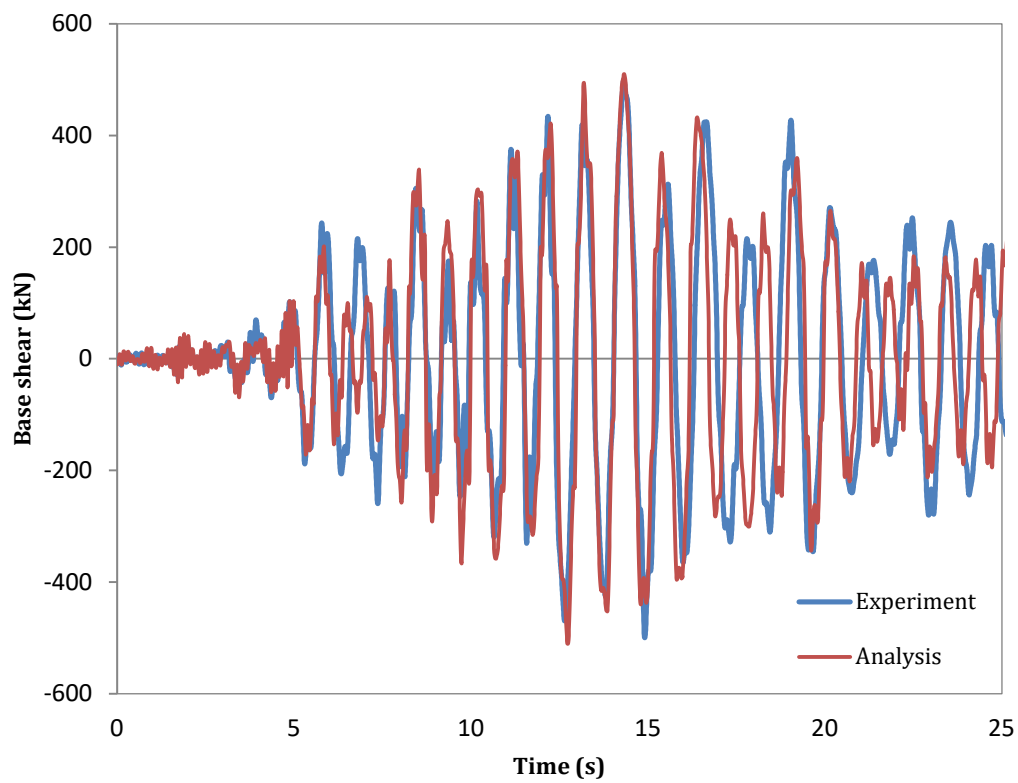


Figure 4.99. Experimental vs. Analytical results - base shear-time (EQ1)

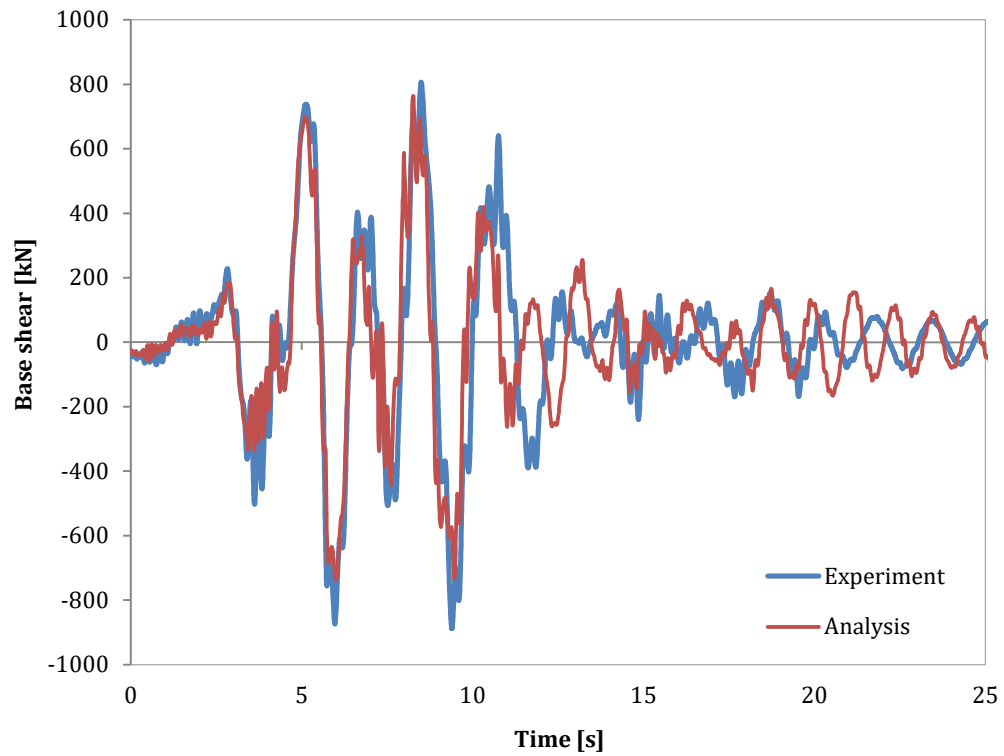


Figure 4.100. Experimental vs. Analytical results – base shear-time (EQ3)

