RWTHAACHEN
STAHLBAU UNIVERSITY

Master Thesis

Seismic behaviour of composite wall systems subjected to different levels of seismic action and with different levels of protection

Autor: Germán Díaz

Tutor: Dipl.-Ing. Hetty Bigelow

INDEX

PICTURES

TABLES

SYMBOLS AND UNITS

The units used for the most used variables in this thesis are:

Magnitude	Unit	Symbol
Length	millimetre	mm
Mass	ton	t, Mg
Time	second	s
Frequency	hertz	Hz, s^{-1}
Force	kilonewton	kN
Stress	megapascal	MPa, N/mm^2
Angle	degree	º

However, if no units are explicit -for a value that is not dimensionless- then the International System of Units applies.

1 INTRODUCTION

Seismic engineers focus not only on designing buildings able to resist an earthquake of a particular intensity, but they also concern about the performance of the structure for different levels of seismicity. This concept has been developed within the Performance Based Earthquake Engineering (PBEE) since many years, and it is slowly being introduced in the existing building codes. In Europe, the Eurocodes are still the reference norm.

Composite systems are considered a good solution in terms of cost/performance relation when designing under earthquake loads. Steel frames with reinforced concrete infill walls (SRCW) are an interesting solution because of the relative reduced cost, and their interesting reparability possibilities.

In chapter 2 of this thesis, the different methods of analysis available in the Eurocodes and within the PBEE will be described and compared. In chapter 3 those composite systems will be described and also the mechanisms by which the energy is dissipated during an earthquake. A particular case study will be the practical part (chapter 4), where an example building will be dimensioned in detail according to the Eurocodes. All the articles of the Eurocodes which are necessary to design using this lateral resisting structural system will be previously explained. The different FE models -developed with ANSYS- used in the static and dynamic analyses will be described, and also the results and the conclusions of the analyses.

The final objective of the project is to obtain a reliable mathematical model, with which incremental dynamic analysis are going to be performed in the future. This model will take into account the different nonlinearities of the system, mainly related to nonlinear materials (steel and concrete) and the modelling of the connection between both materials.

The case study belongs to the INNO-HYCO project, which is being carried out by different European universities, including the RWTH Aachen through the Institute of Steel Structures (Institut und Lehrstuhl für Stahlbau und Leichtmetallbau).

2 METHODS OF SEISMIC DESIGN AND ANALYSIS

2.1 PROCEDURES ACCORDING TO EUROCODE-8

2.1.1 INTRODUCTION

Eurocodes are reference documents for building in Europe. There are ten base documents, each one referring to a special field. Once each document has been approved by the government of a country, it becomes an official norm. The idea is that, in short-time future, Eurocodes become the only documents used to design buildings and structures, harmonising norms and making it easier to build around Europe, using the same rules in every country.

However, Eurocodes also give to each country the possibility to define certain parameters. The document which gives these national determined parameters (NDP) is called the National Annex (NA), where these parameters are fixed following the particular interests, such as security requirements and law, of each country.

Eurocode 8: Design of structures for earthquake resistance [1] (also referenced as EN 1998) focuses on seismic design and earthquake resistant building. This document, as all other Eurocodes, follows the limit state design method, considering two limit states: Ultimate Limit State (ULS) and Serviceability Limit State (SLS).

ULS is a no-collapse requirement, which means that the structure has to be able to resist the peak design load keeping its integrity, without collapsing [1,4]. This scenario is the worst that the structure is expected to be subjected to, according to a probabilistic approach. It is assumed, with a certain level of confidence, that higher levels of seismic action are not likely to happen. After this rare event, the main objective is protection of life, which means that the structure, even if strongly damaged, has to be able to resist vertical actions and possible earthquake replicas, allowing the evacuation of the people inside and all other necessary emergency procedures.

SLS is a damage limitation requirement, and takes into account that the structure has to remain functional when subjected to a situation that is more likely to happen. Under this state it is allowed to have some minor damage in non-structural elements, but only if the building remains functional. This means that users are able to keep doing inside the activities it was designed for, with no limitations of use after an earthquake. No damage is allowed for structural elements, which means that they should not suffer permanent deformations and must keep its stiffness.

The difference between seismic actions for ULS and for SLS comes from different values of the probability of exceedance within a certain period of time. Recommended values are 10% probability of exceedance in 50 years (or return period of 475 years) for ULS; and 10% in 10 years (or 95 years return period) for SLS. The ULS event is less likely to happen, and thus is much stronger than the one corresponding to SLS. Also,

Regarding to methods of analysis and models, Eurocode 8 gives different alternatives to perform calculations. Regarding to methods of analysis, the norm allows four:

- Modal response spectrum analysis, using a linear elastic model. This is the reference method and it is valid for all types of building. The seismic action is transformed into static equivalent forces or displacements, corresponding to each mode of vibration taken into account.

- Lateral force method of analysis, which is a simplified method only valid for buildings that fulfil particular conditions of regularity. An important condition is that the fundamental mode of vibration has to significantly represent the dynamic response of the whole building by itself. This method uses horizontal static forces to represent the seismic action, by performing a linear static analysis.
- Non-linear static analysis, which also uses static forces, but takes into account both geometrical and material non-linearity. This method is also called pushover analysis.
- Non-linear time history dynamic analysis. This is the most accurate method to know the response of the building to a generic dynamic load, but losses some of its precision because of the selection of the excitation function. To represent the seismic action, one or more accelerograms are used, which may not perfectly fit future events. A time-step based analysis is performed with these recorded, artificially generated or simulated accelerograms. It is also important to know that this method requires more computational resources.

Regarding to the model, Eurocode 8 allows the use of:

- A planar model (two-dimensional), when both criteria of regularity in plan and in elevation are fulfilled. Those criteria are explained in EN 1998-1 4.2.3.2 [1]. The use of two planar models, one for each orthogonal direction, is also valid if not all the criteria are satisfied, but only under special conditions.
- A spatial model (three-dimensional), when the building is not considered regular in either plan or elevation.

Regularity in plan	Regularity in elevation	Model	Method of analysis	Behaviour factor
Yes	Yes	Planar (2D)	Lateral force	Reference value
Yes	No	Planar (2D)	Modal	Decreased value
No	Yes	Spatial (3D)	Lateral force	Reference value
No	No	Spatial (3D)	Modal	Decreased value

TABLE 2-1: ALLOWED SIMPLIFICATIONS AND METHODS OF ANALYSIS [1]

For all methods of analysis and models, seismic loads are based on a reference response spectrum, adapted to the area of each study and the materials used in the building (through damping), and also scaled up or down depending on the importance of the building. In both the modal response and the lateral force methods, the spectrum also takes into account the structural type, by using the behaviour factor to obtain the design spectrum.

2.1.2 RESPONSE SPECTRUM

In order to obtain the response spectrum, Eurocode 8 needs certain input data, which are [2,3]:

- Ground type
- Peak acceleration value
- Importance factor

- Damping correction factor
- Spectra type
- Behaviour factor (only in linear methods of analysis)

As earthquake waves travel through soil, it modifies their spectra, acting as a filter. This influence depends on the ground type. Eurocode 8 considers seven different ground types, depending on their mechanical properties:

Type	Description	$v_{s,30}$	N_{SPT}	c_u
A	Rock or rock-like geological formation.	>800	-	-
B	Very dense sand, gravel or stiff clay.	360-800	>50	>250
C	Deep deposits of dense or medium-dense sand, gravel or stiff clay.	180-360	15-50	70-250
D	Deposits of loose-to-medium cohesionless soil, or soft-to-firm cohesive soil.	<180	<15	<70
E	Profile with surface of alluvium (similar to C or D), and then stiffer material.	-	-	-
S_1	Soft clays/silts with high plasticity index and water content.	<100	-	10-20
S_2	Liquefiable soils, or other profiles not included in previous types.	-	-	-

TABLE 2-2: GROUND TYPES AND PARAMETERS [1]

To classify the ground type of a building site, Eurocode 8 gives three parameters in EN 1998-1 3.1.2 [1], which are average shear wave velocity $v_{s,30}$; standard penetration test blow-count N_{SPT} and undrained shear strength of soil c_u (cohesive resistance). After a geological analysis of the ground, those parameters should be sufficient to classify it. Types S_1 and S_2 require additional studies to characterise the seismic action, and soil-building interaction has to be analysed.

The next necessary parameter is the peak value for ground A acceleration or a_{gR}. These values are available for different hazard zones in the National Annex, usually through a hazard map. Ground A is taken as reference for the peak acceleration value, by convention. This value will be modified later on by the soil factor assigned to each ground type.

Depending on the importance of the building, an importance factor γ_I is used to increase or decrease the seismic action. There are four importance classes, from I to IV. Class II is for ordinary buildings, and corresponds to an importance factor γ_I of 1. Actions for less important buildings are decreased, as their possible failure does not affect human lives (e.g. agricultural buildings). On the other hand, buildings of higher importance are those whose failure would affect more human lives (e.g. schools) or whose operability should remain intact after the design event, for instance strategic buildings such as hospitals or military facilities. The other importance factors are defined in the National Annex, but recommended values are:

Class	γ_I	Description
I	0.8	Minor importance buildings.
II	1.0	Ordinary buildings.

Class	γ_I	Description
III	1.2	Important buildings. Schools, institutions...
IV	1.4	Buildings of vital importance for civil protection. Hospitals, power plants...

TABLE 2-3: IMPORTANCE CLASSES AND RECOMMENDED FACTORS [1]

The previously determined peak acceleration value is corrected by this importance factor to obtain design ground peak acceleration:

$$a_g = \gamma_I \cdot a_{gR} \tag{2.1}$$

Damping is also an important parameter, but it is difficult to measure. There are tables which try to approximate the damping ratio of a structure, depending on the materials and structural type. Default damping ratio is 5% in Eurocode 8, so that the corresponding damping correction factor η is 1:

$$\eta = \sqrt{\frac{10}{5+\xi}} \geq 0.55; \quad \text{If} \quad \xi = 5\% \quad \text{then:} \quad \eta = 1 \tag{2.2}$$

Steel structures typically have damping values of 2-3% in the elastic range, while reinforced concrete structures between 3 and 5% [4]. Although 5% damping is taken as reference value, Eurocode 8 allows the use of other damping ratios corresponding to different materials, or for systems with artificial (active) damping devices.

Finally, last classification regards type spectra, which also affects the shape of the final curve. Eurocode 8 considers two types: Type 1 for areas where earthquakes are expected to have a magnitude M_s greater than 5.5 (that is, high and moderate seismicity regions); and Type 2 for smaller values (low seismicity regions).

After having determined previously explained values, response spectrum curves are defined as follows:

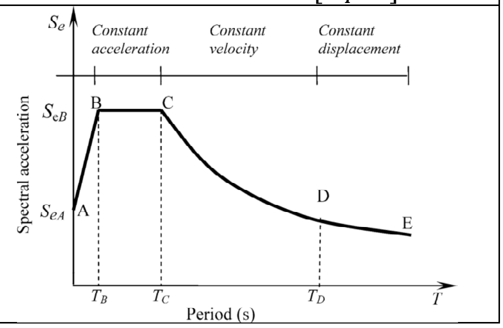
Period range	Horizontal elastic response
$0 \leq T \leq T_B$	$S_e(T) = a_g \cdot S \cdot \left[1 + \frac{T}{T_B} \cdot (\eta \cdot 2.5 - 1)\right]$
$T_B \leq T \leq T_C$	$S_e(T) = a_g \cdot S \cdot \eta \cdot 2.5$
$T_C \leq T \leq T_D$	$S_e(T) = a_g \cdot S \cdot \eta \cdot 2.5 \cdot \left[\frac{T_C}{T}\right]$
$T_D \leq T \leq 4s$	$S_e(T) = a_g \cdot S \cdot \eta \cdot 2.5 \cdot \left[\frac{T_C \cdot T_D}{T^2}\right]$
Shape	

TABLE 2-4: HORIZONTAL ELASTIC RESPONSE SPECTRUM [1]

These curves give spectral acceleration S (in m/s^2) as function of period T, between 0 and 4 seconds. Soil factors and the position of the different areas of the spectrum curve are summarized for both types of spectra in the following table:

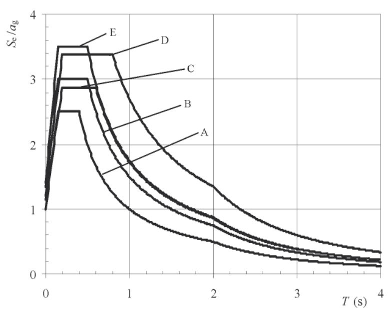
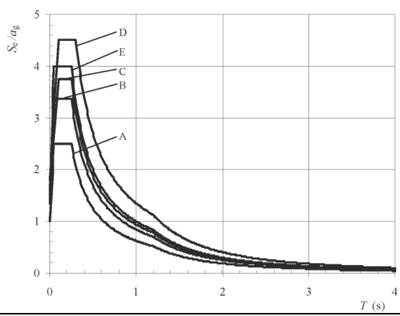
Ground	Type spectra 1				Type spectra 2			
	S	T_B(s)	T_C(s)	T_D(s)	S	T_B(s)	T_C(s)	T_D(s)
A	1.00	0.15	0.40		1.00	0.05	0.25	
B	1.20	0.15	0.50		1.35	0.05	0.25	
C	1.15	0.20	0.60	2.00	1.50	0.10	0.25	1.20
D	1.35	0.20	0.80		1.80	0.10	0.30	
E	1.40	0.15	0.50		1.60	0.05	0.25	
Shape								

TABLE 2-5: PARAMETERS FOR HORIZONTAL RESPONSE [1]

The β parameter in table 2-5 is the lower-bound coefficient, which ensures a minimum value of the seismic action for values higher T_C. This is also a national-determined coefficient, and the recommended value is 0.2.

For the vertical component of the seismic action, S value is taken as 1, and also the formulas are slightly modified:

Period range	Vertical elastic response
$0 \leq T \leq T_B$	$S_{ve}(T) = a_{vg} \cdot \left[1 + \frac{T}{T_B} \cdot (\eta \cdot 3.0 - 1)\right]$
$T_B \leq T \leq T_C$	$S_{ve}(T) = a_{vg} \cdot \eta \cdot 3.0$
$T_C \leq T \leq T_D$	$S_{ve}(T) = a_{vg} \cdot \eta \cdot 3.0 \cdot \left[\frac{T_C}{T}\right]$
$T_D \leq T \leq 4s$	$S_{ve}(T) = a_{vg} \cdot \eta \cdot 3.0 \cdot \left[\frac{T_C \cdot T_D}{T^2}\right]$

TABLE 2-6: VERTICAL ELASTIC RESPONSE SPECTRUM [1]

a_{vg} being the design peak ground value for vertical acceleration, also defined in the National Annex. However, vertical action only has to be taken into account if this value is greater than 0.25g, and only on the structural components detailed in EN 1998-1 4.3.3.5.2(1) [1]:

- Horizontal structural members spanning more than 20m.
- Horizontal cantilever members longer than 5m.
- Pre-stressed horizontal members.
- Beams supporting columns.
- Base-isolated structures.

This action has been traditionally neglected, as it was considered as covered by the permanent and transient loads. However, with the proliferation of new and more accurate seismic records it has been proved that it can also cause important damage to structures [4]. Frequency content of the vertical response spectrum is different, and peak ground acceleration is smaller. Coefficients are, for both Type 1 and 2:

Type spectra 1				Type spectra 2			
a_{vg}/a_g	T_B(s)	T_C(s)	T_D(s)	a_{vg}/a_g	T_B(s)	T_C(s)	T_D(s)
0.90	0.05	0.15	1.00	0.45	0.05	0.15	1.00

TABLE 2-7: PARAMETERS FOR VERTICAL RESPONSE [1]

2.1.3 METHODS OF ANALYSIS

2.1.3.1 Modal response spectrum analysis

When following this method of analysis, all significant modes of vibration of the structures have to be determined and taken into account. Eurocode 8 considers that all the "significant" ones are those, whose modal masses sum at least up to 90% of the total mass of the structure (EN 1998-1 4.3.3.3.1(3) [1]). Moreover, all modes with a modal mass greater than 5% of the total mass shall be taken into account. For each mode of vibration, the intensity of the seismic action comes from the reference design response spectrum, as a spectral acceleration, velocity or displacement.

Design values are used in the two linear methods of analysis explained in this document (see 3.1.3.1 and 3.1.3.2). These values are obtained through the behaviour factor q (q-factor), which takes into account the capacity of the structure to dissipate energy within the inelastic range [16]. The spectrum reduced by the behaviour factor is used in linear analysis, as a simplified but reliable method of taking into account the inelastic response of the structure, but still be able to use an elastic model. The behaviour factor is generally calculated by the ratio α_u/α_1, where α_u is the coefficient by which the seismic action has to be multiplied to reach the collapse of the structure, and α_1 to reach first yield in any of its components (equivalent to the formation of the first plastic hinge). Reference values and upper limits for behaviour factors are found in EN 1998-1 5.2.2, 6.3.2 and 7.3.2 [1], depending on structural types (concrete, steel or mixed, respectively).

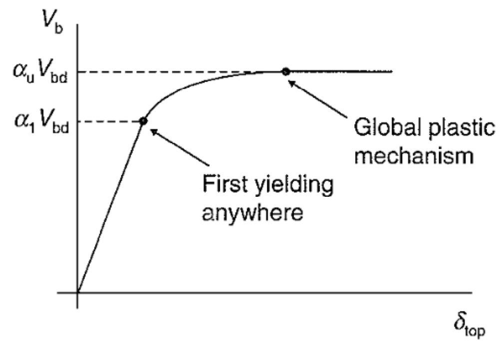


FIGURE 2-1: COEFFICIENTS USED TO CALCULATE THE BEHAVIOUR FACTOR [1]

Formulas for the determination of the design spectrum are:

Period range	Horizontal design response	Vertical design response
$0 \leq T \leq T_B$	$S_d(T) = a_g \cdot S \cdot \left[\frac{2}{3} + \frac{T}{T_B} \cdot \left(\frac{\eta \cdot 2.5}{q} - 1\right)\right]$	$S_{ved}(T) = \frac{S_{ve}(T)}{q}$ q-factor up to 1.5 generally adopted according to EN 1998-1 3.2.2.5(5)
$T_B \leq T \leq T_C$	$S_d(T) = \frac{S_e(T)}{q}$	
$T_C \leq T \leq T_D$	$S_d(T) = \frac{S_e(T)}{q}; \quad \geq \beta \cdot a_g$	
$T_D \leq T \leq 4s$		

TABLE 2-8: HORIZONTAL AND VERTICAL DESIGN SPECTRUM

And the spectral displacement is generally calculated from the spectral acceleration by using:

$$S_{De}(T) = S_e(T) \cdot \left(\frac{T}{2\pi}\right)^2 \tag{2.3}$$

For each mode of vibration, the design spectral displacement is calculated with equation 3.3, but with the design spectrum instead of the elastic. Afterwards, it is multiplied by the eigenvector of the mode and its participation factor to obtain a vector with the displacements of all degrees of freedom (or nodes) considered in the analysis, due to each mode of vibration, shown in equation 3.4.

$$\{U_n\} = S_{Dd}(T_n) \cdot \left(\frac{T_n}{2\pi}\right)^2 \cdot \Gamma_n \cdot \{\phi_n\} \tag{2.4}$$

$$\Gamma_n = \frac{\sum_i \phi_{i,n} m_i}{\sum_i \phi_{i,n}{}^2 m_i} \tag{2.5}$$

Where T_n is the period of mode of vibration n; Γ_n is the participation factor of mode n, $\{\phi_n\}$ is the vector containing the shape of mode of vibration n, normalized to mass matrix; $\phi_{i,n}$ is component i of vector $\{\phi_n\}$ and m_i is the mass of degree of freedom i. Also, figure 2-2 illustrates the process followed to obtain this displacements vector:

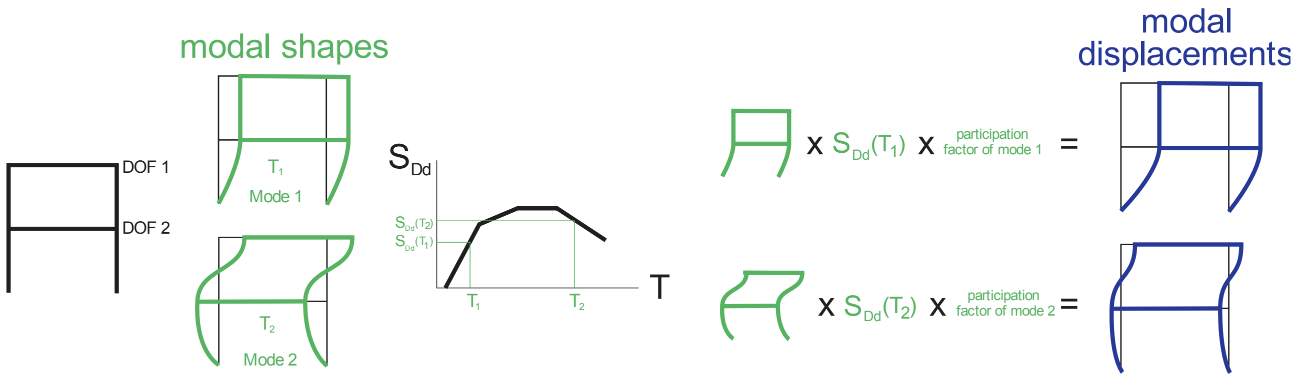


FIGURE 2-2: ILLUSTRATIVE EXAMPLE OF MODAL ANALYSIS

From this modal nodal displacements vector, modal effects are computed by using the elastic model of the structure. The results are internal forces, moments, stresses, etc.

These modal effects are combined between them. Because of the uncertainty of the combination of each mode's peak response value, Eurocode proposes the use of the SRSS rule (square root of the sum of squares):

$$E_E = \sqrt{\sum_i E_{Ei}{}^2} \tag{2.6}$$

Being E_E the global effects and E_{Ei} those effects produced only by mode i. Although the most conservative method to combine these peak modal values would be to sum their absolute values, this approximated formula is considered a good estimation of the peak total response [4,15]. This is the simplest method allowed by Eurocode 8 to combine modes, but more accurate methods are needed if responses of different modes are concluded to be dependent between them.

2.1.3.2 Lateral force method of analysis

When a building is considered regular in plan are in elevation (according to EN 1998-1 4.2.3.3), Eurocode 8 allows the use of the lateral force method of analysis. It considers that only the first mode of vibration (corresponding to the fundamental period, T_1) is significant. In addition to the above mentioned criteria, the following conditions have to be fulfilled:

$$T_1 \leq \begin{cases} T_C \\ 2 \end{cases} \tag{2.7}$$

This method is essentially a simpler version of the previously explained modal analysis, where only the shape of the first mode of vibration is considered. The simplification made in this method also considers floors as rigid members of the structure, which can only move horizontally.

With this method, the base shear force F_b is calculated, and it is the only earthquake load to be considered. If structure is not symmetric, it has to be applied in both positive and negative directions. This base shear force is divided into forces F_i to be applied at the different storeys, proportionally to the mass and height of each (as shown in figure 2):

$$F_b = S_d(T_1) \cdot m \cdot \lambda \tag{2.8}$$

$$F_i = F_b \frac{z_i m_i}{\sum_j z_j m_j} \tag{2.9}$$

Where m is the total mass of the building, m_i and z_i mass and height of storey number i, respectively; and λ a correction factor which takes into account that in short buildings the modal mass of the fundamental mode is higher, in proportion to the total mass, than it is in taller buildings. λ is equal to 0.85 if $T_1 \leq 2 \cdot T_C$ and the number of storeys is three or more, and equal to 1 otherwise.

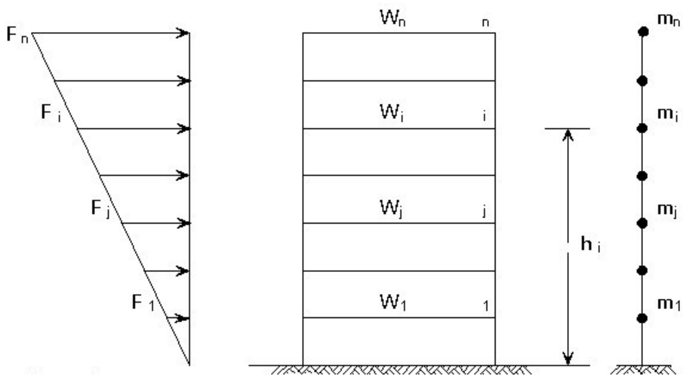


FIGURE 2-3: LATERAL FORCE METHOD LOADS SCHEME [16]

These forces are static equivalent forces representing the seismic action, which is however a dynamic action.

The fundamental period T_1 is an important parameter, but is also sometimes difficult to determine without advanced methods. Eurocode 8 gives three different options, detailed in EN 1998-1 4.3.3.2.2 [1], which are:

1. The use of structural dynamics methods, for example the Rayleigh formula:

$$T_1 = 2\pi \cdot \sqrt{\frac{\sum m_i \cdot \delta_i^2}{\sum F_i \cdot \delta_i}} \tag{2.10}$$

Where m_i is mass of degree of freedom i, and δ_i is lateral drift at degree of freedom i caused by a set of lateral forces F_i. The final period value is independent of the chosen set of forces.

2. An approximated formula for buildings up to 40m high:

$$T_1 = C_t \cdot H^{3/4} \tag{2.11}$$

With C_t being 0.085 for moment resistant steel frames, 0.075 for moment resistant concrete frames and 0.050 for other cases; and H being the height of the building in meters. Also an alternative expression for C_t is given for structures with masonry or concrete shear walls:

$$C_t = \frac{0.075}{\sqrt{\sum A_i \cdot \left(0.2 + \left(\frac{l_{wi}}{H}\right)^2\right)}} \tag{2.12}$$

Being A_i the cross-sectional area of the shear wall i in m^2, and l_{wi} its length, limited by Eurocode 8 to a value of $0.9 \cdot H$.

3. A general (conservative) expression:

$$T_1 = 2 \cdot \sqrt{d} \tag{2.13}$$

Where d is the lateral displacement that the top of the building suffers when subjected to gravity loads applied in the horizontal direction.

In this method, torsional effects have to be taken into account by multiplying the action effects in each load resisting element by a factor δ, according to EN 1998-1 4.3.3.2.4(1) [1]:

$$\delta = 1 + 0.6 \cdot \frac{x}{L_e} \tag{2.14}$$

Where x is the distance from the resisting element to the center of mass of the building, and L_e is the highest distance between two lateral resisting systems (perpendicularly to the direction of the seismic action).

