Software verification against Experimental benchmark data

Numerical evaluation of the seismic response of non - masonry (non - URM) buildings in the Groningen region

Mosayk

Datum July 2015

Editors Jan van Elk & Dirk Doornhof

General Introduction

The structural strength of the buildings in the Groningen area is an important factor for the assessment of risk resulting from induced earthquakes. As the building stock in the Groningen region is quite different from that in other seismically active areas, detailed studies into the structural response to earthquakes are required for the buildings typical for the Groningen area.

While the majority of dwellings in the Groningen area are unreinforced masonry (URM) buildings, many of the larger buildings in the Groningen area, like larger office and industrial buildings, are of different typologies (for instance pre-cast concrete frame structures, cast-in-place concrete structures or steel structure buildings). Prior to modelling the seismic response of these buildings, NAM set up a software verification study. This study is similar to the "URM Modelling and Analysis Cross-Validation study", which was done for modelling of URM buildings. This software verification study for modelling of non-URM buildings was carried out by Mosayk using the SeismoStruct software.

A number of benchmark studies and experiments were chosen for this non-URM software verification. These were based on experiments of non-URM buildings carried out at universities and laboratories in Italy (Pavia, San Michele all'Adige and Ispra), USA (San Diego), Japan (Kyoto and Miki City). The results of these benchmark experiments and modelling were compared and analysed.

In support of the modelling, measurements of the properties of pre-cast building elements is in progress. Typical Groningen building elements are subjected to seismic loading and their response measured.

The results of these non-URM structural modelling activities are then fed into the fragility function development work stream, which is coordinated by the following experts:

External Expert	Affiliation	Role	Main Expertise Area
Helen Crowley	Independent Consultant, Pavia	Collaborator	Building Fragility and Risk
Rui Pinho	University of Pavia	Collaborator	Structural Modeling and Building Fragility

The studies into the fragility of buildings will be reviewed by a panel of independent experts. The following experts have been invited:

External Expert	Affiliation	Role	Main Expertise Area
Jack Baker	Stanford University, US	Independent Reviewer	Building Fragility and Risk
Paolo Franchin	University of Rome "La Sapienza"	Independent Reviewer	Building Fragility and Risk
Michael Griffith	University of Adelaide, Australia	Independent Reviewer	Modeling and Testing of Masonry structures
Curt Haselton	California State University, US	Independent Reviewer	Structural Modeling and Fragility
Jason Ingham	University of Auckland	Independent Reviewer	Modeling and Testing of Masonry structures
Nico Luco	United States Geological Survey	Independent Reviewer	Building Fragility and Risk
Dimitrios Vamvatsikos	NTUA, Greece	Independent Reviewer	Building Fragility and Risk

NAM

<table>
<tr><td rowspan="2">Title</td><td colspan="2" rowspan="2">Software verification against Experimental benchmark data
Numerical evaluation of the seismic response of non-masonry (non-URM) buildings in the Groningen region</td><td>Date</td><td>July 2015</td></tr>
<tr><td>Initiator</td><td>NAM</td></tr>
<tr><td>Author(s)</td><td>Rui Pinho, Federica Bianchi,
Roberto Nascimbene</td><td>Editors</td><td colspan="2">Jan van Elk
Dirk Doornhof</td></tr>
<tr><td>Organisation</td><td>Mosayk</td><td>Organisation</td><td colspan="2">NAM</td></tr>
<tr><td>Place in the Study and Data Acquisition Plan</td><td colspan="4"><u>Study Theme:</u> Building Structural Strength
<u>Comment:</u>
The structural strength of the buildings in the Groningen area is an important factor for the assessment of risk resulting from induced earthquakes. As the building stock in the Groningen region is quite different from that in other seismically active areas, detailed studies into the structural response to earthquakes are required for the buildings typical for the Groningen area.
While the majority of dwellings in the Groningen area are unreinforced masonry (URM) buildings, many of the larger buildings in the Groningen area, like larger office and industrial buildings, are of different typologies (for instance pre-cast concrete frame structures, cast-in-place concrete structures or steel structure buildings). Prior to modelling the seismic response of these buildings, NAM set up a software verification study. This study is similar to the “URM Modelling and Analysis Cross-Validation study”, which was done for modelling of URM buildings. This software verification study for modelling of non-URM buildings was carried out by Mosayk using the SeismoStruct software.</td></tr>
<tr><td>Directliy linked research</td><td colspan="4">(1) Experiments subjecting typical Groningen pre-cast building elements to seismic loading and their response measured.
(2) Preparation of fragility curves for non-URM building typologies.</td></tr>
<tr><td>Used data</td><td colspan="4"></td></tr>
<tr><td>Associated organisation</td><td colspan="4">Mosayk and Eucentre</td></tr>
<tr><td>Assurance</td><td colspan="4">Review by a panel of independent experts.</td></tr>
</table>

D1

DELIVERABLE

Project Information

Project Title: **Numerical evaluation of the seismic response of the main typologies of non-masonry (non-URM) buildings that are found within the Groningen region**

Project Start: 12th May 2014

Duration: 24 months

Technical Point of Contact: Rui Pinho
rui.pinho@mosayk.it

Business Point of Contact: Roberto Nascimbene
roberto.nascimbene@mosayk.it

Deliverable Information

Deliverable Title: **D1. Report on software verification against experimental benchmark data**

Data of Issue: 16th November 2014

Editor(s): Federica Bianchi
federica.bianchi@mosayk.it

Reviewer: Rui Pinho

REVISION: **DRAFT v1**

Table of Contents

1 Introduction

1.1 Objective of report

In the field of **seismic analysis of structures,** the use of computer programs to manage more **complex numerical models** and run more **sophisticated analyses** in very short runtimes has increased significantly in the last 20 years. In order to have confidence in the outputs of these complex analyses, it has become increasingly important to verify and validate the software against literature case studies and experimental data.

The objective of the present deliverable (D1) is precisely to demonstrate that the numerical evaluation of the **seismic response** of the main typologies of **non-masonry (non-URM) buildings** that are found within the **Groningen region** has been performed through a **validated Finite Elements package** able to run accurate analyses, in particular nonlinear static and dynamic analyses.

In the v0 exposure model for the Groningen field, there are currently 54 non-URM building typologies[1], but these can be grouped in terms of structural systems (and associated material types) as shown in Figure 1. The mixed systems are a combination of the systems presented in Figure 1 and include wood frames with steel bracing, precast frames with masonry walls, and precast frames with steel bracing.

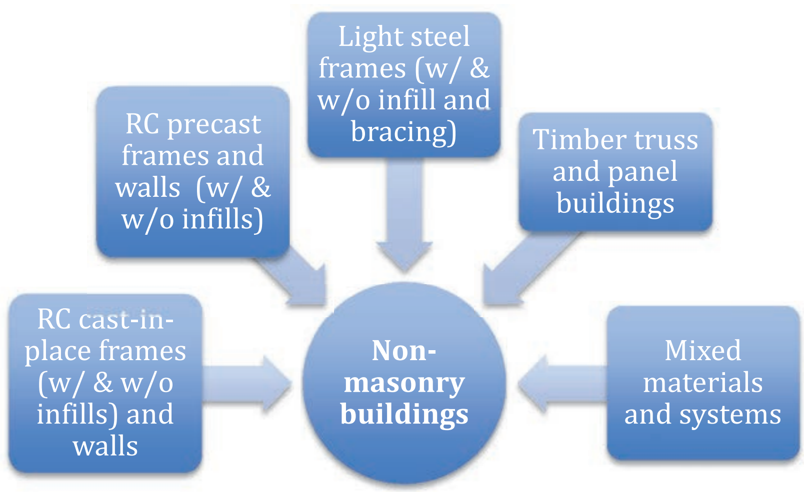


Figure 1: Grouped non-masonry structural systems found in the Groningen region

This report focused on the verification of the numerical modelling of each of the structural systems shown in Figure 1 as follows:

- RC precast frames are covered in Section 2.1. Precast walls are not deal with herein and future planned work for this system is discussed further in Chapter 3.
- Timber panel buildings are presented in Section 2.2; timber trussed roofs are not covered herein due to a lack of experimental test results for this structural system.
- Infill panels (which are present in RC, steel and mixed systems) are covered in Section 2.3.
- RC (reinforced concrete) cast-in-place frames, regular and irregular, are presented in Section 2.4 and 2.5, respectively.

[1] "Exposure model v0 – inference rules", 21st August 2014, Report submitted to NAM

- RC (reinforced concrete) cast-in-place wall buildings are discussed in Section 2.6.
- Steel frame systems, regular and irregular, are covered in Section 2.7 and 2.8, respectively, whilst the connections of these buildings are presented in Section 2.10.
- Composite steel and reinforced concrete frame buildings are presented in Section 2.9, though it is not yet clear whether this typology is found in Groningen; this will be investigated further in the development of the v1 exposure model.

The results of the numerical analyses of the structures in Groningen are currently being used for the development of partial/full collapse fragility functions, and thus the software should be able to model the highly nonlinear response of structures up until collapse. Hence, the remaining sections of this introductory chapter discuss how the software that has been selected for the modelling of non-URM buildings in Groningen is appropriate for this task.

The software employed for this study is **SeismoStruct** (Seismosoft, 2014a), an award-winning Finite Elements program able to accurately predict the large displacement behaviour of space frames under static or dynamic loading (e.g. earthquake strong motion), taking into account both **geometric nonlinearities** and **material inelasticity**. SeismoStruct has been already extensively quality-checked and internationally validated, as described in its "**Verification Report**" (Seismosoft, 2014b). Some of the examples presented herein have been taken from the aforementioned verification report, with a focus on those most relevant to the structural systems presented above, whilst others have been newly developed for the purposes of this deliverable and to ensure that all systems presented in Groningen are covered.

1.2 Geometric nonlinearity and material inelasticity

Enabled by advancements in computing technologies and available test data, **nonlinear analysis** provides the means for the calculation of structural response beyond the elastic range, including strength and stiffness deterioration associated with inelastic material behaviour and large displacements.

Regarding geometric nonlinearity, SeismoStruct takes into account large displacements/rotations and large independent deformations relative to the frame element's chord (also known as *P-Delta effects*) through the employment of a *total co-rotational formulation* developed and implemented by Correia and Virtuoso (2006).

Two examples of geometric nonlinearity are illustrated in Figure 2.

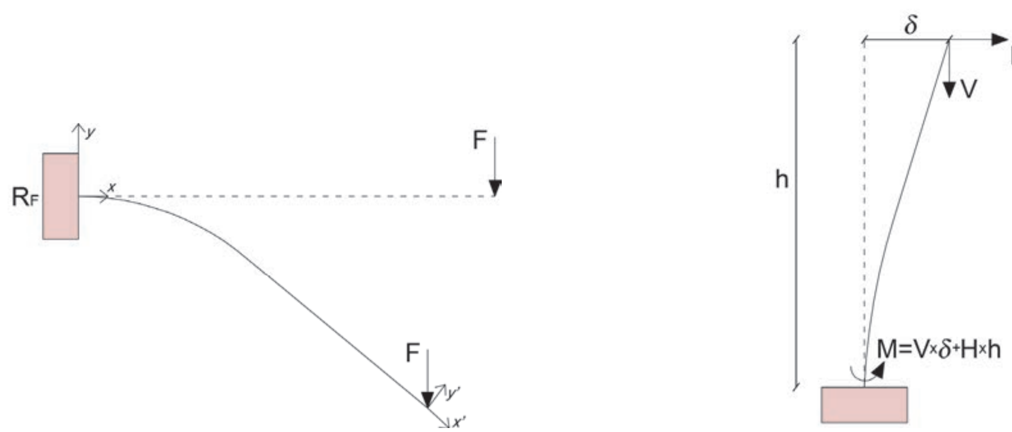


Figure 2: Large displacements/rotations: cantilever subjected to orthogonal load (left); P-delta effects (right)

Inelastic structural component models can be differentiated by the way that plasticity is distributed through the member cross sections and along their length.

In order to take into account the material inelasticity, SeismoStruct makes use of the so-called **fibre approach** (see the drawing on the right of Figure 3) to represent the cross-section behaviour: each fibre is associated with a uniaxial stress-strain relationship; the sectional stress-strain state of beam-column elements is then obtained through the integration of the nonlinear uniaxial stress-strain response of the individual fibres in which the section has been subdivided (see Figure 4). By following this approach (also called "distributed plasticity"), the spread of inelasticity along the member length and across the section depth is explicitly modelled, allowing for accurate estimation of damage distribution.

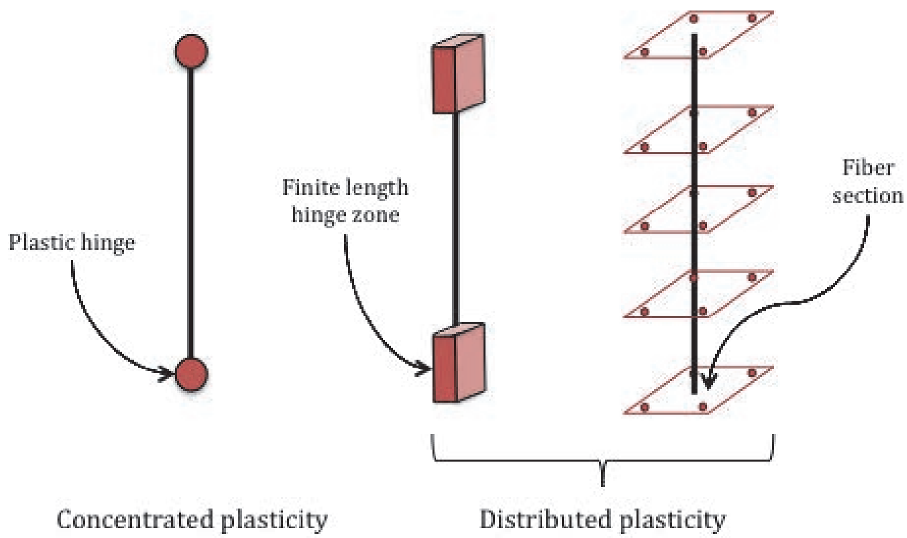


Figure 3: Idealized models of beam-column elements

In the following a **well-known literature case-study** is presented; the results produced by SeismoStruct are compared in graphical form with analytical solutions coming from the literature, as well with other numerical results.

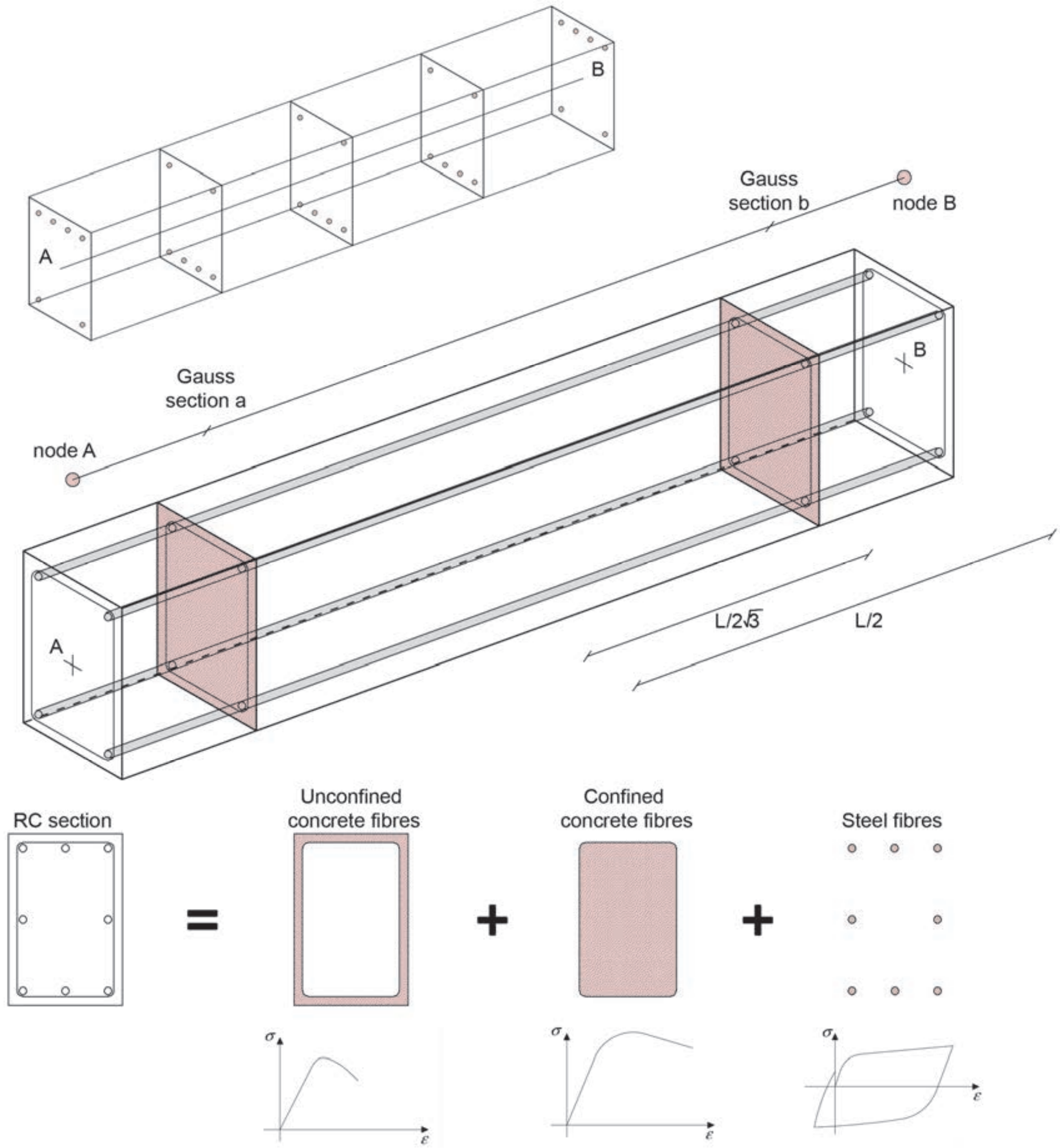


Figure 4: Discretisation of a typical reinforced concrete cross-section

EXAMPLE

A **simply supported beam** (the geometrical properties of which are shown in Figure 5) has been analysed numerically using SeismoStruct, considering both **sources of nonlinearities (geometric nonlinearity and material inelasticity)**.

Two models have been created:

- Model A: Only **geometric nonlinearities** have been taken into account; for these reasons an **elastic material** (with modulus of elasticity, E = 220,000 MPa) has been defined;
- Model B: It takes into account **both sources of nonlinearities;** in this case an **elasto-plastic material** with the same modulus of elasticity of Model A and with a yielding stress equal to 300 MPa has been defined.

The beam-column element employed to model the simply supported beam has been discretised into 4 and 10 inelastic force-based elements, respectively, in Model A and B.

Incremental loads have been applied to each node (with the exception of the supports, in order to simulate the distributed load) in terms of forces in the vertical direction, and a **static pushover analysis** has been run.

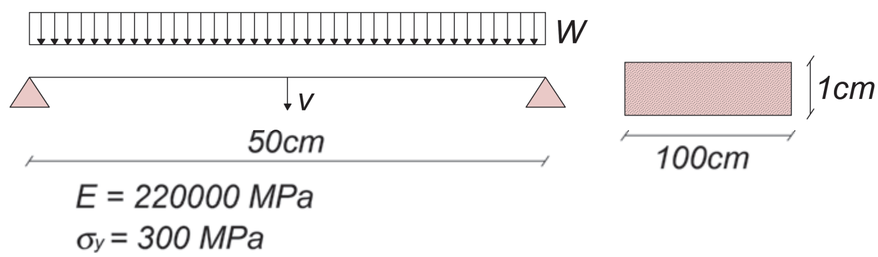
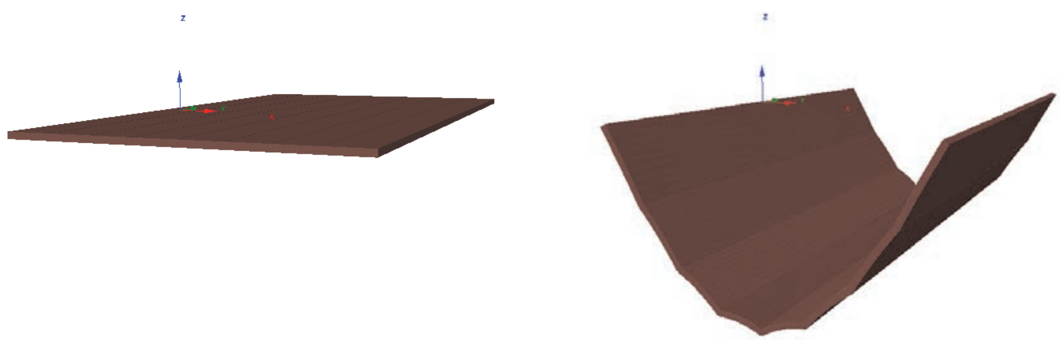


Figure 5: Simply supported beam with uniform load – geometrical and material properties

Figure 6 shows the undeformed and deformed configurations of the analysed simply supported beam.



(a) (b)

Figure 6: Simply supported beam – undeformed (a) and deformed (b) configurations

The “applied load vs. midspan vertical displacement” curve obtained with SeismoStruct by performing the analysis with the first model (**elastic material**) has been compared with the analytical solution presented in Timoshenko and Woinowsky-Krieger (1959), as well as with the numerical results obtained by Magalhães de Souza (2000). The comparison is shown in Figure 7.