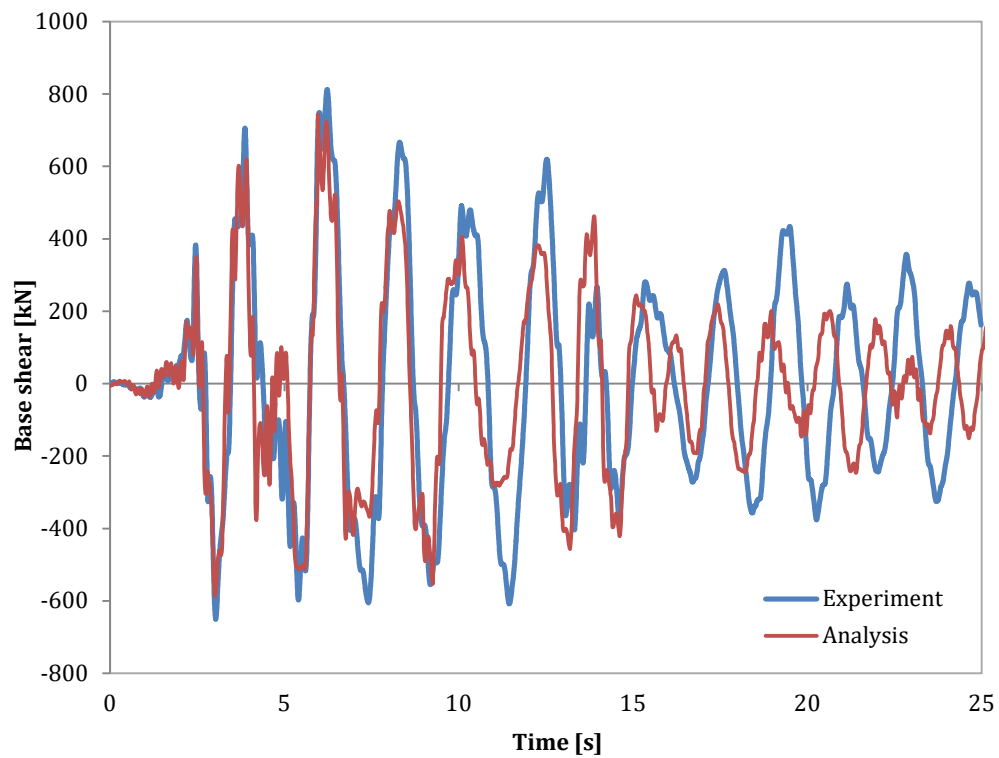


Figure 4.101. Experimental vs. Analytical results – base shear-time (EQ5)

COMPUTER FILE

- Ch4-blind-01.spf

REFERENCES

- Bianchi F., Sousa R., Pinho R. [2011] "*Blind prediction of a full-scale RC bridge column tested under dynamic conditions*", Proceedings of 3rd International Conference on Computational Methods in Structural Dynamics and Earthquake Engineering (COMPDYN 2011), Paper no. 294, Corfu, Greece.

EXAMPLE 2 – Two simple one storey, three-dimensional RC frame structures**DESCRIPTION**

This example describes the modelling of two geometrically identical one-storey, three-dimensional RC frame structures, which were designed for low and high ductility levels according to the EC8 provisions. In each structure the slab does not cover the entire span in one direction and on top of it nine additional masses are placed in a non-symmetrical configuration. Each specimen (i.e. model A and model B, depending on the different steel reinforcement detailing) was tested under dynamic conditions on the LNEC-3D shaking table of Lisbon (Portugal) as part of a Bind Prediction Contest organized at the 15th Word Conference on Earthquake Engineering (15WCEE), by applying four input ground motions of increasing intensity levels in each horizontal direction.

SeismoStruct was employed by the winning team, selected amongst a total of 38 participating entries. The corresponding analytical results are herein compared with the experimental observations. The models are created in the X-Y-Z space.

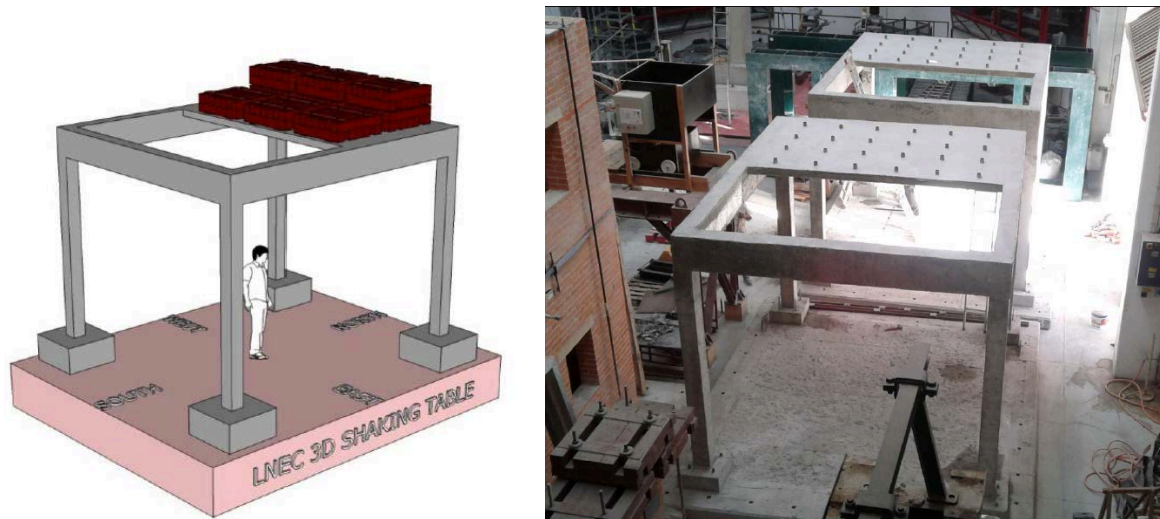


Figure 4.102. One-storey, three dimensional RC frame structures tested on the LNEC-3D shaking table of Lisbon (Portugal) during the 15WCEE [LNEC team, 2012]