2.1.3.3 Non-linear static analysis

This type of analysis is restricted by Eurocode 8 to some cases: the evaluation of the behaviour factor, the design of buildings with seismic isolation the evaluation of the seismic performance of new building designs, and the design of buildings without the use of the behaviour factor [4]. In non-linear static analyses (also called 'pushover' analysis), vertical loads are considered constant while horizontal seismic loads are increased using sub steps (representing the geometrical non-linearity), and also taking into account the material non-linearity.

Eurocode 8 requires the use of two different horizontal patterns. The first one is analogue to the equivalent forces used in the lateral force method (proportional to height and mass of each storey), and the second one follows the shape of the modal response for the fundamental period of the building. The worst solution is taken as valid, also applying loads in both positive and negative direction if the building is not symmetric.

The results obtained through a pushover analysis allow the calculation of the behaviour factor, the analysis of the plastic mechanisms of the structure, and also the generation of the capacity curve, in which the displacement of a control node (usually at roof level) is compared to the base force intensity. The whole system is simplified into a single-degree-of-freedom (SDOF) equivalent system. Although it is a simple graphic, it helps to understand the general inelastic behaviour of the structure. Figure 3 shows an example of a capacity curve of the equivalent SDOF system, and also its elastic-perfectly plastic idealization.

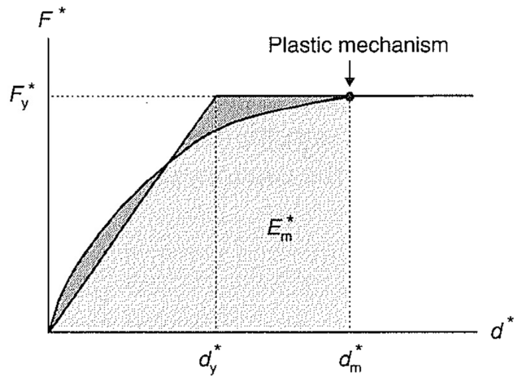


FIGURE 2-4: CAPACITY CURVE AND ELASTIC-PERFECTLY PLASTIC IDEALIZATION OF THE EQUIVALENT SDOF SYSTEM [1]

Where F_y^* is the equivalent shear force, d_y^* the deflection when the elastic limit is reached, and d_m^* the deflection at the yielding point; all parameters referring to the equivalent SDOF system. The determination of the SDOF equivalent magnitudes (force and deflection with asterisk) is done by dividing real values by the participation factor Γ of the mode used in the analysis.

2.1.3.4 Non-linear time history dynamic analysis

When performing a time history dynamic analysis a recorded, generated or simulated accelerogram (acceleration as a function of time) is used for every direction of interest. In a spatial model, three simultaneously acting curves shall be used. The structure will be excited with the accelerograms in order to know its response.

There are three different sources of accelerograms [1,4]:

- Recorded (natural) earthquakes conveniently scaled to fit the peak acceleration.
- Artificially generated accelerograms, fitting a target spectrum.
- Simulated accelerograms, which are generated through physical simulations.

According to Eurocode 8, artificially generated accelerograms should fit the reference spectrum, with a minimum duration of 10s for the stationary part.

For generated, recorded and simulated accelerograms, a minimum of 3 accelerograms must be used for each analysis (EN 1998-1 3.2.3.1.2 and 3). In that case, the worst obtained answer should be considered as the final response. If 7 or more accelerograms are used, Eurocode allows considering the average value of the response as valid (EN 1998-1 4.3.3.4.3(3)).

This method consists on solving the motion differential equation system shown in equation 3.12 (which considers all degrees of freedom of the whole structure) for each time step, by direct integration.

$$[M] \cdot \{\ddot{u}\} + [C] \cdot \{\dot{u}\} + [K] \cdot \{u\} = \{P(t)\} \tag{2.15}$$

Where $[M]$, $[C]$ and $[K]$ are the mass, damping and stiffness matrixes, respectively; $\{\ddot{u}\}$, $\{\dot{u}\}$ and $\{u\}$ are the nodal acceleration, velocity and displacement vectors; and $\{P(t)\}$ is the time varying applied nodal load.

The mass and stiffness matrixes are easily determined in basis of the mathematical model of the structure. However, the damping matrix is usually based on damping ratios, based on experimental data obtained from existing buildings [18], in form of modal damping ratios. For example, Rayleigh damping[1] consists on the linear combination of the mass and the stiffness matrix:

$$[C] = \alpha \cdot [M] + \beta \cdot [K] \tag{2.16}$$

Being α and β two coefficients, which define the importance of each one of the two considered damping mechanisms. If Rayleigh damping is used, the final damping matrix always diagonalises, in terms of:

$$\{\phi_n\}^T \cdot [C] \cdot \{\phi_n\} = 2 \cdot \omega_n \cdot \xi_n \tag{2.17}$$

Where ω_n is the angular velocity of mode n, and ξ_n its damping ratio.

After initial conditions are set (time zero conditions), the values of the ground motion set are taken as excitation to calculate the solution at the end of the first time step. These solutions are taken as initial conditions for the next time step, and so on. The time step must be small enough to represent frequency content accurately and minimize error. If this time step is small enough, error tends to zero for lineal systems. The solution in each time step gives position, velocity and acceleration of each degree of freedom, and then forces, moments and stresses are calculated from those values.

However, non-linear systems do not have an analytical solution, and thus numerical methods are needed. In that case, the stiffness matrix is updated every time step and also material non-linearity (plasticity) is taken into account when computing material's history and stresses. These analyses are performed using adequate finite elements software, which perform the analysis also with different substeps within every time step, to achieve the desired accuracy.

There are basically two approaches used in the direct integration method: explicit and implicit. In the first scheme, responses are expressed as a function of an already known solution, so that $(\{\ddot{u}\}_{t+\Delta t}, \{\dot{u}\}_{t+\Delta t}, \{u\}_{t+\Delta t}) = f(\{\ddot{u}\}_t, \{\dot{u}\}_t, \{u\}_t)$; while in the second, the response if found by solving a system of equations both involving the current state and the next one, so that $G(\{\ddot{u}\}_{t+\Delta t}, \{\dot{u}\}_{t+\Delta t}, \{u\}_{t+\Delta t}, \{\ddot{u}\}_t, \{\dot{u}\}_t, \{u\}_t) = 0$.

An example of an explicit formulation is the central difference method, which uses the following premises:

$$\{\dot{u}\}_{t+\frac{\Delta t}{2}} = \frac{\{u\}_{t+\Delta t} - \{u\}_t}{\Delta t} \; ; \quad \{\dot{u}\}_{t-\frac{\Delta t}{2}} = \frac{\{u\}_t - \{u\}_{t-\Delta t}}{\Delta t} \; ; \quad \{\ddot{u}\}_t = \frac{\{\dot{u}\}_{t+\frac{\Delta t}{2}} - \{\dot{u}\}_{t-\frac{\Delta t}{2}}}{\Delta t} \tag{2.18}$$

[1] Also called "proportional damping".

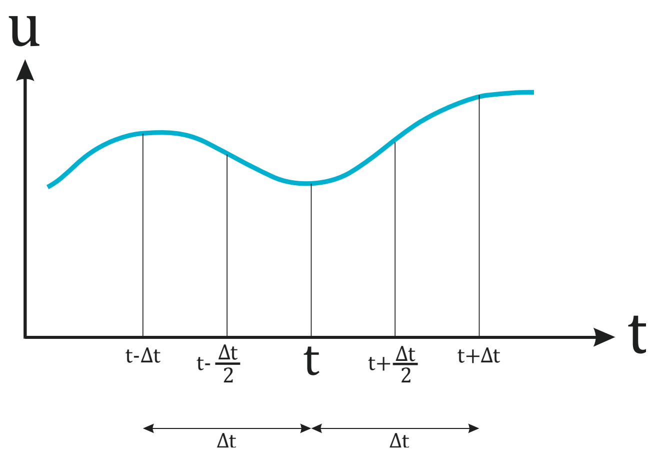


FIGURE 2-5: EXAMPLE OF THE BASIC PREMISES USED IN THE CENTRAL DIFFERENCE METHOD

This leads to:

$$\{\ddot{u}\}_t = \frac{\{u\}_{t+\Delta t} - 2\cdot\{u\}_t + \{u\}_{t-\Delta t}}{(\Delta t)^2}\,;\quad \{\dot{u}\}_t = \frac{\{u\}_{t+\Delta t} - \{u\}_{t-\Delta t}}{2\cdot\Delta t} \qquad \textbf{(2.19)}$$

And substituting $\{\ddot{u}\}_t$ and $\{\dot{u}\}_t$ in equation 3.14 the system obtained is:

$$[\bar{M}]\cdot\{u\}_{t+\Delta t} = \{\overline{P(t)}\} \qquad \textbf{(2.20)}$$

Where $[\bar{M}]$ and $\{\overline{P(t)}\}$ are the effective mass matrix and the effective force vector respectively:

$$[\bar{M}] = \frac{1}{(\Delta t)^2}\cdot[M] + \frac{1}{2\cdot\Delta t}\cdot[C] \qquad \textbf{(2.21)}$$

$$\{\overline{P(t)}\} = \{P(t)\} - \left([K] - \frac{2}{(\Delta t)^2}\cdot[M]\right)\cdot\{u\}_t - \left(\frac{1}{(\Delta t)^2}\cdot[M] - \frac{1}{2\cdot\Delta t}\cdot[C]\right)\cdot\{u\}_{t-\Delta t} \qquad \textbf{(2.22)}$$

Solution is found on basis of the available solutions in t and t-Δt, by inverting the effective mass matrix.

Regarding to implicit methods, one of the most used is the Newmark Beta method. This is the method used by ANSYS by default. It defines velocity and displacement as:

$$\{\dot{u}\}_{t+\Delta t} = \{\dot{u}\}_t + [(1-\gamma)\cdot\{\ddot{u}\}_t + \gamma\cdot\{\ddot{u}\}_{t+\Delta t}]\cdot\Delta t \qquad \textbf{(2.23)}$$

$$\{u\}_{t+\Delta t} = \{u\}_t + \{\dot{u}\}_t\cdot\Delta t + \left[\left(\frac{1}{2}-\delta\right)\cdot\{\ddot{u}\}_t + \delta\cdot\{\ddot{u}\}_{t+\Delta t}\right]\cdot(\Delta t)^2 \qquad \textbf{(2.24)}$$

Where γ and δ are parameters[2] which determine both the stability and accuracy of the method. For example, if γ=1/6 and δ=1/2, acceleration is assumed to vary linearly within an interval. Combining these two equations with the general equilibrium equation 2.15 evaluated at t+Δt, the solution is found by iteration. For linear problems, however, it is possible to find a direct solution in terms of the effective mass matrix and the effective force vector (which are different from the ones used in the central difference method).

Finally, it is important to notice that the use of non-linear time history analyses is restricted in Eurocode 8 to the evaluation of new building designs and the design of buildings with seismic isolation.

[2] In literature, Newmark parameters are normally refered as α and β. To avoid confusion with the α and β damping parameters, the greek letters γ and δ will be used in this document, respectively.

2.1.3.5 Combination with other actions and safety verification

The general procedure used in Eurocodes when combining actions [19] is:

$$\sum \gamma_{G,j} \cdot G_{k,j} \ \text{"+"} \ \gamma_{Q,1} \cdot Q_{k,1} \ \text{"+"} \ \sum \gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i} \tag{2.25}$$

Where G are permanent actions (self weight) and Q variable actions (use loads, wind, snow...), affected by the coefficients detailed in EN 1991 and National Annexes. A main $Q_{k,1}$ action is chosen, and combined with secondary Q actions in a semi-probabilistic approach to risk management. Characteristic actions (e.g. G_k) are increased by the γ coefficients, and thus design actions are obtained. The operation "+" means "to be combined with". The whole process leads to generate all possible combinations between loads, by selecting a different primary action, in order to find the different possible scenarios. This is a normal procedure to be followed independently on the method of analysis chosen.

In either accidental (impact or fire) or seismic events, different combination rules apply. In seismic cases, common loads are combined with the seismic action as follows [1,4]:

$$\sum G_{k,j} \ \text{"+"} \ A_{Ed} \ \text{"+"} \sum \psi_{2,i} \cdot Q_{k,i} \tag{2.26}$$

Being A_{Ed} the design seismic action, and recommended values for partial factor $\psi_{2,i}$:

- $\psi_{2,i} = 0$ for wind, temperature and snow over 1000m above sea.
- $\psi_{2,i} = 0.3$ for live loads in residential or office buildings (but $\psi_{2,i} = 0$ on the roof).

For each structural element being verified, when subjected to a load combination, the resistance condition for Ultimate Limit State (non-collapse criteria) is:

$$E_d \leq R_d \tag{2.27}$$

This means that the design value for resistance (calculated using characteristic values of material strengths divided by the corresponding partial factor γ_M) has to be greater than the design value of the action effect. In simple words, that is equivalent to ensure that, with a certain level of confidence, each element is able to resist more than it needs to.

The partial factors γ_M to be considered in ULS are those corresponding to the accidental design situation. However, Eurocode 8 [1] does not allow to use those factors if uncertainties about the strength degradation of the materials are present, when subjected to cyclic deformations. In that case, the standard partial factors for persistent and transient design situations must be used, which can be found in Eurocode 2 for concrete and reinforcing steel:

Design situation	γ_c (concrete)	γ_s (reinforcing steel)
Persistent and transient	1.5	1.15
Accidental	1.2	1.00

TABLE 2-9: PARTIAL FACTORS FOR MATERIALS FOR ULTIMATE LIMIT STATES [20]

Finally, for Serviceability Limit State (damage limitation) Eurocode 8 only establishes limitation of interstorey drift:

Interstorey limit	Case
$d_r \cdot \nu \leq 0.005 \cdot h$	Buildings having non-structural elements attached to the structure.

Interstorey limit	Case
$d_r \cdot \nu \leq 0.0075 \cdot h$	Buildings having ductile non-structural elements.
$d_r \cdot \nu \leq 0.010 \cdot h$	Buildings having non-structural elements not attached to the structure.

TABLE 2-10: LIMITATION OF INTERSTOREY DRIFT (SLS)

Where d_r is design value for interstorey drift (difference between lateral displacement on the top and the bottom of a storey, according to EN 1998-1 4.3.4), and ν is the reduction factor by which ULS seismic action is multiplied to obtain the ULS seismic action. Those values are NDP, but Eurocode 8 recommends 0.4 for importance classes III and IV, and 0.5 for classes I and II.

2.2 PERFORMANCE-BASED EARTHQUAKE ENGINEERING

2.2.1 INTRODUCTION

In some projects, the two limit states given by Eurocodes are not found to be sufficient to give an accurate solution for a building. In those cases it is not only important to know if the building fulfils the requirements of the code, but also to know how good its performance is in different load situations. Eurocodes are sufficient to fulfil the requirements of the European market, but are not intended to satisfy the requirements of the customer, which promotes and economically supports the design of the building. It becomes important to be able to measure and quantify how well a structure behaves when subjected to seismic loads of different magnitude, taking into account not only security reasons, but also economical [5,6,7,8,9,10]. In order to do this, the Performance-Based Earthquake Engineering (PBEE, and also PBSD[3]) considers two general variables: performance levels and seismic hazards. Performance levels are ranged between no damage and the complete collapse of a structure, while seismic hazards are defined in terms of return period (from 'frequent' to 'very rare' events).

The SEAOC Vision 2000 Committee defined in 1995 four reference performance levels [23]:

- Fully operational (FO) or Fully Functional. It means no significant damage in structural and non-structural elements, which happens when all elements remain in the elastic range.
- Operational (O) or Immediate Occupancy (IO). When the building is able to recover its functionality immediately, even though minor reparations have to be carried out.
- Life safe (LS). The structure keeps its integrity, so that life safety is protected and the building can be evacuated. It is possible to repair the building, but probably not economically viable.
- Collapse Prevention (CP) or Near Collapse. The structure does not collapse, but suffers from severe damage in both structural and non-structural elements.

This description of the four performance levels is qualitative and very general, but it is afterwards translated into technical parameters.

[3] Performance-Based Seismic Design

At the beginning of the building design process, performance objectives are established by coupling a level of expected performance with a particular level of seismic ground motion. An example of the performance objectives for three different levels of importance is shown in this graphic, also based on the conventions from Vision 2000:

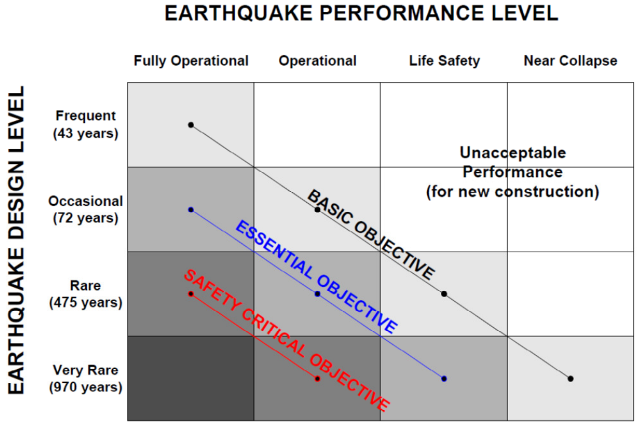


FIGURE 2-6: DIFFERENT PERFORMANCE OBJECTIVES [12]

This graphic helps to understand how different levels of protection may be given to buildings with different importance. The three lines show three different possible criteria for a normal building (Basic Objective), an important one (Essential Objective) and a strategic building (Safety Critical Objective). However, it is not necessary to follow any of these lines, as every building is particularly designed to accomplish with the performance objectives of its expected life-cycle.

It is also possible to show these performance objectives in a load-deflection curve:

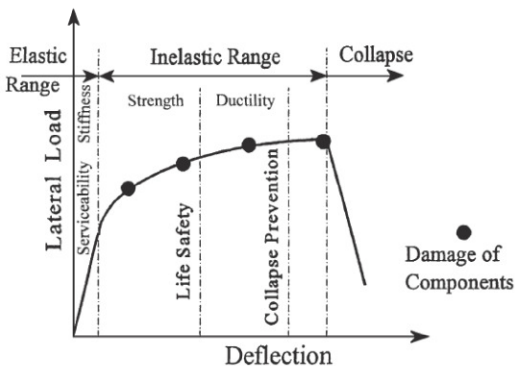


FIGURE 2-7: SIMPLIFICATED REPRESENTATION OF THE DIFFERENT PERFORMANCE LEVELS [24]

PBEE does not have a defined methodology, but many different approaches still in development. Also, a next-generation of PBEE is being discussed in organizations like the Pacific Earthquake Engineering Research Center. Basics of the PBEE methodology, which most of the particular methods share, will be explained.

2.2.2 METHODOLOGY

An analysis following the PBEE methodology starts by establishing performance objectives, always related to these three types of losses [11]:

- Life loss and serious injuries (casualties).
- Economic loss (reparation costs and damaged equipment).
- Service loss (periods of downtime).

These objectives are defined in an agreement between the owners or promoters of the building and the engineers responsible for the project, and then must be converted into measurable parameters related directly to the design of the structure.

After that, a preliminary design is done and the building analysed, for example, by simulation with an increasing seismic load. If the desired goals are fulfilled, the design is complete. If not, it has to be modified until it accomplishes the established objectives. This iterative process leads to a final appropriate design:

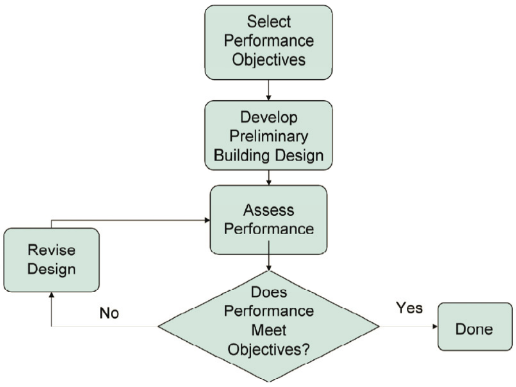


FIGURE 2-8: FLOW DIAGRAM OF PBEE ITERATIVE METHODOLOGY [5]

PBEE offers sufficient resources to achieve a particular solution to each project, which perfectly fits what it is expected from it. The last generation of this methodology is based on a complex and accurate probabilistic analysis. There are four types of parameters involved in this process:

- Intensity Measure (IM) is a parameter that characterises the seismic action, and it is the result of a hazard analysis. It is normally a peak ground acceleration value (PGA) or the elastic spectral pseudo-acceleration $S_a(T_1)$ for the fundamental period of the building.
- Engineering Demand Parameter (EDP), which represents the response of the building, in terms of accelerations, velocities, displacements, interstorey drifts, inelastic deformations, internal forces, etc. These parameters are the result of the structural analysis, which in most cases tends to be a non-linear dynamic analysis.
- Damage Measure (DM), which characterises the consequences of the damage after an earthquake. These are related to the three objectives explained before, but are qualitative.
- Decision Variable (DV). Those are quantitative variables, translation of the DM into numbers: economical units, life loss, and downtime.

Each parameter plays a different role in the methodology, based in four steps [7]:

- Hazard analysis (characterisation of the seismic hazard in terms of an IM).
- Structural analysis (relation between EDP and IM).
- Damage analysis (relation between DM and EDP).
- Loss analysis (relation between DM and DV or losses).

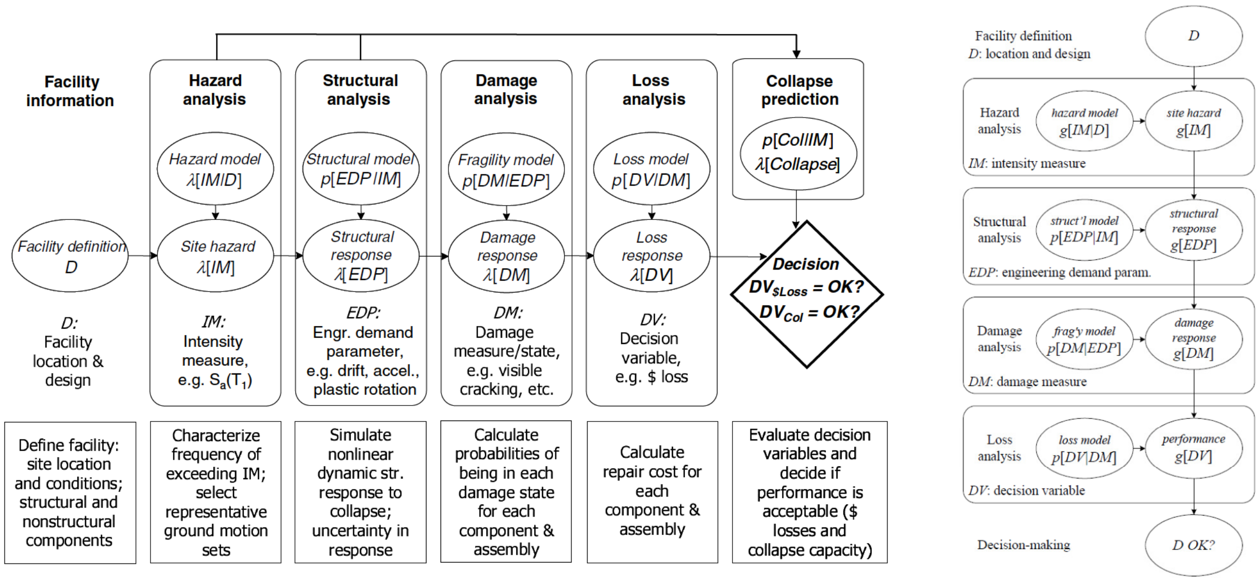


FIGURE 2-9: TWO GRAPHIGS SHOWING FRAMEWORK OF PBEE PROPOSED BY PEER [25]

The characterisation of the seismic load is the first step, and takes into account the location, soil properties and type of the structure. Different levels of seismic action are considered, defined in terms of an IM. The peak acceleration value and the spectral acceleration for the fundamental period of the structure are the most common parameters used for that purpose.

After that, an important part of the engineer's work is to find the exact relation between all those parameters (IMs, EDPs, DMs and DVs) in each case. After the structural analysis, the relation between IM and EDPs is found, and gives the probability of reaching a certain level of an EDP (for example, interstorey drift) for a given intensity of the seismic value.

In the EDP-DM relation (damage analysis), qualitative values are considered. For example, if a DM is defined as "loss of electricity" in a building (in this case, a non-structural damage), it is important to evaluate which value of an EDP (same variable: interstorey drift) will cause this, and even more precisely, which is the probability of an electric failure with a certain value of this displacement. This information is shown by fragility curves, which represent the probability of exceeding a particular DM for a given value of an EDP.

For both damage and structural analysis, it is necessary to generate a model of the structure and analyse its response to differently scaled seismic loads. Models may be lineal or non-linear, and also the use of static or dynamic analyses is permitted. However, the actual tendency is to perform non-linear dynamic analysis (time-history simulations) with accelerograms representing the seismic action. Incremental Dynamic Analyses are performed by scaling a given (or generated) record in order to fit each value of an IM, and determine the response of the system. This method is also called "dynamic pushover analysis" (DPO), because of its similarities to the pushover method described in the Eurocode section, although this one uses dynamic excitation instead of static loads. The final result of this process is a curve which expresses an EDP as a function of a chosen IM parameter.

Finally, once relation between IM-EDP-DM is known, losses are computed. Those are monetary units (dollars, Euros...), downtime (hours, days...) and casualties (injures, deaths...). The result is the mean value of the probability of exceeding a given value of a DV (losses) in a year, according to the *Pacific Earthquake Engineering Center* [7,12,13]:

$$\lambda(DV) = \iiint G(DV|DM) \cdot |dG(DM|EDP)| \cdot |dG(EDP|IM)| \cdot |d\lambda(IM)| \quad \textbf{(2.28)}$$

Where:

$$G(x|y) \text{ is } P(X > x \mid given\ Y = y) \quad \textbf{(2.29)}$$

This integral is in fact the product of all the probabilities of exceedance of each parameter being higher than a reference value, so that at the end

This conclusion leads to check if the response is reasonable, according to the needs of the client (risk-management). In the previously given example, the amount of money which would be lost by the owners of the building -because of loosing electricity during a certain period of time- would be calculated. If performance objectives are not achieved, the iterative process starts again by improving the previous design. Sometimes, performance objectives are not only decided by the owners of the building, but imposed by insurance companies, in order to have more reduced insurance fees [26].