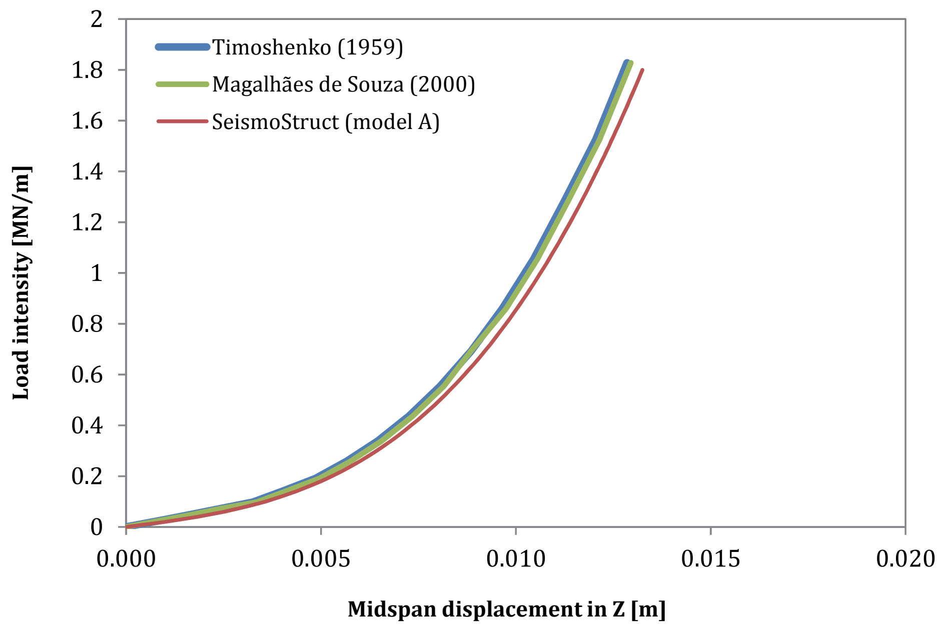


Figure 7: Simply supported beam – load intensity vs. midspan displacement in Z (elastic material)

As in the previous case, the "applied load vs. midspan vertical displacement" curve obtained with the second model (**elasto-plastic material**) has been compared with the analytical results provided in Backlund (1976), as wells as Magalhães de Souza (2000), as shown in Figure 8.

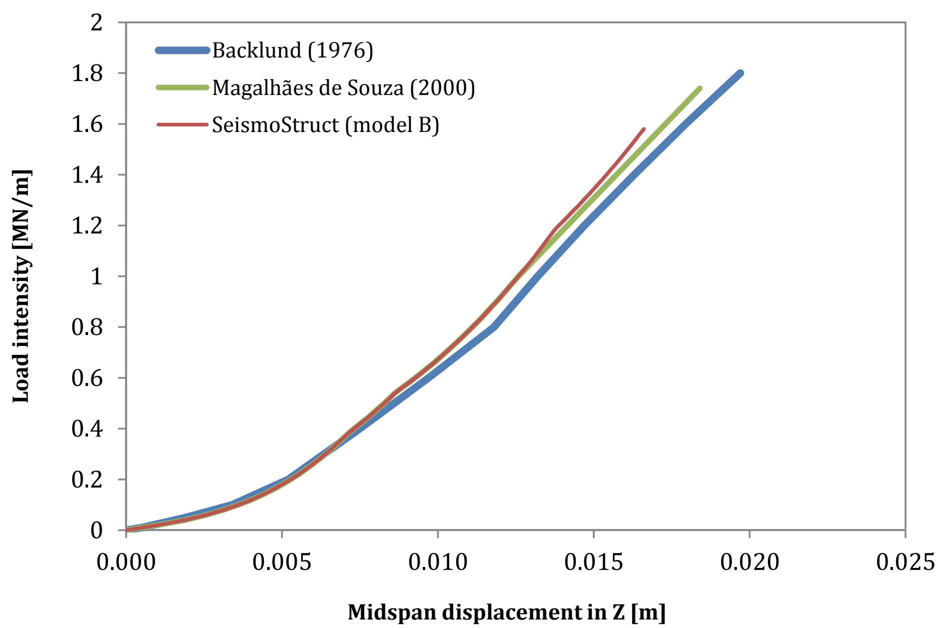


Figure 8: Simply supported beam – load intensity vs. midspan displacement in Z (elasto-plastic material)

1.3 Modelling of progressive collapse

In order to ensure resistance against progressive collapse phenomena, a structure should respect the requirement of robustness, as well as integrity, continuity, redundancy and ductility.

It may happen that extreme situations or abnormal loads destroy a member of the structure. In these cases, the progressive collapse could be modelled either through a "member removal" or by assigning to such member only a residual capacity.

In the following an example of modelling progressive collapse of reinforced concrete (RC) buildings subjected to extreme loading is presented, using three different software capable of modelling member removals.

EXAMPLE

The prototype structure consists of a 5-storey 6 x 6-bay RC moment resisting frame. Bay length and storey height are 6 m and 3 m, respectively, as shown in Figure 9. Column and beam cross-sections are 400 x 400 mm^2 and 500 x 300 mm^2, respectively. A uniform longitudinal reinforcement, consisting of 8 Ø18, is provided in the columns of each floor, while 7 Ø18 at the top and 5 Ø18 at the bottom of the cross-sections are used to reinforce all the beams; the transverse reinforcement is composed of Ø8 stirrups, spaced by 20 cm, both in beams and columns.

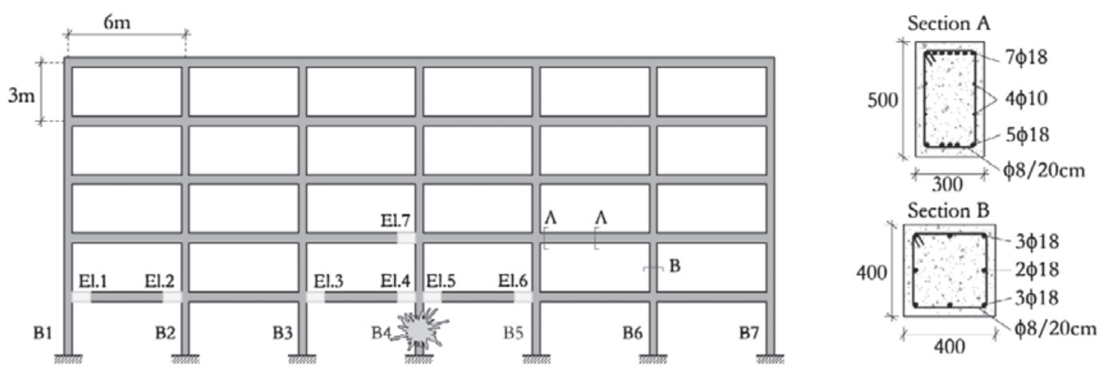


Figure 9: Prototype frame for progressive collapse modelling– geometry, elements, cross-sections and reinforcement details

The "vertical displacement vs. time" curves obtained with SeismoStruct (S) have been compared with other FE programs, such as Opensees (O) and a local detailed FE program (L). The comparison is shown in the following figure, wherein similar response of the three programs is observed.

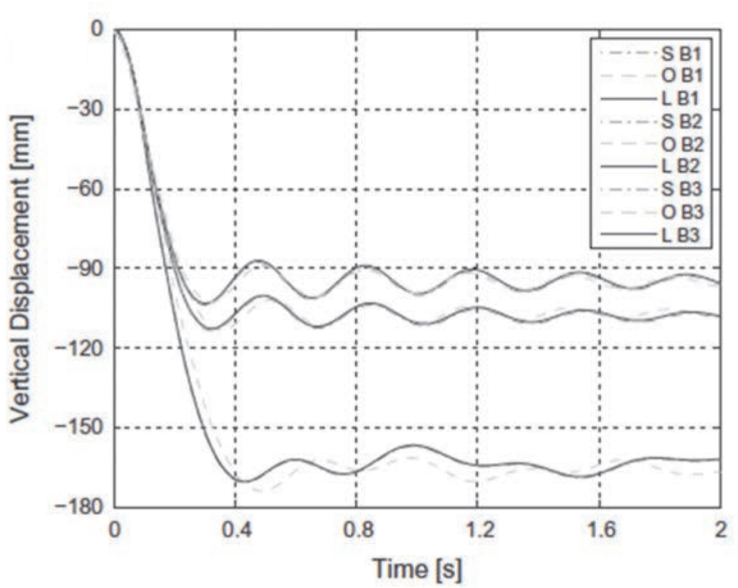


Figure 10: Progressive collapse modelling – vertical displacement vs. time

2 Numerical results against experimental benchmark data

As anticipated in the Introduction, the aim of the present deliverable (D1) is to demonstrate that the numerical evaluation of the **seismic response** of the main typologies of **non-masonry (non-URM) buildings** that are found within the **Groningen region** has been performed through a **validated FE analysis program**.

For these reasons, a comprehensive collection of examples will be presented in the following paragraphs.

2.1 Precast concrete frame structures

GENERAL

This example describes the modelling of a **3/4-scaled two-bay three-storey precast frame** tested at the EUCENTRE of Pavia (Italy) under a quasi-static cyclic top displacement history, consisting of symmetric stepwise increasing drift cycles. The pinned beam-to-column connections are made of neoprene pads with vertical steel dowels. The majority of the Italian and European industrial facilities consist of precast RC frames, composed of continuous monolithic columns and pin-ended beams. These present high flexibility and low resistance in beam-to-column and panel-to-structure connections. Seismic response greatly depends on the connection system behavior.

In the following, the analytical results will be compared with the experimental benchmark data.

NOTE: Further information about the test specimen and the experimental campaign may be found in Brunesi et al. (2014).

Figure 11: Three-dimensional precast reference building tested at the EUCENTRE of Pavia (Italy)

STRUCTURAL CONFIGURATION AND MATERIAL PROPERTIES

The model consists of a 3:4 scaled two-bay three-storey RC precast frame. All the structural dimensions are indicated in Figure 12 (right):

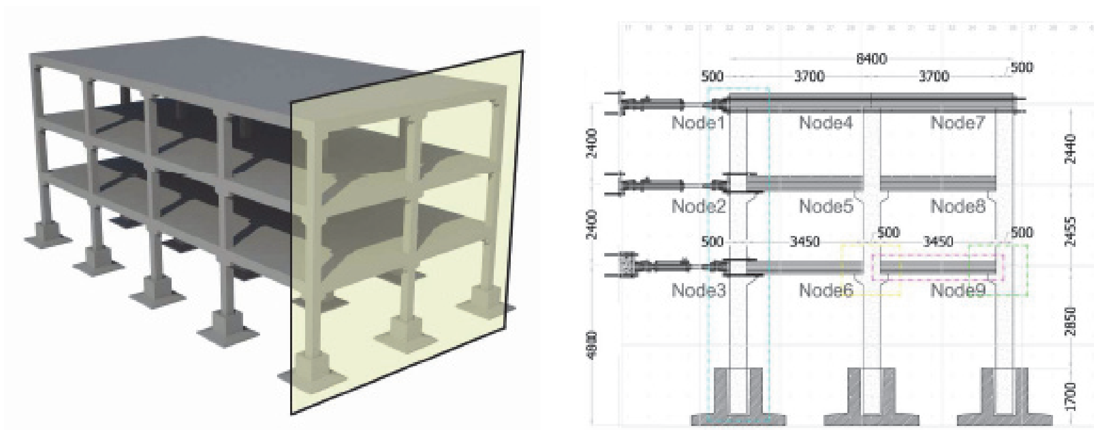


Figure 12: Three-dimensional precast reference building – geometry (elevation)

Regarding the material properties, *Mander et al. concrete model* and *uniaxial bilinear stress-strain model with kinematic strain hardening* have been employed for defining the concrete and the steel material, respectively.

The characteristic parameters for each constitutive model are listed below:

- Beam Concrete: f_c = 48,000 kPa; f_t = 4800 kPa; ε_c = 0.002 m/m;
- Column Concrete: f_c = 53,000 kPa; f_t = 5300 kPa; ε_c = 0.002 m/m;
- Steel: E_s = 2.00E+008 kPa; f_y = 430,000 kPa; μ = 0.005.

MODELLING AND LOADING

Columns have been modelled through 3D force-based inelastic frame elements (infrmFB) with 5 integration sections. The number of fibres used in the section equilibrium computations has been set to 150.

The pinned gravity beams, that have been designed in order to remain elastic, have been modelled through elastic frame elements (elfrm) with the following properties:

- Beam 1) EA = 5.980E+006 [kN]; EI (axis2) = 6.1127E+04 [kNm^2]; EI (axis3) = 1.2475 E+05 [kNm^2]; GJ = 5.4314E+04 [kNm^2].
- Beam 2-3) EA = 7.698E+006 [kN]; EI (axis2) = 1.2991E+05 [kNm^2]; EI (axis3) = 1.6039 E+05 [kNm^2]; GJ = 9.4483E+04 [kNm^2].

The modelling of the beam-column connection of neoprene pads with vertical steel dowels has been realized through link elements (for details refer to the input files of the model).

The masses have been computed starting from the values of dead and live loads imposed on the beams and resulting from the transformation of the applied loads. The values are summarized in the table below:

Table 1: RC precast frame – applied permanent loads

Floor	Floor Height [mm]	Beam Height [mm]	Beam weight [kN]	Dead load [kN]	Live load [kN]	Total distributed [kN/m]
1	7900	450	18.6	20.0	10.0	8.70
2	5500	450	18.6	27.0	20.0	13.6
3	3100	350	16.6	42.0	25.0	19.4

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

A quasi-static cyclic loading with increasing force amplitude is loaded in the "Time-history Curves" dialog box. The used output sampling time interval is set to 0.01 seconds. The cyclic loading is applied at the three lateral nodes (as in the figure below) as a static time-history load in terms of forces in the X direction.

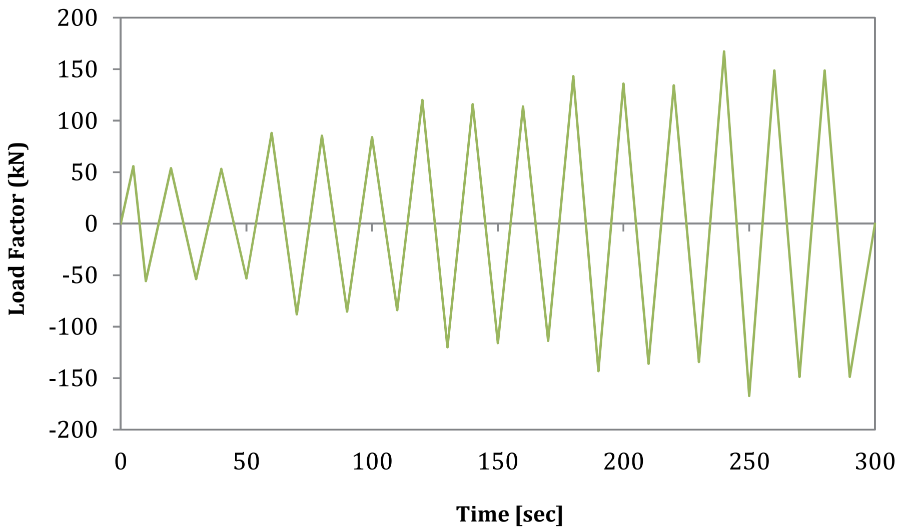


Figure 13: RC precast frame – applied cyclic loading

The FE model is presented below:

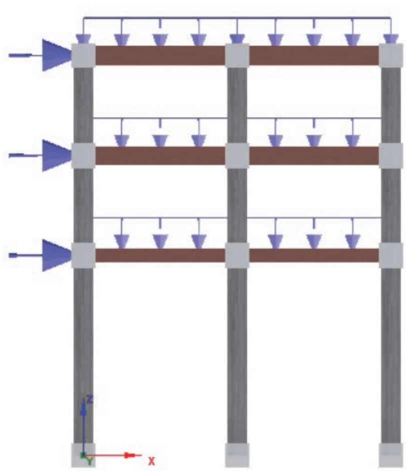


Figure 14: RC precast frame – FE model view

Analysis Type

Static time-history analysis.

Results Comparison

The comparison between numerical and experimental results is shown below:

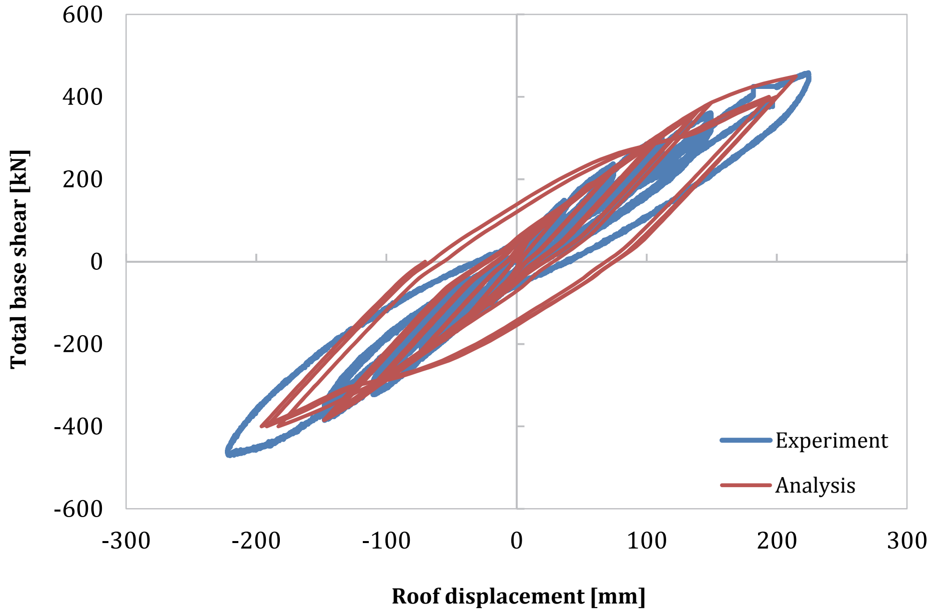


Figure 15: RC precast frame – Experimental vs. Analytical results

2.2 Timber panels

GENERAL

This example describes the modelling of a **full-scale timber panel** connected to the foundation with three angle brackets and two hold-downs at both ends. The panel has been tested at CNR-IVALSA Italian National Research Council, Trees and Timber Institute in San Michele all'Adige (Italy) by imposing a cyclic lateral displacement on the top of the panel, after applying a vertical load of 18.5 kN.

The analytical results will be compared with the experimental benchmark data.

NOTE: Monotonic and cyclic tests have been performed in accordance with EN26891 (12) and EN12512 (13), respectively.

Figure 16: Example of timber (X-lam) panel (left) and wall panel with angle brackets and hold-down connectors tested at Ivalsa CNR (right - copyright IVALSA-CNR)

STRUCTURAL CONFIGURATION

The model consists of a timber panel. All the dimensions are indicated in Figure 17:

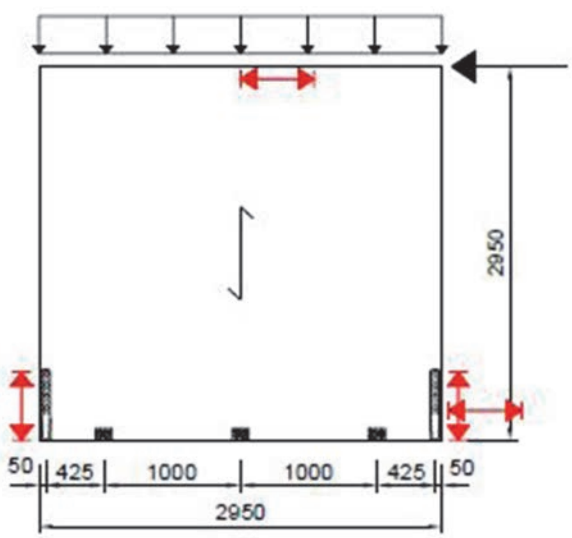


Figure 17: Angle brackets and hold-down connectors and their location in the wall panel (dimensions in mm) (copyright IVALSA-CNR)

MODELLING AND LOADING

The timber panel has been modelled as rigid through horizontal and vertical elastic frame elements (elfrm) with the following properties: EA = 1.0000E+07 [kN]; EI (axis2) = 1.0000E+07 [kN-m^2]; EI (axis3) = 1.0000E+07 [kN-m^2]; GJ = 1.0000E+07 [kN-m^2].

In order to model the nonlinear behavior of the connections located at the base of the panel (i.e. three angle brackets and two hold-downs, as shown in Figure 17), two "equivalent" springs (one at each end) have been inserted in the model. They have been modelled through the definition of link elements, in which for the F_1, F_3, M_1, M_2 and M_3 degrees of freedom a linear symmetric response curve has been chosen with a value of stiffness k0 equal to 1.00E+08, whereas for the F_2 degree-of-freedom a modified Richard-Abbott response curve has been chosen.

All the nodes at the supports have been pinned.

The vertical loads assigned for the frame have been applied as distributed permanent loads in terms of forces in the Z direction (for details refer to the input file of the model).