GEOMETRY AND PROPERTIESStructural Geometry

Each model consists of four RC square columns (0.2 x 0.2 m) and four rectangular beams (0.2 x 0.4 m). The longitudinal reinforcing details of the structural elements are given in the following tables:

Table 4.16. Longitudinal reinforcing details of columns

MODEL	Column_section
A	0.2x0.2 (8 ϕ 10)
B	

Table 4.17. Longitudinal reinforcing details of beams

MODEL	Beam (1 st and 5 th section)	Beam (2 nd and 4 th section)	Beam (3 rd section)
A	0.2x0.4 (6 ϕ 10 upper, 2 ϕ 12 lower)	0.2x0.4 (3 ϕ 10 upper, 3 ϕ 12 lower)	0.2x0.4 (2 ϕ 10 upper, 3 ϕ 12 lower)
B	0.2x0.4 (4 ϕ 10 in the corners)	0.2x0.4 (4 ϕ 10 in the middle of the section)	0.2x0.4 (4 ϕ 10 in the corners)

Material Properties of the structural elements

The Mander et al. concrete model is employed for defining the concrete materials. The characteristic parameters for model A are listed below:

- Concrete for columns: $f_c = 34000$ kPa; $f_t = 4000$; $\varepsilon_c = 0.002$ m/m $\gamma = 24$ kN/m³
- Concrete for beams: $f_c = 28500$ kPa; $f_t = 3500$; $\varepsilon_c = 0.002$ m/m $\gamma = 24$ kN/m³

NOTE: The characteristic parameters for model B are the same except for the confinement factors.

The Menegotto-Pinto steel model is employed for defining the steel material. The characteristic parameters for model A are listed below:

- $E_s = 2.0000E+008$ kPa; $f_y = 560000$ kPa; $\mu = 0.003$; $\gamma = 78$ kN/m³

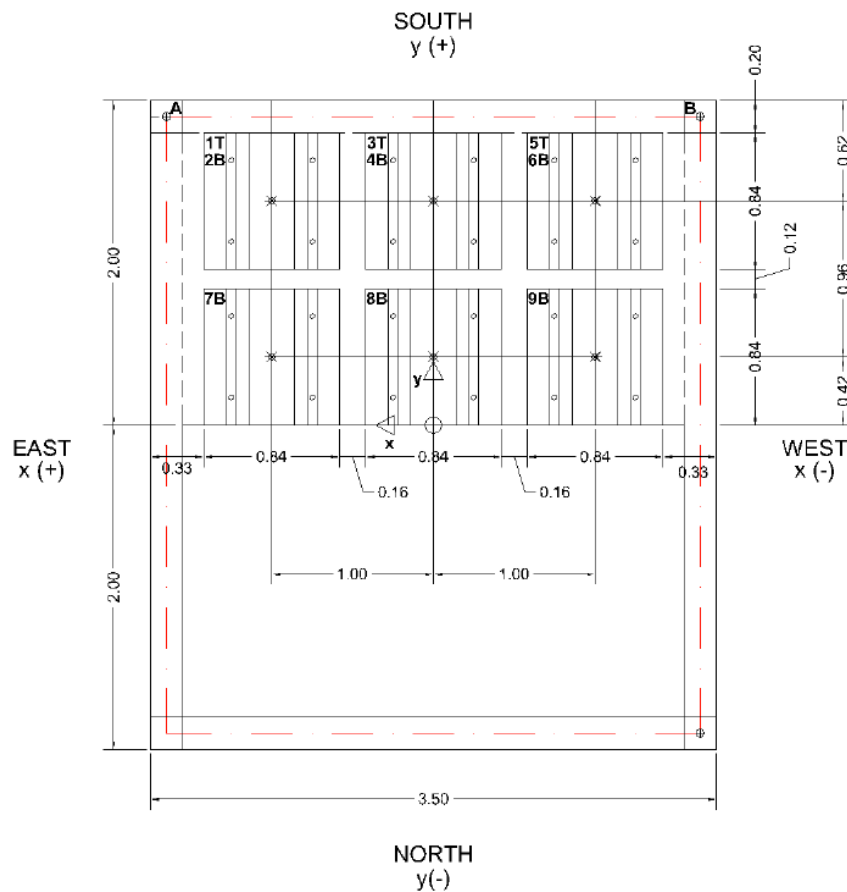


Figure 4.103. Plan view of the prototype model with the localization of the masses [LNEC team, 2012]

MODELLING AND LOADING

Columns and beams are modelled through 3D inelastic force-based frame elements (infrmFB) with 3÷5 integration sections depending on the element subdivision (refer to the SeismoStruct models for details). The number of fibres used in section equilibrium computations is set to 200.

Regarding the mass distribution, the additional masses on top of the slab are applied in a lumped fashion, considering also the rotational inertia. For the exact values, please refer to the SeismoStruct frame model.