Although this is the most recent methodology proposed by the PEER, it is also possible to analyse and design buildings using part of the resources available. For example, there are PBEE methodologies which only consider three quantitative variables (DV, DM and IM), with an analogue procedure.

2.3 COMPARISON BETWEEN EUROCODE 8 AND PBEE

Eurocodes and the PBEE propose different philosophies of designing and analysing buildings. In this section, similarities and differences between them will be described.

First of all, it is important to clarify that Eurocodes give a clear answer whether a building does or does not fulfil its requirements. It is a code, and it has a tendency, like other European norms, to be conservative. The PBEE is not a document, but a methodology still in development since many years, without a particular authority behind it. However, agencies and organisations like the Federal Emergency Management Agency (FEMA) or the Pacific Earthquake Engineering Research Center (PEER) edit books and detailed recommendations on the use of this methodology.

As seen in both sections explaining Eurocode 8 and the PBEE methodology, the first one defines two limit states, while the second proposes four different performance levels. However, this performance levels are only indicators of the performance of the building. PBEE focuses on finding a particular solution for each project, by establishing performance objectives which may depend on the purpose of the building, and normally decided by its owners. Within this context, two buildings made according to Eurocode 8 may be equally considered as “acceptable” if they accomplish all the requirements, but maybe one has a better performance than the other. PBEE gives resources to quantify this performance, while Eurocodes do not.

Both methods are able to consider the importance of the building. Eurocode does this by using the importance factors and the PBEE by setting stricter objectives for buildings of special importance, for example by using the reference performance objectives proposed by Vision 2000.

Probabilistic methods are more explicit in PBEE than in Eurocode 8, whose partial factors method could be defined as a semi-probabilistic approach to risk analysis. While in the European norms probabilities of exceedance have been given by the National Annex, by following the PBEE methodology it is possible to answer questions about the probability of having any kind of damage, and also define particular objectives in terms of probabilities or return periods. For example, in an industry downtime leads to economical losses, therefore a company may decide to ensure that downtime after a relative frequent earthquake shall be less than 2 days, defining this with a determinate confidence level.

Regarding to more technical aspects, PBEE (and, in particular, performing an IDA) also allows knowing which parts of the structure will fail first. By following the described iterative design process, it is possible to make improvements on the design which maybe do not increase the cost too much, for the advantages they give in case of seismic events, as reparations may cost a lot more if those improvements would not have been carried out. When designing according to Eurocode 8, it is also possible to set dissipative structural elements which will fail first, but it does not focus on the economical advantage of having all damage concentrated in an easy-repairable fusible element, for instance. Both philosophies allow the final technical solution, but with a different approach. Within the PBEE it becomes easier to show the economical advantages of a particular design in terms of performance, than it is by using the Eurocodes.

Finally, Eurocodes are prescriptive [14], and thus allow less flexibility to solutions. It is possible to design for higher performance than the minimum required by the codes (by making the R_d value several times higher than E_d, for example), but they were not originally intended for that. For example, two different designs for the same building, one having two time the stiffness of the other, but both fulfilling the requirements in Eurocode, would be equally considered as valid. This means that, after the safety verification of ULS, no advantage is given to the one which has more stiffness.

On the other hand, because of that, it is easier to make decisions within Eurocodes, as most of them are already given directly by the norm and its National Annexes. At the same time this reduces flexibility, it increases security by minimizing possible errors committed by engineers, acting also as a shield for them in case of unpredicted damage. Because of that, more resources are needed when designing within the PBEE methodology than for Eurocodes. The PBEE tends to share decisions between engineers and clients during the design process, by evaluating which amount of risk are decision-makers willing to allow, while the philosophy of Eurocodes does not consider non-technical parameters and its relation with the design. All these analyses increase the cost of the design process. However, this may possibly reduce afterwards the costs derived from reparations or downtime after a seismic event, if the performance objectives have been carefully chosen with the intention of minimizing them.

3 COMPOSITE WALL SYSTEMS

3.1 INTRODUCTION

Buildings are designed to resist not only gravitational loads, related to self-weight and the use of the building, but also horizontal loads such as wind or earthquakes. These are even more important and can cause huge damage or lead to collapse. Each structural system has its own mechanism to resist horizontal forces. Examples of lateral resistance systems are shown in the following figure:

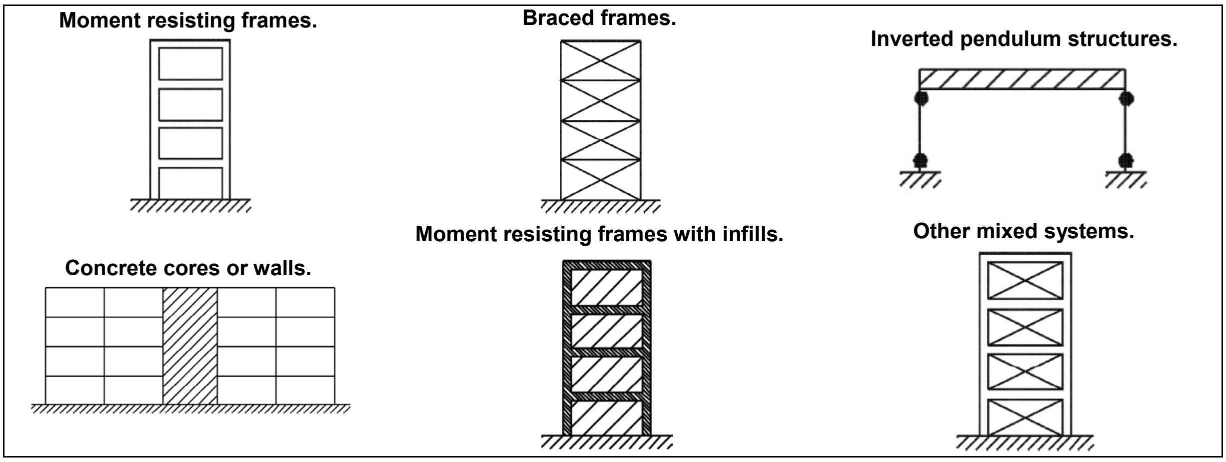


FIGURE 3-1: DIFFERENT LATERAL LOAD RESISTING STRUCTURAL SYSTEMS [1]

In particular, during an earthquake, it is not only important to resist horizontal forces, but also to dissipate energy. This energy is dissipated in different parts of the structure due to mechanisms such as friction (when transformed into heat) or plastic deformation. For example, in moment resisting frames with strong columns, the main dissipative system is the formation of plastic hinges in beams.

Infilled frames are dual systems formed by a steel or concrete framed structure, the reinforced concrete or steel plate infill, and the connection between them (also called interface). The frame structure is formed by vertical and horizontal boundary elements, being those columns and beams respectively.

Those hybrid[4] systems have a good ductile behaviour when subjected to cyclic loads, and they are also an economical solution for earthquake-resistant buildings [27]. As they only provide stiffness in the plane of the wall, it is necessary to have those lateral resistance elements placed in, at least, two orthogonal directions.

Steel frames with reinforced concrete infills (also called SRCW[5]) will be described in this chapter.

[4] They are called “hybrid” because they use two different resistant elements (the steel profiles and the reinforced concrete infill). Composites, however, are materials which are formed by two or more materials, for example, reinforced concrete.

[5] Steel frames with Reinforced Concrete infill Walls.

3.2 STEEL FRAMES WITH REINFORCED CONCRETE INFILLS (SRCW)

Steel frames with reinforced concrete infills are formed by a reinforced concrete wall, with a boundary steel frame. Depending on the quality of the interface, three types of infilled frames are defined [17]:

- Non-integral infilled frames are those, where there is no connection between the frame and the infill. For example, a steel frame filled with a masonry wall.
- In fully-integral infilled frames, a perfect connection between the steel columns and beams is guaranteed, so that both elements work together. In those cases, shear connectors are normally used.
- Semi-integral infilled frames, where the connection between both materials is present only in discrete points, so it is not continuous.

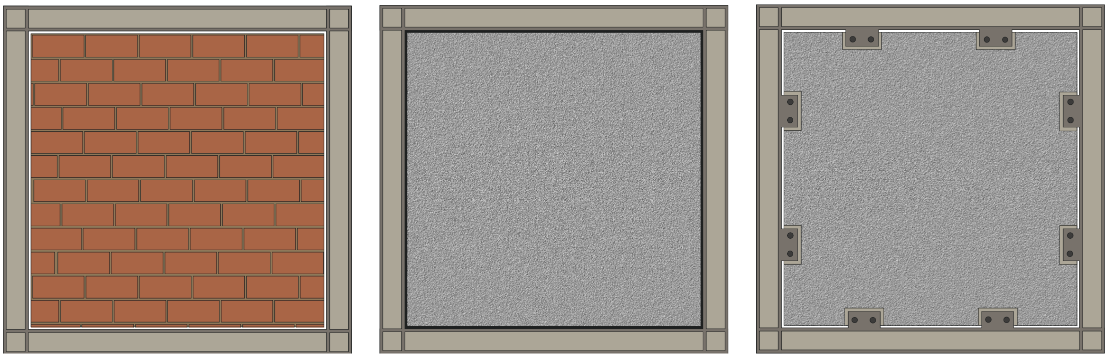


FIGURE 3-2: NON-INTEGRAL, FULLY-INTEGRAL AND SEMI-INTEGRAL INFILLED FRAMES

Fully-integral infilled frames are the ones with the highest stiffness, in comparison with the other two types.

There are also two different ways of connecting the horizontal beams to the columns, which results into two different structural systems:

- Shear walls inside simply-supported frames. In this structural system, the infill walls are expected to provide the whole lateral resisting capacity. Although it is possible that the boundary elements also reach the plastic region, they are designed as a gravity structure only, while the walls are designed to provide all the lateral resistance. This system is more economic, because of the relative inexpensive simply-supported connections, compared to the moment resisting ones.
- Shear walls inside or in parallel with moment frames (dual systems). Here the walls and the boundary frames together resist the lateral load. However, infill panels are normally designed to resist the entire horizontal load, and the frames to work as back-up systems, for example by having them dimensioned for the 25% of the whole lateral load [28].

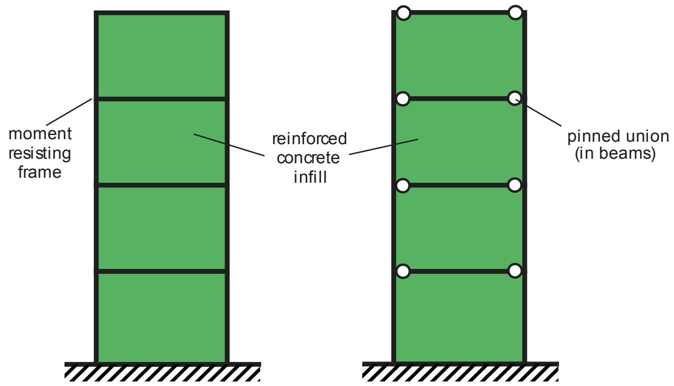


FIGURE 3-3: MOMENT FRAMES VS SIMPLY-SUPPORTED FRAME

When subjected to lateral loads, infilled walls behave essentially as shown in figure 13:

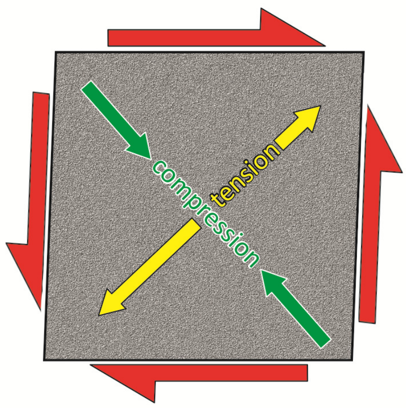


FIGURE 3-4: BEHAVIOUR OF CONCRETE INFILLS WHEN SUBJECTED TO LATERAL LOAD

The concrete is strong in compression, while tension has to be held by the steel reinforcement. The red arrows symbolise the shear connection between concrete and steel.

As mentioned in the introduction, these systems have a good behaviour during seismic events, which consist on a lateral load changing its direction several times (that is, a cyclic load). Dissipative mechanisms in SRCW are [27]:

1. Minor cracking in the concrete wall, due to inelastic deformation, when subjected to low or moderate seismic events. Those cracks allow relative easy and economical reparation with epoxy resin, for example.
2. Inelastic response of the whole composite system, with severe cracking and crushing of concrete combined with yielding of the reinforcement steel and the stud connectors of the interface. As long as the frame remains intact (that is, still in the elastic range), the reparation of the wall is possible by replacing the whole infill.
3. Yielding of the boundary elements, with no possible reparation. Therefore, as it is not the desired behaviour, these elements are usually over-dimensionated to prevent this scenario.

The behaviour of the system when subjected to the seismic cyclic load is hysteretic, due to effect of permanent deformations when materials enter the inelastic range. Each cycle deteriorates the properties and the stiffness of the resisting system, until its complete failure. This hysteretic behaviour of consecutive load cycles is shown in load-drift graphics (figure 14 is an example). Cyclic shear loads cause cracking of the concrete, and this is called the “pinching effect”.

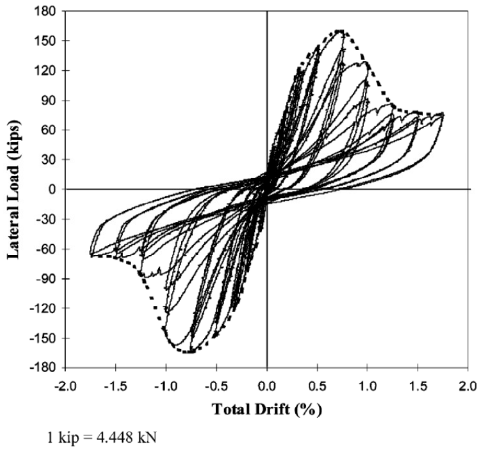


FIGURE 3-5: EXAMPLE OF THE HYSTERETIC BEHAVIOUR OF A SRCW SPECIMEN [27]

Depending on the ductility of the wall, which determines its capacity to dissipate energy within load cycles, the graphic has different shapes. In walls with higher ductility, this curve covers a larger area and show higher values of displacement. On the other hand, more fragile walls have thinner graphics. The area within each cycle load gives an idea of how much energy has been dissipated.

3.3 SRCW WITHIN EUROCODE 8

Infilled frames are not treated in a separate chapter in Eurocode 8. The basic specific information needed to design them is shared between these three parts:

- EN 1998-1 [1] Chapter 5 : Specific rules for steel buildings
- EN 1998-1 [1] Chapter 6: Specific rules for concrete buildings
- EN 1998-1 [1] Chapter 7: Specific rules for composite steel-concrete buildings

And also references to those other documents are given:

- EN 1990 [19] (Basis of structural design)
- EN 1992-1-1 [20] (Eurocode 2: Design of concrete buildings)
- EN 1993-1-1 [21] (Eurocode 3: Design of steel structures)
- EN 1994-1-1/2 [22] (Eurocode 4: Design of composite steel and concrete structures)

For example, the steel profiles should be chosen according to Eurocode 3 –complemented by references in Eurocode 8–, the concrete infills must be dimensioned according to Eurocode 8 and some parts of Eurocode 2, while the connection between the steel and concrete is treated in Eurocode 4.

However, the basic general information about the composite structural system is given in the specific composite section of Eurocode 8 (EN 1998-1 7 [1]). In this section, different composite structural types are classified. The one which fits best for the case examined within the scope of this thesis is “composite structural systems, type 1” (EN 1998-1 7.3.1e [1]), which is defined as a “steel or composite frame working together with concrete infill panels connected to the steel structure”. Concrete elements are classified in the Eurocode in three ductility classes: low (DCL), medium (DCM) and high (DCH), depending on their capacity to dissipate energy. For each class, different building requirements and constructive details are given. Higher ductility classes are reached by fulfilling more strict requirements mainly related to the different reinforcements of the concrete wall. This document will focus on DCM design, which is the ductility class chosen for the case study. Eurocode 8 gives in this chapter upper limits for the behaviour factor (explained in 3.1.3.1) of this type of composite structure, depending on the ductility class of the walls:

Ductility Class Medium	Ductility Class High
$q \leq 3 \cdot \frac{\alpha_u}{\alpha_1}$	$q \leq 4 \cdot \frac{\alpha_u}{\alpha_1}$

TABLE 3-1: UPPER LIMITS FOR BEHAVIOUR FACTORS FOR COMPOSITE STRUCTURAL SYSTEMS, TYPE 1 [1]

For composite structural system the default value for $\frac{\alpha_u}{\alpha_1}$ is given equal to 1.1.

Once seismic loads are determined from the analysis, the system has to be dimensioned and completely described. In order to have a complete final design, it is necessary to determine:

- The steel profiles (columns and beams) and their material properties.
- The dimensions of the concrete infill wall and the strength of the concrete used.
- The concrete reinforcement and its properties, including:
  - Horizontal shear reinforcement
  - Vertical reinforcement (resisting bending, shear and normal forces)
  - Confining reinforcement of boundary elements
  - Additional reinforcement surrounding the stud connectors (against splitting)
- The connection between concrete and steel (headed stud connectors)

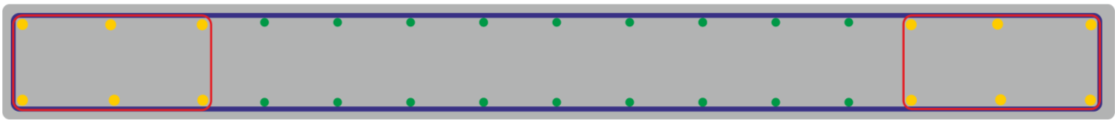


FIGURE 3-6: CROSS-SECTION SHOWING FOUR (OF FIVE) TYPES OF REINFORCEMENT PLACED IN A WALL. BLUE FOR HORIZONTAL REINFORCEMENT, RED FOR CONFINING REINFORCEMENT, YELLOW FOR VERTICAL REINFORCEMENT INSIDE THE CONFINING ZONE AND GREEN FOR THE VERTICAL REINFORCEMENT IN THE INNER ZONE OF THE WALL

All this parameters are subjected to the prescriptions given in the Eurocodes. The different sections directly related to the design of infilled frames, which contain the general requirements and also the particular conditions to be fulfilled, are summarized in the following table. Each verification will be explained in this chapter and then exemplified with a particular case study in the next chapter. The table shows an internal index to be used in this document, a brief description and the collection of sections of different Eurocodes which are related to each verification.

ID	Verifications	EC	Clauses related
0a	Measurements of the wall. Minimum value for b_{wo} and b_w.	8	5.4.1.2.3 5.4.3.4.2(10)
0b	Minimum concrete and steel classes.	8 2 3	7.2.1(1) 7.2.2(1) 7.2.3(1) Annex C 6.2(1)
0c	Concrete cover and spacing of bars	2	4.4.1.2 8.2
1	Resistance of the wall and the boundary elements (moment, shear and axial loads).	8	6.10.3(1) 7.10.1(3) 7.10.2(2)-(3) 5.4.2.4(2)-(7)

ID	Verifications	EC	Clauses related
1a	Axial loads on boundary elements.	3	6.2
1b	Bending and axial resistance of the wall (ULS, plastic).	4 8	 5.4.3.4.1
1c	Shear resistance of the wall. Minimum shear reinforcement.	8 2	5.5.3.4.2 5.5.3.4.3(3)-(5) 6.2.3.1(3) 9.6.2-3
1d	Maximum value of normalized axial load.	8	5.4.3.4.1(2)
1e	Local ductility check. Minimum curvature ductility factor to be provided. Critical region and confining reinforcement. Length of the confined part.	8	5.4.3.4.2(2)-(9) 5.4.3.2.2(9);(11) 5.4.3.4.2(5);(10) 5.2.3.4(3)-(4)
2a	Requirements of the connection between steel and concrete.	8	7.10.1(2) 7.10.3(5)
2b	Requirements for headed studs.	4-2	6.6.3 6.6.4 6.6.1.1(8) Annex C

TABLE 3-2: VERIFICATIONS IN THE DIFFERENT EUROCODES RELATED TO INFILLED WALLS

Where:

- ID 0 is for those related to preliminary requirements.
- ID 1 is for those related to resistance of the composite wall system.
- ID 2 is for those related to the connection between steel and concrete.

3.3.1 VERIFICATIONS 0: PRELIMINARY REQUIREMENTS

3.3.1.1 Verifications 0a: measurements of the wall

The three main dimensions of the wall are height (h_w), lenght (l_w) and thickness (b_w). In other cases it would be possibly necessary to disguise between b_w, which is the thickness of the confined parts of the wall, and b_{w0} (thickness of the central part of the wall). In this case study, however, $b_w = b_{w0}$ applies. There are two conditions which affect the thickness of the wall:

- In EN-1998-1 8 5.4.1.2.3 [1]: $b_{w0} \geq \max\left\{0.15, {h_s}/{20}\right\}$. Where h_s is the clear storey height.
- In EN-1998-1 5.4.3.4.2(10) [1]: $b_w \geq 200\text{mm}$ in the confined parts of the wall, but works here also as general minimum value.

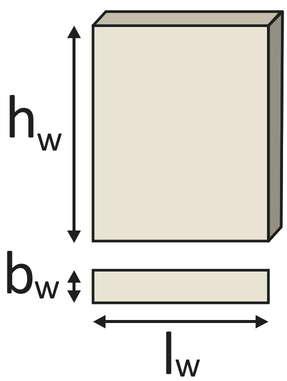


FIGURE 3-7: MEASUREMENTS OF THE WALL

3.3.1.2 Verifications 0b: concrete and steel classes.

According to EN-1998-1 7.2.1(1) [1], the concrete class should not be lower than C20/25. Also, classes higher than C40/50 are not governed by the norm.

Regarding to steel reinforcement, its class has to be greater than B or C in dissipative zones for DCM and greater than class C for DCH, according to EN-1998-1 7.2.2(2) [1]. Properties of the steel reinforcement of different classes are available in Annex C of EN 1992-1-1 [20].

Finally, the steel profiles chosen must be at least class 2 to be able to develop plastic hinges, when needed, according to EN-1993-1-1 6.2.1(8).

3.3.1.3 Verifications 0c: concrete cover and spacing of bars

Concrete cover and the spacing of reinforcement bars are treated in Eurocode 2. The minimum cover depends on many parameters, such as the environment (corrosion, humidity, etc.), the quality controls during the building process, the desired fire resistance, etc. The final value is calculated according to EN-1992-1-1 4.1.1.2:

$$\begin{aligned} c_{nom} &= c_{min} + \Delta c_{dev} \\ c_{min} &= max\{c_{min,b};\ c_{min,dur} + \Delta c_{dur,\gamma} - \Delta c_{dur,st} - \Delta c_{dur,add};\ 10\ mm\} \end{aligned} \tag{3.1}$$

Where $c_{min,b}$ is the minimum cover with regard to bond (the diameter of the bar in case of separated bars; and if nominal aggregate size is higher than 32mm, the value is increased in 5mm); $c_{min,dur}$ is a value which depends both on the structural class and the exposure class; $\Delta c_{dur,\gamma}$ and $\Delta c_{dur,add}$ are factors for additional protection, but recommended values are 0mm for both; and Δc_{dev} recommended value is 10mm.

The structural classes go from S1 to S6, while exposure classes from X0 to XA3 (there are 18 different levels). Moreover, depending on different criteria such as concrete class or special design conditions, exposure classes can be increased or decreased.

In order to allow fresh concrete to be brought in through the reinforcement bars, a minimum spacing has to be provided between them. The minimum distance is calculated according to EN-1992-1-1 8.2:

$$s_{min} = max\{k_1 \cdot diameter; d_g + k_2; 20mm\} \tag{3.2}$$

Where d_g is the maximum size of aggregate and $k_1 = 1$; $k_2 = 5$ (recommended values).

3.3.2 VERIFICATIONS 1: RESISTANCE OF THE WALL AND THE BOUNDARY ELEMENTS

Eurocode 8 gives some rules regarding to how the loads have to be carried by the composite system:

- Horizontal shear forces must be carried by shear in the wall and in the interface between the wall and beams. (EN 1998-1 7.10.1(3)P [1]).
- It shall be assumed that the seismic action effects in vertical boundary elements are axial forces only. (EN 1998-1 7.10.2(2)P [1]).
- It shall be assumed that shear forces are carried by the reinforced concrete wall, and the gravity loads by the wall acting composedly with the vertical boundary elements. (EN 1998-1 7.10.2(3) [1]).

This leads to check for moment, shear and axial load in ULS:

- **Axial loads:** $N_{Ed} \leq N_{Rd(wall)} + N_{Rd(boundary\ elements)}$
- **Moment:** $M_{Ed} \leq M_{Rd(wall)} + M_{Rd(boundary\ elements)}$
- **Shear:** $V_{Ed} \leq V_{Rd(wall)}$

Those verifications are shared between different clauses of the Eurocodes, and are dependent on the reinforcement, the measurements of the wall, the steel profiles chosen, etc. However, the verifications for moment and axial loads will be checked together by calculating the plastic resistant moment and reserving one part of the section for the axial gravity load N_{Ed}.

3.3.2.1 Verifications 1a: axial loads on boundary elements.

Axial loads on boundary elements help to carry both moment and gravity loads. The maximum axial force which a profile is able to carry in the ULS is calculated as follows:

$$N_{Rd} = \frac{f_y \cdot A}{\gamma_{M0}} \tag{3.3}$$

Where γ_{M0} partial factor is 1.0 for the most National Annexes, A is the area of the steel profile and f_y is yield strength of the steel used.

The maximum moment carried by the two profiles depends on the distance between them:

$$M_{Rd(boundary\ elements)} = N_{Rd} \cdot d_{columns} \tag{3.4}$$

This can be schematized in the following figure showing only the two vertical steel profiles:

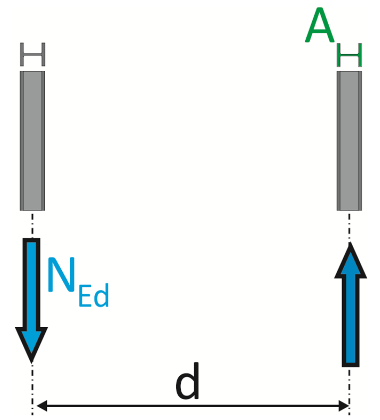


FIGURE 3-8: RESISTANT MOMENT OF THE BOUNDARY ELEMENTS

3.3.2.2 Verifications 1b: bending and axial resistance of the wall

For ULS, the calculation of the plastic neutral axis of the section x_{pl} gives maximum value of moment carried by the concrete wall, after removing the part of the wall destined to hold the vertical load N_{Ed}.