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 has been set to 0.01 seconds. The cyclic loading has been applied at the top lateral node as a static time-history load in terms of displacements in the Y direction. It is defined as follows:

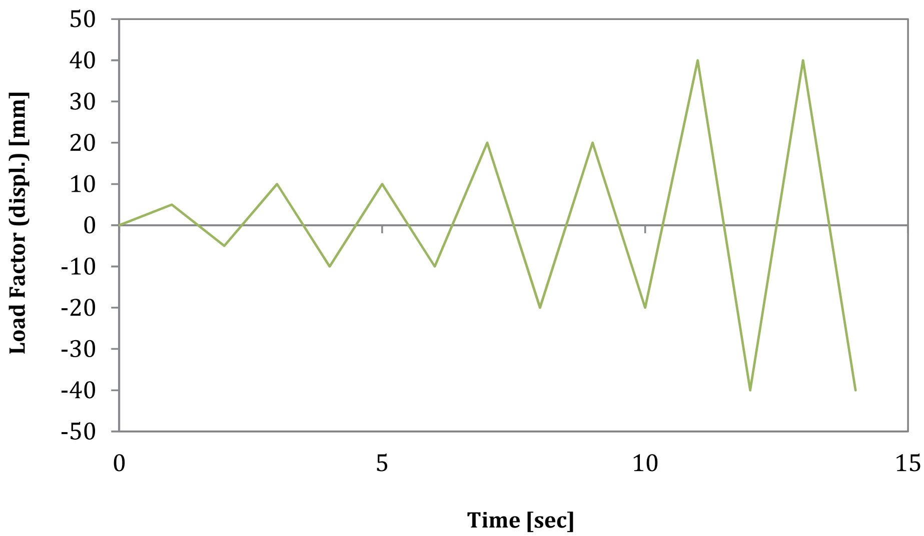


Figure 18: Timber panel – applied cyclic loading

The FE model is shown below:

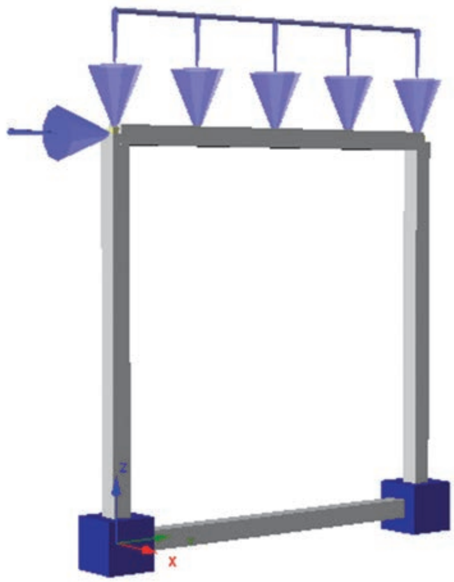


Figure 19: Timber panel – FE model view

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

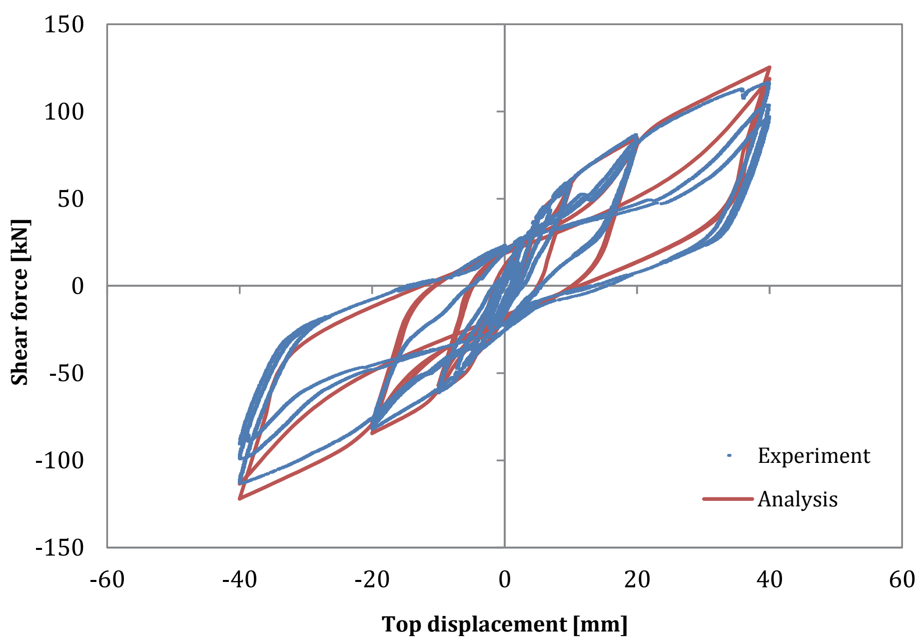


Figure 20: Timber panel – Experimental vs. Analytical results

2.3 Masonry infill panels

GENERAL

This example describes the modelling of a **half scale, three-storey RC building**, similar to SPEAR structure, with nonstructural masonry infilled walls (made of cellular concrete Gasbeton RDB). The prototype building has been 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 has been tested at the EUCENTRE of Pavia (Italy) under pseudo-dynamic conditions using the Herceg-Novi accelerogram registered during the Montenegro 1979 earthquake.

In the following, the analytical results for scaled Herceg-Novi record with moderate intensity (PGA = 0.3g) will be compared with the experimental benchmark data.

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

Figure 21: Half-scale three-storey RC irregular building with infill panels tested at the EUCENTRE of Pavia (Italy)

STRUCTURAL CONFIGURATION AND MATERIAL PROPERTIES

The model consists of a 3D half-scale three-storey RC irregular frame with infill panels. The interstorey height is equal to 1.5 m at each floor. The slab thickness is 0.08 m. All beams have equal dimensions of 0.125 x 0.25m (125 x 250mm). The cross section dimension of column C2 is 0.125 x 0.375m (125 x 375mm) whilst all remaining columns are 0.125 x 0.125m (125 x 125mm). The frame is filled by nonstructural masonry infilled walls.

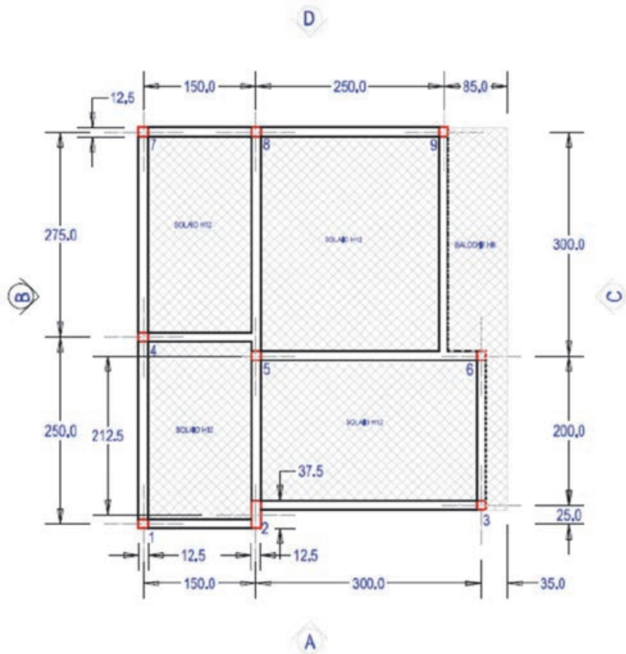


Figure 22: Half-scale three-storey RC irregular building with infill panels – plan view (Ali, 2009)

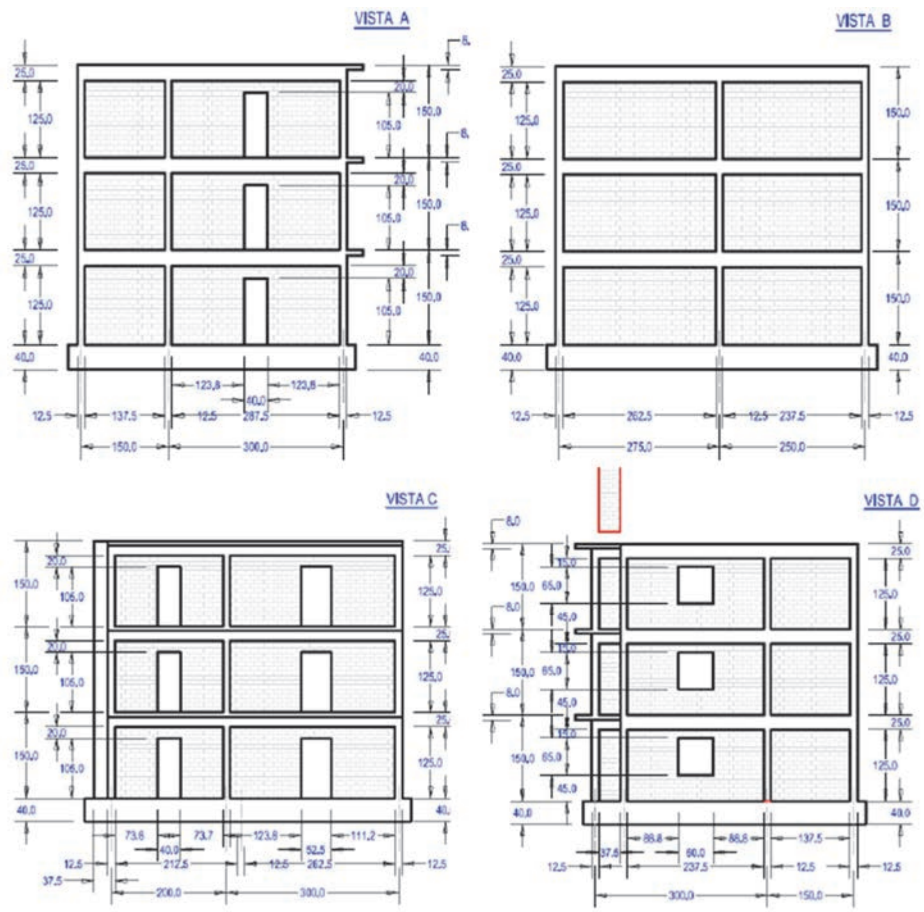


Figure 23: Half-scale three-storey RC irregular building with infill panels – elevation (Ali, 2009)

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

- (beams 1 and 2): f_c = 43 MPa; f_t = 0 MPa; ε_c = 0.002 mm/mm
- (beam 3): f_c = 36 MPa; f_t = 0 MPa; ε_c = 0.002 mm/mm
- (columns 1 and 2): f_c = 33 MPa; f_t = 0 MPa; ε_c = 0.002 mm/mm
- (column 3): f_c = 27 MPa; f_t = 0 MPa; ε_c = 0.002 mm/mm

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

- (steel for 6mm bars): E_s = 200000 MPa; f_y = 364 MPa; μ = 0.004
- (steel for 10mm bars): E_s = 200000 MPa; f_y = 293 MPa; μ = 0.004

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.

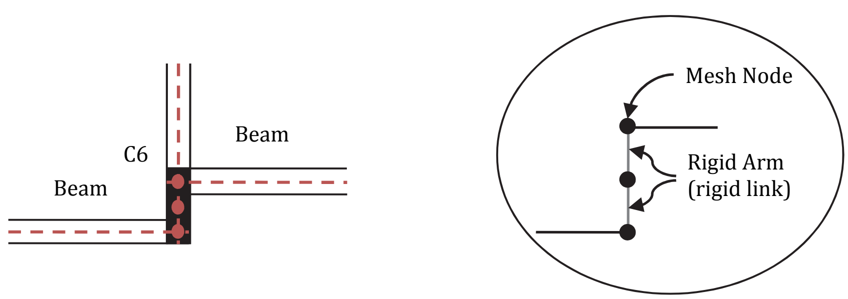


Figure 24: Half- scale three-storey RC irregular building with infill panels – modelling of column C6

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-Novi record with moderate intensity (PGA = 0.3g) is loaded in the "Time-history Curves" dialog box. The applied curve is presented below:

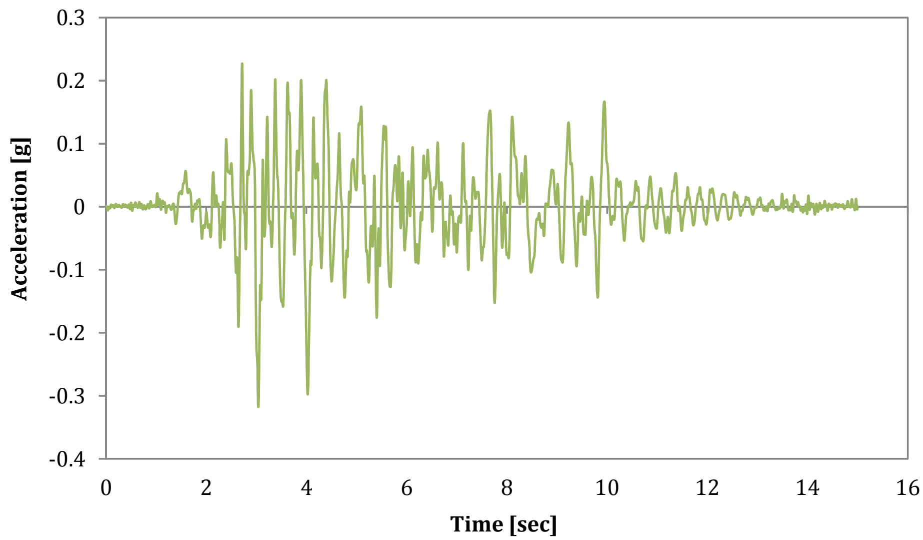


Figure 25: Half- scale three-storey RC irregular building with infill panels – scaled Herceg-Novi record

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:

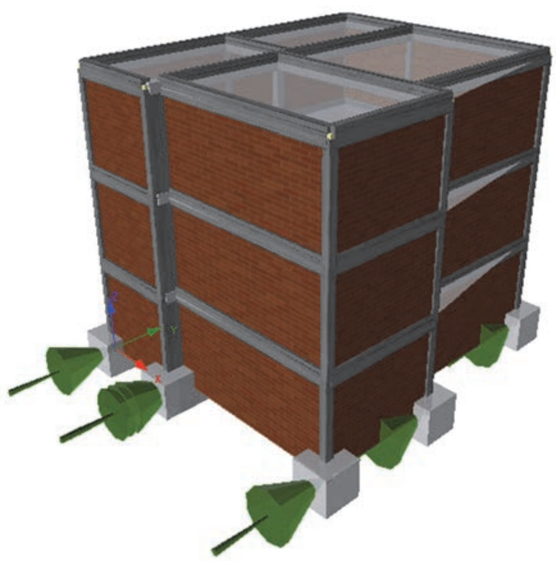


Figure 26: Half- scale three-storey RC irregular building with infill panels – FE model view

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

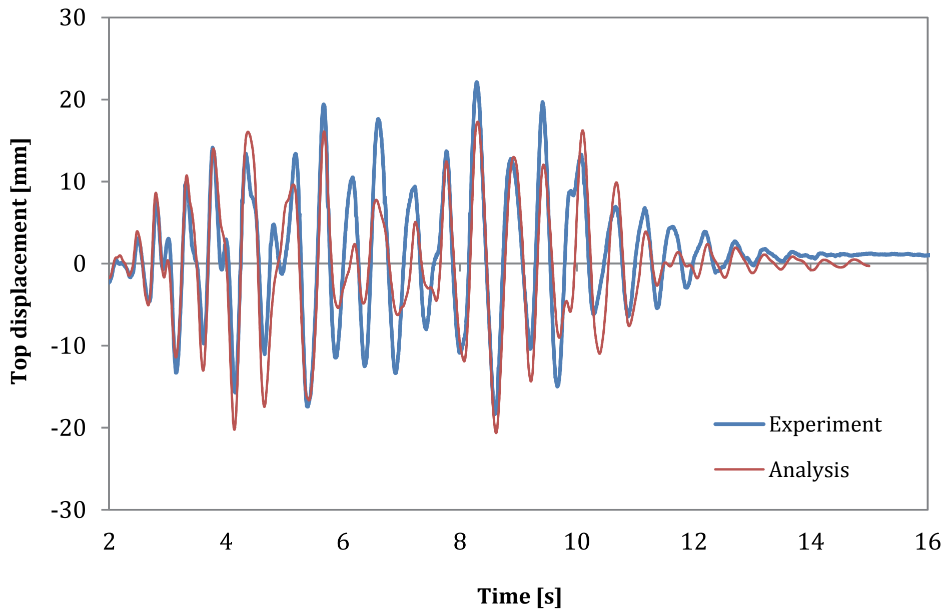


Figure 27: Half- scale three-storey RC irregular building with infill panels – Experimental vs. Analytical results (top displacement-time)

2.4 Concrete regular frame buildings

GENERAL

This example describes the numerical modelling of a **full-scale four-storey reinforced concrete cast-in-place frame**, which has been 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 Europe in the 1950's and 1960's. The frame has been tested at the ELSA laboratory (Joint Research Centre, Ispra, Italy) under a set of subsequent pseudo-dynamic loadings (i.e. hazard-consistent accelerograms artificially generated to fit the spectra for return periods of 100, 475, 975 and 2000 years).

In the following, the analytical results for the 475 and 2000 years return period motion respectively will be compared with the experimental benchmark data.

NOTE: Further information about the test specimen and the experimental campaign can be found in Pinto et al. (1999), Carvalho et al. (1999), Pinho and Elnashai (2000) and Varum (2003).

Figure 28: Full-scale four-storey RC frame tested at the ELSA laboratory of Ispra (Italy)

STRUCTURAL CONFIGURATION AND MATERIAL PROPERTIES

The model consists of a four-storey three-bay reinforced concrete frame. Each floor has the same storey height, which is equal to 2.7 m. The bay lengths are equal to 5, 5 and 2.5 m, respectively, as indicated in Figure 29.

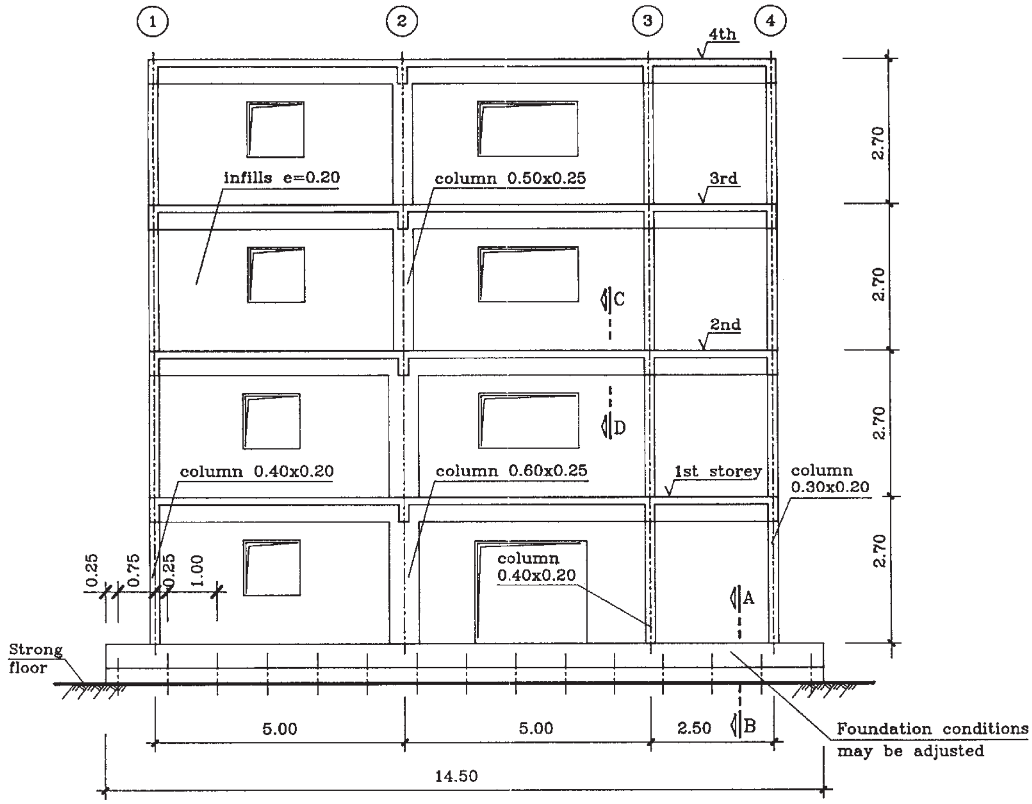


Figure 29: Full-scale four-storey RC frame – geometry (elevation and plan views, after Carvalho et al., 1999)

The geometrical details of the structural elements, i.e. columns and beams, are shown in the figure above and are also summarized the following tables:

Table 2: Full-scale four-storey RC frame – geometrical details of columns

Storey	Col1	Col2	Col3	Col4
3-4	0.4x0.2 (6 ϕ12)	0.25x0.5 (4 ϕ16 + 2 ϕ12)	0.4x0.2 (6 ϕ12)	0.3x0.2 (6 ϕ12)
1-2	0.4x0.2 (6 ϕ12)	0.25x0.6 (8 ϕ16 + 2 ϕ12)	0.4x0.2 (8 ϕ12)	0.3x0.2 (6 ϕ12)

Table 3: Full scale four-storey RC frame – geometrical details of beams

Floor	Beam (1st and 2nd bay)	Beam (3rd 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		

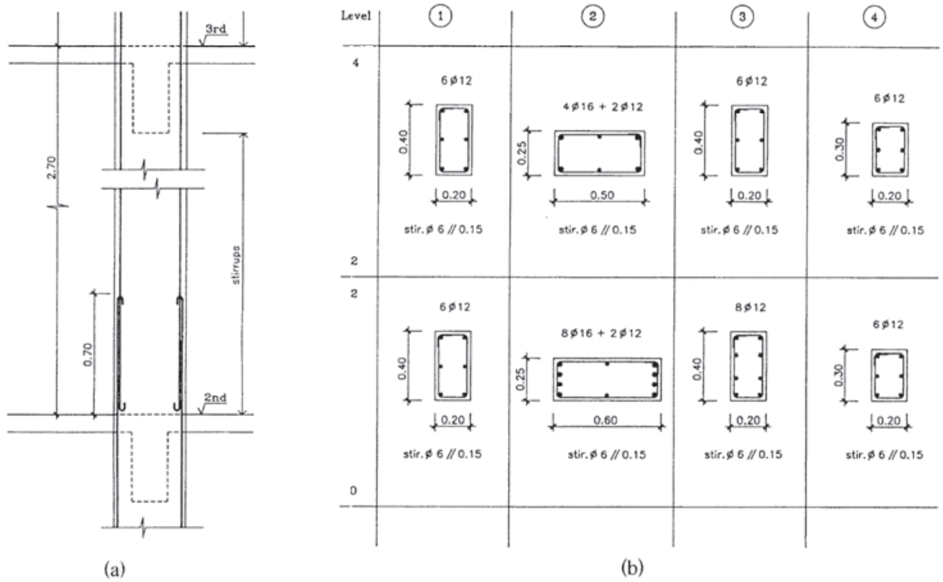


Figure 30: Full-scale four-storey RC frame – lap-splicing detail (a) and reinforcement details of the columns (b), after Carvalho et al. (1999)

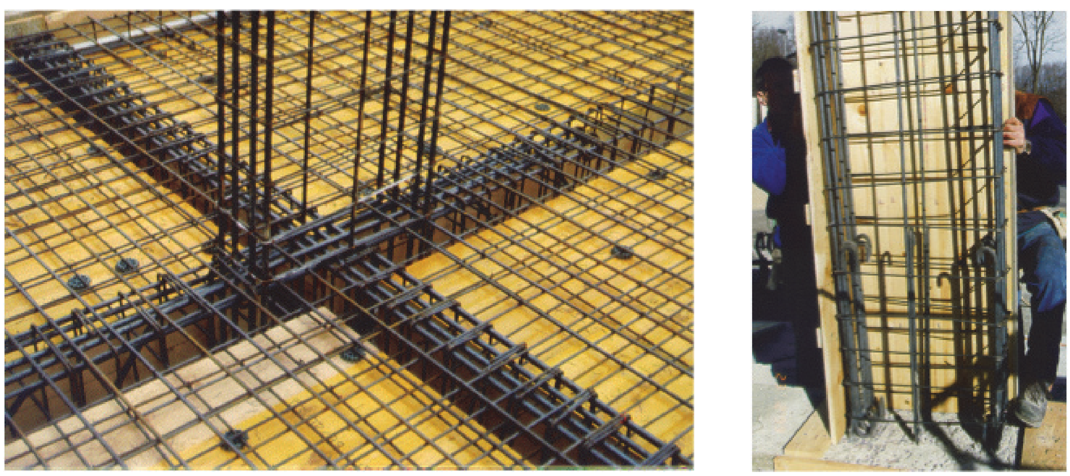


Figure 31: Full-scale four-storey RC frame – construction details

Regarding the materials, *Mander et al. concrete model* and *Menegotto-Pinto steel model* have been employed for defining the concrete and the steel material properties, respectively, used in the RC rectangular sections for modelling columns and beams. The following calibrating parameters have been assigned to each constitutive model:

- Concrete: f_c = 16,300 kPa; f_t = 1900 kPa; ε_c = 0.002 m/m;
- Steel: E_s = 2.00E+008 kPa; f_y = 343,000 kPa; μ = 0.0024.