The additional masses are connected to each other and to the adjacent beams through the use of elastic frame elements (elfrm) with the section properties of the slab.

All the foundation nodes are considered as fully restrained against rotations and translations.

NOTE: In each model the height of the columns is slightly greater in order to model the strain-penetration length.

In order to run a nonlinear dynamic analysis, two time-history curves are loaded in the “Time-history Curves” dialog box, one for each direction (NS and EW components). Each curve includes a 10 s interval with no acceleration (needed to damp out the structure motion after each earthquake run). The applied ground motions are shown below:

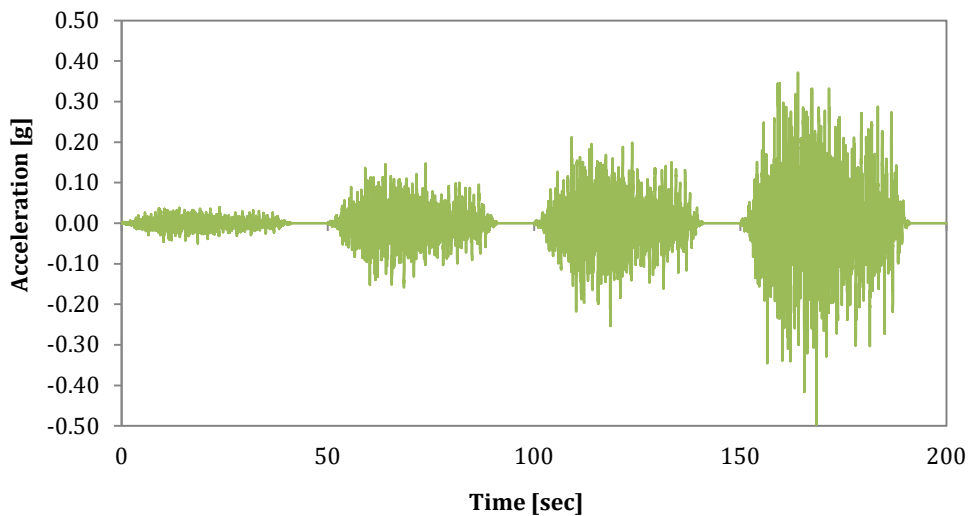


Figure 4.104. Input ground motion in the X direction (East-West component)

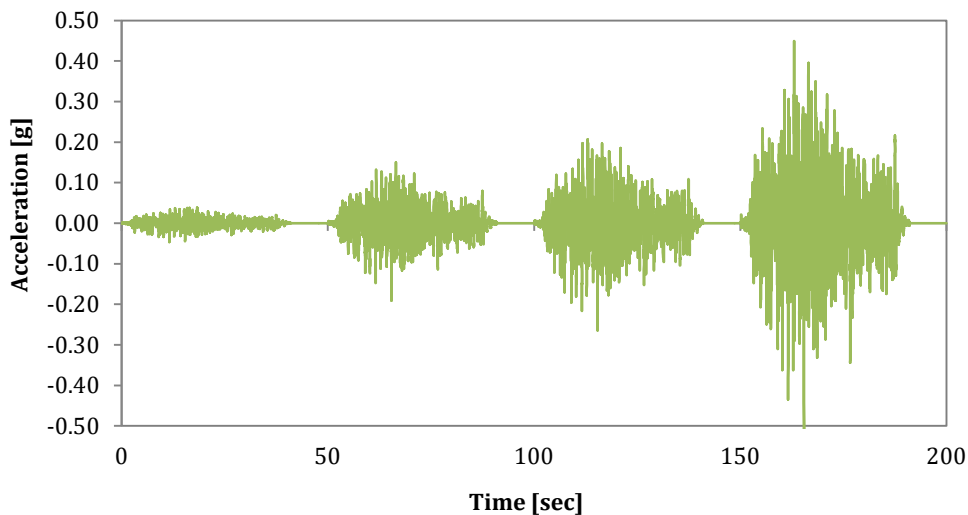


Figure 4.105. Input ground motion in the Y direction (North-South component)

The time step for the dynamic time-history analysis is set to 0.005 s.

Dynamic time-history loads are applied at the base nodes, in terms of accelerations in the X and Y directions.

NOTE: A 1.5% mass-proportional damping is applied as global damping in the “Project Settings” menu.

The FE model of the building is presented below:

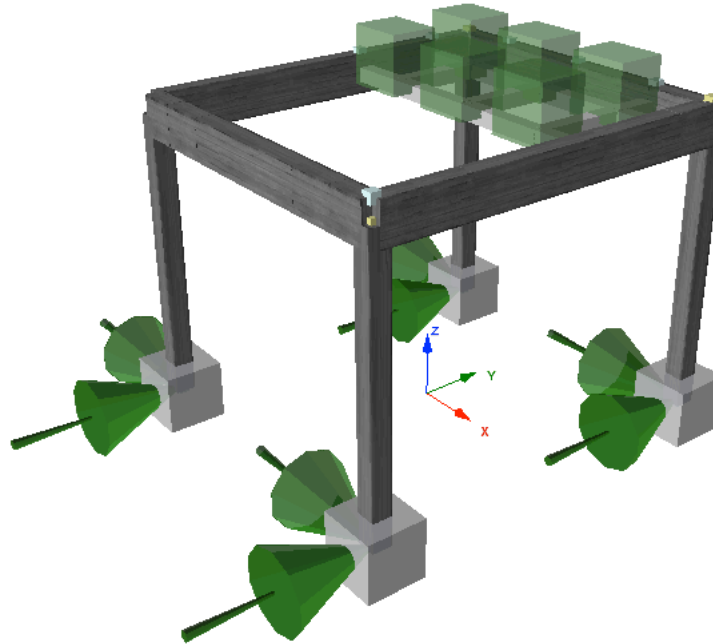


Figure 4.106. FE model of the structure

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results for model A is shown hereafter:

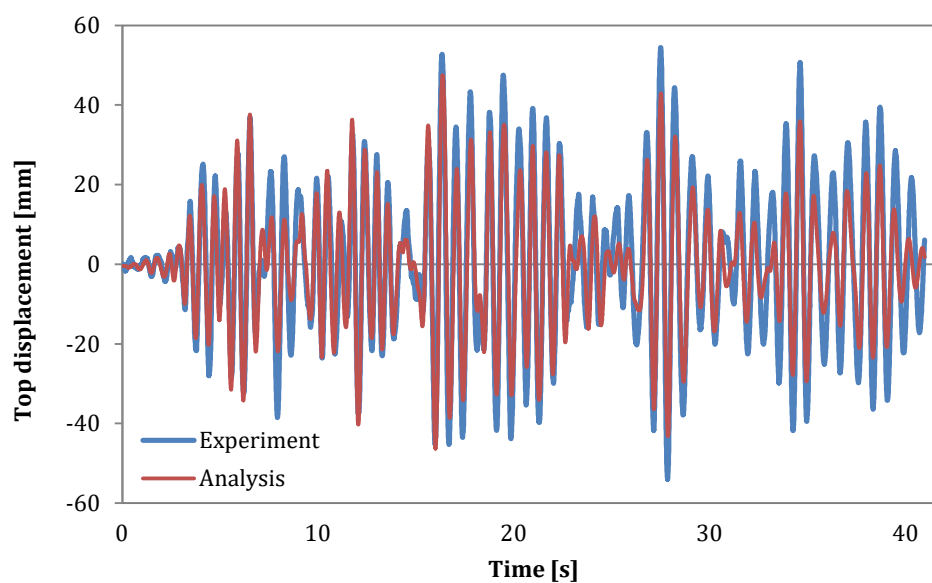


Figure 4.107. Experimental vs. Analytical results – top displacement-time (EQ3_REF)_Comp X

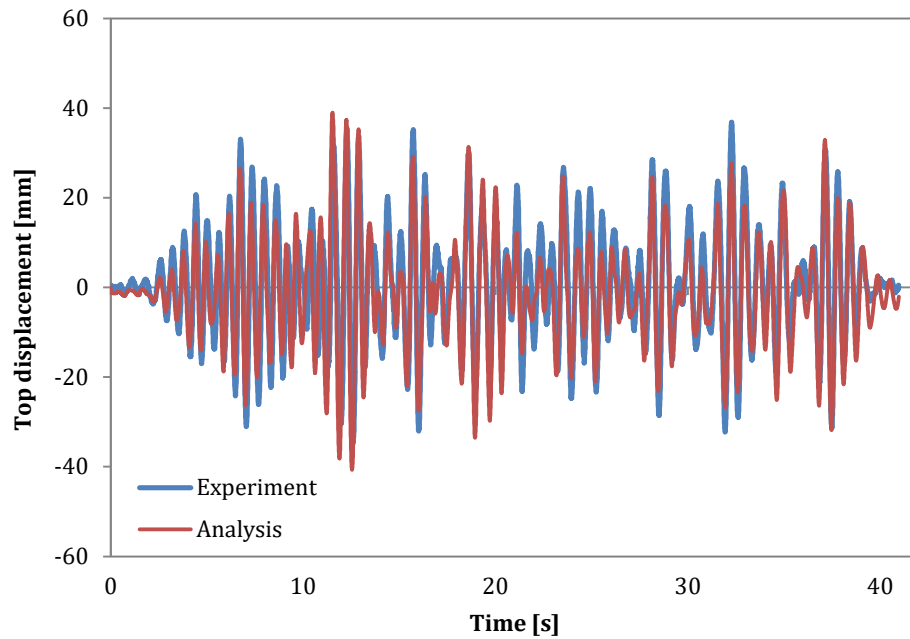


Figure 4.108. Experimental vs. Analytical results – top displacement-time (EQ3_REF)_Comp Y

The comparison between experimental and analytical results for model B is shown hereafter:

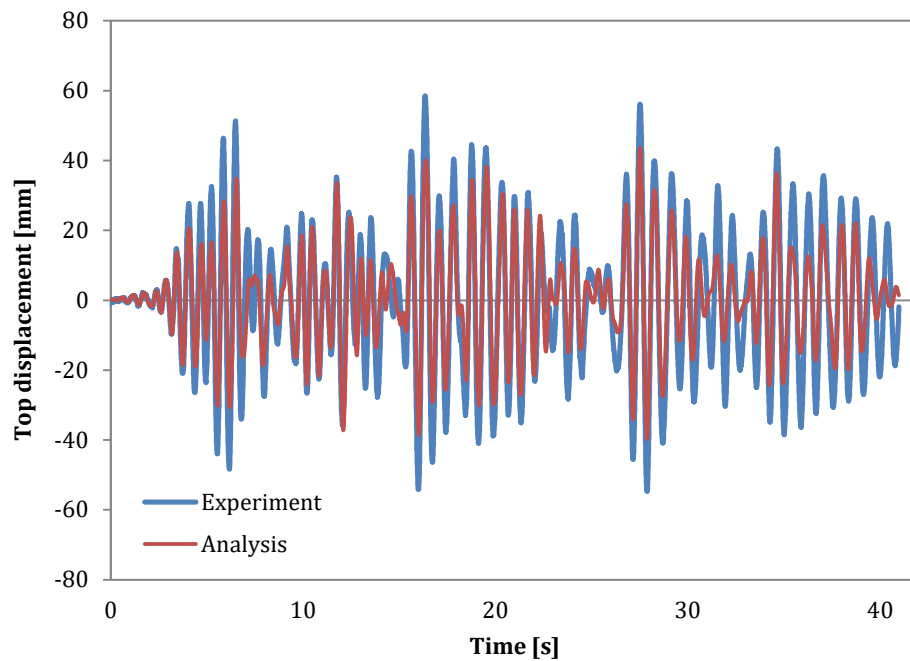


Figure 4.109. Experimental vs. Analytical results – top displacement-time (EQ3_REF)_Comp X

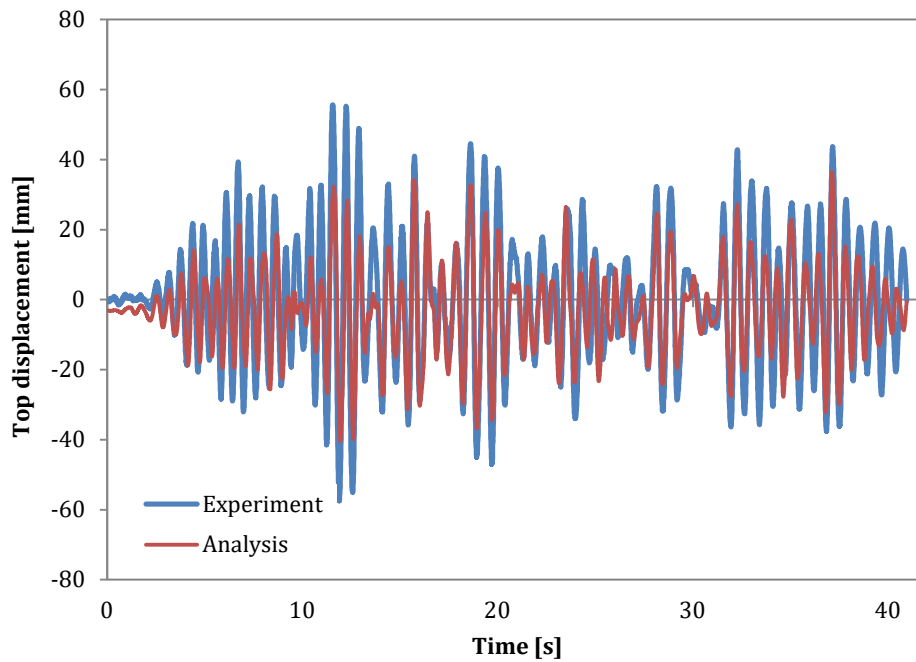


Figure 4.110. Experimental vs. Analytical results – top displacement-time (EQ3_REF)_Comp Y

COMPUTER FILES

- Ch4-blind-02A.spf
- Ch4-blind-02B.spf

REFERENCES

- LNEC team [2012] “*Blind Test Challenge Report*”, Report, LNEC (National Laboratory for Civil Engineering), Portugal.

EXAMPLE 3 – Half-scale three-storey RC frame

DESCRIPTION

This example describes the modelling of a half-scale, three-storey, three bays RC frame. The specimen has been prepared at the Tsinghua University (China) to be tested under quasi-static conditions, under the framework of Blind Prediction Contest organised by the Earthquake Resistance and Disaster Prevention Branch of the Architectural Society of China. Actuators have been used to apply lateral and vertical forces (as shown in the figure below).

SeismoStruct was employed by the winner of this competition, who took part in this exercise together with 30 other teams (details in here [hyperlink to <http://www.collapse-prevention.net/show.asp?ID=13&adID=2>]). The analytical results, obtained with the FE analysis program SeismoStruct in terms of top displacements and shear forces at the bottom, are compared with the experimental results. The FE model is defined in the X-Z plane.



Figure 4.111. Half-scale, three-storey reinforced concrete frame tested under pseudo-static cyclic loadings

GEOMETRY AND PROPERTIES

Structural Geometry

The model consists of a three-storey, three-bay RC frame (see Figure 4.112). All the columns have a square 200x200 mm section, whilst the beams are rectangular with dimensions 125x250 mm and a slab effective width equal to 845 mm.