In order to find the neutral plastic axis of the concrete section, two possible cases are considered: the neutral axis lies on the boundary area (case 1) or in the central area of the wall (case 2). For both zones, an average degree of reinforcement is used to simplify the expressions. This means that the reinforcement will be treated as continuum steel reinforcement, instead of considering each discrete bar. By solving the static equilibrium between both sides of the neutral axis, a value for x_{pl} is found. Two hypotheses have to be made for each possible case, and if the result is coherent with the hypothesis, then it is taken as correct.

It is important to notice that, as the calculations are made for the plastic neutral axis, which corresponds to the ULS limit state, the concrete is supposed to resist 0.85·f_{cd} times the area (only in compression), while for the steel the average reinforcement multiplied by f_{sd}.

In the first case, depicted in figure 18, where the plastic neutral axis is placed in the boundary area ($x_{pl} \leq l_b$), the equation is:

$$b_w \cdot f_{sd} \cdot \left[l_b \cdot \rho_{sv,b} + (l_w - 2 \cdot l_b) \cdot \rho_{sv,i} + \left(l_b - x_{pl}\right) \cdot \rho_{sv,b}\right] = \\ = b_w \cdot x_{pl} \cdot \left[0.85 \cdot f_{cd} + f_{sd} \cdot \rho_{sv,b}\right] \tag{3.5}$$

Which leads to:

$$x_{pl} = \frac{f_{sd} \cdot \left[\rho_{sv,b} \cdot 2 \cdot l_b + \rho_{sv,i} \cdot (l_w - 2 \cdot l_b)\right]}{0.85 \cdot f_{cd} + 2 \cdot f_{sd} \cdot \rho_{sv,b}} \tag{3.5}$$

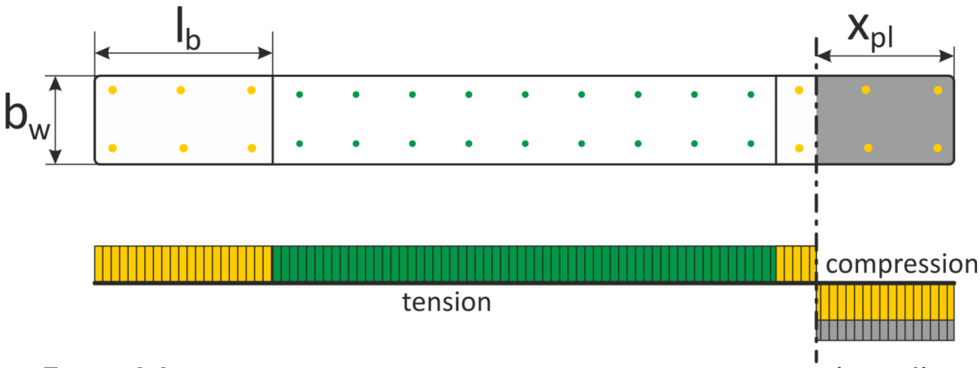


FIGURE 3-9: AND DISTANCES USED TO FIND THE PLASTIC NEUTRAL AXIS (CASE 1)

And if it is in the central area ($x_{pl} \geq l_b$), figure 19:

$$b_w \cdot f_{sd} \cdot \left[l_b \cdot \rho_{sv,b} + \left(l_w - x_{pl} - l_b\right) \cdot \rho_{sv,i}\right] = \\ = b_w \cdot \left[x_{pl} \cdot 0.85 \cdot f_{cd} + f_{sd} \cdot \left(\left(x_{pl} - l_b\right) \cdot \rho_{sv,i} + l_b \cdot \rho_{sv,b}\right)\right] \tag{3.6}$$

$$x_{pl} = \frac{f_{sd} \cdot \rho_{sv,i} \cdot l_w}{0.85 \cdot f_{cd} + 2 \cdot f_{sd} \cdot \rho_{sv,i}} \tag{3.6}$$

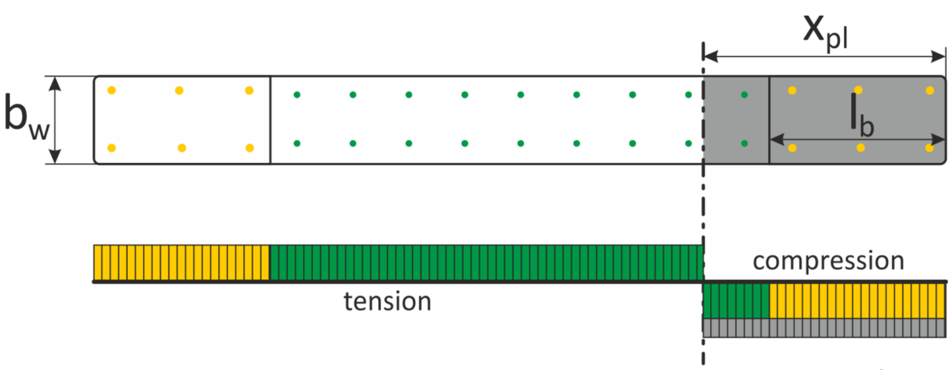


FIGURE 3-10: FORCES AND DISTANCES USED TO FIND THE PLASTIC NEUTRAL AXIS (CASE 2)

Where the average degree of reinforcement is calculated as follows for both areas:

$$\rho_{sv,b} = \frac{A_{sv,b}}{b_w \cdot l_b} \tag{3.7}$$

$$\rho_{sv,i} = \frac{A_{sv,i}}{b_w \cdot (l_w - 2 \cdot l_b)} \tag{3.8}$$

After the plastic neutral axis is found, some area has to be reserved on both sides of the neutral axis to hold the axial force N_{Ed}. For each of the previous cases, two possible cases exist. For a better understanding, in the following figure there is a summary of all possible cases.

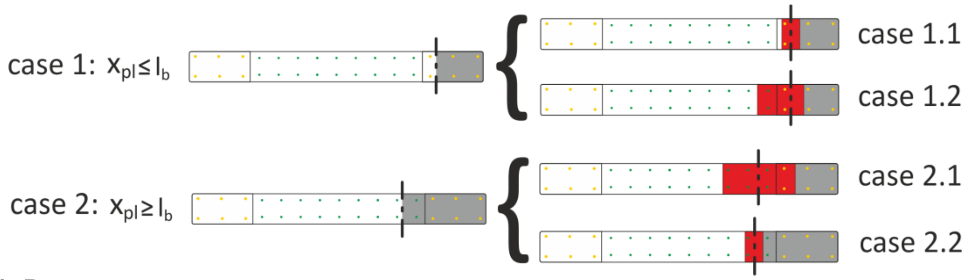


FIGURE 3-11: DIFFERENT CASES FOR THE DETERMINATION OF THE RESISTANT MOMENT OF THE CONCRETE WALL

In order to find the dimensions of the area holding the axial force (in red), it is necessary to solve a different system of equations for each case. With a similar procedure done before, the solution for each case is taken as correct if it matches the conditions required in each one. Once these dimensions are known, the resistant plastic moment of the section is calculated on the basis of all present forces multiplied by their distance to the neutral axis.

Schemes on each of the four possible cases are shown in the following figures. In case 1.1, the area destined to the axial force is contained inside the boundary area of the wall.

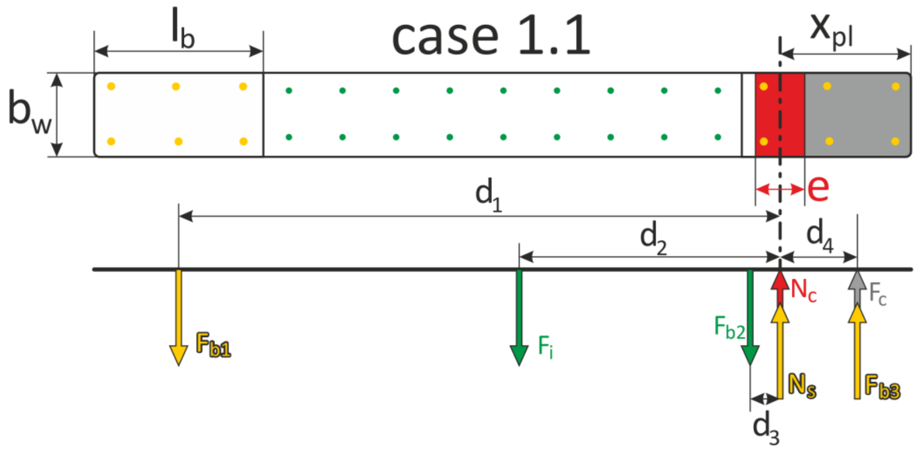


FIGURE 3-12: SCHEME ON THE FORCES AND DISTANCES FOR CASE 1.1

In case 1.2, it takes part of the central area and part of the boundary area. Since the reinforcement ratios of the two areas are different, the red area does not have to be symmetric.

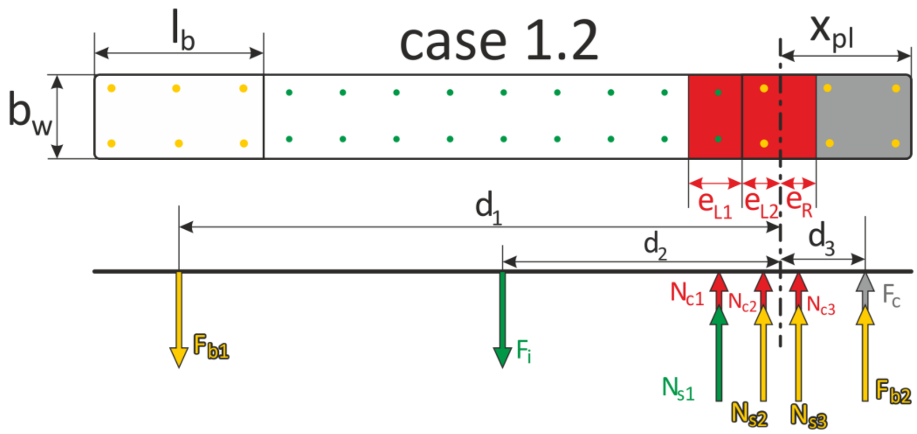


FIGURE 3-13: SCHEME ON THE FORCES AND DISTANCES FOR CASE 1.2

Case 2.1 is similar to the previous one, but corresponds to the neutral axis being outside the boundary area.

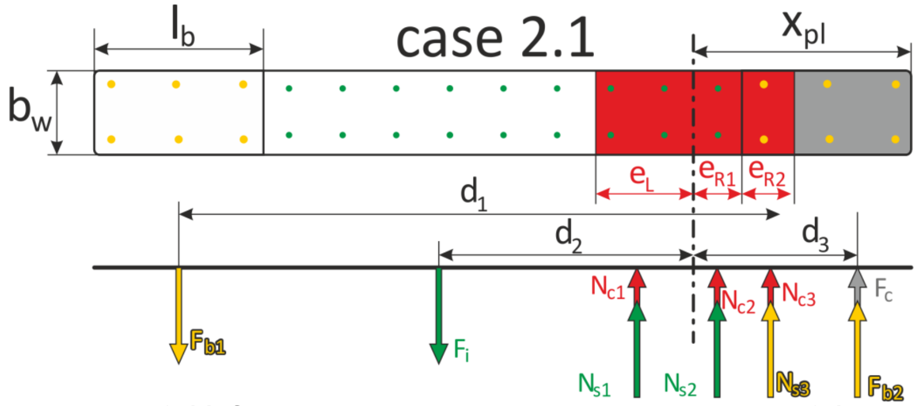


FIGURE 3-14: SCHEME ON THE FORCES AND DISTANCES FOR CASE 2.1

Finally, in case 2.2 the whole axial load is carried within the central area of the wall.

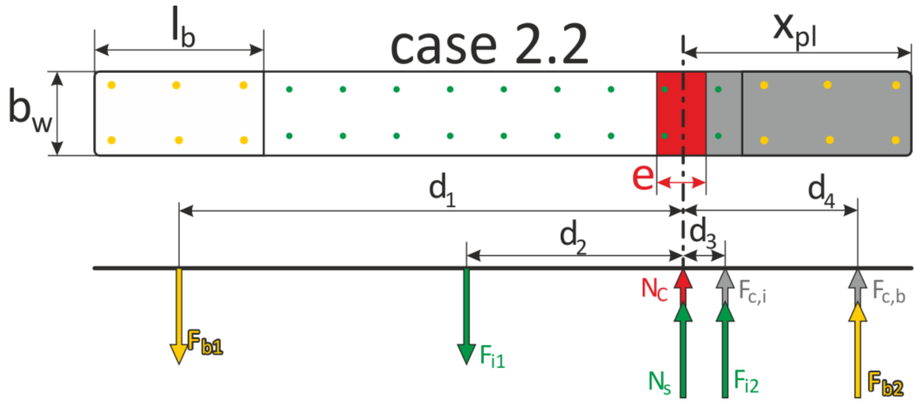


FIGURE 3-15: SCHEME ON THE FORCES AND DISTANCES FOR CASE 2.2

The systems of equations used to determine the dimensions of the "red" area, and its conditions are detailed in table 12:

Case	Equations	Conditions
1.1	$\{N_{Ed} = N_c + N_s$ Where: $N_c + N_s = b_w \cdot e \cdot (0.85 \cdot f_{cd} + \rho_{sv,b} \cdot f_{sd})$	$x_{pl} < l_b$ $l_b - x_{pl} > \frac{e}{2}$
1.2	$\begin{cases} N_{Ed} = N_{c1} + N_{s1} + N_{c2} + N_{s2} + N_{c3} + N_{s3} \\ e_{L2} = l_b - x_{pl} \\ (N_{c1} + N_{s1}) \cdot \frac{e_{L1}}{2} = (N_{c2} + N_{s2}) \cdot \frac{e_{L2}}{2} + (N_{c3} + N_{s3}) \cdot \left(\frac{e_R}{2} + e_{L2}\right) \end{cases}$ Where: $N_{c1} + N_{s1} = b_w \cdot e_{L1} \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,i} \cdot f_{sd}\right)$ $N_{c2} + N_{s2} = b_w \cdot e_{L2} \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,b} \cdot f_{sd}\right)$ $N_{c3} + N_{s3} = b_w \cdot e_{L3} \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,b} \cdot f_{sd}\right)$	$x_{pl} < l_b$ $e_{L1} + e_{L2} > l_b - x_{pl}$

Case	Equations	Conditions
2.1	$\begin{cases} N_{Ed} = N_{c1} + N_{s1} + N_{c2} + N_{s2} + N_{c3} + N_{s3} \\ e_{R1} = x_{pl} - l_b \\ (N_{c1} + N_{s1}) \cdot \frac{e_L}{2} = (N_{c2} + N_{s2}) \cdot \frac{e_{R1}}{2} + (N_{c3} + N_{s3}) \cdot \left(e_{R1} + \frac{e_{R2}}{2}\right) \end{cases}$ Where: $N_{c1} + N_{s1} = b_w \cdot e_L \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,i} \cdot f_{sd}\right)$ $N_{c2} + N_{s2} = b_w \cdot e_{R1} \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,i} \cdot f_{sd}\right)$ $N_{c3} + N_{s3} = b_w \cdot e_{R2} \cdot \left(0.85 \cdot f_{cd} + \rho_{sv,b} \cdot f_{sd}\right)$	$x_{pl} > l_b$ $e_{R1} + e_{R2} > x_{pl} - l_b$ $l_b > e_{R2}$
2.2	$\{N_{Ed} = N_c + N_s$ Where: $N_c + N_s = b_w \cdot e \cdot (0.85 \cdot f_{cd} + \rho_{sv,i} \cdot f_{sd})$	$x_{pl} > l_b$ $x_{pl_b} - \frac{e}{2} > l_b$

TABLE 3-3: SYSTEMS AND CONDITIONS NEEDED FOR THE CALCULATION OF THE AREA DESTINED TO THE AXIAL LOAD

After that, the plastic resistant moment can be calculated, depending on the case:

$$M_{pl,case\ 1.1} = F_{b1} \cdot d_1 + F_i \cdot d_2 + F_{b2} \cdot d_3 + (F_{b3} + F_c) \cdot d_4 \quad (3.9)$$

$$M_{pl,case\ 1.2} = M_{pl,case\ 2.1} = F_{b1} \cdot d_1 + F_i \cdot d_2 + (F_{b2} + F_c) \cdot d_3 \quad (3.10)$$

$$M_{pl,case\ 2.2} = F_{b1} \cdot d_1 + F_{i1} \cdot d_2 + (F_{i2} + F_{ci}) \cdot d_3 + \left(F_{c,b} + F_{b2}\right) \cdot d_4 \quad (3.11)$$

Where:

Case	Forces	Distances
1.1	$F_{b1} = b_w \cdot l_b \cdot \rho_{sv,b} \cdot f_{sd}$ $F_i = b_w \cdot (l_w - 2 \cdot l_b) \cdot \rho_{sv,i} \cdot f_{sd}$ $F_{b2} = b_w \cdot (l_b - x_{pl} - \frac{e}{2}) \cdot \rho_{sv,b} \cdot f_{sd}$ $F_{b3} = b_w \cdot (x_{pl} - \frac{e}{2}) \cdot \rho_{sv,b} \cdot f_{sd}$ $F_c = b_w \cdot (x_{pl} - \frac{e}{2}) \cdot 0.85 \cdot f_{cd}$	$d_1 = l_w - \frac{l_b}{2} - x_{pl}$ $d_2 = \frac{l_w}{2} - x_{pl}$ $d_3 = \frac{l_b - x_{pl} + \frac{e}{2}}{2}$ $d_4 = \frac{e}{2} + \frac{x_{pl} - \frac{e}{2}}{2}$
1.2	$F_{b1} = b_w \cdot l_b \cdot \rho_{sv,b} \cdot f_{sd}$ $F_i = b_w \cdot (l_w - 2 \cdot l_b - e_{L1}) \cdot \rho_{sv,i} \cdot f_{sd}$ $F_{b2} = b_w \cdot \left(x_{pl} - e_R\right) \cdot \rho_{sv,b} \cdot f_{sd}$ $F_c = b_w \cdot \left(x_{pl} - e_R\right) \cdot 0.85 \cdot f_{cd}$	$d_1 = l_w - \frac{l_b}{2} - x_{pl}$ $d_2 = \frac{l_w - 2 \cdot l_b - e_{L1}}{2} + e_{L1} + e_{L2}$ $d_3 = x_{pl} - \frac{x_{pl} - e_R}{2}$

Case	Forces	Distances
2.1	$F_{b1} = b_w \cdot l_b \cdot \rho_{sv,b} \cdot f_{sd}$ $F_i = b_w \cdot (l_w - 2 \cdot l_b - e_L - e_{R1}) \cdot \rho_{sv,i} \cdot f_{sd}$ $F_{b2} = b_w \cdot (x_{pl} - e_{R1} - e_{R2}) \cdot \rho_{sv,b} \cdot f_{sd}$ $F_c = b_w \cdot (x_{pl} - e_{R1} - e_{R2}) \cdot 0.85 \cdot f_{cd}$	$d_1 = l_w - \frac{l_b}{2} - x_{pl}$ $d_2 = \frac{l_w - 2 \cdot l_b - e_L - e_{R1}}{2} + e_L$ $d_3 = \frac{x_{pl} - e_{R1} - e_{R2}}{2} + e_{R1} + e_{R2}$
2.2	$F_{b1} = b_w \cdot l_b \cdot \rho_{sv,b} \cdot f_{sd}$ $F_{i1} = b_w \cdot \left(l_w - l_b - x_{pl} - \frac{e}{2}\right) \cdot \rho_{sv,i} \cdot f_{sd}$ $F_{i2} = b_w \cdot \left(x_{pl} - l_b - \frac{e}{2}\right) \cdot \rho_{sv,i} \cdot f_{sd}$ $F_{c,i} = b_w \cdot \left(x_{pl} - l_b - \frac{e}{2}\right) \cdot 0.85 \cdot f_{cd}$ $F_{c,b} = b_w \cdot l_b \cdot 0.85 \cdot f_{cd}$ $F_{b2} = b_w \cdot \rho_{sv,b} \cdot f_{sd}$	$d_1 = l_w - \frac{l_b}{2} - x_{pl}$ $d_2 = \frac{l_w - l_b - x_{pl} - \frac{e}{2}}{2} + \frac{e}{2}$ $d_3 = \frac{l_w - l_b - \frac{e}{2}}{2} + \frac{e}{2}$ $d_4 = x_{pl} - \frac{l_b}{2}$

TABLE 3-4: VARIABLES USED FOR THE CALCULATION OF THE PLASTIC RESISTANT MOMENT OF THE WALL

Finally, the resistant moment of the concrete wall is directly the plastic moment calculated with the previously detailed equations, so that $M_{Rd(wall)} = M_{pl}$.

3.3.2.3 Verifications 1c: shear resistance of the wall

The concrete wall alone has to resist the horizontal seismic load V_{Ed}. Therefore, shear reinforcement is necessary in both vertical and horizontal directions. Eurocode 8 disguises between different mechanisms of failure, in order to calculate the resistance of the wall to shear forces (V_{Rd}) by different complementary procedures. All of them must be greater than the design shear force (V_{Ed}). For medium ductility class (DCM), Eurocode checks for the shear resistance of the wall by using the expressions in EN-1992-1-1 6.2.3 [20], as explained in EN-1998-1 5.3.4.1(1):

The method in Eurocode 2 gives V_{Rd} as the smaller value of $V_{Rd,max}$ and $V_{Rd,s}$, and using according to Eurocode 8 the parameters $z = 0.8 \cdot l_w$ and $\tan\theta = 1$.

$$V_{Rd} = \min\{V_{Rd,s};\ V_{Rd,max}\} \tag{3.12}$$

$$V_{Rd,s} = \frac{A_{sw}}{s} \cdot z \cdot f_{ywd} \cdot \cot\theta \tag{3.13}$$

$$V_{Rd,max} = \frac{\alpha_{cw} \cdot b_w \cdot z \cdot \nu_1 \cdot f_{cd}}{cot\theta + tan\theta} \tag{3.14}$$

Where A_{sw} is the cross sectional area of the shear reinforcement; f_{ywd} is the design yield strength of the reinforcement steel ($=0.8 \cdot f_{wk}$); s is the spacing of the stirrups; α_{cw} is 1.0 for non-prestressed structures; and ν_1 is:

$$\begin{gathered} 0.6 \text{ for } f_{ck} \leq 60MPa \\ 0.9 - \frac{f_{ck}}{200} > 0.5 \text{ for } f_{ck} > 60MPa \end{gathered} \tag{3.15}$$

However, the formula for compressive strength higher than 60 MPa will not be used since, as explained in 4.3.1.2, concrete classes higher than C40/50 are not within the scope of Eurocode 8.

The full value of this shear resistance can be taken outside the critical region of the wall, while only 40% of this value may be considered inside. The critical region of the wall is defined as the area where the formation of the plastic hinge is expected to be. This area is located at the base of the wall, with a height of:

$$h_{cr} = \max\left\{l_w, \frac{h_w}{6}\right\}; \quad h_{cr} \leq \begin{cases} 2 \cdot l_w \\ \begin{cases} h_s \; for \; n \leq 6 \; storeys \\ 2 \cdot h_s \; for \; n \geq 7 \; storeys \end{cases} \end{cases} \tag{3.16}$$

This criterion in Eurocode 8 was designed for high walls, but in the case of infilled walls, as storeys are independent from each other, it results in the critical area usually covering most of the wall:

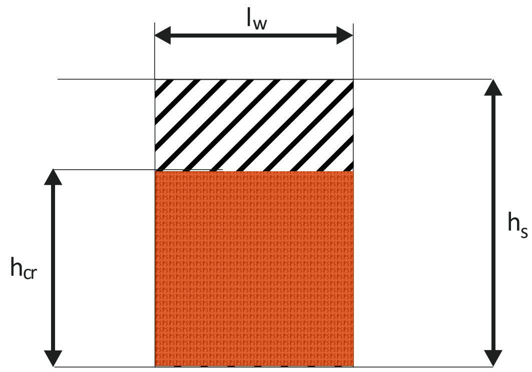


FIGURE 3-16: CRITICAL REGION OF THE WALL

Moreover, minimum and maximum reinforcement conditions for walls detailed in EN 1992-1 9.6.2 and 3 have to be fulfilled:

$$\begin{aligned} \rho_{v,min} &= 0.002 \\ \rho_{v,max} &= 0.004 \\ \rho_{h,min} &= \max\{0.001 \cdot A_c; 0.25 \cdot \rho_v\} \end{aligned} \tag{3.17}$$

The value of the thickness of the wall (b_w) is the maximum allowed distance between two adjacent vertical bars.

3.3.2.4 Verifications 1d: maximum value of normalized axial load

Eurocode 8 normalizes the axial load by dividing it by the resistance of the concrete area where it is applied. According to EN 1998-1 5.4.3.4.1(2) [1] in primary seismic DCM walls the value of the normalised axial load must be:

$$\nu_d = \frac{N_{Ed}}{A_c \cdot f_{cd}} = \frac{N_{Ed}}{b_w \cdot l_w \cdot f_{cd}} \leq 0.4 \tag{3.18}$$

3.3.2.5 Verifications 1e: local ductility check

Verifications regarding local ductility of the wall affect the reinforcement of the boundary elements and also the vertical reinforcement, in order to achieve a determined value of the curvature ductility factor, which depends on the behaviour factor itself. This is explained in

EN 1998-1 5.4.3.4.2 [1]. However, it is necessary before to calculate the value of the necessary ductility, with the expression in EN 1998-1 5.2.3.4(3) [1]:

$$\mu_\phi = 1.5 \cdot \begin{cases} 2 \cdot q_0 - 1 \; if \; T_1 \geq T_C \\ 1 + 2 \cdot (q_0 - 1) \cdot \dfrac{T_C}{T_1} \; if \; T_1 < T_C \end{cases} \tag{3.19}$$

Where q_0 is the basic value of the behaviour factor, in these systems, equal to the behaviour factor q. The 1.5 factor applies to the critical region of the wall, according to EN 1998-1 5.2.3.4(4) [1].