For the remaining parameters, required to fully describe the mechanical characteristics of the materials, the default values have been set.

Modelling and Loading

Two different models have been created:

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

The masses, proportional to the tributary areas, have been 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: Full-scale four-storey RC frame – lumped and distributed masses (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, has been loaded.

The time step for the dynamic analysis has been set 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:

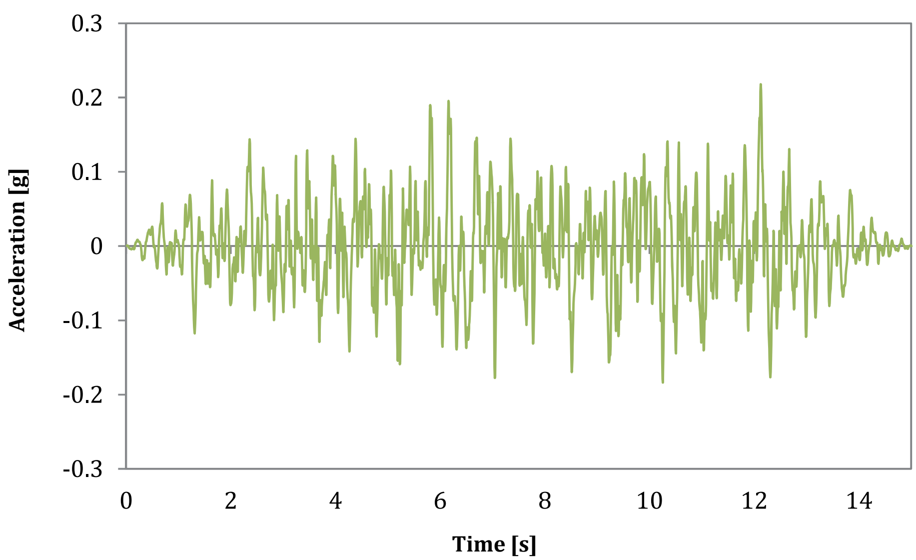


Figure 32: Full-scale four-storey RC frame – artificial acceleration time history for 475 years return period (Acc-475)

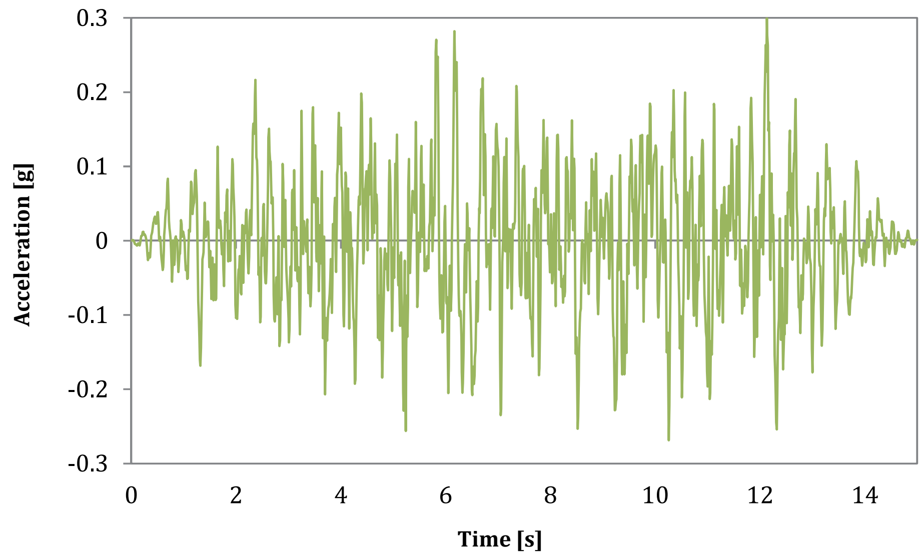


Figure 33: Full-scale four-storey RC frame – artificial acceleration time history for 975 years return period (Acc-975)

The vertical loads have been automatically computed by the program (they are derived from the applied masses, based on the 'g' value).

Dynamic time-history loads have been applied at the base nodes, in terms of accelerations, as indicated in the following sketch of the FE model:

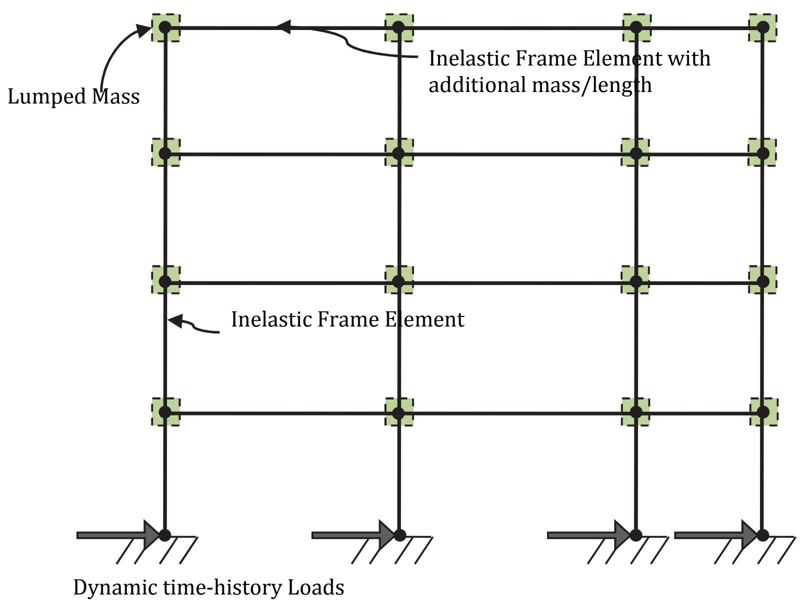


Figure 34: Full-scale four-storey RC frame – FE model sketch

Analysis Type

Nonlinear dynamic time-history analysis.

Results Comparison

The comparison between experimental and analytical results is shown below:

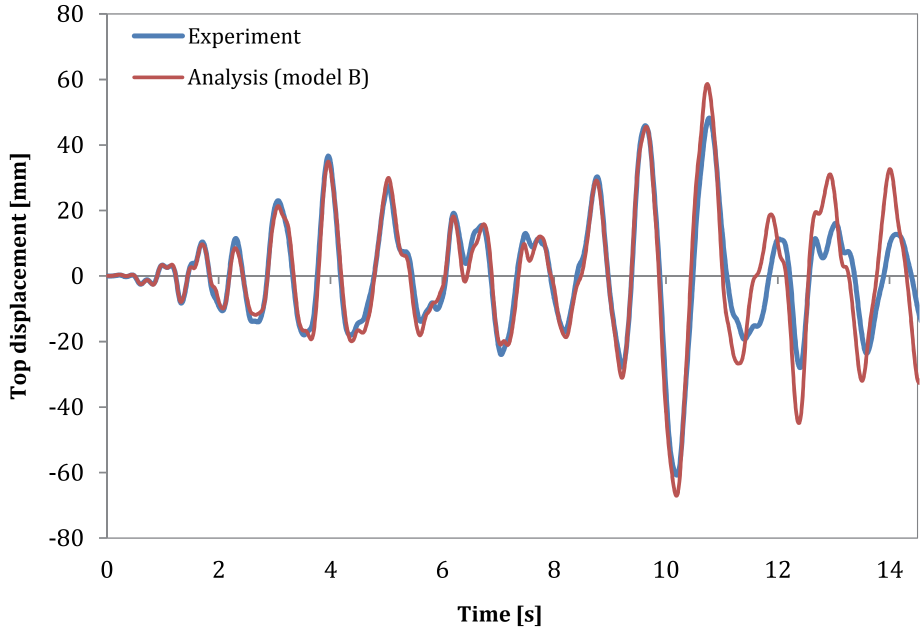


Figure 35: Four-storey RC frame – Experimental vs. Analytical results (top displacement-time (475yrp))

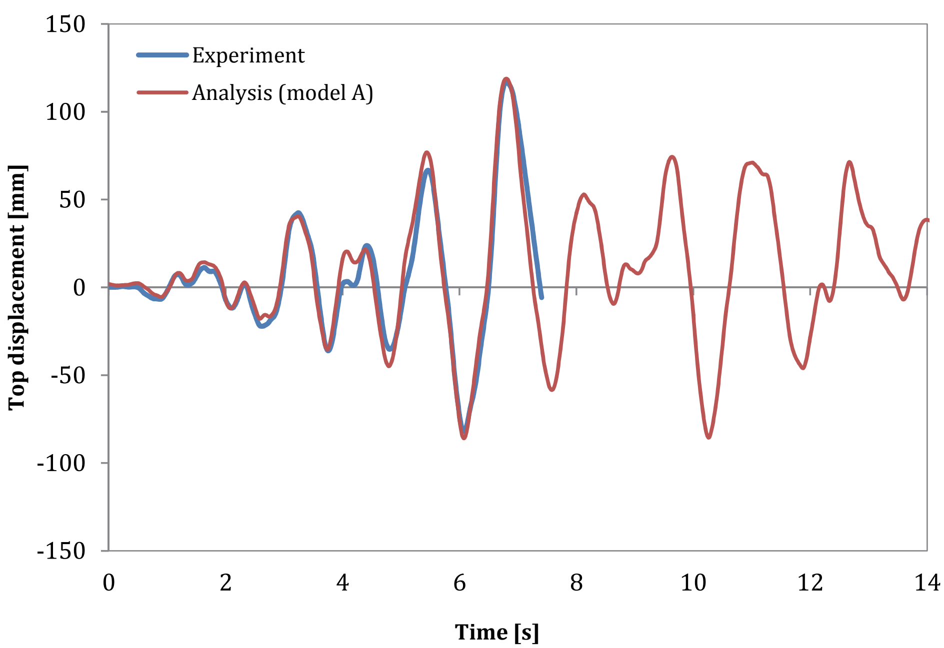


Figure 36: Four-storey RC frame – Experimental vs. Analytical results (top displacement-time (975yrp))

2.5 Concrete irregular frame buildings

GENERAL

This example describes the modelling of a **full-scale three-storey three-dimensional RC building**, which has been designed for gravity loads only, according to the 1954-1995 Greek Code. The prototype building has been 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 has been 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.

In the following, the analytical results will be compared with the experimental benchmark data.

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

Figure 37: Three-storey full-scale RC irregular building tested at the ELSA laboratory of Ispra (Italy)

STRUCTURAL CONFIGURATION AND MATERIAL PROPERTIES

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).

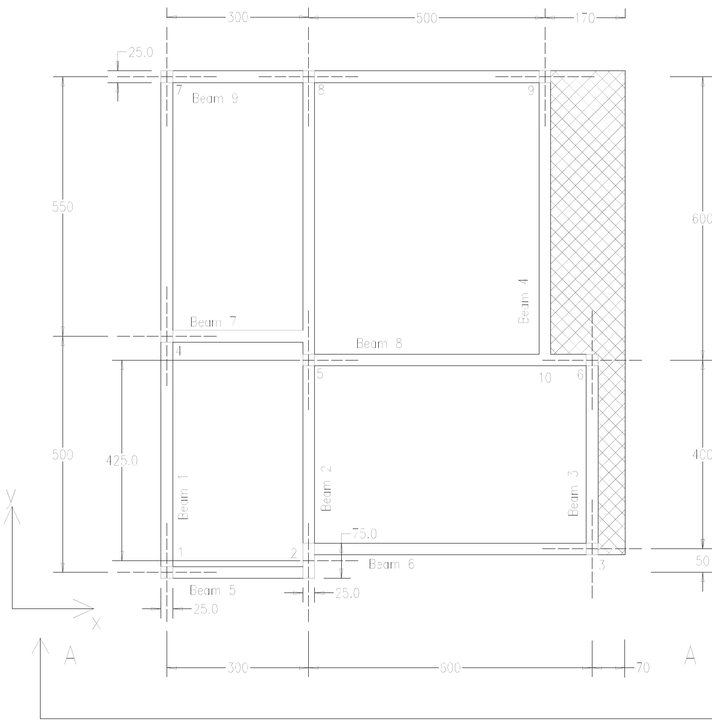


Figure 38: Three-storey full-scale RC irregular building – plan view (Lanese et al., 2008)

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

- f_c = 26.5 MPa; f_t = 0.001 MPa; ε_c = 0.002 mm/mm

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

- E_s = 200,000 MPa; f_y = 459 MPa; μ = 0.004

Modelling and Loading

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

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

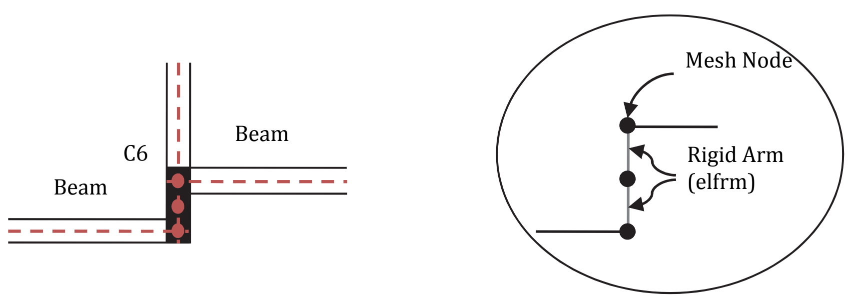


Figure 39: Three-storey full-scale RC irregular building – modelling of column C6 and its connection to the adjacent beams

Regarding the applied masses, they have been 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 have been considered as fully restrained against rotations and translations.

The slabs have been 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 have been loaded in the "Time-history Curves" dialog box (one for X direction, another for Y direction, respectively). The two curves are defined as follows:

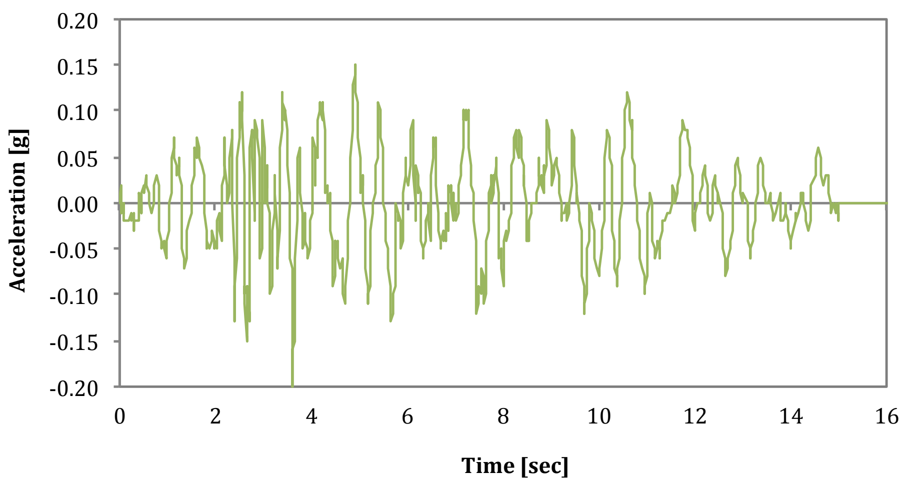


Figure 40: Three-storey full-scale RC irregular building – H-BCR140 accelerogram in the X direction

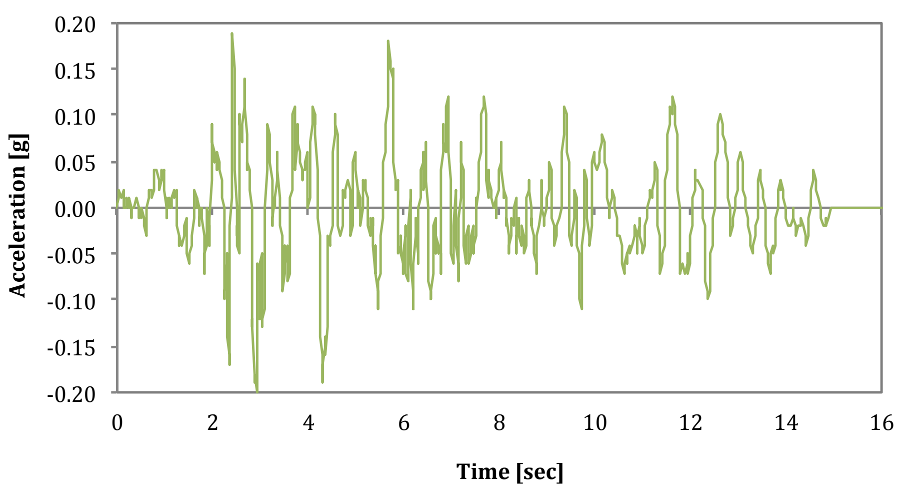


Figure 41: Three-storey full-scale RC irregular building – H-BCR230 accelerogram in the Y direction

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

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

The FE model of the building is presented below:

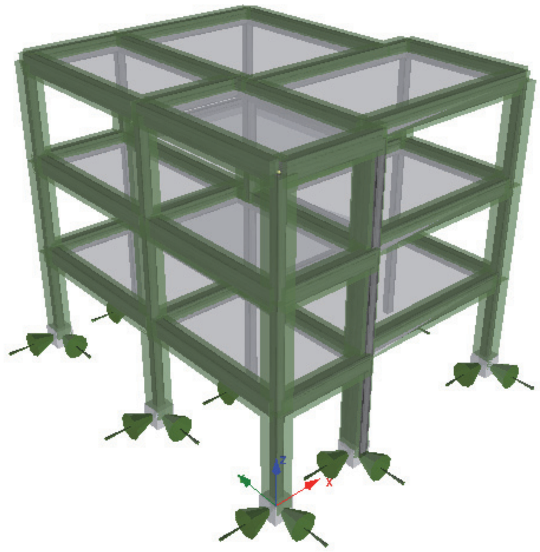


Figure 42: Three-storey full-scale RC irregular building – FE model view

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

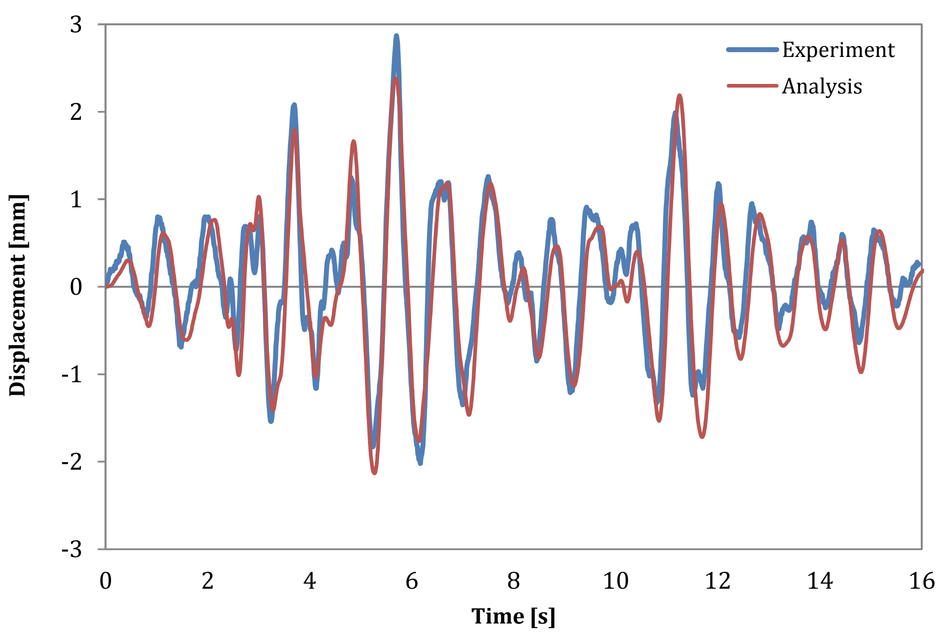


Figure 43: Three-storey full-scale RC irregular building – Experimental vs. Analytical results (displacement-time

2.6 Concrete wall

GENERAL

This example describes the modelling of a **full-scale seven-storey RC structural wall** which has been 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 has been 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 for the strongest input motion (EQ4 in the following) will be compared with the experimental benchmark data.