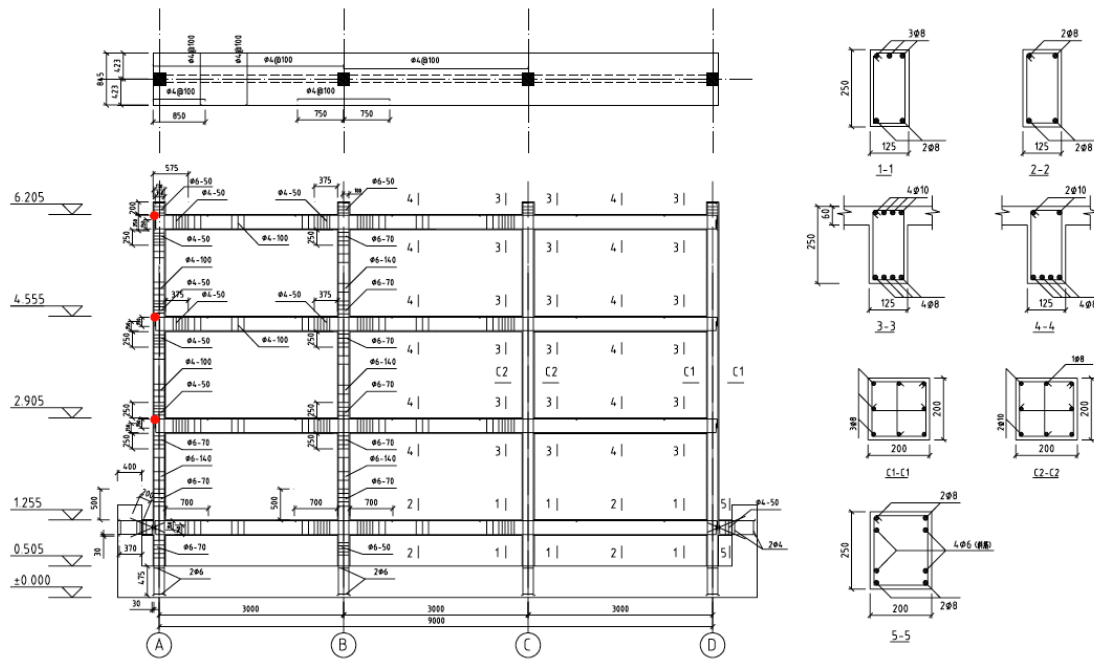


Figure 4.112. Frame and section dimensions

Material Properties of the column

The Mander et al. concrete model has been employed for defining the concrete material used for columns and beams. The material's parameters are listed below:

- (concr0): $f_c = 24168 \text{ kPa}$; $f_t = 3000$; $\epsilon_c = 0.002 \text{ m/m}$; $\gamma = 24 \text{ kN/m}^3$
- (concr1): $f_c = 27512 \text{ kPa}$; $f_t = 3000$; $\epsilon_c = 0.002 \text{ m/m}$; $\gamma = 24 \text{ kN/m}^3$
- (concr2): $f_c = 26372 \text{ kPa}$; $f_t = 3000$; $\epsilon_c = 0.002 \text{ m/m}$; $\gamma = 24 \text{ kN/m}^3$
- (concr3): $f_c = 25536 \text{ kPa}$; $f_t = 3000$; $\epsilon_c = 0.002 \text{ m/m}$; $\gamma = 24 \text{ kN/m}^3$

The Menegotto-Pinto steel model has been employed for defining the steel materials with the following properties:

- (steel $\phi 8$) $E_s = 2.8985\text{E}+008 \text{ kPa}$; $f_y = 582000 \text{ kPa}$; $\mu = 0.01$; fracture/buckling strain = 0.0962; $\gamma = 78 \text{ kN/m}^3$
- (steel $\phi 10$) $E_s = 2.65433\text{E}+008 \text{ kPa}$; $f_y = 481000 \text{ kPa}$; $\mu = 0.01$; fracture/buckling strain = 0.1013; $\gamma = 78 \text{ kN/m}^3$

MODELLING AND LOADING

Columns and beams are modelled through a 3D inelastic force-based frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 200.

At the top of the frame, the 'link beams' connecting horizontally the columns have been modelled with rigid links.

All foundation nodes are considered as fully restrained against rotations and translations.

The vertical load has been applied as permanent loads in terms of forces in the negative Z direction to the beam-column joints at the top level. In particular, loads equal to 163 and 326 kN have been assigned to the external and internal columns, respectively.

A quasi-static cyclic loading with increasing displacement amplitude has been loaded in the "Time-history Curves" dialog box. The used output sampling time interval is set to 1 second. The cyclic loading has been applied at the lateral nodes of each floor as a static time-history load in terms of

displacements in the X direction, considering a 'scale' factor equal to 0.333 at the first floor, 0.666 at the second floor and 1 at the third floor.