As explained before, this ductility factor is provided through confinement in the boundary areas of the wall (where the bending reinforcement $\rho_{sv,b}$ is placed). The neutral axis depth at ultimate curvature (within the elastic range) is calculated according to EN 1998-1 5.4.3.4.2(5)(a) [1] with the approximate formula:

$$x_u = (\nu_d + \omega_v) \cdot \frac{l_w \cdot b_c}{b_0} \tag{3.20}$$

Where ν_d is already explained in (4.15); ω_v is the mechanical ratio of vertical reinforcement. An average degree of reinforcement for all the wall can be calculated by taking into account all vertical bars and the dimensions of the whole concrete section. Then, if $x_u \leq l_c$ (being l_c the length of the confined zone. This value must be greater than $0.15 \cdot l_w$ and than $1.5 \cdot b_{w0}$), and thus the provisions in EN 1998-1 5.2.3.4.2(4) apply to calculate the necessary confining reinforcement:

$$\alpha \cdot \omega_{wd} \geq 30 \cdot \mu_\phi \cdot (\nu_d + \omega_v) \cdot \varepsilon_{sy,d} \cdot \frac{b_c}{b_0} - 0{,}035 \tag{3.21}$$

$$\alpha = \alpha_n \cdot \alpha_{sr} = \left(1 - \sum_{1 \leq i \leq n} \frac{{b_i}^2}{6 \cdot b_0 \cdot h_0}\right) \cdot \left(1 - \frac{s}{2 \cdot b_0}\right) \cdot \left(1 - \frac{s}{2 \cdot h_0}\right) \tag{3.22}$$

Where $\varepsilon_{sy,d}$ is the design value of steel strain at yield; s is the spacing of the stirrups; n is the total number of bars engaged by the confining hoops; ω_v is the mechanical ratio of vertical reinforcement; and the other variables are shown in the following scheme:

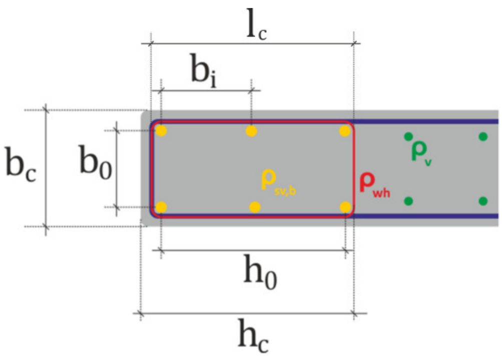


FIGURE 3-17: SCHEME OF THE BOUNDARY AREA OF THE WALL

If $x_u > l_c$, then clause EN 1998-1 5.4.3.4.2(5)(b) applies, and the ductility factor is calculated with a different method:

$$\mu_\phi = \frac{\phi_u}{\phi_y} = \frac{\varepsilon_{cu2,c}/x_u}{\varepsilon_{sy}/(d - x_y)} \tag{3.23}$$

Where $\varepsilon_{sy} = 0.0025$ normally; d is the effective depth of the section of the wall ($= l_w - concrete\ cover - \frac{Ø_{sh}}{2}$); $x_y = d - x_u + \frac{Ø_{sv,b}}{2}$ and $\varepsilon_{cu2,c}$ is calculated as detailed in EN 1992-1-1 3.1.9:

$$\varepsilon_{cu2,c} = 0.0035 + 0.1 \cdot \alpha \cdot \omega_{wd} \tag{3.24}$$

With this new value of the curvature ductility factor, the conditions regarding to the confining reinforcement are checked with the same expression in 3.21.

Finally, in all cases the confining cross-ties must have a greater diameter than 6mm.

3.3.3 VERIFICATIONS 2: INTERFACE

3.3.3.1 Verification 2a: requirements of the connection between steel and concrete

Headed stud connectors are used to transfer shear forces between steel and concrete. Those connectors are typically used in composite construction, and are typically welded by using special semi-automatic equipment. The connectors have high ductility to avoid a brittle failure of the connection. Also, in order to ensure a good connection, it is important to confine the concrete near the connectors, by using cross-ties. This is even more important in shear walls, because of the small distance between the connectors and the concrete surface.

In order to dimensionate the shear connection between the steel profiles and the concrete wall, it is necessary to know which forces have to be transferred between the two surfaces. Eurocode gives two general rules:

- The infills shall be tied to the boundary elements to prevent separation (EN 1998-1 7.10.1(2) [1]).
- Headed stud connectors should be provided to transfer vertical and horizontal shear forces between the structural steel of the boundary elements and the reinforced concrete (EN 1998-1 7.10.3(5) [1]).

The maximum horizontal forces to be carried by the shear connectors (between beams and the wall) are the shear forces which every wall has to resist, named V_{Ed}. Vertical forces in the connection between columns and the wall will be noted as V_{Ed}* and calculated by equilibrium of moments:

$$V_{Ed} \cdot h_w = V_{Ed}^* \cdot l_w \tag{3.25}$$

Where distances are shown in the following scheme:

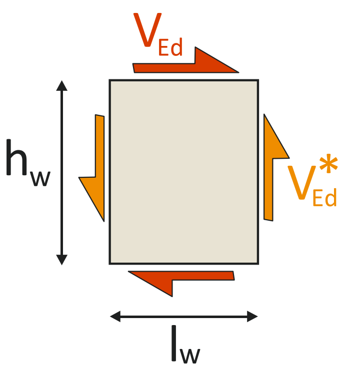


FIGURE 3-18: SHEAR FORCES TO BE CARRIED BY THE SHEAR CONNECTION

3.3.3.2 Verification 2b: dimensioning of headed studs

Once the maximum force to be transferred is known, it is necessary to calculate the number of headed stud connectors which have to be placed in each side of the wall. This includes its measurements (length, diameter, etc.) and material properties, and also the spacing and the necessary reinforcement which ensures the effectiveness of the connection. In general terms:

$$V_{Ed} \leq P_{Rd,h} \cdot n_h$$
$$V_{Ed}^{*} \leq P_{Rd,v} \cdot n_v \quad \textbf{(3.26)}$$

Where n_h and n_v are the number of headed stud connectors in the horizontal direction and in the vertical, and $P_{Rd,h}$ and $P_{Rd,v}$ the resistance of one connector, respectively for the ones used in each direction.

If the conditions detailed in EN 1994-2 6.6.4 are fulfilled, the value of P_{Rd} shall be taken from the formulas in EN 1994-2 6.6.3 (those two values will be referred here as $P_{Rd,1}$ and $P_{Rd,2}$. If not, but the criteria in Annex C of the same document are followed, then the minimum value of $P_{Rd,1}$, $P_{Rd,2}$ and the value calculated with the expressions in Annex C ($P_{Rd,C}$) is taken.

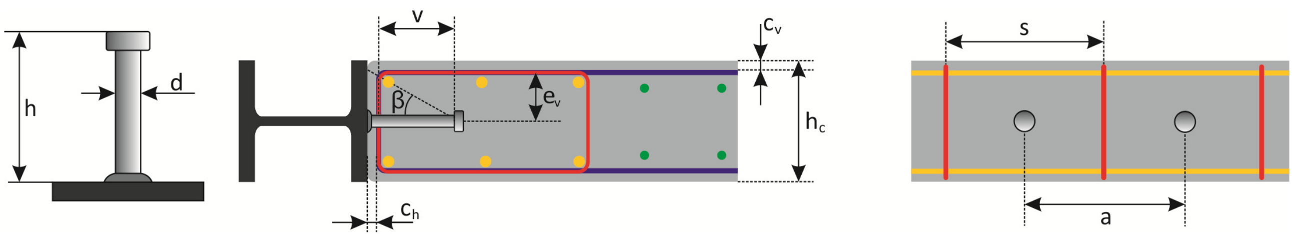


FIGURE 3-19: MEASUREMENTS AND DISTANCES OF THE HEADED STUD CONNECTORS

The first two values are:

$$P_{Rd,1} = \frac{0.8 \cdot f_u \cdot \pi \cdot \frac{d^2}{4}}{\gamma_V} \quad \textbf{(3.27)}$$

$$P_{Rd,2} = \frac{0.29 \cdot \alpha \cdot d^2 \cdot \sqrt{f_{ck} \cdot E_{cm}}}{\gamma_V} \quad \textbf{(3.28)}$$

Where f_u is the ultimate tensile strength of the stud (must be < 500MPa); h is the nominal height of the stud; d is the diameter of the shank of the stud (must be between 19 and 25mm to ensure a ductile connection); γ_V is the partial factor for connections (recommended value is 1.25); f_{ck} is the concrete characteristic strength; E_{cm} is its modulus of elasticity and α:

$$\alpha = \begin{cases} 0.2 \cdot \left(\frac{h}{d} + 1\right) & if\ \ 3 \leq h/d \leq 4 \\ 1 & if\ \ h/d > 4 \end{cases} \quad \textbf{(3.29)}$$

Additional conditions are that $v \geq 14 \cdot d$; $s < 18 \cdot d$; $e_v \geq 6 \cdot d$ and that the tensile force does not exceed 0.1 times the shear resistance of the stud connector.

On the other side, the expression from Annex C is:

$$P_{Rd,C} =\cdot \frac{1.4 \cdot k_V \cdot (f_{ck} \cdot d \cdot e_v)^{0.4} \cdot \left(\frac{a}{s}\right)^{0.3}}{\gamma_V} \quad (in\ kN) \quad \textbf{(3.30)}$$

Where k_V is 1 for shear connectors in an edge position and 1.14 if they are in a middle position; a is the spacing of the connectors; s the spacing of the stirrups and the conditions are that $\frac{a}{2} \leq s \leq a$; $e_v \geq 50mm$; $19 \leq d \leq 50mm$; $\frac{h}{d} \geq 4$; $110 \leq a \leq 440mm$; $\frac{s}{e_v} \leq 3$; minimum diameter for the stirrups being 8mm and for the longitudinal reinforcement, 10mm, and also:

$$\begin{array}{lll} \beta \leq 30° \quad or & v \leq \max\{110; 1.7 \cdot e_v; 1.7 \cdot \frac{s}{2} & for\ uncracked\ concrete \\ \beta \leq 23° \quad or & v \leq \max\{160; 2.4 \cdot e_v; 2.4 \cdot \frac{s}{2} & for\ cracked\ concrete \end{array} \tag{3.31}$$

In investigated case it would be convenient to use the expression for cracked concrete, as the walls are expected to work after several cyclic loads, which may produce premature cracking of the concrete surrounding the stud connectors.

If all those conditions are fulfilled, the minimum value from the three available can be taken for the shear resistance of one headed stud connector, using the corresponding values of the previous expressions for the vertical and the horizontal connectors:

$$P_{Rd} = \min\{P_{Rd,1};\ P_{Rd,2};\ P_{Rd,C}\} \tag{3.32}$$

Finally, the stud connectors must be able to resist an axial load of 0.3 times their shear resistance, according to Annex C of EN 1994-2.

It is important to note that, for the vertical shear connection, the confining reinforcement also works for the studs. However, on the top and bottom of the wall, there is no such reinforcement. This leads to set additional reinforcement stirrups, which are called A_{swv} in this document.

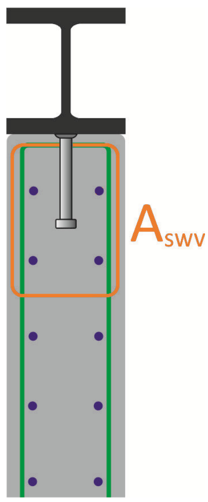


FIGURE 3-20: DETAILS ON THE STUD CONNECTORS IN THE HORIZONTAL DIRECTION AND ITS REINFORCEMENT

4 CASE STUDIES

This thesis will analyse the behaviour and performance of fully-integral reinforced concrete infilled walls in simply-supported frames, by analysing one of the case studies within the INNO-HYCO[6] research project. This European program analyses composite steel-concrete structural solutions for building in seismic areas, and it is being developed by different universities and institutions[7] in Germany, Greece, Belgium and Italy.

In particular, in this thesis a 4 storey building will be analysed.

4.1 DESCRIPTION OF THE STRUCTURAL SYSTEM

The structural system of the building consists of steel frames (HEB and IPE standard profiles) working as gravity-resistant structure, combined with infilled walls placed instead of columns, mainly near the perimetral area of the building. This is done to maximize its effect against the torsion of the building.

The situation of the walls and the two most significative plain vertical sections of the structure are detailed in the following figures:

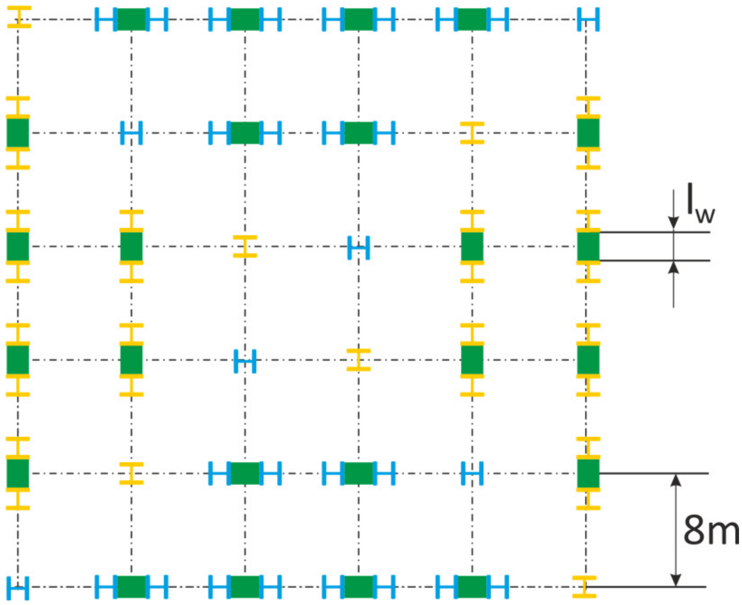


FIGURE 4-1: TOP VIEW OF THE STRUCTURAL SYSTEM (PROFILES AND WALLS NOT SCALED).

The lateral section with "outer shear walls":

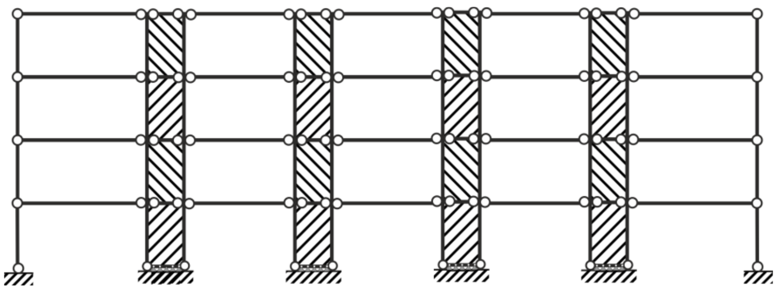


FIGURE 4-2: SECTION OF THE STRUCTURE (FOUR OUTER SHEAR WALLS)

[6] INNOvative HYbrid and COmposite steel-concrete structural solutions

[7] University of Camerino, Italy; Universiti of Liège, Belgium; Consorzio Pisa Ricerche, Italy; Shelter S.A., Greece; Ocam s.r.l., Italy; Dezi Steel Design s.r.l, Italy; University of Thessaly Research Comittee, Greece; and the RWTH Aachen University, Germany.

And the section with “inner shear walls“:

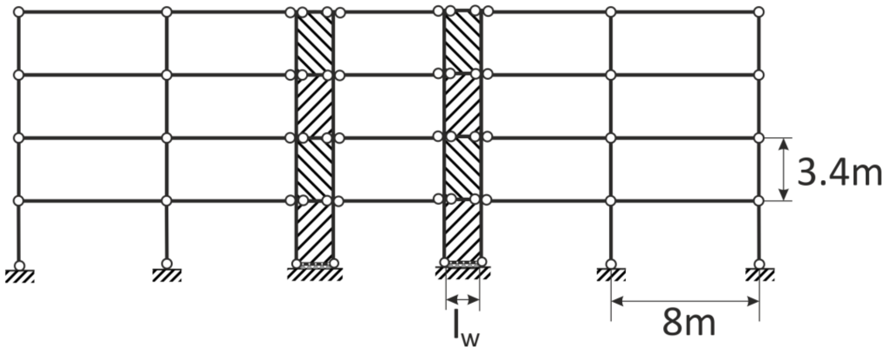


FIGURE 4-3: SECTION OF THE STRUCTURE (TWO INNER SHEAR WALLS)

4.1.1 STEEL PROFILES AND MATERIALS

The materials used and its properties and partial factors (for persistent and transient situation) are:

	E_{cm}	ν	f_{ck}	γ_C	f_{cd}
Concrete	35000 MPa	0.2	40 MPa	1.5	26.66 MPa
	E		f_{yk}	γ_{M0}	f_{yd}
Steel S235	210 000 MPa	0.3	235 MPa	1.0	235 MPa
Steel S355	210 000 MPa	0.3	355 MPa	1.0	355 MPa
			f_{syk}	γ_S	f_{syd}
Reinforcement steel B500B	210 000 MPa	0.3	500 MPa	1.15	435 MPa

TABLE 4-1: PROPERTIES OF THE MATERIALS USED IN THE CASE STUDY

Table 16 shows the steel profiles chosen for each part of the structure:

Element	Profile	Steel
Horizontal beams (gravity structure)	IPE 500	S235
Horizontal beams (boundary system)	HEB 200	S235
Columns 1st storey (boundary system)	HEB 200	**S355**
Columns 2nd storey (boundary system)	HEB 200	S235
Columns 3rd storey (boundary system)	HEB 160	S235
Columns 4th storey (boundary system)	HEB 100	S235
Columns 1st and 2nd storey (gravity structure)	HEB 220	S235
Columns 3rd and 4th storey (gravity structure)	HEB 300	S235

TABLE 4-2: STEEL PROFILES USED IN EACH ELEMENT OF THE STRUCTURAL SYSTEM

4.1.2 DIMENSIONS OF THE WALLS

The dimensions of the walls (in mm) are, therefore:

Floor	Wall	l_w	$b_w=b_{w0}$	h_w	l_c	l_b
4	inner	2000	200	3300	300	325
3	inner	1940	200	3300	300	325
2	inner	1900	200	3300	300	325
1	inner	1900	200	3200	300	325
4	outer	2000	200	3300	300	325
3	outer	1940	200	3300	300	325
2	outer	1900	200	3300	300	325
1	outer	1900	200	3200	300	325

TABLE 4-3: DIMENSIONS OF THE WALLS

A general illustrative image of one wall and its boundary elements is shown in the following figure:

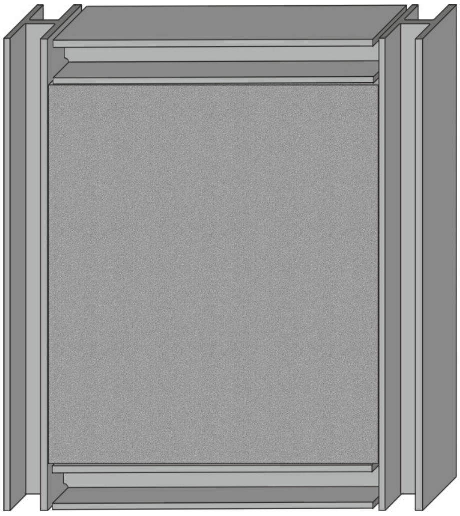


FIGURE 4-4: CONCRETE WALL AND BOUNDARY ELEMENTS

4.1.3 DETAILS OF THE WALLS

As explained in the previous chapter, the composite wall is formed by a steel frame with reinforced concrete infill. The details for each type of reinforcement used inside the concrete wall are summarized in the following figures and tables, and also the headed stud connectors which ensure the bonding between the concrete and the steel profiles.

4.1.3.1 Bending and vertical shear reinforcement ($A_{sv,b}$ and $A_{sv,i}$)

The vertical reinforcement of the wall is divided in two different types. The bars inside the boundary area of the wall ($A_{sv,b}$) which are mainly destined to resist the overturning moment, and the bars situated outside the boundary area ($A_{sv,i}$), which contribute both to bending moment and shear resistance.

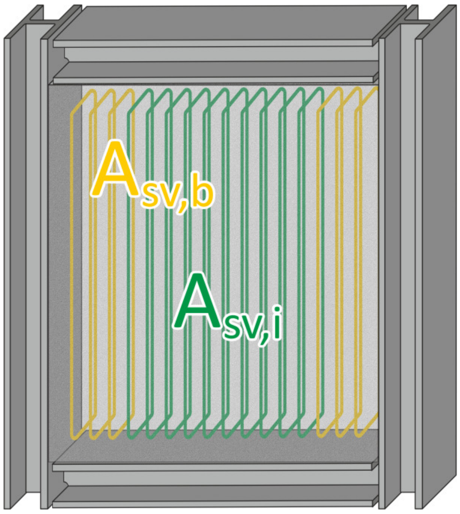


FIGURE 4-5: BENDING AND VERTICAL SHEAR REINFORCEMENT OF THE WALL

The details of the bending reinforcement are:

Floor	Wall	$A_{sv,b}$	$Ø_{sv,b}$	n	$n_{v,b}$	$s_{v,b}$
4	inner	314	10	2	2	280.0
3	inner	471	10	2	3	140.0
2	inner	2011	16	2	6	54.8
1	inner	2513	20	2	4	90.0
4	outer	314	10	2	2	280.0
3	outer	628	10	2	4	93.3
2	outer	2413	16	2	6	54.8
1	outer	3142	20	2	5	67.5
units		**mm^2**	**mm**			**mm**

TABLE 4-4: BENDING REINFORCEMENT

Where $Ø_{sv,b}$ is the diameter of the reinforcement bars, n is the number of bars in each section (number of bars in parallel), $n_{v,b}$ is the number of bar sets (each one containing n bars) used in the design and $s_{v,b}$ its spacing.

The details of the vertical shear reinforcement are:

Floor	Wall	$A_{sv,i}$	$Ø_{sv,i}$	n	$n_{v,i}$	$s_{v,i}$	$s_{v,i,b}$
4	inner	402	8	2	4	400	85.0
3	inner	603	8	2	6	220	105.0
2	inner	1106	8	2	11	120	38.0
1	inner	4423	16	2	11	110	90.0
4	outer	402	8	2	4	400	85.0
3	outer	905	8	2	11	115	80.0
2	outer	2941	12	2	13	100	38.0
1	outer	6434	16	2	16	75	77.5
units		**mm^2**	**mm**	**-**	**-**	**mm**	**mm**

TABLE 4-5: VERTICAL SHEAR REINFORCEMENT

Where $s_{v,i,b}$ is the distance between the centre of the last bar in the boundary area (the last from $A_{sv,b}$) and the first bar of the $A_{sv,i}$ reinforcement.

4.1.3.2 Horizontal shear reinforcement

The horizontal shear reinforcement of the wall (A_{sh}) is set to hold the horizontal forces.

FIGURE 4-6: HORIZONTAL SHEAR REINFORCEMENT OF THE WALL

The details of the reinforcement used in the analysed walls:

Floor	Wall	A_{sh}	$Ø_{sh}$	n	n_{sh}	s_h
4	inner	1257	10	2	8	400
3	inner	1257	10	2	8	400
2	inner	1414	10	2	9	400
1	inner	2513	10	2	16	200
4	outer	1257	10	2	8	400
3	outer	1257	10	2	8	400
2	outer	2042	10	2	13	250
1	outer	3299	10	2	21	150
units		**mm^2**	**mm**	**-**	**-**	**mm**

TABLE 4-6: HORIZONTAL SHEAR REINFORCEMENT

4.1.3.3 Confining reinforcement in boundary zones of the wall

The necessary confining reinforcement of the two boundary zones of the wall is named A_{swh}:

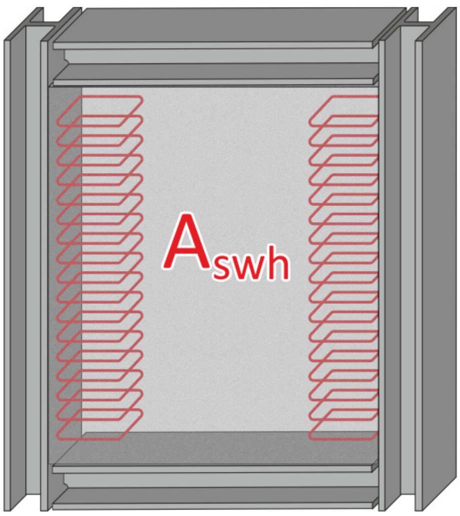


FIGURE 4-7: CONFINING REINFORCEMENT OF THE BOUNDARY ZONES OF THE WALL

Floor	Wall	A_{swh}	$Ø_{swh}$	n	n_{swh}	s_{wh}
4	inner	5027	10	2	32	100
3	inner	5027	10	2	32	100
2	inner	5027	10	2	32	100
1	inner	5027	10	2	32	100
4	outer	5027	10	2	32	100
3	outer	5027	10	2	32	100
2	outer	5027	10	2	32	100
1	outer	5027	10	2	32	100
units		**mm²**	**mm**	**-**	**-**	**mm**

TABLE 4-7:REINFORCEMENT IN THE COFINING ZONE OF THE WALL

4.1.3.4 Reinforcement needed for the horizontal shear connection

As explained in the previous chapter, the stud connectors placed on the top and bottom of the wall need of an additional reinforcement in order to guarantee their correct performance:

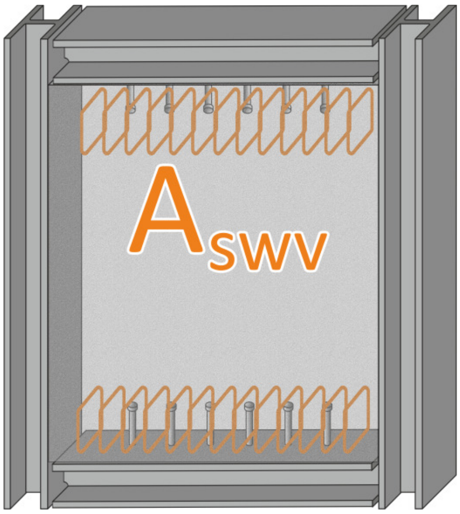


FIGURE 4-8: REINFORCEMENT NEEDED BY THE STUD CONNECTORS RESISTING HORIZONTAL SHEAR

Floor	Wall	A_{swv}	$Ø_{swv}$	n	n_{swv}	s_{wv}
4	inner	3142	10	2	20	90
3	inner	3142	10	2	20	90
2	inner	3142	10	2	20	90
1	inner	3142	10	2	20	90
4	outer	3142	10	2	20	90
3	outer	3770	10	2	24	90
2	outer	3142	10	2	20	90
1	outer	3142	10	2	20	90
units		**mm²**	**mm**	**-**	**-**	**mm**

TABLE 4-8: REINFORCEMENT NEEDED BY THE HORIZONTAL SHEAR CONNECTORS

4.1.3.5 Shear connection between steel and concrete

Two different rows of headed stud connectors are placed in the wall to ensure the adequate behaviour of the wall. The stud connectors placed in both lateral sides hold the vertical shear V_{Ed}*, while the ones placed in the top and the bottom of the wall are carrying the horizontal shear forces V_{Ed}.