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

Figure 44: Full-scale seven-storey RC shear wall building tested at the NEES Large High-Performance Outdoor Shake Table at UCSD's Englekirk Structural Engineering Center

STRUCTURAL CONFIGURATION AND MATERIAL 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 dimensions in elevation and in plan are indicated in the figures below.

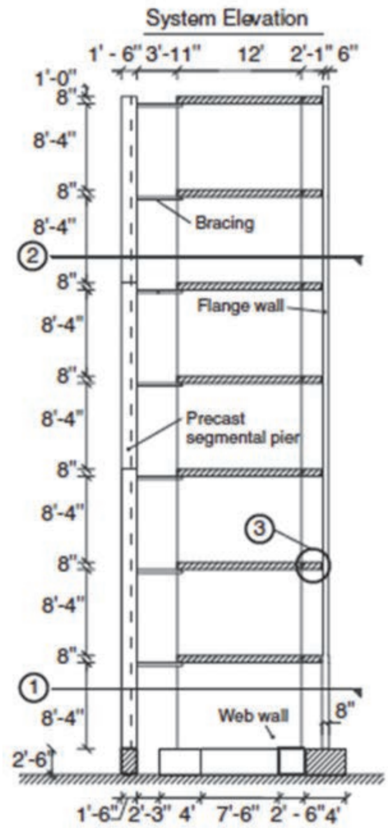


Figure 45: Full-scale seven-storey RC shear wall building – frame geometry (elevation) (Martinelli and Filippou, 2009)

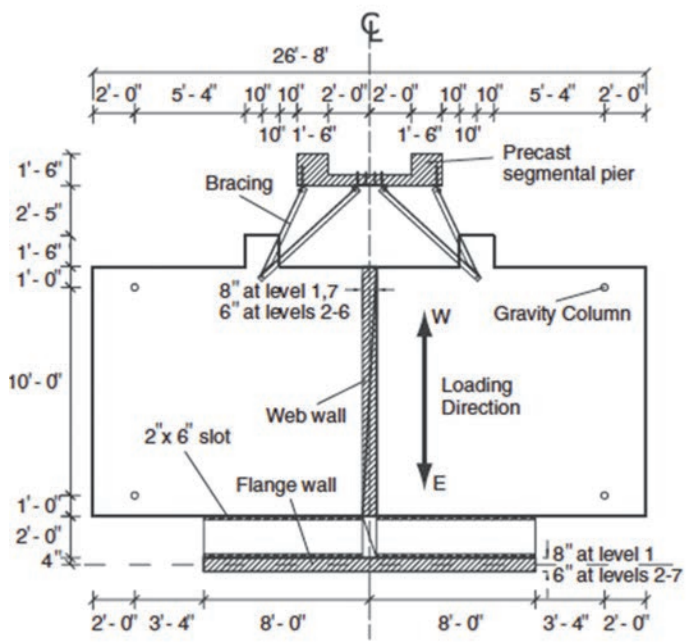


Figure 46: Full-scale seven-storey RC shear wall building – frame geometry (floor plan view) (Martinelli and Filippou, 2009)

The geometrical details of the structural elements, i.e. walls and gravity columns, are given in the following table, whilst in Figure 47 is shown the walls reinforcement at different levels.

Table 5: Full-scale seven-storey RC shear wall building – geometrical details of walls and gravity columns

Storey	Web Wall	Flange Wall	Gravity Columns
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

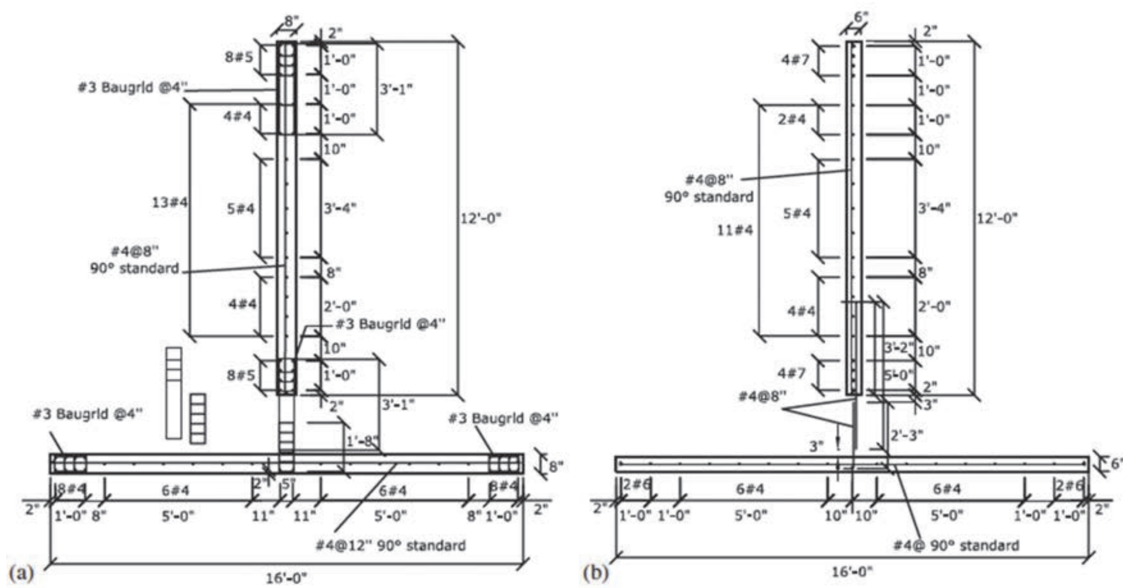


Figure 47: Full-scale seven-storey RC shear wall building – wall reinforcement at the first level (left); wall reinforcement at levels 2 to 6 (right) (Martinelli and Filippou, 2009)

Regarding the materials, *Mander et al. concrete model* has been employed for defining the concrete material properties used in the RC rectangular sections for modelling the walls. The following calibrating parameters have been assigned:

- Concrete: f_c = 5426.39 psi; f_t = 542.71 psi; ε_c = 0.00269 in/in.

Menegotto-Pinto steel model (with the calibrating parameters listed below) has been employed for defining the steel material used for the wall reinforcement:

- Steel: E_s = 2.9007E+007 psi; f_y = 66497.42 psi; μ = 0.014.

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

For the remaining parameters, required to fully describe the mechanical characteristics of each material, the default values have been set.

MODELLING AND LOADING

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

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

The precast column, since it has been designed in order to remain elastic, has been modelled through an elastic frame element (elfrm) with the following properties: EA = 3.9720E+009 [lb]; EI (axis2) = 3.3035E+012 [lb-in^2]; EI (axis3) = 1.7548 E+011 [lb-in^2]; GJ = 5.7887E+010 [lb-in^2].

The modelling of the slabs has been realized through the definition of nodal constraints (i.e. rigid diaphragms).

The masses have been assigned in a lumped fashion to each floor node. The applied values are summarized in the table below:

Table 6: Full-scale seven-storey RC shear wall building – lumped masses (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 have been 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) has been 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:

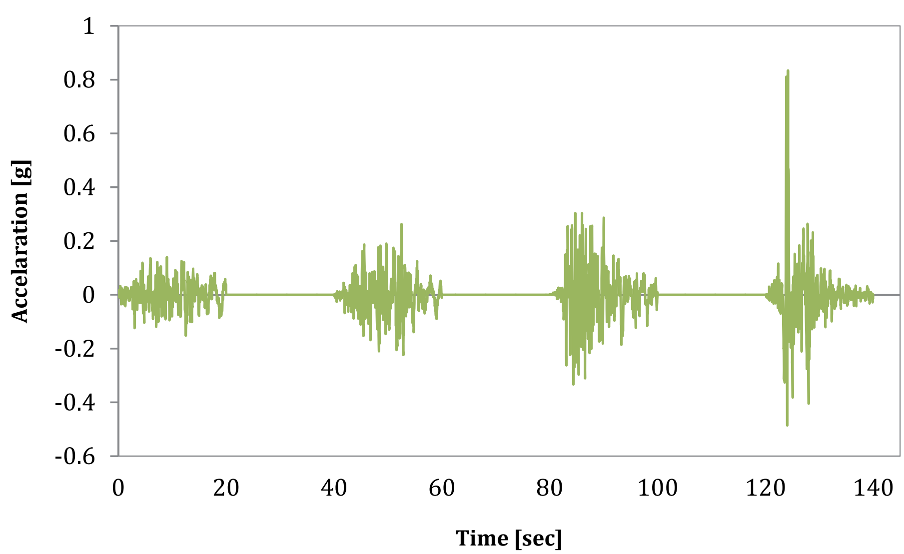


Figure 48: Seven-storey full-scale RC shear wall building – input ground motion

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

NOTE: A 1% tangent stiffness-proportional damping has been applied to the structural model.

Two different views of the FE model are presented below:

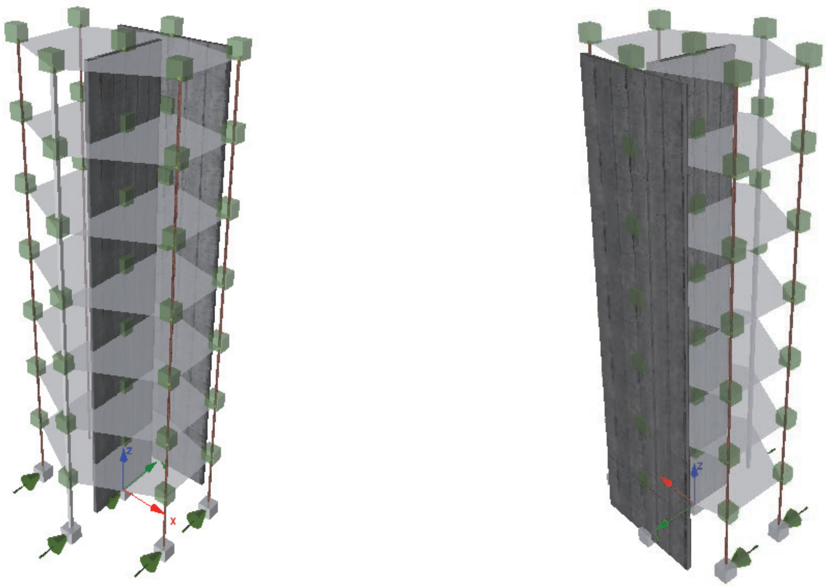


Figure 49: Seven-storey full-scale RC shear wall building – FE model views

ANALYSIS TYPE

Nonlinear dynamic time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

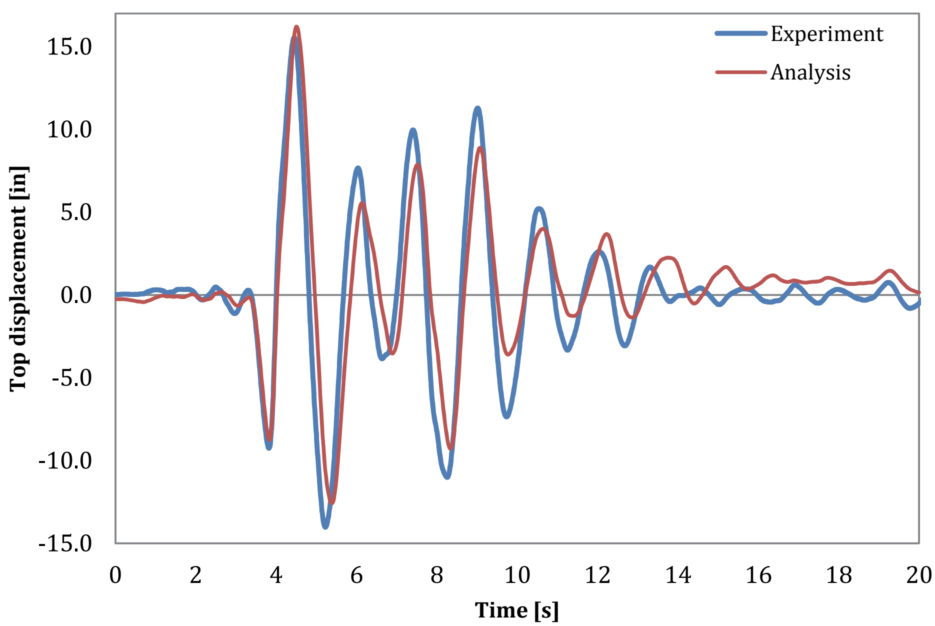


Figure 50: Seven-storey full-scale RC shear wall building – Experimental vs. Analytical results (top displacement-time (EQ4))

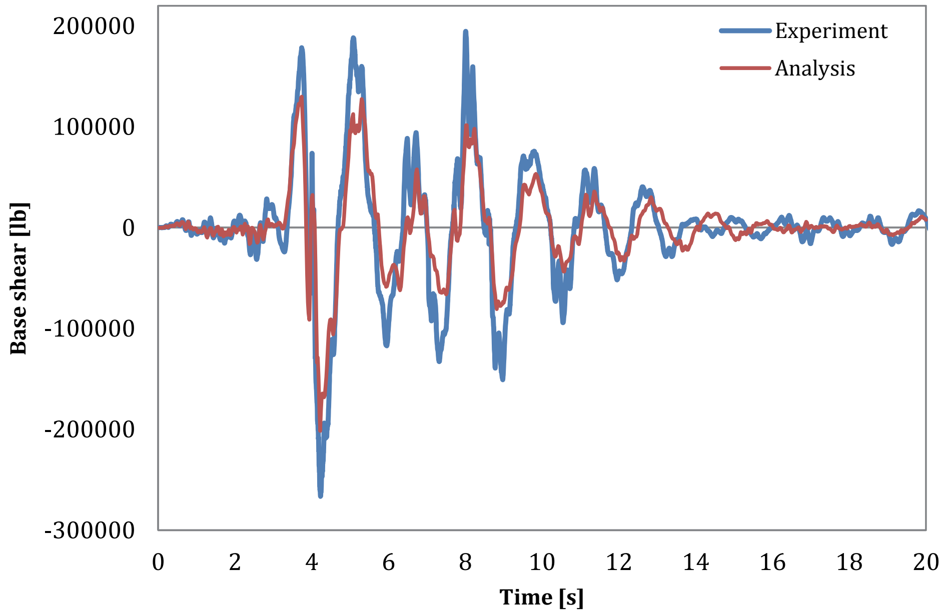


Figure 51: Seven-storey full-scale RC shear wall building – Experimental vs. Analytical results (base shear-time (EQ4))

2.7 Steel regular frame buildings

GENERAL

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 has been 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 have been connected horizontally to the columns by high strength bolts through gusset plates to allow for the rotation at the column top.

The analytical results will be compared with the experimental benchmark data.

NOTE: Further information about the test specimen and the experimental campaign can be found in Matsumiya et al. (2004) and Nakashima et al. (2006).

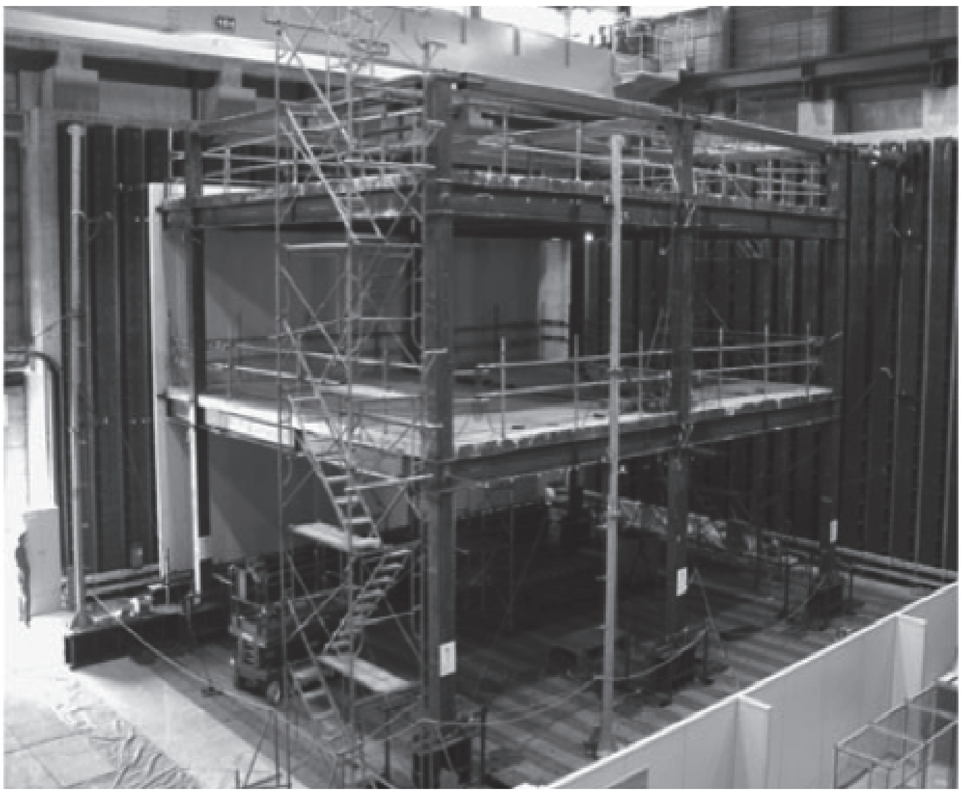


Figure 52: Three-storey, three-dimensional steel moment frame (Nakashima et al., 2006)

STRUCTURAL CONFIGURATION AND MATERIAL 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”).

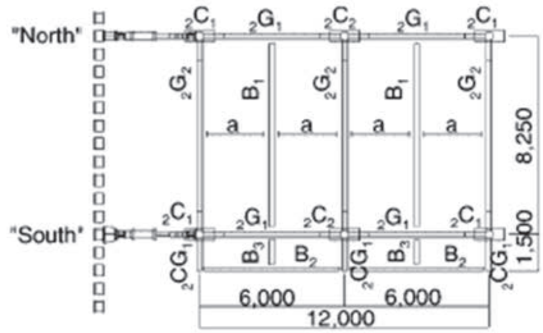


Figure 53: Three-storey, three-dimensional steel moment frame – plan view (mm) (Nakashima et al., 2006)

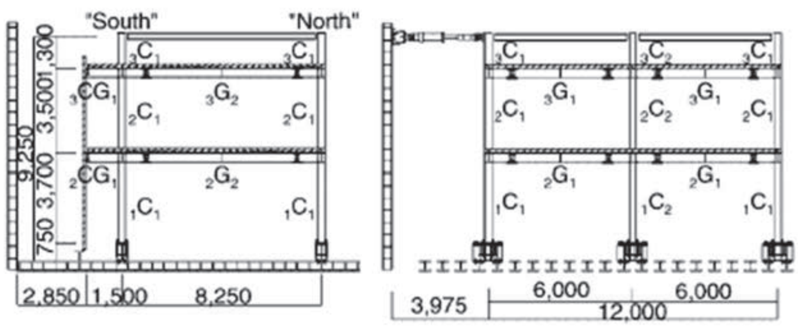


Figure 54: Three-storey, three-dimensional steel moment frame – elevations (mm) (Nakashima et al., 2006)

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.

The figure below shows the two connection details.

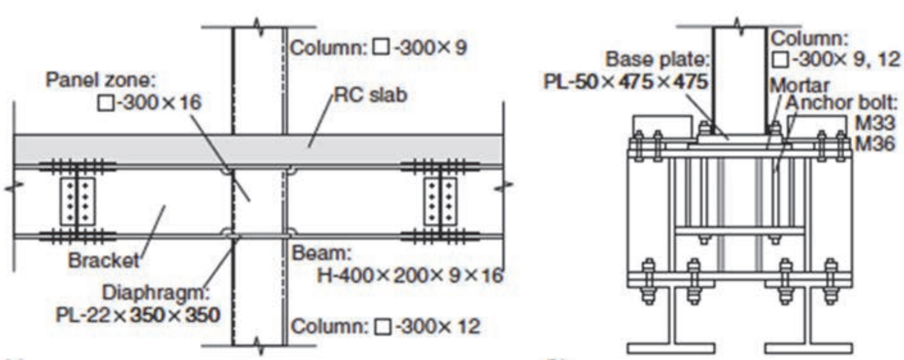


Figure 55: Three-storey, three-dimensional steel moment frame – beam-to-column (left) and column base (right) connection (Nakashima et al., 2006)

Regarding the materials, a *bilinear steel model*, according with the measured material properties after testing (Nakashima et al., 2006), has been employed for defining the steel material properties. The following calibrating parameters have been assigned to each constitutive model:

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

For the remaining parameters, required to fully describe the mechanical characteristics of the materials, the default values have been set.

MODELLING AND LOADING

The lateral resisting frame has been modelled through steel beam and column elements. Each structural member has been modelled through a 3D force-based inelastic frame element (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations is set to 100.

Panel zones have been 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 have been modelled with an elastic frame element (elfrm) with the following properties: EA = 1.0000E+010 [kN]; EI (axis2) = 1 [kN-m^2]; EI (axis3) = 1 [kN-m^2]; GJ = 1 [kN-m^2].

Rotational springs have been inserted at the bottom of the first storey columns to allow for the rotational flexibility of the column base. They have been modelled with link elements, in which for the F_1, F_2, F_3, M_1, and M_3 degrees of freedom a linear symmetric response curve has been chosen with a value of stiffness k0 equal to 1.00E+09, whereas for the M_2 degree-of-freedom a trilinear symmetric response curve has been chosen.

Regarding the restraining conditions, all the base nodes have been fully fixed.