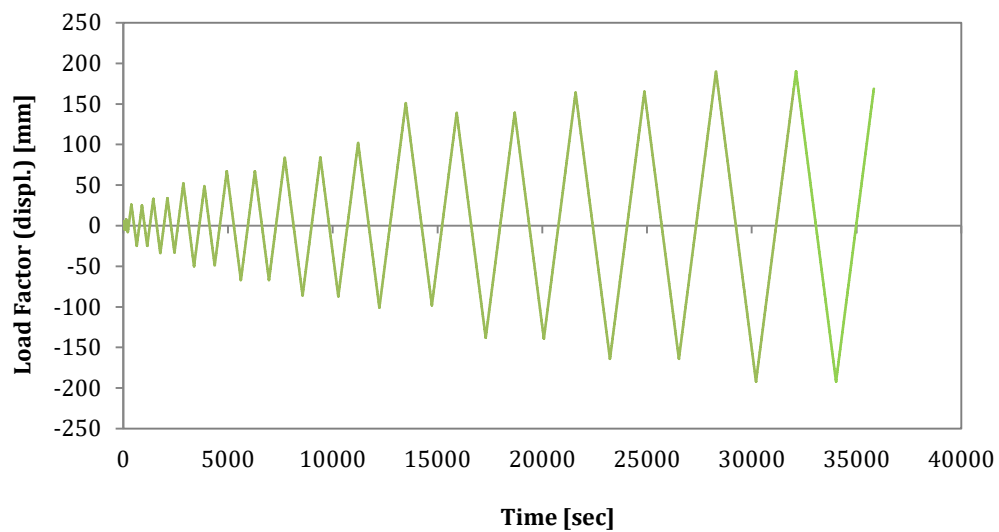


Figure 4.113. Cyclic loading

The FE model of the frame is shown below:

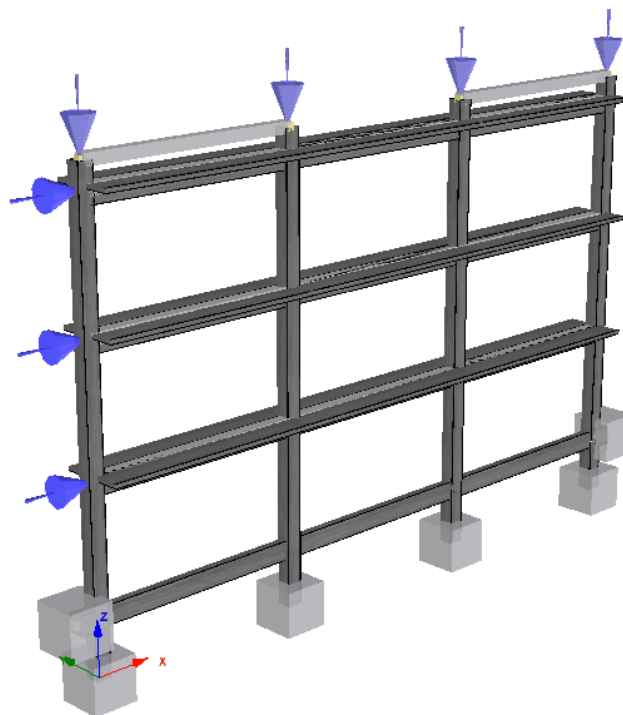


Figure 4.114. FE model of the frame

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between experimental and analytical results is shown below:

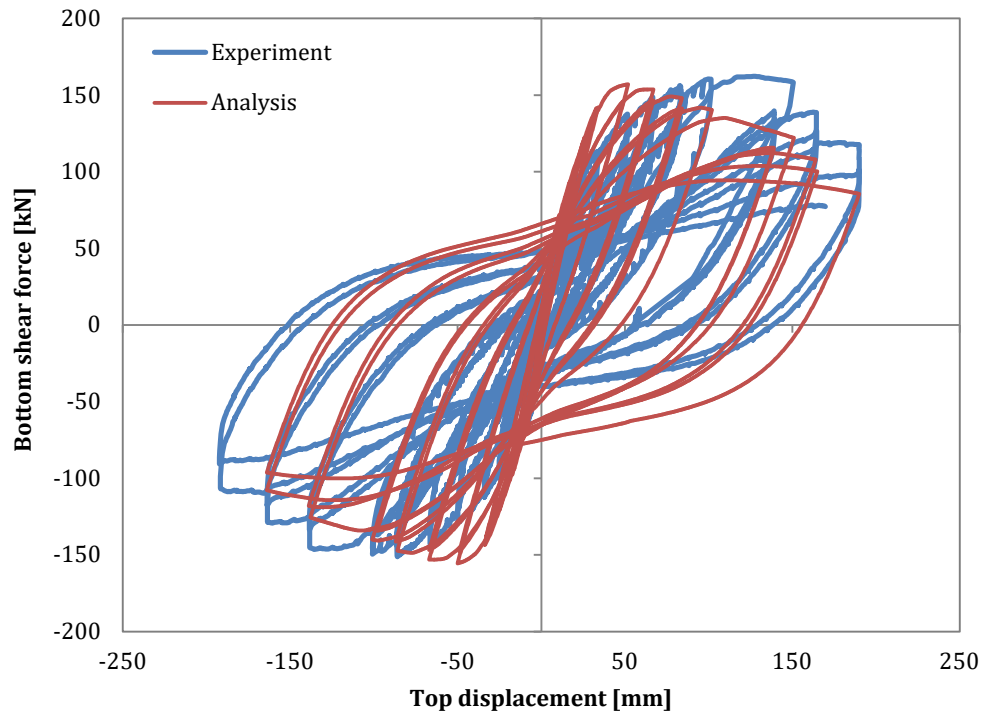


Figure 4.115. Base shear vs. Top displacement

COMPUTER FILE

- Ch4-blind-03.spf

REFERENCES

- Peizhou Zhang, Jinping Ou. "Numerical Simulation for Pseudo-Static Test of R.C. Frame Structure Using the Fiber Model". Building Structure, 2012 (under review, in Chinese)
- Xinzheng Lu, Lieping Ye, Peng Pan, et al. "Pseudo-static collapse experiments and numerical prediction competition of RC frame structure I: RC frame experiment". Building Structure. 2012, 42(11): 19-22. (in Chinese)