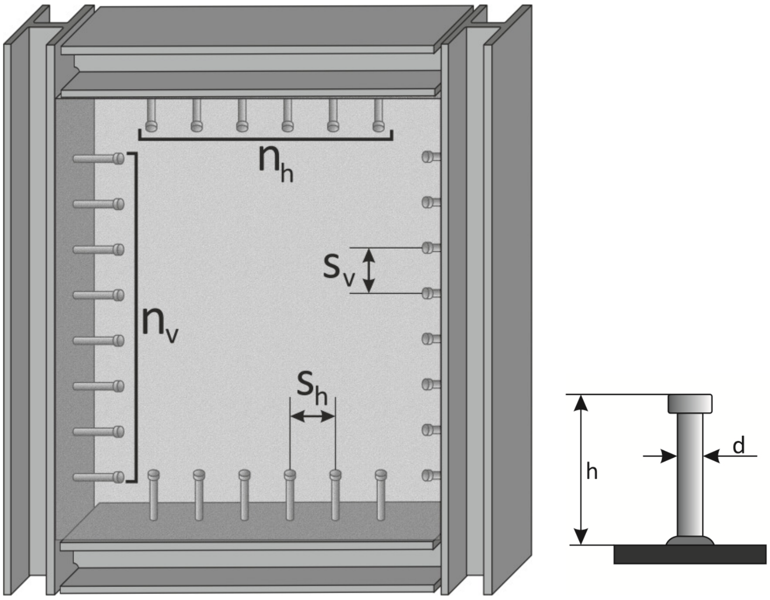


FIGURE 4-9: HEADED STUD CONNECTORS

Floor	Wall	d_v	n_v	h_v	s_v	d_h	n_h	h_h	s_h
4	inner	19	17	225	143	19	10	225	125
3	inner	19	17	225	139	19	10	225	120
2	inner	19	17	225	136	19	10	225	117
1	inner	19	17	225	136	19	10	225	117
4	outer	19	18	225	143	19	11	225	125
3	outer	19	18	225	139	19	11	225	120
2	outer	19	18	225	136	19	11	225	117
1	outer	19	18	225	119	19	11	225	117
units		**mm**	**-**	**mm**	**mm**	**mm**	**-**	**mm**	**mm**

TABLE 4-9: HEADED STUD CONNECTORS

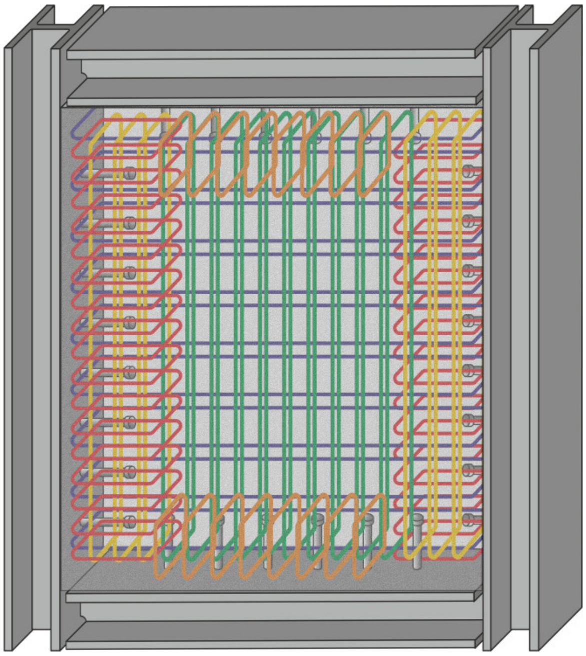
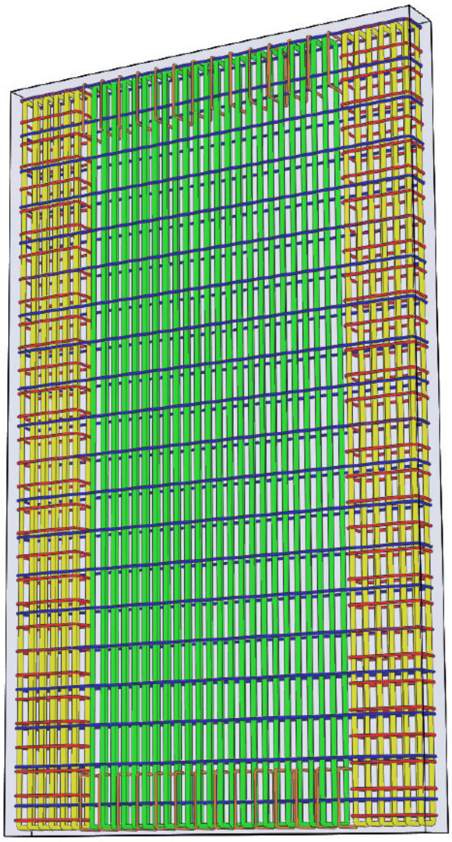


FIGURE 4-10: WALL WITH REINFORCEMENT AND HEADED STUD CONNECTORS

Also a 3D model has been developed (for one wall) with SolidWorks to check for interferences between the different reinforcement bars.



4.2 CALCULATION OF THE GRAVITY AND SEISMIC LOADS

The basic information used for the determination of the gravity and seismic loads of the building is given by the specifications of the project. The structural elements will have to carry two vertical linear loads:

$$G_k = 6.5\ {}^{kN}/_{m^2} \qquad Q_k = 3.0\ {}^{kN}/_{m^2} \tag{4.1}$$

Where the resulting vertical load is calculated by the following combination, according to the load combinations explained in 3.1.3.5:

$$G_k + \psi_2 \cdot Q_k = 6.5 + 0.3 \cdot 3.0 = 7.4\ {}^{kN}/_{m^2} \tag{4.2}$$

This leads to a mass per storey of:

$$m_{total} = \left(\frac{G_k + \psi_2 \cdot Q_k}{g}\right) \cdot A_{total} = \frac{7.4 \cdot 160}{9.81} = 1206{,}93t \tag{4.3}$$

In order to use it for the determination of the design spectrum, it is necessary to have a reference value for the behaviour factor. The approximated behaviour factor for the composite system DCM is:

$$q = 3 \cdot \frac{\alpha_u}{\alpha_1} = 3 \cdot 1.1 = 3.3 \tag{4.4}$$

The natural period of the building is approximated with the expression in Eurocode 8:

$$T_1 = C_t \cdot H^{3/4} = 0.050 \cdot 13.6^{3/4} = 0.3541\ s \tag{4.5}$$

Regarding to the calculation of the spectrum, for type spectra 1, ground type C and a value a_g=0.25g (α=0.25):

S	T_B	T_C	T_D
1.15	0.20	0.6	2.0

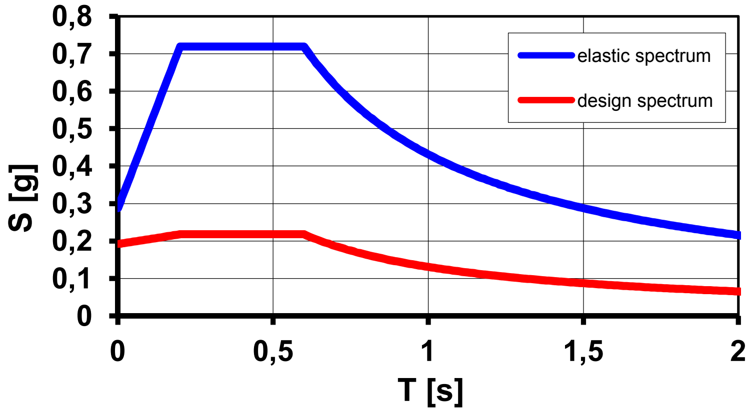
TABLE 4-10: PARAMETERS RELATED TO THE SPECTRUM

Finally, as $T_B < T_1 < T_C$, and with $\eta = 1$:

$$S_e(T) = a_g \cdot S \cdot \eta \cdot 2.5 = 0.25 \cdot 9.81 \cdot 1.15 \cdot 2.5 = 7.05\ m/s^2 \tag{4.6}$$

$$S_d(T) = \frac{S_e(T)}{q} = 2.14\ m/s^2 \tag{4.7}$$

The elastic and the design spectrum have the following shape:



The lateral force method of analysis is used, and therefore the value of the total lateral force, and also the forces to be carried by each storey, are calculated on basis of the total mass of the building and the expressions detailed in the explanation of the method. This leads to:

Floor	F_b	z_i	m_i	$z_i \cdot m_i$	F_i	total shear
4	8767.88	13.6	1206.93	3507.15	3507.15	3507.15
3		10.2	1206.93	2630.36	2630.36	6137.52
2		6.8	1206.93	1753.58	1753.58	7891.09
1		3.4	1206.93	876.79	876.79	8767.88
units	**kN**	**m**	**t**	**m·t**	**kN**	**kN**

TABLE 4-11: TOTAL SHEAR FORCES

The shear forces for each floor have to be distributed to different walls. However, after that, the correction factor (δ) due to torsion effects is applied, with L_e=40m:

Floor	Wall	x	δ	walls/floor	V_{Ed}	total shear (storey)
4	inner	12	1.18	4	344.87	3507.15
3	inner	12	1.18	4	603.52	6137.52
2	inner	12	1.18	4	775.96	7891.09
1	inner	12	1.18	4	862.17	8767.88
4	outer	20	1.30	8	379.94	3507.15
3	outer	20	1.30	8	664.90	6137.52
2	outer	20	1.30	8	854.87	7891.09
1	outer	20	1.30	8	949.85	8767.88
units		**m**	**-**	**mm**	**kN**	**kN**

TABLE 4-12: SHEAR FORCES IN EACH WALL

On the other hand, the vertical forces over each composite wall system are calculated in basis of the area which corresponds to each wall, and adding the loads of the above storeys:

Floor	Wall	Area	vertical load	N_{Ed}
4	inner	64	7.4	473.60
3	inner	64	7.4	947.20
2	inner	64	7.4	1420.80
1	inner	64	7.4	1894.40
4	outer	32	7.4	236.80
3	outer	32	7.4	473.60
2	outer	32	7.4	710.40
1	outer	32	7.4	947.20
units		**m^2**	**kN/m^2**	**kN**

TABLE 4-13: VERTICAL GRAVITY LOADS

And the moments of each wall are calculated by multiplying the storey lateral force Fi of all the storeys above by its respective distance to the base of the current wall, and also by the corresponding delta factor, and dividing by the total number of walls in each floor (=12):

Floor	Wall	Area	δ	M_{Ed}
4	inner	3507.15	1.18	1172.56
3	inner	2630.36	1.18	3224.53
2	inner	1753.58	1.18	5862.79
1	inner	876.79	1.18	8794.18

Floor	Wall	Area	δ	M_{Ed}
4	outer	3507.15	1.30	1291.80
3	outer	2630.36	1.30	3552.45
2	outer	1753.58	1.30	6459.00
1	outer	876.79	1.30	9688.51
units		**kN**	**-**	**kN·m**

TABLE 4-14: MOMENTS IN EACH WALL

4.3 VERIFICATIONS WITHIN THE EUROCODES

4.3.1 VERIFICATIONS 0A

The basic measurements of the wall must fulfil:

Floor	Wall	$b_w=b_{w0}$	$h_s/20$	h_w/l_w	$b_w>\max\{0.15; h_s/20; 200\}$
4	inner	200	170	1.65	✓
3	inner	200	170	1.70	✓
2	inner	200	170	1.74	✓
1	inner	200	170	1.68	✓
4	outer	200	170	1.65	✓
3	outer	200	170	1.70	✓
2	outer	200	170	1.74	✓
1	outer	200	170	1.68	✓
units		**kN**	**mm**	**-**	**-**

TABLE 4-15: VERIFICATIONS RELATED TO THE MEASUREMENTS OF THE WALL

It is also important to notice that in no wall the ratio h_w/l_w is lower than 2, so none of the walls must be considered as “slender”. In that case, different provisions are given by the norm.

4.3.2 VERIFICATIONS 0B

Concrete class is C40/50, higher than C20/25 but not higher than C40/50, so. ✓

Steel reinforcement is class B, sufficient for DCM. ✓

All steel profiles are all at least class 2 (in fact, all of them are class 1) ✓:

Profile	Steel	Class
IPE 500	S235	1
HEB 200	S235	1
HEB 200	S355	1
HEB 160	S235	1
HEB 100	S235	1

TABLE 4-16: SECTION CLASS OF THE STEEL PROFILES

4.3.3 VERIFICATIONS 0C

The general value for cover in the walls is calculated for exposure class XC1 (Dry or permanently wet. Concrete inside buildings with low air humidity); structural class S4. The class is decreased because of having concrete class C40/50, so final class is S3. This leads to $c_{min,dur}$=10mm, and therefore:

$$c_{min} = max\{10;\ 10 + 0 - 0 - 0;\ 10\ mm\} = 10\ mm$$
$$c_{nom} = c_{min} +\ \Delta c_{dev} = 10 + 10 = 20\ mm \tag{4.8}$$

And the spacing of bars, calculated for the bar with highest diameter:

$$s_{min} = max\{1 \cdot 20; 20mm\} = 20\ mm \tag{4.9}$$

4.3.4 VERIFICATIONS 1A

Moment resistance of boundary elements:

Floor	Wall	profile	steel	A	N_{Rd}	$M_{Rd(boundary\ elements)}$
4	inner	HEB 100	S235	2600	611.00	1283.10
3	inner	HEB 160	S235	5430	1276.05	2679.71
2	inner	HEB 200	S235	7810	1835.35	3854.24
1	inner	HEB 200	S355	7810	2772.55	5822.36
4	outer	HEB 100	S235	2600	611.00	1283.10
3	outer	HEB 160	S235	5430	1276.05	2679.71
2	outer	HEB 200	S235	7810	1835.35	3854.24
1	outer	HEB 200	S355	7810	2772.55	5822.36
units		-	-	mm^2	kN	kN·m

TABLE 4-17: CALCULATION OF THE RESISTANT MOMENT OF THE TWO STEEL PROFILES

4.3.5 VERIFICATIONS 1B

In order to determine the resistant moment of the concrete wall, taking into account that it carries also a vertical axial load, the different cases explained in the previous chapter are considered. In this particular case:

Floor	Wall	$\rho_{v,b}$	$\rho_{sv,i}$	l_w	l_c	l_b	x_{pl}	region
4	inner	0.004833	0.001489	2000	300	325	83.37	boundary
3	inner	0.007249	0.002337	1940	300	325	115.98	boundary
2	inner	0.030932	0.004423	1900	300	325	224.87	boundary
1	inner	0.038665	0.017693	1900	300	325	384.11	outside
4	outer	0.004833	0.001489	2000	300	325	83.37	boundary
3	outer	0.009666	0.003506	1940	300	325	151.22	boundary
2	outer	0.037119	0.011762	1900	300	325	307.27	boundary
1	outer	0.048332	0.025735	1900	300	325	471.97	outside
units		-	-	mm	mm	mm	mm	-

TABLE 4-18: DETERMINATION OF THE X_{PL}

Where f_{sd}=435 N/mm^2; f_{cd}=26.67N/mm^2. After that, the different subcases are analysed to find the final resistant moment:

Floor	Wall	case	e	e_L	e_{L1}	e_{L2}	e_R	e_{R1}	e_{R2}	$M_{Rd(wall)}$
4	inner	1.1	96	-	-	-	-	-	-	432.96
3	inner	1.1	183	-	-	-	-	-	-	631.21
2	inner	1.1	197	-	-	-	-	-	-	2126.52
1	inner	2.1	-	152	-	-	-	59	77	3060.97
4	outer	1.1	48	-	-	-	-	-	-	426.26
3	outer	1.1	88	-	-	-	-	-	-	896.12
2	outer	1.2	-	-	59	18	32	-	-	2681.69
1	outer	2.2	139	-	-	-	-	-	-	4019.76
units		**-**	**mm**	**mm**	**mm**	**mm**	**mm**	**mm**	**mm**	**kN·m**

TABLE 4-19: CALCULATION OF THE RESISTANT MOMENT OF THE WALL

And the verification of the moment is:

Floor	Wall	$M_{Rd(boundary\ elements)}$	$M_{Rd(wall)}$	M_{Ed}	M_{Rd}	$M_{Ed} < M_{Rd}$
4	inner	1283.10	432.96	1172.56	1716.06	✓
3	inner	2679.71	631.21	3224.53	3310.91	✓
2	inner	3854.24	2126.52	5862.79	5980.76	✓
1	inner	5822.36	3060.97	8794.18	8883.32	✓
4	outer	1283.10	426.26	1291.80	1709.36	✓
3	outer	2679.71	896.12	3552.45	3575.83	✓
2	outer	3854.24	2681.69	6459.00	6535.92	✓
1	outer	5822.36	4019.76	9688.51	9842.11	✓
units		**kN·m**	**kN·m**	**kN·m**	**kN·m**	**-**

TABLE 4-20: MOMENT VERIFICATION

4.3.6 VERIFICATIONS 1C

Shear resistance of the walls, according to Eurocode 2, taking into account also that, in the critical region, only the 40% of the whole shear resistance is considered:

Floor	Wall	ν_d	s	z	$V_{Rd,s}$	$V_{Rd,max}$	V_{Ed}	$V_{Ed}<0.4\cdot\min\{V_{Rd,s}; V_{Rd,max}\}$
4	inner	0.04	400	1600	2010.62	5120.00	344.87	✓
3	inner	0.09	400	1552	1950.30	4966.40	603.52	✓
2	inner	0.14	400	1520	2148.85	4864.00	775.96	✓
1	inner	0.19	200	1520	7640.35	4864.00	862.17	✓
4	outer	0.02	400	1600	2010.62	5210.00	379.94	✓
3	outer	0.05	400	1552	1950.30	4966.40	664.90	✓
2	outer	0.07	250	1520	4966.23	4864.00	854.87	✓
1	outer	0.09	150	1520	13370.62	4864.00	949.85	✓
units		**-**	**mm**	**mm**	**kN**	**kN**	**kN**	**-**

TABLE 4-21: SHEAR VERIFICATION

With f_{ywd}=435 N/mm^2; cotθ=1; α_{cv}=1; v_1=0.6; f_{cd}=26.67 N/mm^2 and b_w=200mm.

4.3.7 VERIFICATIONS 1D

The maximum value of the normalised axial load is verificated as follows:

Floor	Wall	N_{Ed}	A_c	f_{cd}	ν_d	$\nu<0.40$
4	inner	473.60	400 000	26.67	0.04	✓
3	inner	947.20	388 000	26.67	0.09	✓
2	inner	1420.80	380 000	26.67	0.14	✓
1	inner	1894.40	380 000	26.67	0.19	✓
4	outer	236.80	400 000	26.67	0.02	✓
3	outer	473.60	388 000	26.67	0.05	✓
2	outer	710.40	380 000	26.67	0.07	✓
1	outer	947.20	380 000	26.67	0.09	✓
units		kN	mm^2	N/mm^2	-	-

TABLE 4-22: VERIFICATION OF THE NORMALISED AXIAL LOAD

4.3.8 VERIFICATIONS 1E

The verification of the ductility requirements in the boundary zones of the wall depends on the location of the elastic neutral axis of the wall:

Floor	Wall	ν_d	ω_v	x_u	$x_u \leq l_c$	μ_ϕ	α	ω_{wd}
4	inner	0.04	0.04	230	yes	3.64	-0.0897	0.248
3	inner	0.09	0.06	405	no	17.20	0.3136	0.248
2	inner	0.14	0.22	912	no	5.16	0.4590	0.248
1	inner	0.19	0.41	1501	no	1.42	0.4222	0.256
4	outer	0.02	0.04	171	yes	5.06	-0.0897	0.248
3	outer	0.05	0.09	353	no	23.07	0.4122	0.248
2	outer	0.07	0.33	1022	no	5.28	0.4814	0.248
1	outer	0.09	0.55	1619	no	0.96	0.4618	0.256
units		-	mm	mm	-	-	-	-

TABLE 4-23: LOCAL DUCTILITY VERIFICATIONS

And the requirements regarding to diameter and spacing of the reinforcement bars:

Floor	Wall	$Ø_{swh}$	$Ø_{swh} \geq 8$	$Ø_{swh} \leq b_w/8=25$	s_{wh}	$s_{wh}<250$	$s_{wh}<25\cdot Ø_{swh}$
4	inner	10	✓	✓	10	✓	✓
3	inner	10	✓	✓	10	✓	✓
2	inner	10	✓	✓	10	✓	✓
1	inner	10	✓	✓	10	✓	✓
4	outer	10	✓	✓	10	✓	✓
3	outer	10	✓	✓	10	✓	✓
2	outer	10	✓	✓	10	✓	✓
1	outer	10	✓	✓	10	✓	✓
units		mm	-	-	mm	-	-

TABLE 4-24: CONDITIONS ON THE REINFORCEMENT OF THE BOUNDARY ZONES OF THE WALL

4.3.9 VERIFICATIONS 2A

Regarding to the shear connectors:

Floor	Wall	V_{Ed}	P_{Rd1}	P_{Rd2}	$P_{Rd,L}$	P_{Rd}	$n_h \cdot P_{Rd} > V_{Ed}$
4	inner	344.87	90.73	99.41	113.76	90.73	✓
3	inner	603.52	90.73	99.41	112.38	90.73	✓
2	inner	775.96	90.73	99.41	111.43	90.73	✓
1	inner	862.17	90.73	99.41	111.43	90.73	✓
4	outer	379.94	90.73	99.41	113.76	90.73	✓
3	outer	664.90	90.73	99.41	112.38	90.73	✓
2	outer	854.87	90.73	99.41	111.43	90.73	✓
1	outer	949.85	90.73	99.41	111.43	90.73	✓
units		kN	kN	kN	kN	kN	-

TABLE 4-25: SHEAR CONNECTORS IN THE HORIZONTAL DIRECTION

Floor	Wall	V_{Ed}*	P_{Rd1}*	P_{Rd2}*	$P_{Rd,L}$*	P_{Rd}*	$n_v \cdot P_{Rd} > V_{Ed}$
4	inner	569.04	92.54	99.41	118.41	90.73	✓
3	inner	1026.61	92.54	99.41	117.34	90.73	✓
2	inner	1347.72	92.54	99.41	116.61	90.73	✓
1	inner	1452.08	92.54	99.41	116.61	90.73	✓
4	outer	626.90	92.54	99.41	118.41	90.73	✓
3	outer	1131.01	92.54	99.41	117.34	90.73	✓
2	outer	1484.77	92.54	99.41	116.61	90.73	✓
1	outer	1599.75	92.54	99.41	112.03	90.73	✓
units		kN	kN	kN	kN	kN	-

TABLE 4-26: SHEAR CONNECTORS IN THE VERTICAL DIRECTION

And the conditions to be fulfilled by the Annex C:

Floor	Wall	$\frac{a}{2} \leq s \leq a$	$e_v \geq 50$	$19 \leq d \leq 50$	$\frac{h}{d} \geq 4$	$110 \leq a \leq 440$	$\frac{s}{e_v} \leq 3$	$\beta \leq 23°$
4	inner	✓	✓	✓	✓	✓	✓	✓
3	inner	✓	✓	✓	✓	✓	✓	✓
2	inner	✓	✓	✓	✓	✓	✓	✓
1	inner	✓	✓	✓	✓	✓	✓	✓
4	outer	✓	✓	✓	✓	✓	✓	✓
3	outer	✓	✓	✓	✓	✓	✓	✓
2	outer	✓	✓	✓	✓	✓	✓	✓
1	outer	✓	✓	✓	✓	✓	✓	✓
4*	inner	✓	✓	✓	✓	✓	✓	✓
3*	inner	✓	✓	✓	✓	✓	✓	✓
2*	inner	✓	✓	✓	✓	✓	✓	✓
1*	inner	✓	✓	✓	✓	✓	✓	✓
4*	outer	✓	✓	✓	✓	✓	✓	✓
3*	outer	✓	✓	✓	✓	✓	✓	✓
2*	outer	✓	✓	✓	✓	✓	✓	✓
1*	outer	✓	✓	✓	✓	✓	✓	✓

TABLE 4-27: CONDITIONS NEEDED IN ANNEX C

4.4 DESCRIPTION OF THE DIFFERENT FEM MODELS

By means of a numerical model the following characteristics will be studied:

- The general behaviour of a hybrid system based on a steel frame and concrete infills.
- The particular need to study the behaviour and performance of the discrete connection between steel and concrete.
- The interest of studying the dynamic behaviour of the system, through transient dynamic analyses, where ductility plays an important role.
- The interest of including the different sources of nonlinearities:
  - Nonlinear steel frame
  - Nonlinear concrete
  - Nonlinear head stud connectors
  - The geometrical nonlinearity

Therefore, different models have been created to study and isolate the different nonlinearities, and to help making decisions and reaching useful conclusions for further research.

The FEA[8] software used in this case study is the ANSYS simulation software, in its mechanical version. A brief description of each one of the used elements will be provided in the next section.

4.4.1 ANSYS ELEMENTS USED IN THE NUMERICAL MODELS

The demand to model beams (corresponding to the steel frames), concrete walls and the connection between them, and also the need to run dynamic analysis leads to the combined use –depending on the model– of different elements: BEAM188, PLANE182, SOLID65, COMBIN39 and MASS21. All of them are capable of simulating large deflection, large strain and plasticity.

The BEAM188 element is used to model beams (boundary elements). Each element consists of two nodes, each one having six degrees of freedom, three translations (UX, UY and UZ) and three rotations (ROTX, ROTY and ROTZ).