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

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 has been set to 0.1 seconds. The cyclic loading has been applied at the top lateral node as a static time-history load in terms of displacements in the X direction.

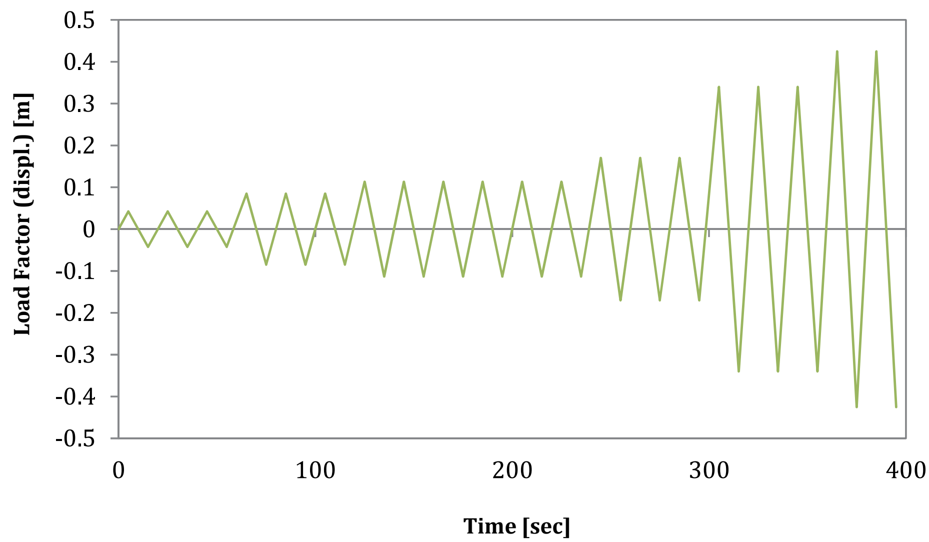


Figure 56: Three-storey, three-dimensional steel moment frame – applied cyclic loading

The FE model of the steel frame is presented below:

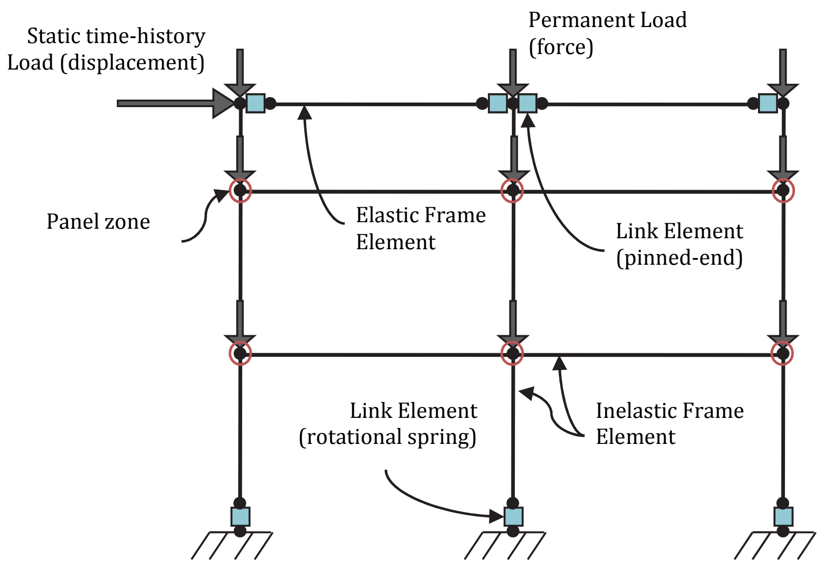


Figure 57: Three-storey, three-dimensional steel moment frame – FE model sketch

ANALYSIS TYPE

Nonlinear static time-history analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

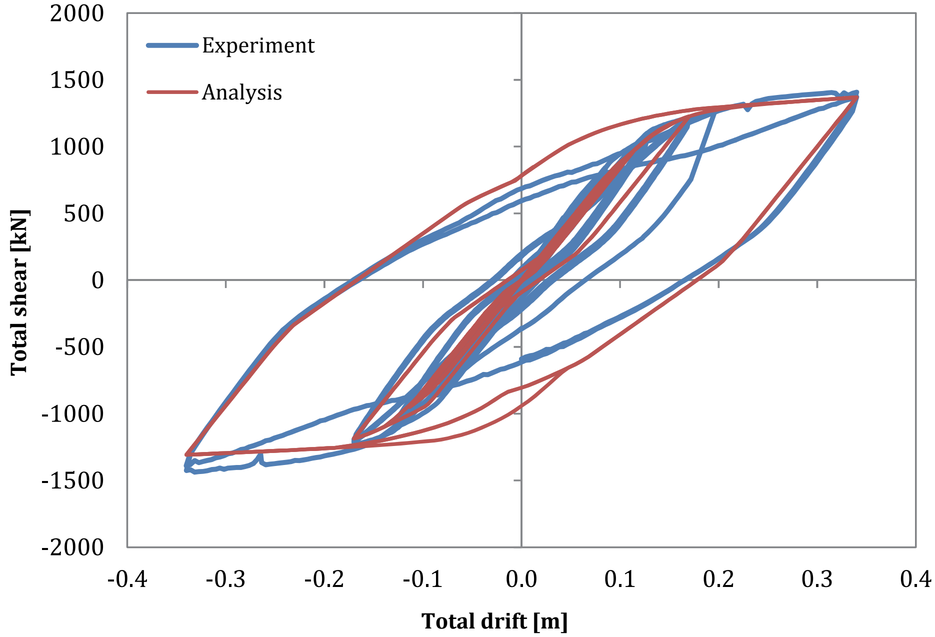


Figure 58: Three-storey, three-dimensional steel moment frame – Total shear vs. Total drift

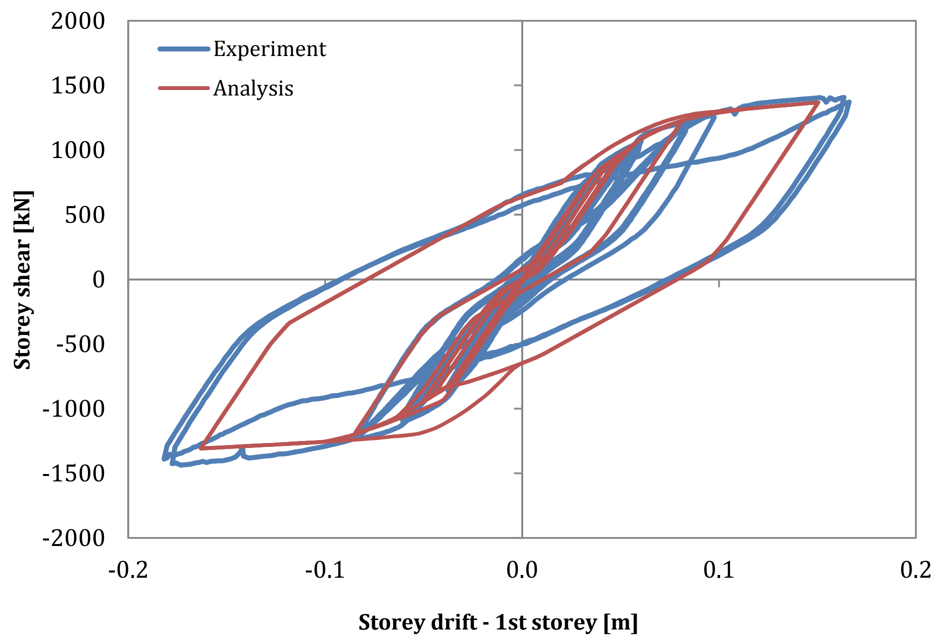


Figure 59: Three-storey, three-dimensional steel moment frame – Storey shear vs. Storey drift – 1st storey

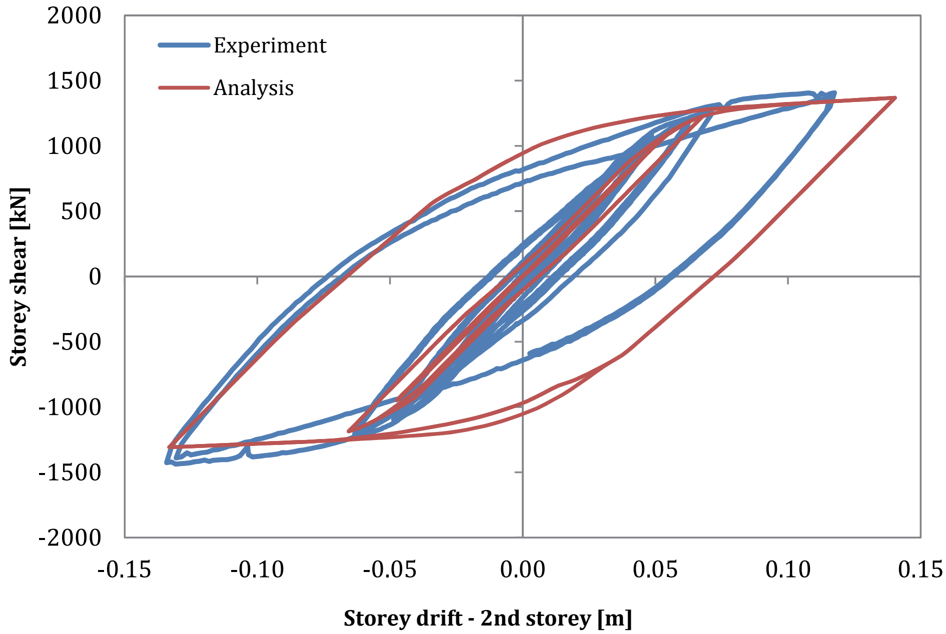


Figure 60: Three-storey, three-dimensional steel moment frame – Storey shear vs. Storey drift – 2nd storey

2.8 Steel irregular frame buildings

General

This example describes the modelling of a **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 will be compared with the experimental benchmark data.

NOTE: Full details on structural geometry and material characteristics may be found in Pavan et al. (2008).

Figure 61: Four-storey 3D steel moment resisting frame (NRIESDP, 2007)

Structural Configuration and Material Properties

The model consists of two steel frames composed by two bays 5 m long, along NS (i.e. X) direction, and three steel frames (with one bay 6 m long) in EW (i.e. Y) direction, as shown in Figure 62.

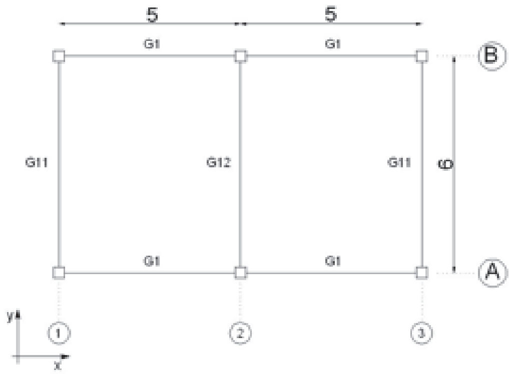


Figure 62: Four-storey 3D steel moment resisting frame – plan view (from Pavan et al., 2008)

The interstorey height is equal to 3.5 m, with the exception of the ground floor, the height of which is equal to 3.875 m.

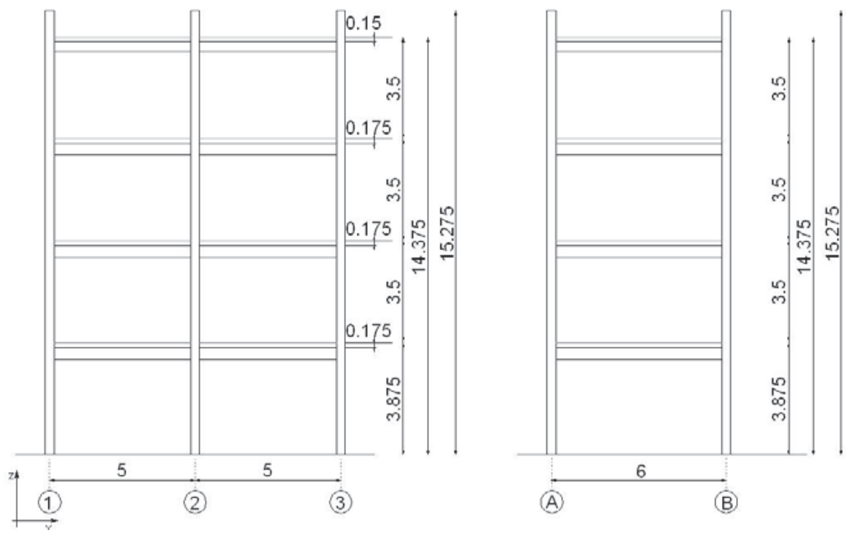


Figure 63: Four-storey 3D steel moment resisting frame – elevation views (from Pavan et al., 2008)

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

Table 7: Four-storey 3D steel moment resisting frame – geometrical details of structural elements

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 of composite deck floors (at the second, third and fourth level) and reinforced concrete (at the roof), each beam section has been modelled as a composite I-section.

The *Mander et al. concrete model* has been employed for defining the concrete material properties used in the composite I-sections for modelling the slabs. The calibrating parameters are listed in Table 8.

Table 8: Four-storey 3D steel moment resisting frame – calibrating parameters for concrete materials

PARAMETERS	2nd Level	3rd Level	4th 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, a *bilinear model* has been employed for defining the steel material properties. In particular, eight different kind of steel materials have been defined. The calibrating parameters are listed in the following tables:

Table 9: Four-storey 3D steel moment resisting frame – calibrating parameters for steel materials (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 10: Four-storey 3D steel moment resisting frame – calibrating parameters for steel materials (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 have been modelled through 3D force-based inelastic frame elements (infrmFB) with 4 integration sections. The number of fibres used in section equilibrium computations has been set to 100;
- Model B: Columns and beams have been modelled through 3D displacement-based inelastic frame elements (infrmDB). The number of fibres used in section equilibrium computations has been set to 100.

The masses have been computed starting from the values of weights given by the organizing committee and have been assigned to each beam section as additional mass.

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

The slabs have been modelled by introducing a rigid diaphragm for each floor level.

Table 11: Four-storey 3D steel moment resisting frame – applied masses

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

In order to run a nonlinear dynamic time-history analysis, three time-history curves have been 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 dampen out the structural motion after each earthquake run).

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

The three input motions are given in the figures below:

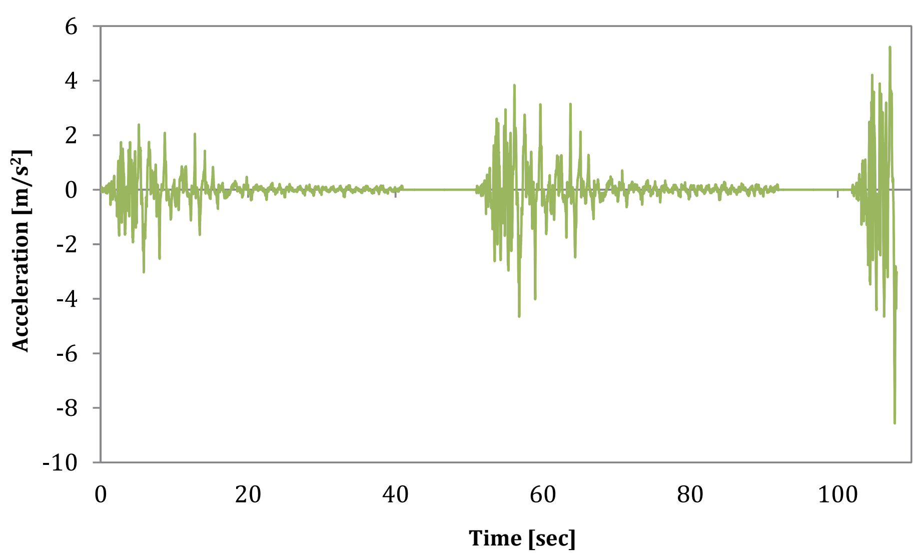


Figure 64: Four-storey 3D steel moment resisting frame – post-test shaking table acceleration time-history (NS-component)

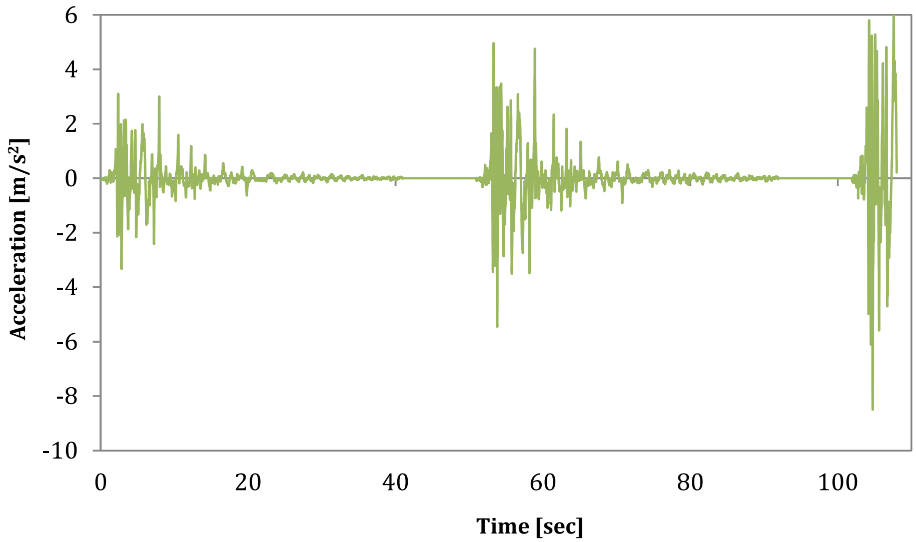


Figure 65: Four-storey 3D steel moment resisting frame – post-test shaking table acceleration time-history (EW-component)

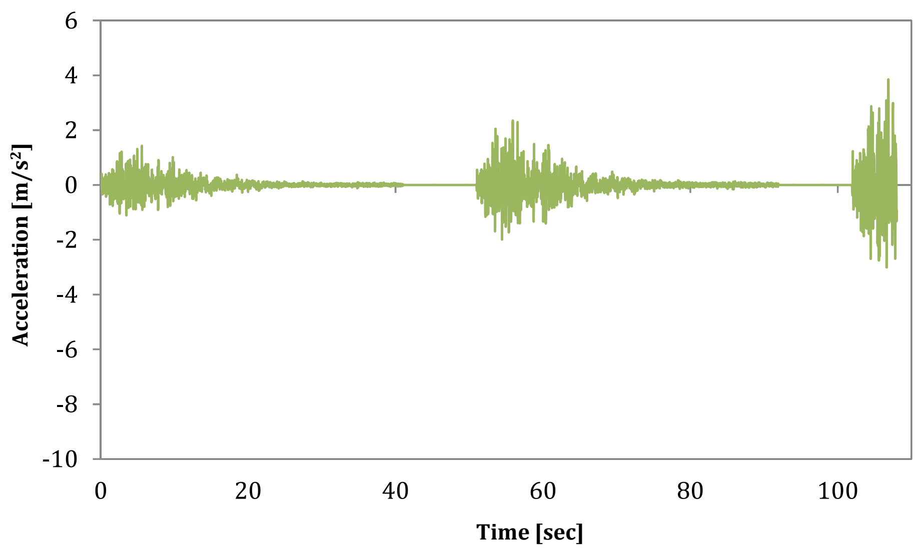


Figure 66: Four-storey 3D steel moment resisting frame – post-test shaking table acceleration time-history (vertical component)

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

NOTE: A 0.5% tangent stiffness-proportional damping has been applied to the structural model.