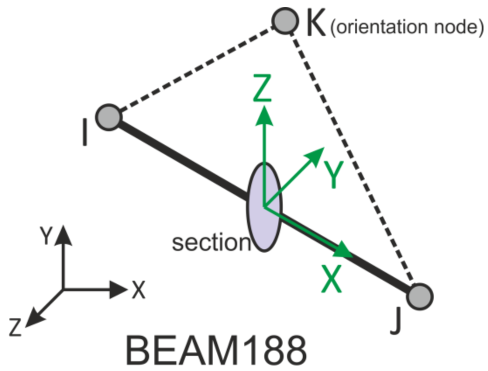


FIGURE 4-11: SCHEME OF THE BEAM188 ELEMENTS OF ANSYS

[8] Finite Element Analysis

A custom section can be selected in basis of different parameters, and in this case only I-shaped cross-sections will be used, to simulate standard HEB profiles. The parameters which define a generalized I-shaped section are shown in the following figure:

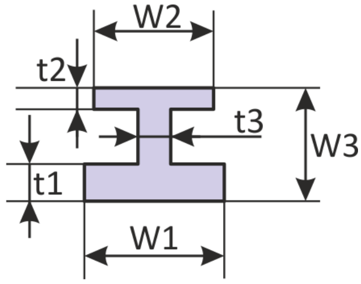


FIGURE 4-12: INPUT DATA FOR AN I-SHAPED PROFILE IN BEAM188 ELEMENTS

The PLANE182 element is a plane element (exists only in a two-dimensional plane) defined by 4 nodes with two degrees of freedom each (UX and UY). The element will be used with the *plane stress with thickness* option, as it will be used to model the concrete wall. This option corresponds to KEYOPT(3)=3, and a value for the thickness has to be provided as real constant.

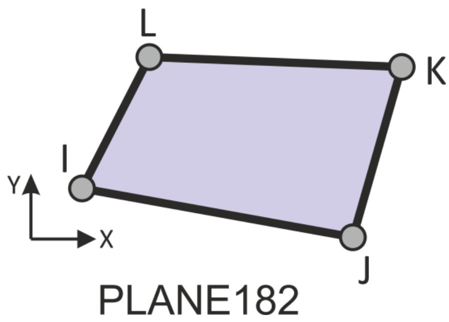


FIGURE 4-13: SCHEME OF THE PLANE182 ELEMENT IN ANSYS

The SOLID65 element is a particular version of a general solid element (such as SOLID45), specially designed to work with the concrete material model of ANSYS. It has eight nodes with three degrees of freedom each. It allows the modelling of concrete cracking –in three different planes– and crushing, and it is possible to set reinforcement in three different directions (defined by two angles ϕ and θ). However, this reinforcement is set as volume ratio, and it is homogenously distributed ("smeared") all over the element, with a perfect bonding between the two materials. The material of the reinforcement may also be nonlinear.

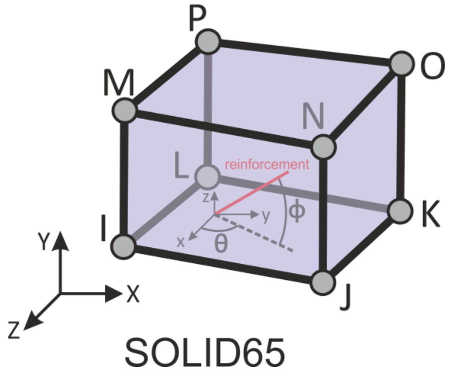


FIGURE 4-14: SCHEME OF THE SOLID65 ELEMENT IN ANSYS

The cracking/crushing capability allows cracks to open and close, but four parameters have to be given for this to work: the uniaxial tensile cracking stress, the uniaxial crushing stress and two shear transfer coefficients for open and closed cracks. This coefficients (β) must be between 0 (no shear transfer in the cracked plane) and 1 (full shear transfer even if cracked). Cracks may happen in one of the integration points, by following a 3D failure surface defined by the two failure stresses (tensile and compressive):

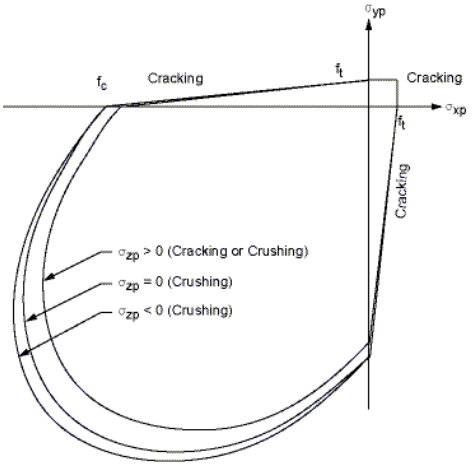


FIGURE 4-15: FAILURE SURFACES OF THE CONCRETE MATERIAL MODEL OF ANSYS

The COMBIN39 element is a nonlinear spring with multiple capabilities. In this project it will be used to simulate the headed stud connectors. The spring works in either 2D or 3D dimensions, but the force-deflection relation is always one-dimensional. A custom curve can be set, but the slope of the curve has to be always greater or equal to zero.

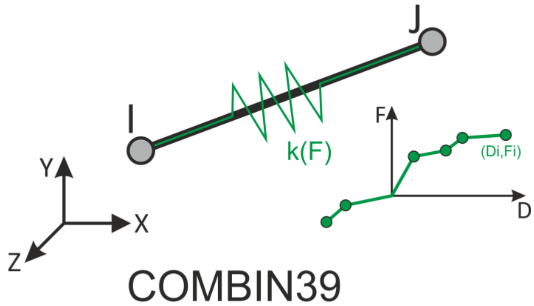


FIGURE 4-16: SCHEME OF THE COMBIN39 ELEMENTS OF ANSYS

The element also allows to set different configurations for the return path of the spring, for example if a hysteretic behaviour is desired.

Finally, the MASS21 element is used in the modal and dynamic analysis to represent the mass of the different storeys. It has only one node, and different numbers of degrees of freedom depending on the options. In this case, it will have only UX, UY and UZ, and no rotational inertias.

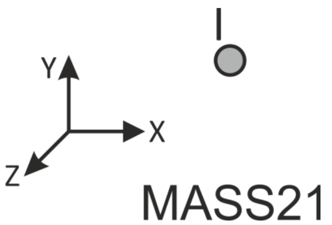


FIGURE 4-17: SCHEME OF THE MASS21 ELEMENTS OF ANSYS

4.4.2 NUMERICAL MODELS

As explained at the beginning of the chapter, different models have been developed for different purposes. In the following sections the different conclusions about these analyses will be explained. The modelling of the complete 3D building would require huge computational resources –this topic will be also discussed in some of the developed models also–. Instead of modelling the whole building, a set of four walls will be the reference model adopted, as this scheme is repeated 12 times in the building. Also a one-wall option has been used for checking purposes.

In order to disguise each one of the models used, they are summarized in the following table:

Model name	Walls	Steel (boundary elements)	Concrete model	Model of the shear connection	Mass
FREQ	4/4	Elastic	Elastic	Coupling DOF	Equivalent mass
COUPL4	4/4	Elastic	Elastic	Coupling DOF	No mass
LIN4	4/4	Elastic	Elastic	Linear springs	No mass
PLAST4	4/4	Elastic	Elastic	Plastic nonlinear springs	No mass
BILIN4	4/4	Elastic	Elastic	Bilinear nonlinear springs	No mass
PLAST4BKIN	4/4	Nonlinear (bilinear)	Elastic	Plastic nonlinear springs	No mass
CONCR	1/4	-	Nonlinear with smeared reinforcement	-	No mass
DYN1	1/4	Nonlinear (bilinear)	Elastic	Plastic nonlinear springs	Equivalent mass
DYN4	4/4	Nonlinear (bilinear)	Elastic	Plastic nonlinear springs	Equivalent mass
DYN4CONCR	4/4	Nonlinear (bilinear)	Nonlinear with smeared reinforcement with two different ratios	Plastic nonlinear springs	Equivalent mass

TABLE 4-28: DIFFERENT FE MODELS USED IN THE CASE STUDIES

The characteristics of the models will be discussed and explained in the corresponding analyses where each model is used.

4.4.2.1 Natural frequency of the system

One of the first interesting verifications to make is to check if the approximate value of the natural frequency of the structure calculated by following the Eurocode 8 procedures is near the actual real value. It is also interesting to run a modal analysis of the system in order to know which are the first modes of vibration and their frequencies. This has been done with a simplified model of the 4-wall system, the FREQ model. This model uses the real geometry, and elastic properties for materials. The equivalent mass at the top of each wall is calculated on the basis of the assumption that each behaves as a solid block. Because of that, we can calculate the mass of each of our four degrees of freedom (one for each floor) as the total mass of a storey divided by the number of walls acting in parallel. There are 12 walls in each direction, which leads to a value of 100.5775 tons. This mass will be distributed along the length of the beam.

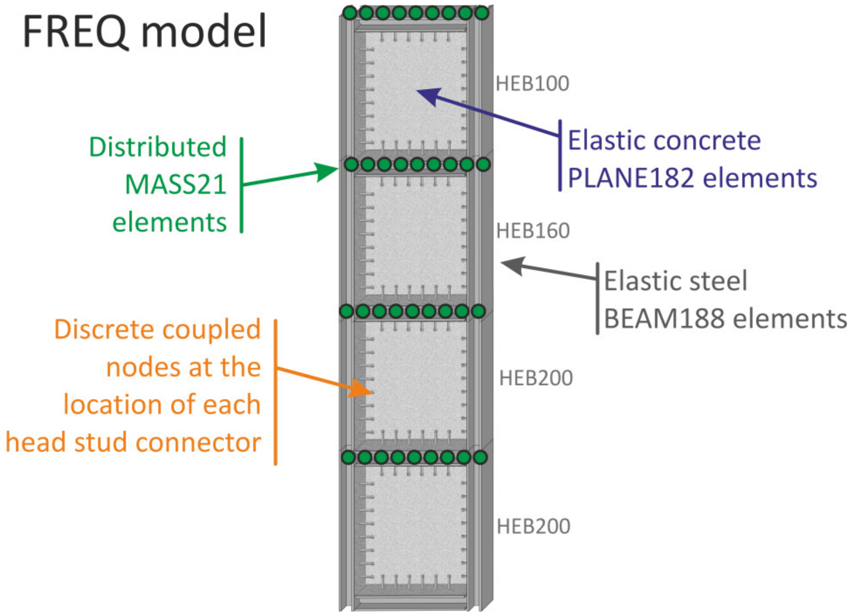
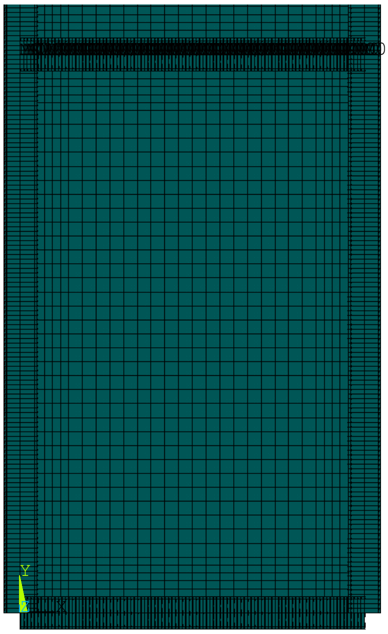


FIGURE 4-18: SCHEME OF THE FREQ MODEL

A mapped mesh has been used for the four areas. A detail view on the mesh used for the first wall:



4-19: DETAIL ON THE MESH USED IN THE FIRST WALL

With this model, the results for the first 5 modes of the structure are:

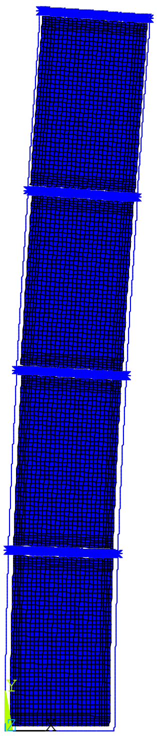
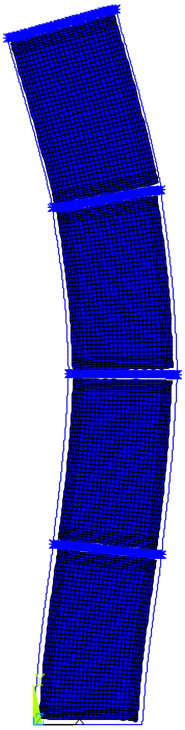
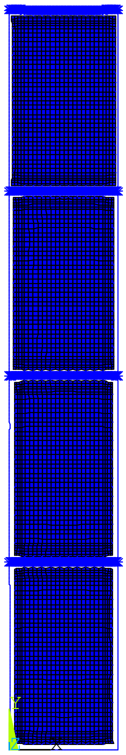
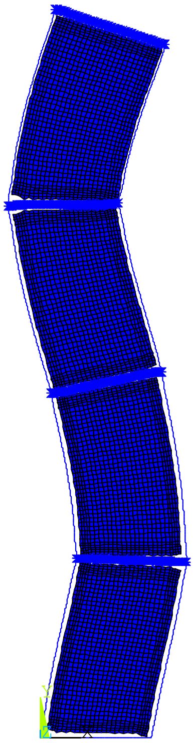
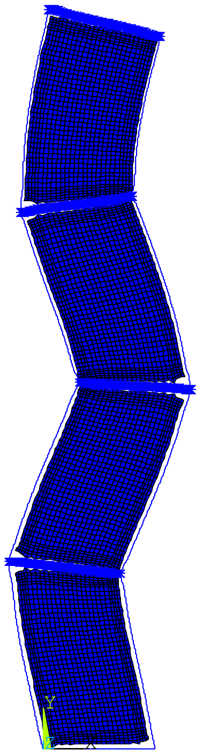
Mode	1	2	3	4	5
Frequency (Hz)	1.1727	6.3961	11.613	15.036	23.403
Period (s)	0.85273301	0.15634527	0.08611039	0.06650705	0.04272956
Omega (rad/s)	7.36829141	40.1878815	72.966631	94.4739743	147.045386
Shape					

TABLE 4-29: NATURAL MODE SHAPES OF THE ANALYSED SYSTEM

The system has 12 walls in each direction, so it is necessary to transform this value. As these walls are the only lateral resisting system of the structure, they provide the whole horizontal stiffness. When multiplying the stiffness matrix by 12, but also the mass. So as ω^2 are the eigenvalues of the matrix $[M]^{-1} \cdot [K]$, the result is directly an approximation of the complete natural period of the building.

The natural period of the building, according to Eurocode 8 is:

$$T_1 = C_t \cdot H^{3/4} = 0.050 \cdot 13.6^{3/4} = 0.3541\ s \tag{4.10}$$

4.4.2.2. Analysis of the alternatives to model the shear connection

Head stud shear connectors are one of the best solutions available, but they are not a perfect bonding between steel and concrete in the shear direction. Different studies and laboratory experiments have been made within this field, with a general accepted conclusion that head stud connectors behave linearly until a certain level of deflection s_e (sometimes called "slip", instead of deflection), where they start yielding. Failure is considered to be after 6mm deflection. In the case of the studs used in the INNO-HYCO project, which have a diameter of 19mm, the following curve has been taken into consideration [29]:

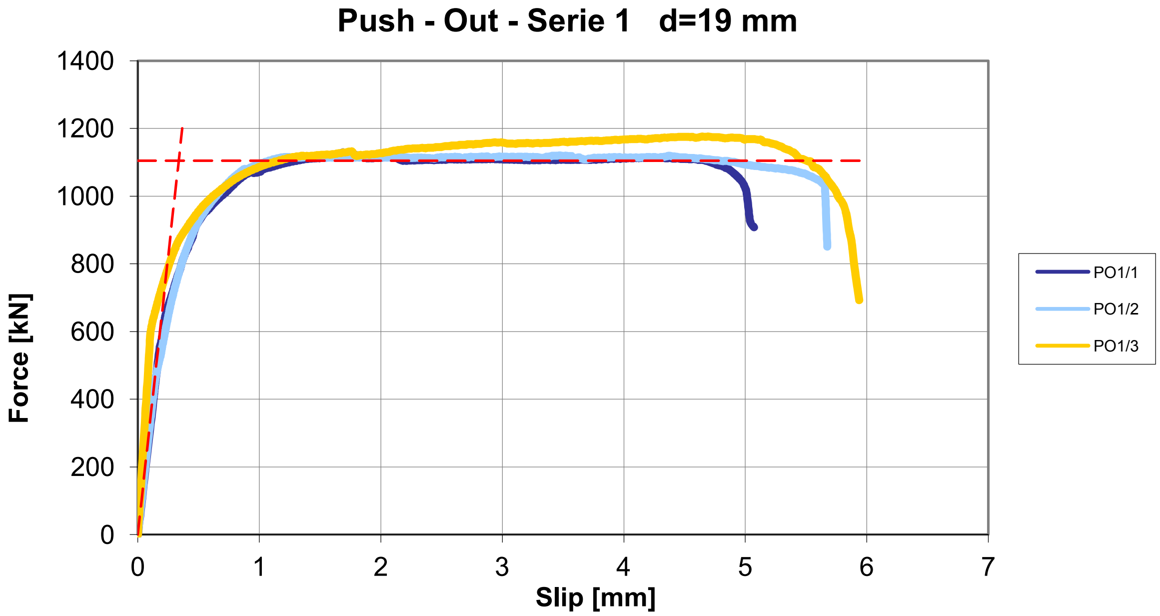


FIGURE 4-20: LOAD-SLIP CURVE FOR A 19MM DIAMETER HEADED STUD SHEAR CONNECTOR [29]

The two lines in red show the approximate point of slip which will be considered as the end of the elastic range. As the head stud connectors from the graphic are not exactly the ones used within this project (they have different resistance), it is necessary to calculate a correspondent value of slip to be used in the analyses. The value of the slope of the elastic part will be used to calculate the intersection between the elastic line and the real resistance of the head stud:

$$s_e(curve) = 0.34mm \qquad P_{Rd}(curve) = 1105000N \qquad slope = {}^{P_e}\!/_{s_e}$$

$$s_e(case\ study) = \frac{P_{Rd}}{slope} = \frac{90730N}{slope} = 0.0279mm$$

In order to decide which is the best alternative to model this connectors, and also study the differences between the alternatives within the load range of the systems under study, four different alternatives have been considered.

In the first model, called COUPL4, the nodes located at the position of each head stud connector have been coupled. This means that the displacement solution will be the same for all nodes coupled in a particular direction. In this model both vertical and horizontal direction are coupled at the location of a head stud connector, between the nodes corresponding to the concrete wall (PLANE182 elements) and the nodes corresponding to the boundary elements (BEAM188 elements).

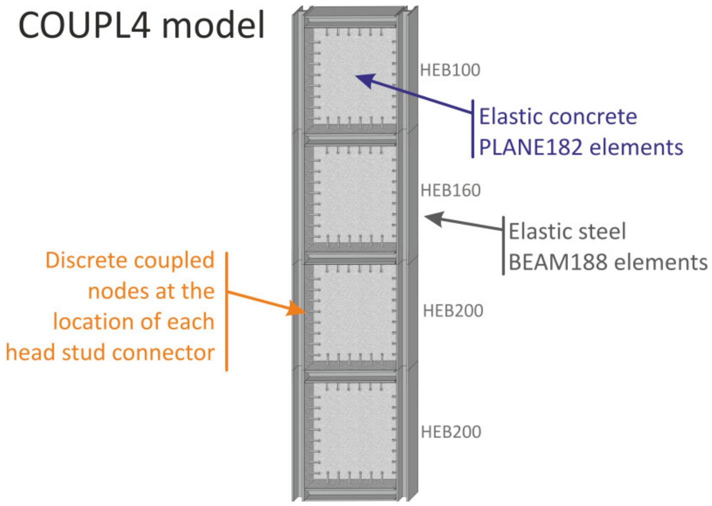


FIGURE 4-21: SCHEME OF THE COUPL4 MODEL

In the next three models the displacement in the direction of the shear transfer is ruled by a force-displacement or force-slip curve, by using a COMBIN39 spring element. By setting different curves, three different models are LIN4 (for a linear law), PLAST4 (for perfect plasticity) and BILIN4 (for a bilinear plasticity curve).

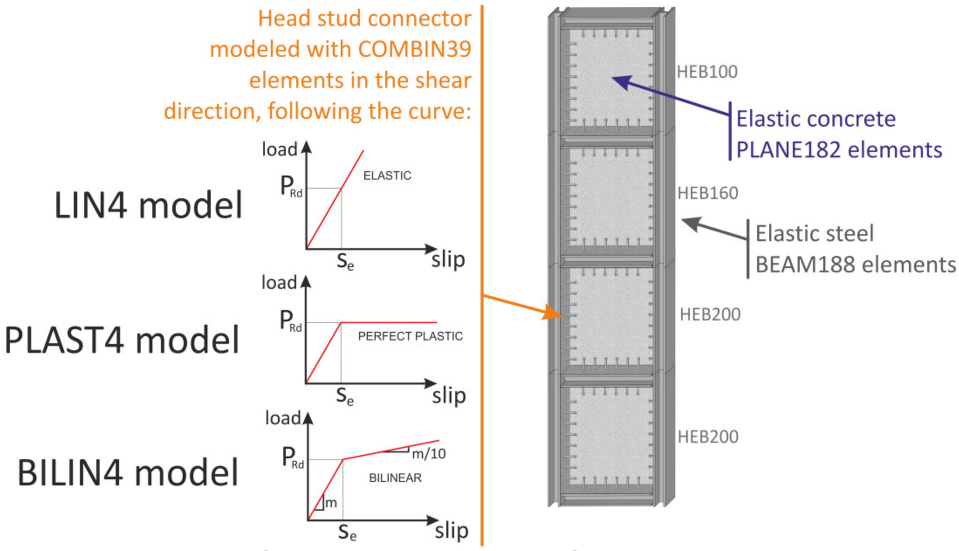


FIGURE 4-22: SCHEME OF THE LIN4, PLAST4 AND BILIN4 MODELS

In order to be sure on the differences that the use of different s_e values brings, each model has been tested with three values: 0.25, 0.5 and 1mm.

The model of the shear connection is schematized in the following figure:

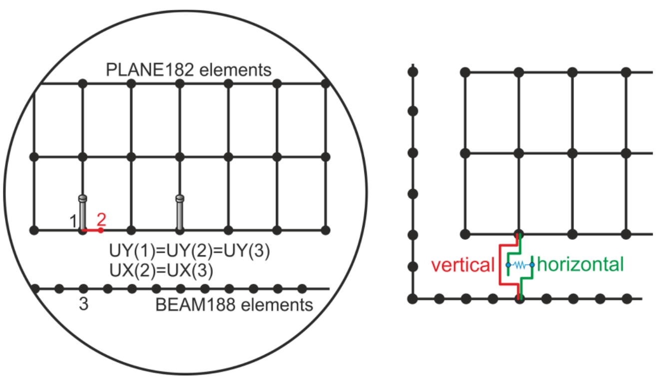


FIGURE 4-23: DETAIL OF THE MODELING OF THE SHEAR CONNECTORS

Where the node numbered as 1 is the node corresponding to the PLANE182 element (concrete wall), the node 3 is the one corresponding to a BEAM188 element (the boundary elements). The COMBIN39 spring is created between node 1 and a new node (2) created at an offset of 10mm from node 1.

In order to compare the accuracy of the different models, pushover analyses have been performed. Four horizontal forces have been set, one in each storey, following a pattern which is proportional to mass and height of each storey (following the fundamental period of vibration of the system). The load will be increased gradually. ANSYS uses in this static analysis the variable TIME to represent the amount of load, being TIME=0 the initial substep and TIME=1 the final substep (maximum value of load). It is important to notice that TIME=0.344868 corresponds to the design load (for a inner wall, in this case):

TIME (portion of load)	F_1	F_2	F_3	F_4	Total shear
0	0	0	0	0	0
0.1	100	75	50	25	250
...	...	...	...	...	...
0.344868	344.86	258.65	172.43	86.217	862.17
...	...	...	...	...	...
1	1000	750	500	250	2500
units	**kN**	**kN**	**kN**	**kN**	**kN**

TABLE 4-30: EXAMPLE OF THE MEANING OF THE TIME VARIABLE IN THE PUSHOVER ANALYSES

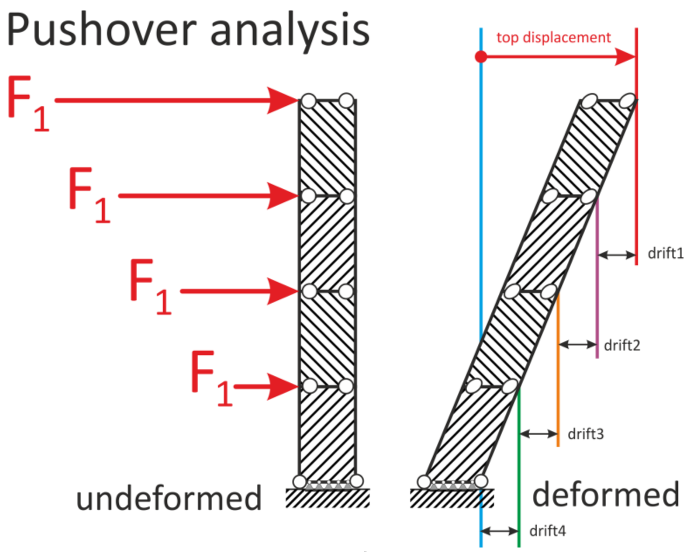


FIGURE 4-24: PUSHOVER ANALYSES (LOADS AND STUDIED VARIABLES)

The different variables which will be considered as answer will be the displacement at the top of the fourth wall, and the different interstorey drifts available.

If we look at the results of the top displacement value, a first conclusion is that all models remain within the elastic range when the load applied is under the design load. This could be considered obvious, but it is important to check that the assumptions made when predimensioning the shear connection are reasonably valid. After the elastic range of each connector, the solutions begin to diverge.