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

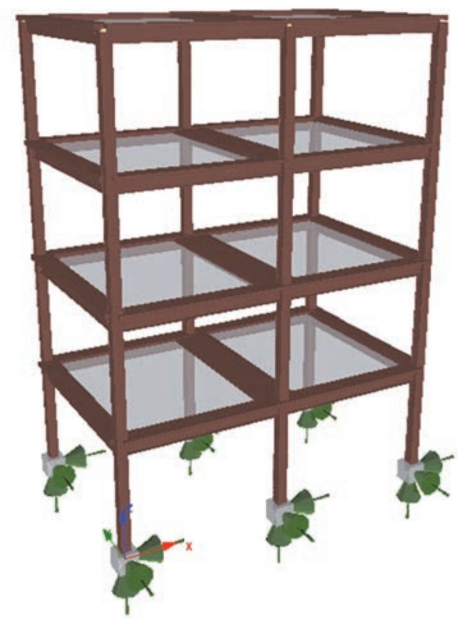


Figure 67: Four-storey 3D steel moment resisting frame – FE model view

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:

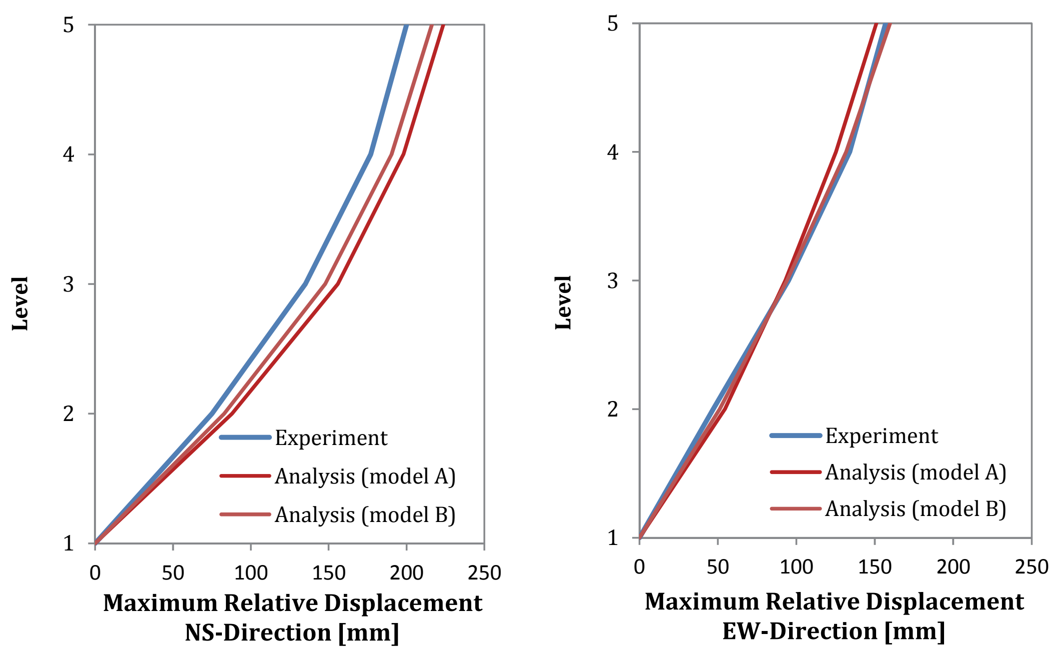


Figure 68: Four-storey 3D steel moment resisting frame – Experimental vs. Analytical results (maximum relative displacement-floor level)

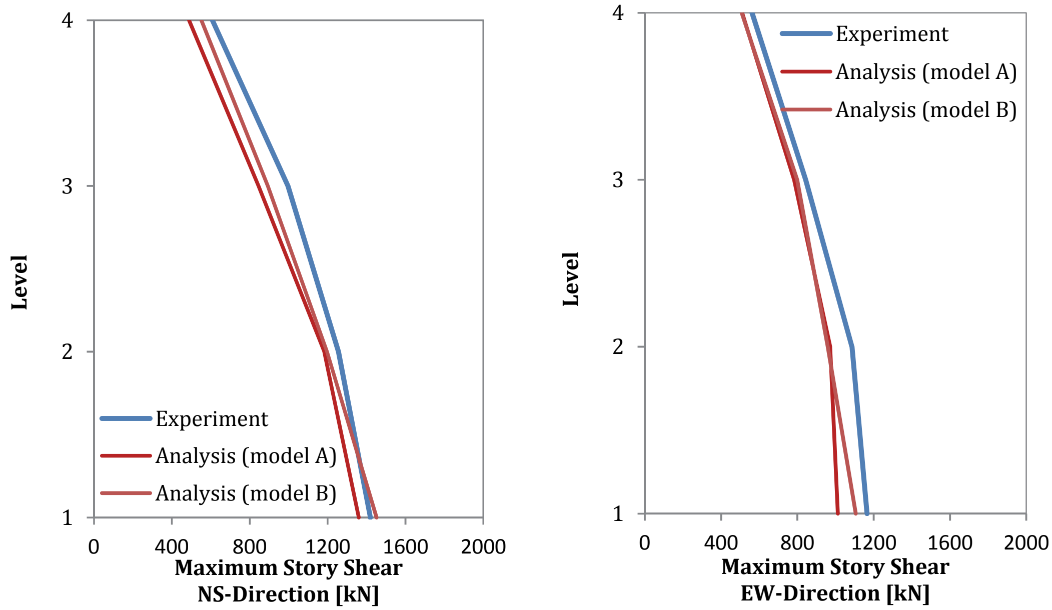


Figure 69: Four-storey 3D steel moment resisting frame – Experimental vs. Analytical results (maximum storey shear-floor level)

2.9 Composite steel/RC buildings

General

This example describes the modelling of a **1/3-scale one-storey composite frame** with rigid connections, which has been designed in accordance to current design standards. The steel-concrete frame has been tested under incremental loadings.

In the following, the analytical results will be compared with the experimental benchmark data.

NOTE: Further information about the test specimen and the experimental campaign can be found in Lanhui Guo et al (2013).

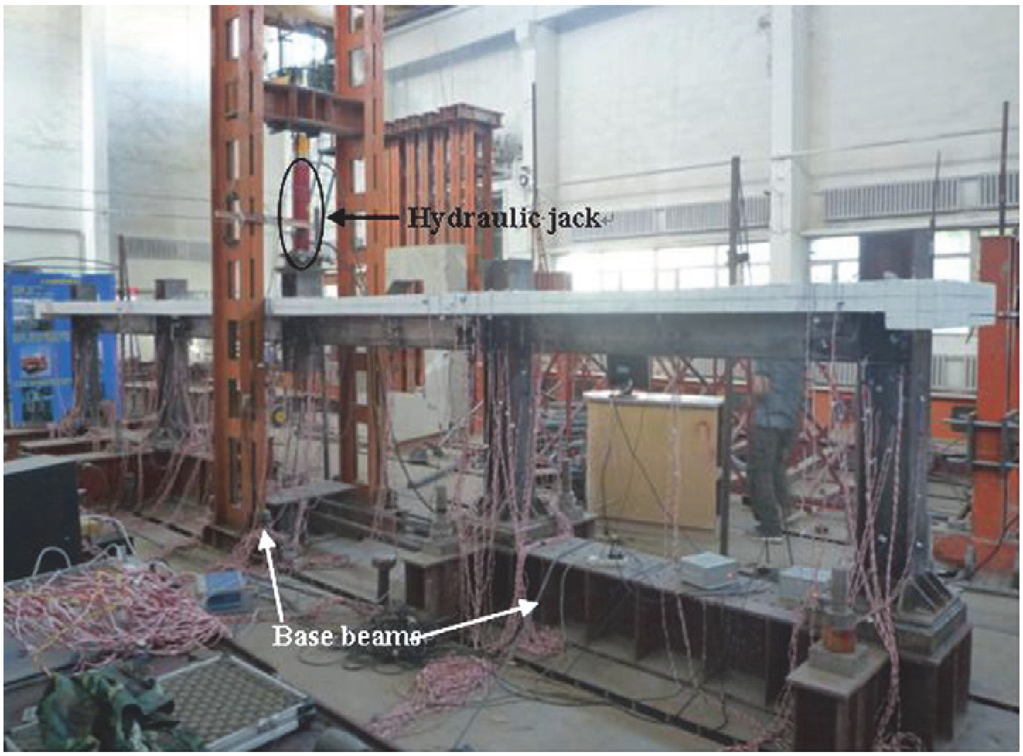


Figure 70: 1/3-scale one-storey composite frame

Structural Configuration and Material Properties

The model consists of a one-storey composite frame with four bays. The column in the middle has been not supported, in order to simulate the loss of a column. Since the frame is 1/3-scale, the storey height is equal to 1.2 m, whilst the bay length is 2 m, as indicated in Figure 71. The steel beams have been fully welded to the flanges of the steel columns, in order to guarantee rigid connections. The cross sections of beams and columns have the following dimensions respectively, H200x100x5.5x8 and H200x200x8x12. The RC slabs have been designed with a depth and a width of 100 mm and 800 mm, respectively.

The design of the shear studs for the composite joints in the frame has been based on the Chinese Code for Design of Steel Structures (GB 50017-2003) following full composite design assumption. The shear studs with diameter of 16 mm have been welded to the steel beam with a spacing of 100 mm.

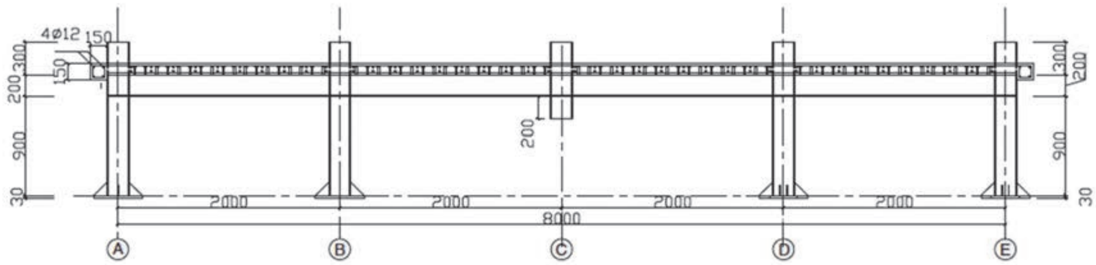


Figure 71: 1/3-scale one-storey composite frame – frame geometry

The reinforcement mesh ratio for the RC slab is 0.85%. Longitudinal plane reinforcements with the diameter of 12 mm are placed in two layers with equal spacing along the width of the slab. Two layers of 8-mm-diameter plane bars are supplied as transverse reinforcement to prevent longitudinal splitting failure of the concrete slab, as shown in the figure below.

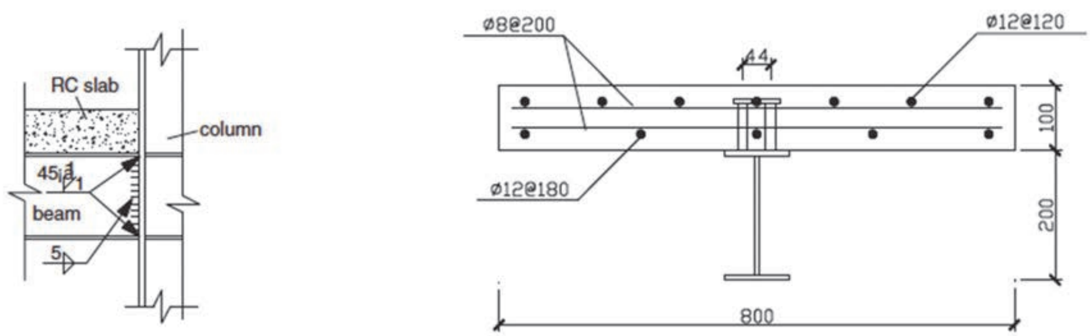


Figure 72: 1/3-scale one-storey composite frame – full-welded beam-to-column connection (left); cross section of composite beam (right)

Regarding the materials, *Mander et al. concrete model* and *bilinear model with kinematic strain-hardening* have been employed for defining the concrete and the steel material properties used in the sections for modelling the composite beams and the steel columns, respectively. The following calibrating parameters have been assigned to each constitutive model:

- Concrete: f_c = 22,000 kPa; f_t = 2640 kPa; ε_c = 0.002 m/m;
- Steel (for beam reinforcement): E_s = 1.95E+008 kPa; f_y = 330000 kPa; μ = 0.005;
- Steel (for beams): E_s = 2.10E+008 kPa; f_y = 300,000 kPa; μ = 0.005;
- Steel (for columns): E_s = 2.10E+008 kPa; f_y = 275,000 kPa; μ = 0.005.

Modelling and Loading

Two models, respectively named *model A* and *model B*, have been created. Model A employs materials with the properties listed in the previous paragraph, whilst model B employs the same properties of model A with the exception of the steel material used for modelling the beams: in this case the yielding strength has been set to 275,000 kPa.

In both cases, columns and beams have been modelled through 3D displacement-based inelastic frame elements (infrmDB). The number of fibres used in section equilibrium computations has been set to 150.

The modelling of the steel-concrete composite connection of vertical steel shear studs was realized through link elements (for details refer to the input files of the model).

All foundation nodes have been considered as partially restrained against rotations and translations realized through link elements (for details refer to the input files of the model).

In order to run a static pushover analysis, an incremental load has been applied to the midspan node in the vertical direction.

The vertical loads have been automatically computed by the program (they are derived from the applied masses, based on the value assigned to acceleration due to gravity).

The FE model of the composite frame is presented below:

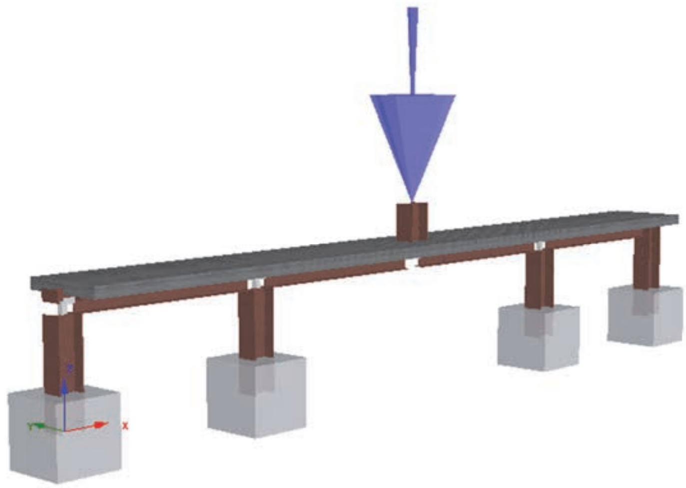


Figure 73: 1/3-scale one-storey composite frame – FE model view

ANALYSIS TYPE

Static pushover analysis.

RESULTS COMPARISON

The comparison between numerical and experimental results is shown hereafter:

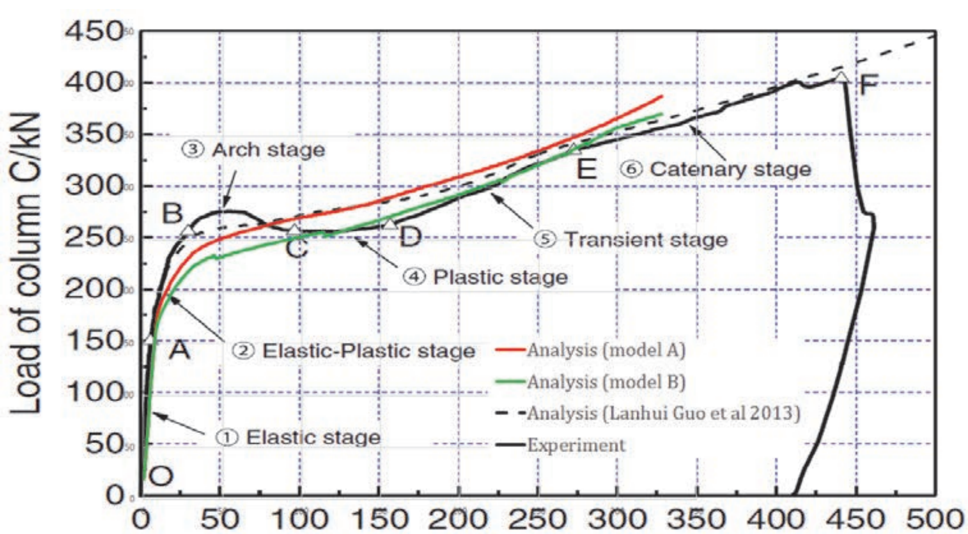


Figure 74: 1/3-scale one-storey composite frame – experimental vs. analytical results

2.10 Steel connections

2.10.1 "Simplified" modelling

GENERAL

This example describes the modelling of three steel joints (respectively named JB1-3A, A2 and EP2 in the following) tested by Bursi et al. (2002), Korol et al. (1990) and Broderick & Thomson (2002), respectively, under different cyclic loadings.

The analytical results, obtained by employing the modified Richard-Abbot hysteretic model, which is able to simulate the cyclic behaviour of steel and composite joints, is compared with the experimental results.

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

STRUCTURAL CONFIGURATION

The test specimen JB1-3A (Bursi et al., 2002) consists of an extended end-plate connection with 18 mm plate thickness (see Figure 75 (a)). Specimen A2 (Korol et al., 1990) consists of an extended end-plate connection with stiffeners in the column web as well as in the end-plate (see Figure 75 (b)), whilst specimen EP2 (Broderick & Thomson, 2002) is a flush end-plate joint (see Figure 75 (c)). The geometrical dimensions are indicated in the figure below.

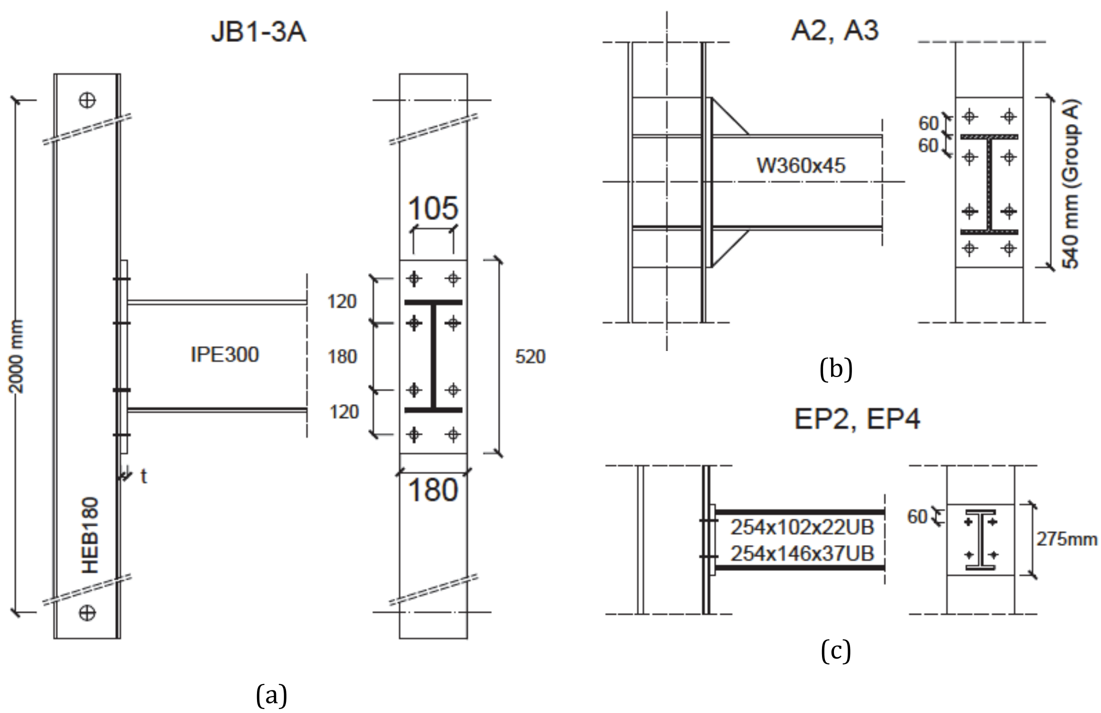
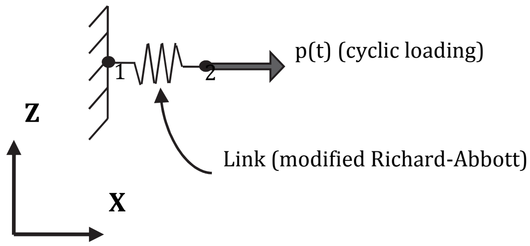


Figure 75: Steel joints – geometric representation of JB1-3A (a), A2 (b) and EP2 (c)

MODELLING AND LOADING

Each model consists of two coincident joints connected by a link element.



<u>Link Properties</u>
(see paragraph *Modelling and Loading*)

Figure 76: Steel joints – FE model sketch (1)

The link elements have been defined with the following properties: for the F_1, F_2, F_3, M_1 and M_2 degrees of freedom a linear symmetric response curve has been chosen with a value of stiffness k0 equal to 1.00E+012 kN/m, whereas for the M_3 degree-of-freedom a modified Richard-Abbott response curve has been chosen with the properties summarized in the tables below:

Table 12: Steel joints – modified Richard-Abbott response curve parameters (ascending branches)

PARAMETERS	JB1-3A (Bursi)	A2 (Korol)	EP2 (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 13: Steel joints – modified Richard-Abbott response curve parameters (descending branches)

PARAMETERS	JB1-3A (Bursi)	A2 (Korol)	EP2 (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 has been loaded in the “Time-history Curves” dialog box. It has been defined as follows:

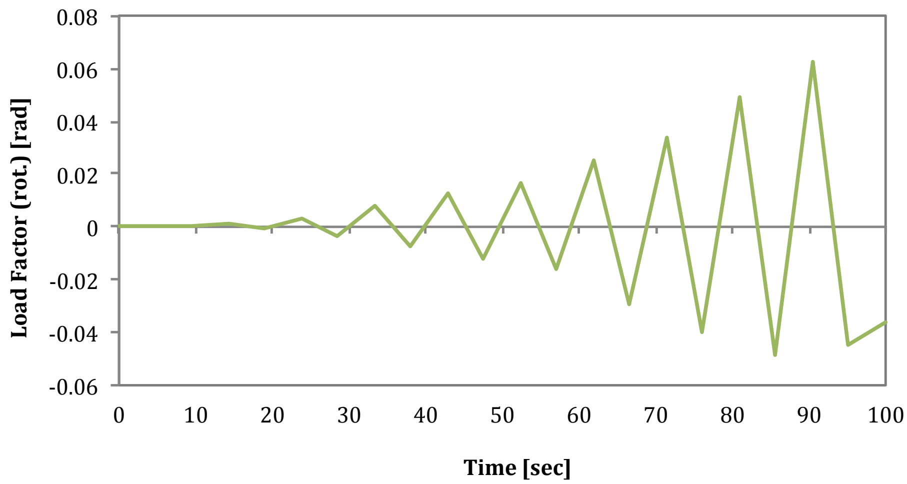


Figure 77: Steel joints – applied cyclic loading

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

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

A sketch of the FE model is shown below:

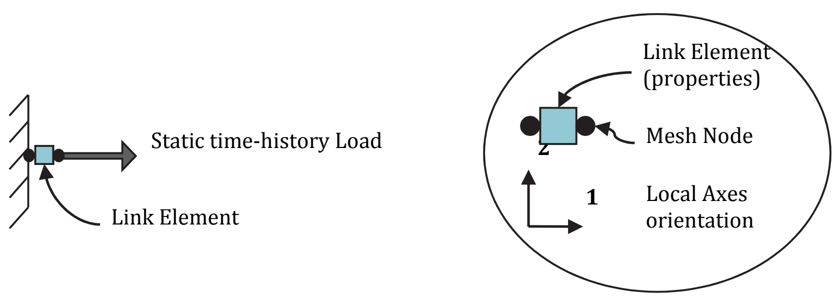


Figure 78: Steel joints – FE model sketch (2)

Analysis Type

Nonlinear static time-history analysis.

Results Comparison

The comparison between numerical and experimental results is shown hereafter:

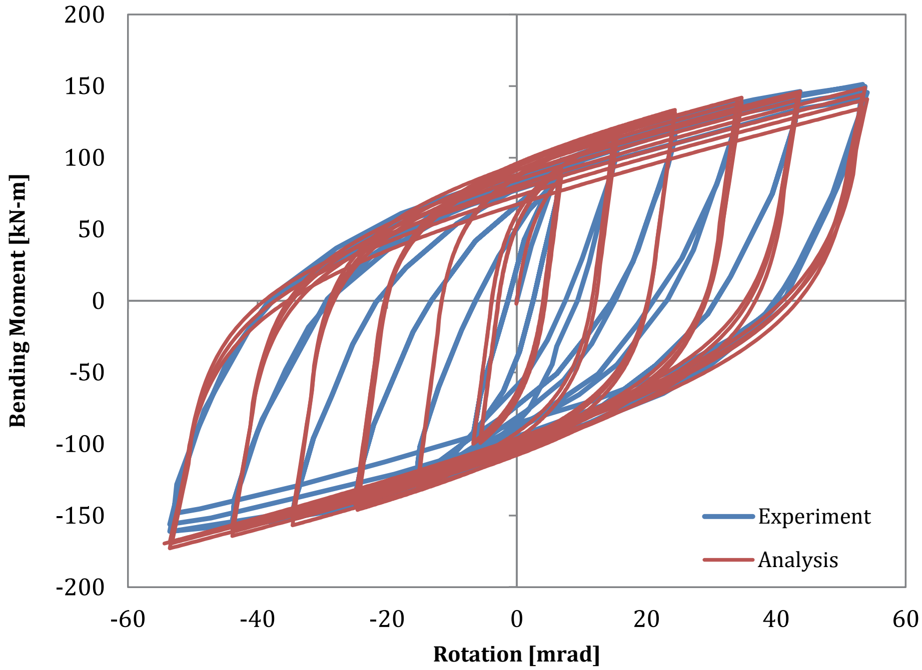


Figure 79: Steel joints – Experimental vs. Analytical results (moment-rotation – Bursi et al., 2002)

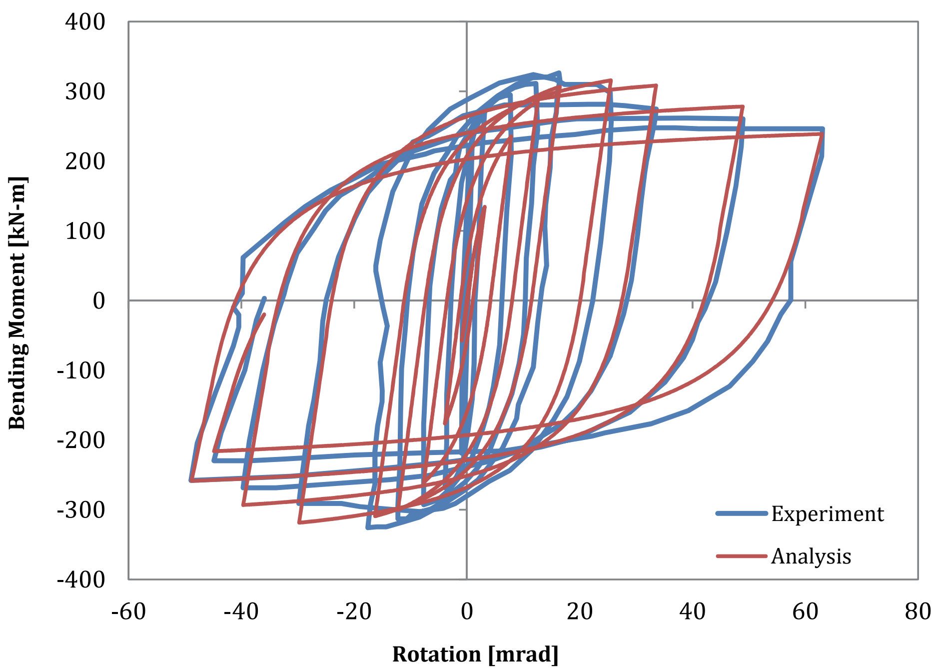


Figure 80: Steel joints – Experimental vs. Analytical results (moment-rotation – Korol et al., 1990)

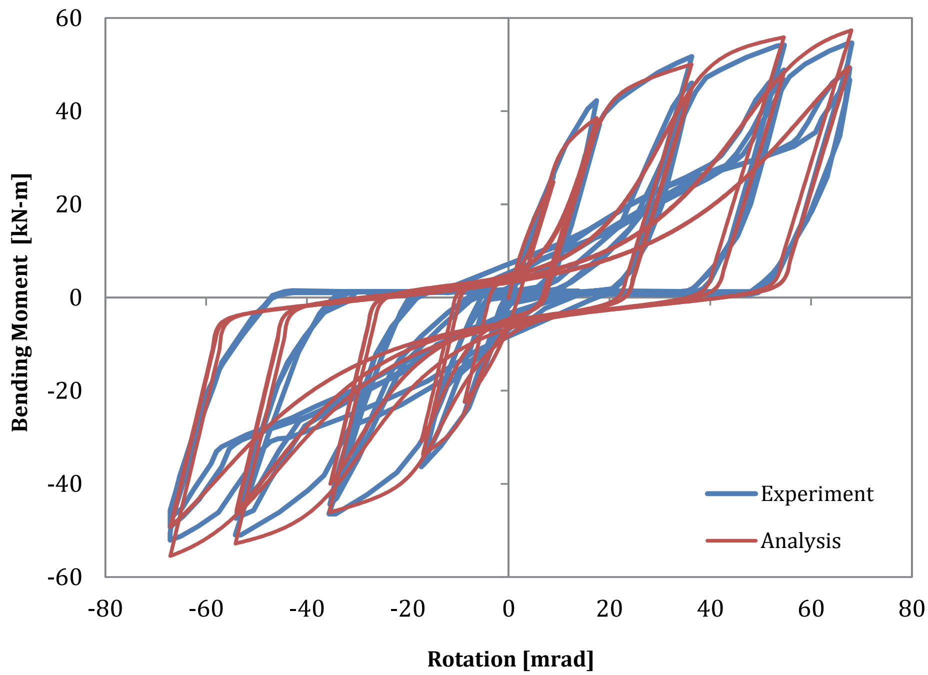


Figure 81: Steel joints – Experimental vs. Analytical results (moment-rotation – Broderick et al., 2002)

2.10.2 "Detailed" modelling

GENERAL

This example describes the modelling of two **full-scale partially-restrained bolted beam-to-column connections** (respectively named FS-01(CA-02) and FS-02(CA-04) in the following) tested at the Georgia Institute of Technology by Schrauben (1999) under cyclic loading.

The analytical results will be compared with the experimental benchmark data.

NOTE: A comprehensive discussion both of the experimental test set up and cyclic loading history is furnished in Schrauben (1999) and Brunesi et al. (2013).

STRUCTURAL CONFIGURATION AND MATERIAL PROPERTIES

The test specimen FS-01(CA-02) consists of a W18 x 40 (W460 x 60) beam bolted to a W14 x 145 (W360 x 216) column by a 9" x 3-1/8" x 5/16" (229 x 79 x 8 mm) shear tab and clip angles cut from a L8 x 6 x 1 (L203 x 152 x 25). Two 7/8" (22 mm) diameter, 3-1/4" (83 mm) long A490 high-strength bolts with one washer have been used to fasten one leg of each clip angle to the flange of the column. The gage distance of these 'tension' bolts is 2-1/2" (63 mm). Four 7/8" (22 mm) diameter, 3" (76 mm) long A490 high-strength bolts with two washers have been used to fasten the other leg to the beam flange and three 7/8" (22mm) diameter, 2" (51mm) long A490 high-strength bolts with one washer have been used to fasten the beam web to the shear tab.

Specimen FS-02(CA-04) is similar to the previous, since the same components have been adopted, except the gage distance of the tension bolts which is 4" (102 mm).

The geometry of each connection is sketched in the figure below.

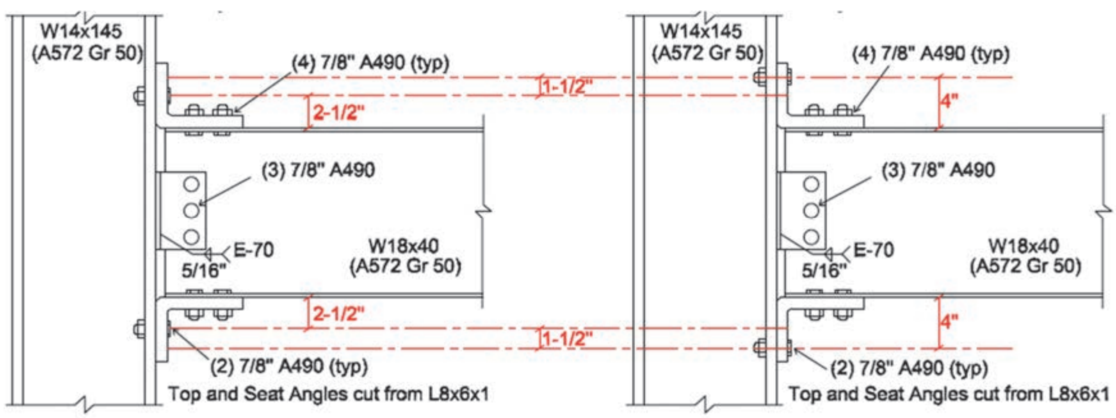


Figure 82: Partially-restrained bolted beam-to-column connections – connection characteristics of specimens FS-01 (left) and FS-02 (right)

Regarding the materials, a *bilinear model with kinematic strain-hardening* have been employed for defining the steel material properties used for modelling columns, beams and top and seat angles. The steel classes which have been considered are indicated in Figure 82.

MODELLING AND LOADING

Columns, beams and top and seat angles have been modelled through 3D force-based inelastic frame elements (infrmFB) with an appropriate number of fibres.

In order to represent the effects of the nonlinear contact algorithms in an equivalent manner, zero-length tri-linear links have been combined with the previous elements, as shown in Figure 83.

A sketch of the FE model is shown below:

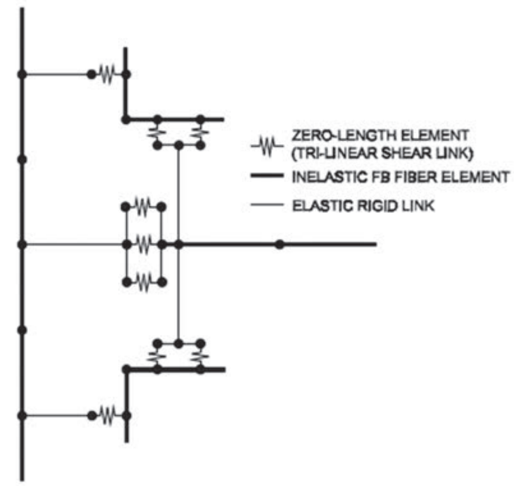


Figure 83: Partially-restrained bolted beam-to-column connections – FE model sketch

A cyclic loading, consisting of symmetric stepwise increasing deformation cycles, has been loaded in the “Time-history Curves” dialog box. The used output sampling time interval has been set to 0.1 seconds. The cyclic loading has been applied at the tip of the beam as a static time-history load in terms of displacements in the vertical direction.

The FE analysis model is presented below:

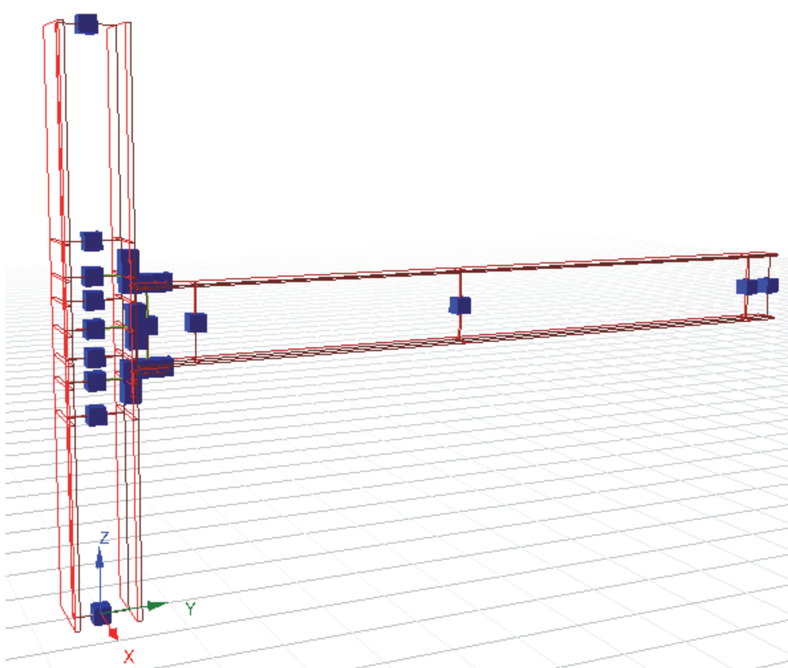


Figure 84: Partially-restrained bolted beam-to-column connections – FE model view

ANALYSIS TYPE

Nonlinear static time-history analysis.

Results Comparison

The comparison between numerical and experimental results is shown hereafter.

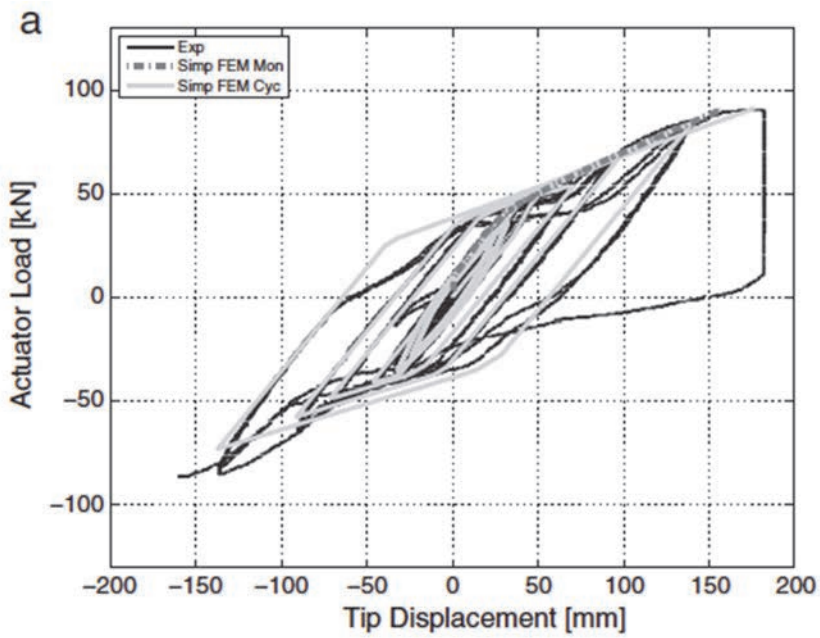


Figure 85: Partially-restrained bolted beam-to-column connections – Experimental vs. Analytical results (actuator load-top displacement – FS-01)

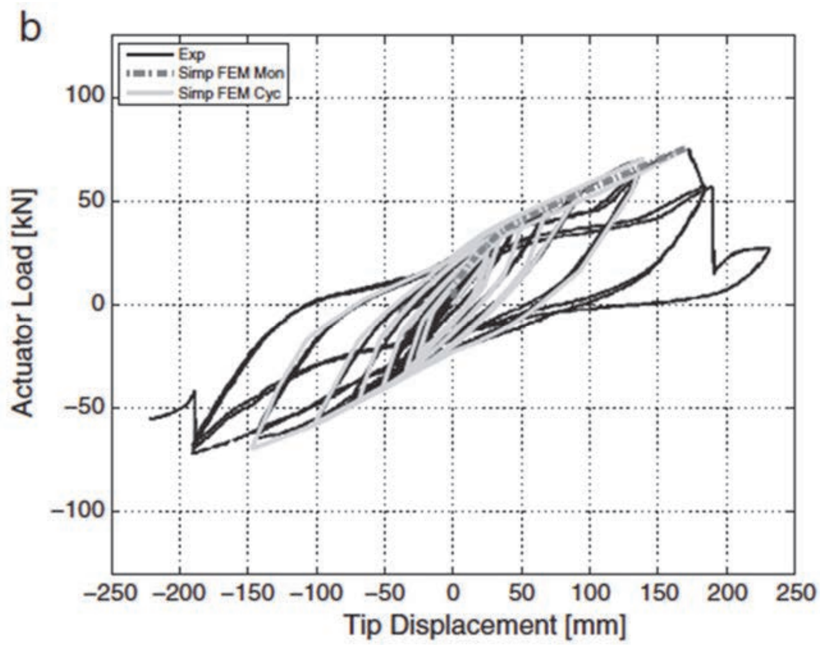


Figure 86: Partially-restrained bolted beam-to-column connections – Experimental vs. Analytical results (actuator load-top displacement – FS-02)

3 Closing remarks and next steps

The numerous examples presented in this deliverable have demonstrated the capability of the selected software, SeismoStruct, to model the nonlinear response of structures with material types and structural systems similar to those that have been used to construct the non-URM buildings in Groningen.

There are a large number of precast wall structures in Groningen, used for residential, industrial and commercial buildings, and although this report has demonstrated that SeismoStruct can be used to model precast structures, the experimental example presented was for a frame building, and the focus of the test was not on the response and failure of the precast connections.

In fact, the weakest links of the seismic response of multi-storey RC precast structures and one storey industrial buildings are the connections. In particular, due to the combined effects of the horizontal and the vertical components of the earthquake, the connections between elements may fail, so the structural elements may experience relative movements, lose their support and possibly fall down. Although the wall-panels usually do not suffer heavy damages, the connections, which are not always designed for seismic loads, may fail due to the high level of the seismic demand in terms of displacement, so the panels may suffer rigid overturning and fall down.

The characteristics of the connections used in precast construction can vary significantly from one country to another and thus the use of an experimental example with connection failure would not necessarily provide the confidence needed in the capability of SeismoStruct to model precast connections in Groningen. Hence, given the abundance of these structures in Groningen, an experimental and modelling campaign has been planned, to be carried out by EUCENTRE, which will involve:

1) Detailed numerical analyses on the response of the connectors-wall concrete sub-system using a solid-elements finite element package;
2) Laboratory testing of Groningnen-specific precast panels/connections;
3) Calibration of a phenomenological model appropriate for the seismic assessment of full structures.

The phenomenological model will then be included in SeismoStruct and the analytical results will be compared against the experimental test data (in a similar manner to that done for the steel connections in Section 2.10). An updated version of this deliverable will then be produced.

References

GENERAL AND INTRODUCTION

Seismosoft [2014a]. "*SeismoStruct v7 – A computer program for static and dynamic nonlinear analysis of framed structures*", available from http://www.seismosoft.com

Seismosoft [2014b]. "*SeismoStruct Verification Report, for version 7*," available from http://www.seismosoft.com

Remo Magalhães de Souza [2000] "*Force-based Finite Element for Large Displacement Inelastic Analysis of Frames",* PhD Thesis in Civil and Environmental Engineering, University of California, Berkeley.

Brunesi E, Nascimbene R. [2014] "*Extreme response of reinforced concrete buildings through fiber force-based finite element analysis*", Engineering Structures, No. 69, pp. 206–215.

EXAMPLE 2.1

Brunesi, E., Nascimbene, R., Bolognini, D. and Bellotti, D. [2014] "*Experimental investigation of the cyclic response of RC precast framed structures*", PCI Journal (accepted for publication).

EXAMPLE 2.2

Rinaldin G., Amadio C., Fragiacomo M. [2011] "*A component approach for non-linear behavior of cross-laminated timber panels",* Proceedings of the 14th Italian Conference on Earthquake Engineering (ANIDIS), Bari, Italy.

EXAMPLE 2.3

Ali H. [2009] "*Half Scale Three-Storey Infilled RC Building; A Comparison of Experimental and Numerical Models*", MSc Dissertation, ROSE School, Pavia, Italy.

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.

EXAMPLE 2.4

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.

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.

Varum, H. [2003]. "*Seismic Assessment, Strengthening and Repair of Existing Buildings*", PhD Thesis, Department of Civil Engineering, University of Aveiro.

EXAMPLE 2.5

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)

EXAMPLE 2.6

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.

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.

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).

EXAMPLE 2.7

Guedes J. [1997] "*Seismic behaviour of reinforced concrete bridges. Modelling, numerical analysis and experimental assessment*", PhD Thesis, Univ. of Porto, Porto, Portugal.

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.

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.

Pinto A.V., Verzeletti G., Pegon P., Magonette G., Negro P., and Guedes J. [1996] "*Pseudo-Dynamic Testing of Large-Scale R/C Bridges*", ELSA Lab, Report EUR 16378 EN.

EXAMPLE 2.8

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

EXAMPLE 2.9

Lanhui G., Shan G., Feng F., Yuyin W. [2013] "*Experimental study and numerical analysis of progressive collapse resistance of composite frames*", Journal of Constructional Steel Research, Vol. 89, pp. 236-251.

EXAMPLE 2.10.1

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.

EXAMPLE 2.10.2

Brunesi E, Nascimbene R, Rassati G.A. [2014] “*Response of partially-restrained bolted beam-to-column connections under cyclic loads*”, Journal of Constructional Steel Research, Vol. 97, pp. 24–38.

Brunesi E, Nascimbene R, Rassati G.A. [2013] “*Evaluation of the response of partially restrained bolted beam-to-column connection subjected to cyclic pseudo-static loads*”, Structures Congress 2013: Bridging Your Passion with Your Profession - Proceedings of the 2013 Structures Congress 2013:2310–21.

Schrauben CS. [1999] “*Behavior of full-scale bolted beam-to-column T-stub and clip angle connections under cyclic loading*”, MS thesis, Georgia Institute of Technology.

Appendix: Benchmark data and corresponding structural models

Benchmark data and corresponding structural models are enclosed, in digital format.