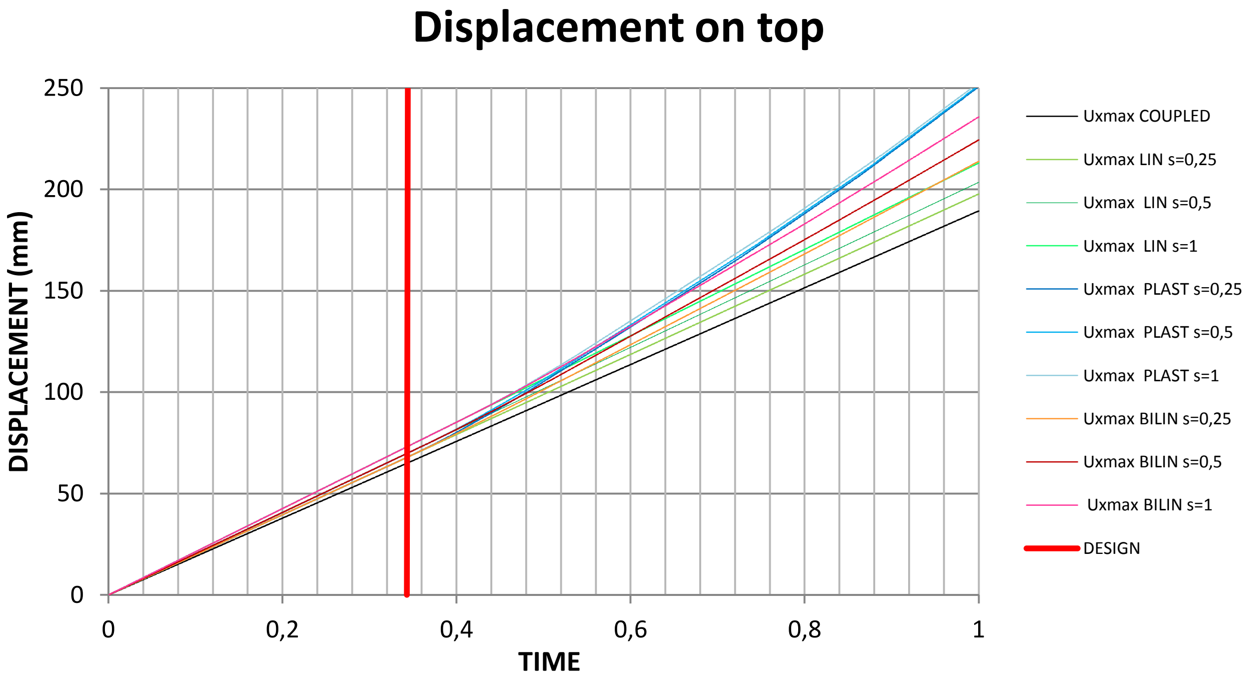


FIGURE 4-25: DISPLACEMENTS ON TOP OF THE DIFFERENT MODELS

The model with highest stiffness is the coupled one (COUPL, in black colour), and it is perfectly elastic of course. The LIN models (green colours) behave also linearly in all the range, but with different slopes depending on the s_e parameter used. The three PLAST models (blue colours) have an elastic limit, and after that, they start yielding with a constant value of force. This makes them the ones with highest values of displacement on top. Finally, the BILIN models have two different slopes: at the beginning, where still in the elastic range and after it they still behave linearly, but with much less stiffness.

It is easier to notice those changes of slope when comparing the curves for a constant value of s_e:

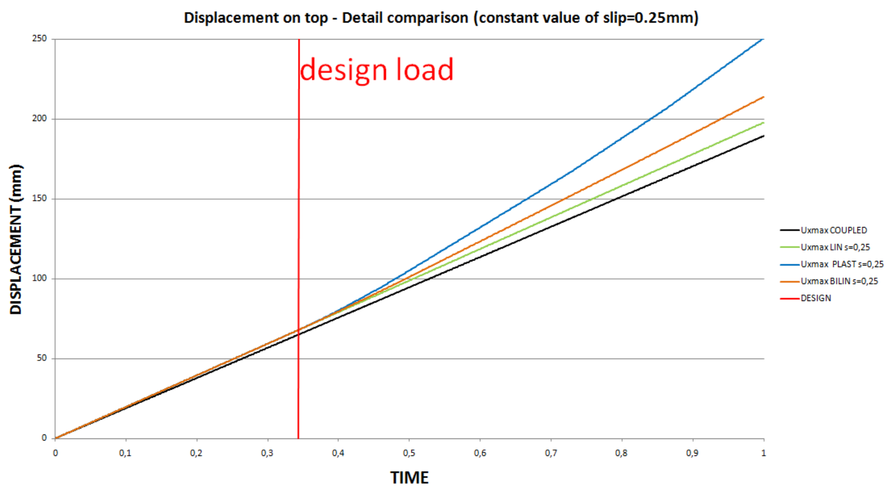


FIGURE 4-26: DISPLACEMENTS ON TOP OF THE DIFFERENT MODELS WITH CONSTANT VALUE OF SLIP

It is also interesting to check that the different values for s_e successfully change the yielding point of the head stud connectors. A more detailed graphic has been made, for the PLAST model, in order to show the loads where this happens:

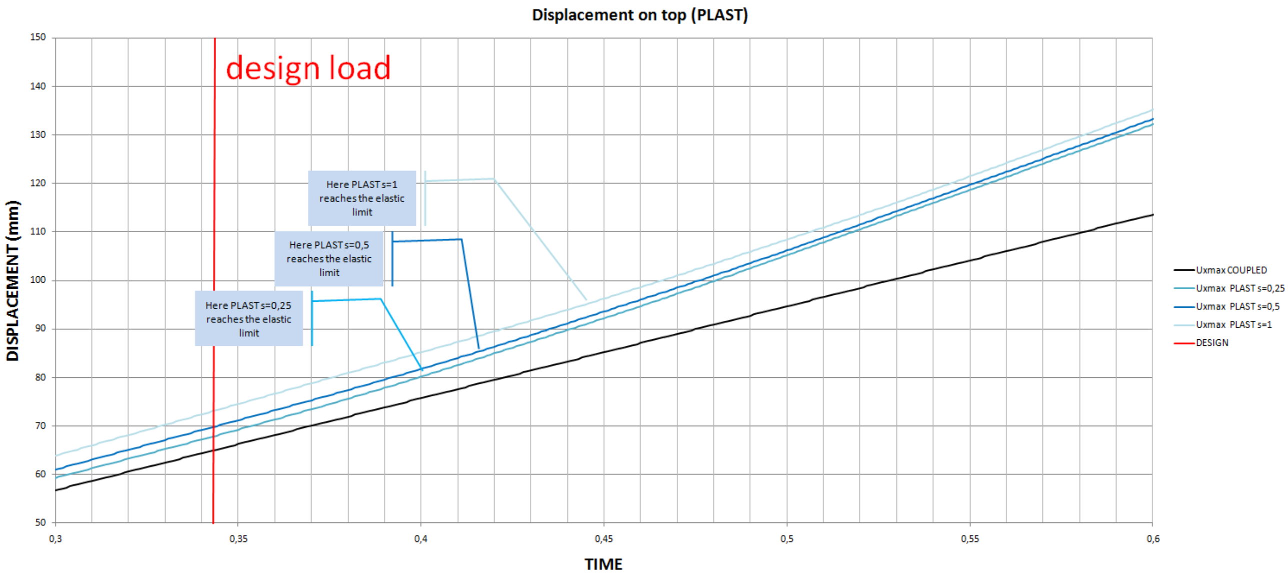


FIGURE 4-27: DETAIL ZOOM ON THE DIFFERENT POINTS WHERE THE STUD CONNECTORS START YIELDING

When looking at the interstorey drift, similar conclusions are made. The two most different interstorey drifts have been compared, and solutions also diverge in a similar way, but with more intensity in drift 4, as values are higher:

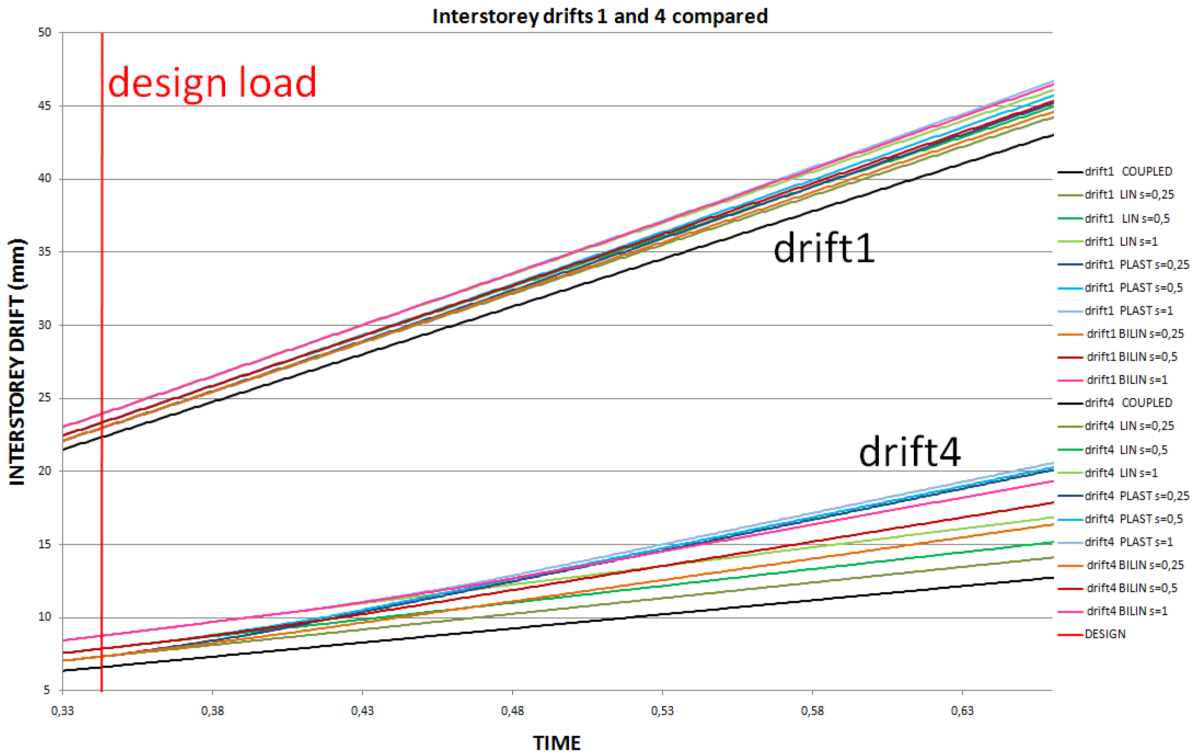


FIGURE 4-28: ZOOM ON THE INTERSTOREY DRIFTS FOR THE DIFFERENT MODELS

Also the load intensities where yielding of the columns is expected have been found:

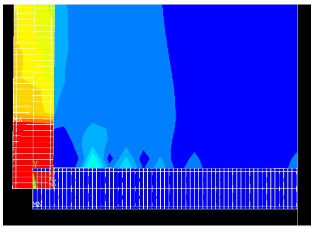
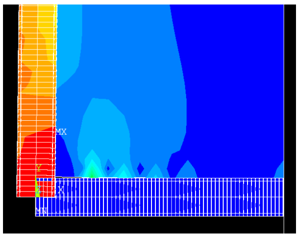
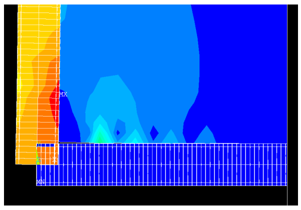
	COUPL4	LIN4	PLAST4
TIME	0.406	0.459	0.423
Figure	 Red zone is von Mises equivalent stress 355MPa	 Red zone is von Mises equivalent stress 355MPa	 Red zone is von Mises equivalent stress 355MPa

TABLE 4-31: YIELDING POINTS IN THE LEFT COLUMN

The difference between the LIN4 and the PLAST4 makes sense, as the linear springs can still carry a high portion of the load (contributing to lateral stiffness). However, the coupled model should theoretically yield with higher loads. The reason of this contradiction is that the coupled model forces the section rotate just at the location of the last head stud connector. The connection is perfect, and this forces the column to carry a bigger amount of load.

Another point of interest is to focus on the internal loads in each head stud connector (that is, in each COMBIN39 element). It is possible to read two variables from each spring: force and slip. If the force in a head stud is plotted against slip, the curve should look exactly as the curve that has been set. For example, for one of the head stud connectors in the bottom-central part of the first wall:

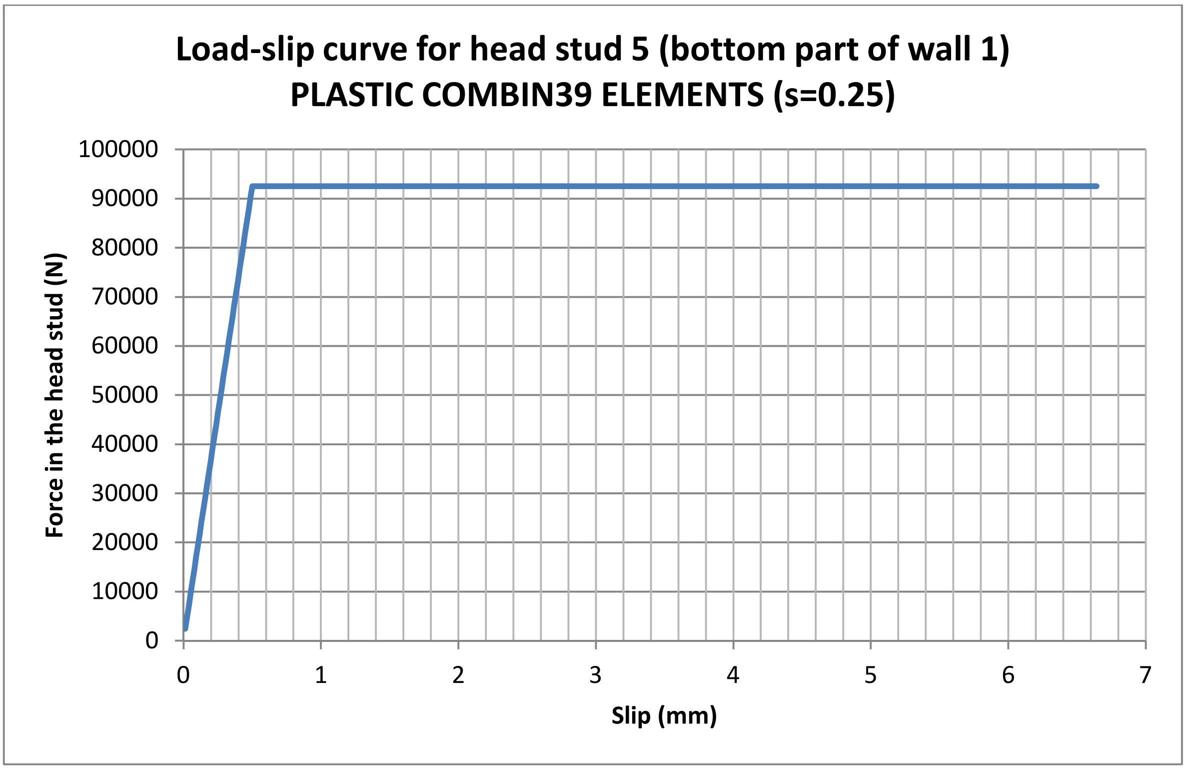


FIGURE 4-29: LOAD-SLIP CURVE FOR HEADED STUD 5

The value where the slope changes to horizontal is s_e.

It is possible to plot also the loads that each stud connector is carrying at each load intensity (TIME). This is interesting to analyse how the horizontal shear is transferred to the stud connectors and then to the beams. For a constant value of slip, the different force distributions will be analyses. At the bottom of the first wall, for linear springs (model LIN4; s_e=0.25mm):

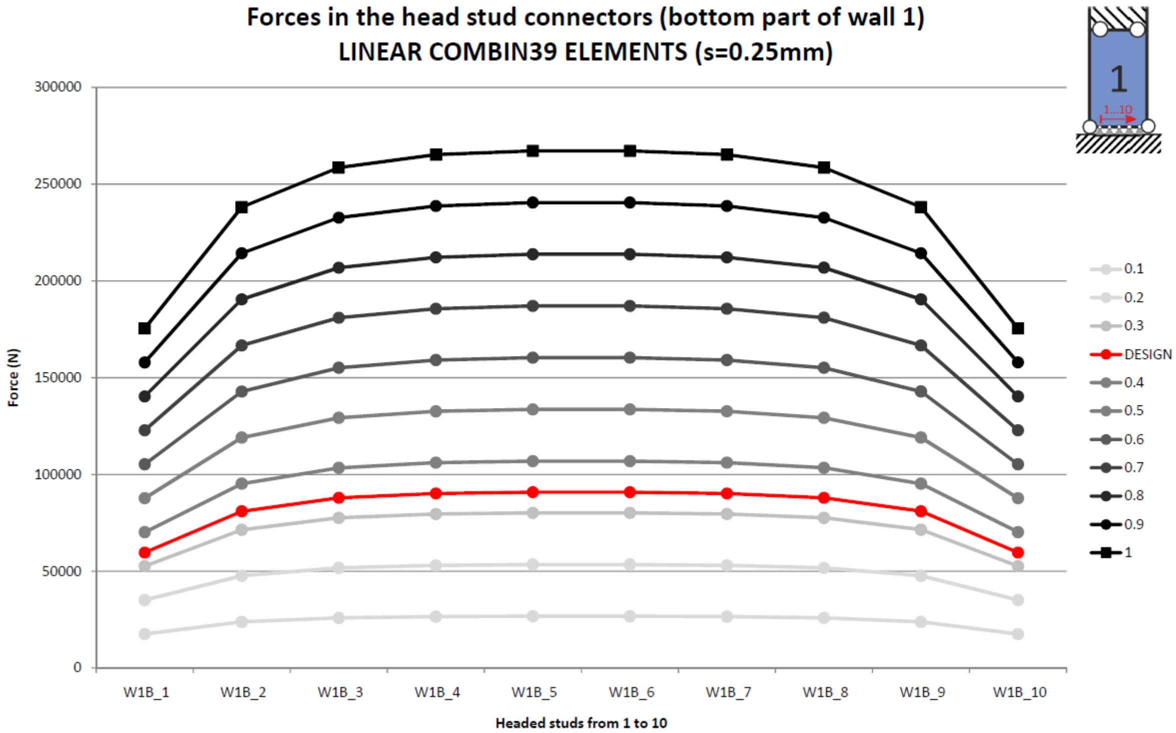


FIGURE 4-30: FORCES IN THE DIFFERENT SHEAR CONNECTORS AT THE BOTTOM OF WALL 1

It is important to notice that the connectors on both extremes carry a smaller proportion of the total horizontal load. In this case, with elastic springs, the curves only increase their size when increasing the load. However, if a nonlinear model is used, each stud connector has a maximal load, and after that point the load is transferred to the nearby headed studs.

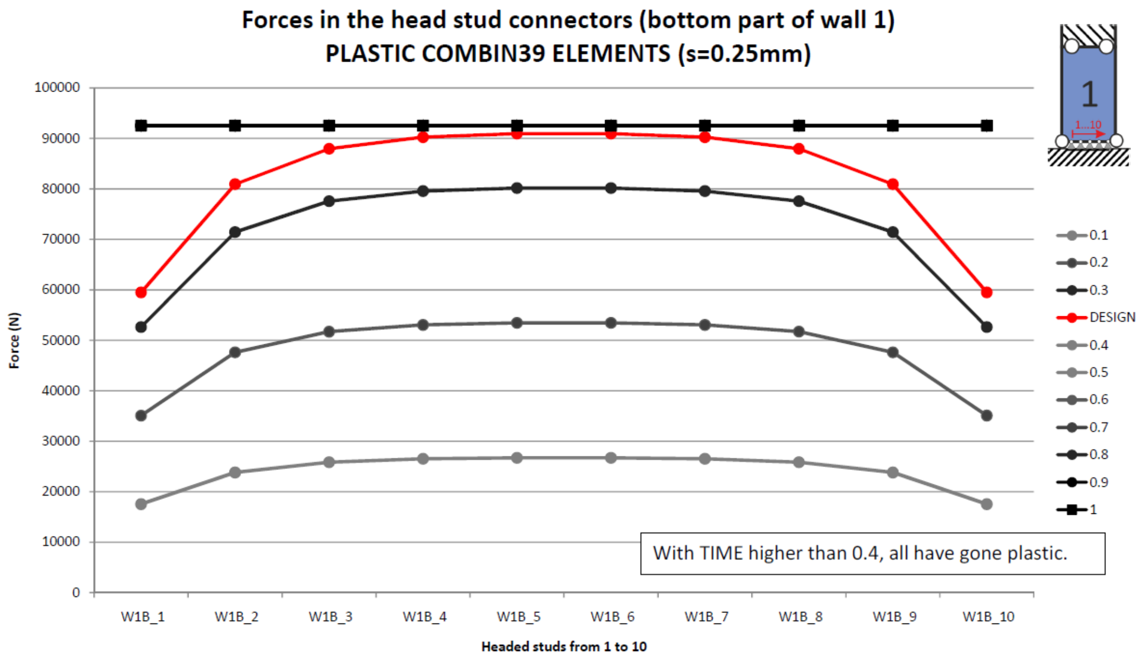


FIGURE 4-31: FORCES IN THE DIFFERENT SHEAR CONNECTORS AT THE BOTTOM OF WALL 1

This happens until all ten connectors at the base have already entered the plastic region. After that, all increasing horizontal loads are carried by the columns. The values of the internal forces in each stud connectors must correspond to the horizontal reactions in each of the end nodes of a spring, as they are fixed to the basement. Also, the total shear must be equal to the sum of the forces in each stud connector plus the reactions of each one of the columns. It is also interesting to notice how the force is transferred to the columns when all studs have reached its elastic limit, but how they carry a higher proportion of the load when they have not. This is interesting, because it means that the first parts to suffer when under a seismic load would be the stud connectors, which will directly transfer the load to the concrete wall.

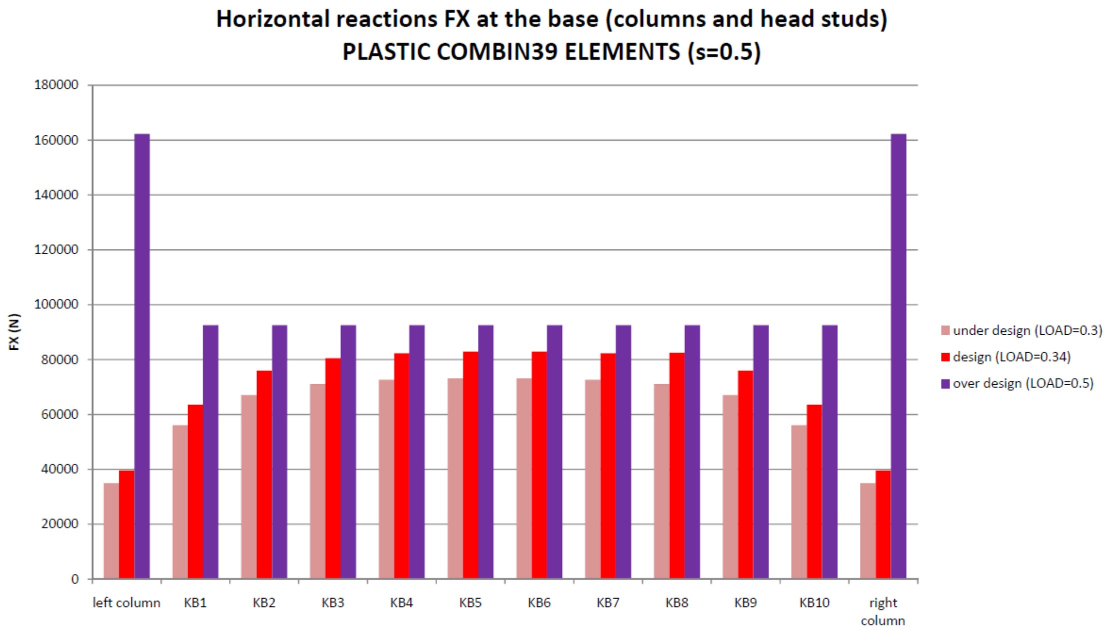


FIGURE 4-32: HORIZONTAL REACTIONS FOR DIFFERENT LOADS

Vertical reactions are also plotted and follow a similar increasing pattern, but this time without yielding, as those nodes are simply coupled.

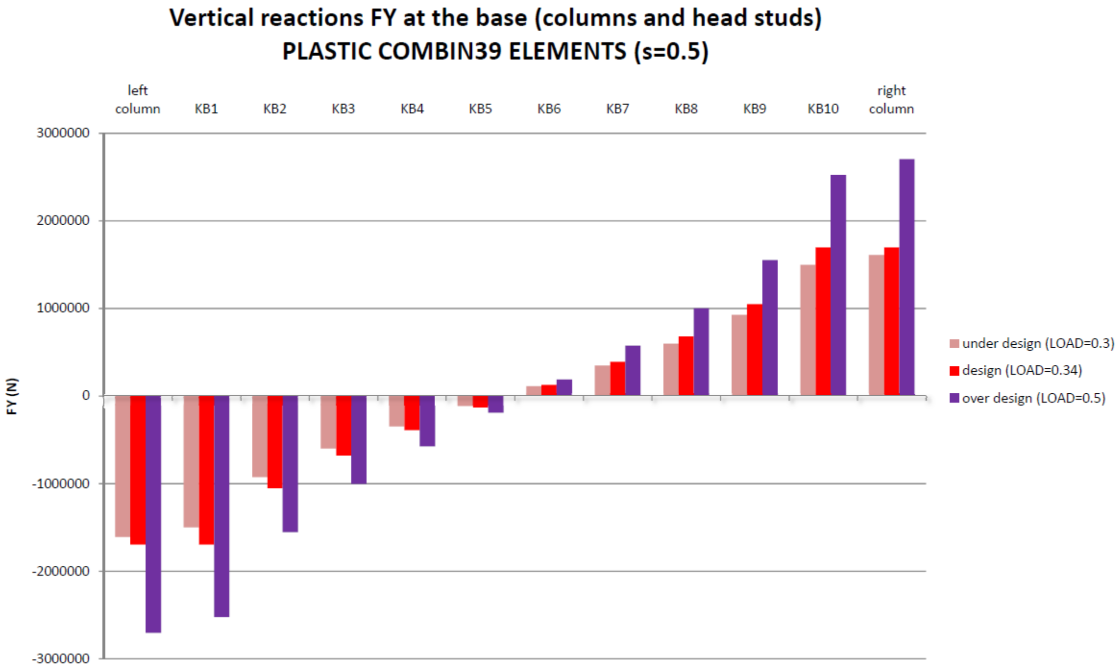


FIGURE 4-33: VERTICAL REACTIONS FOR DIFFERENT LOADS

The distribution of the horizontal shear is different at the top of the wall. This happens because the first wall is fixed at the basement, so it has a particular behaviour, whether all other horizontal interfaces are isolated. In this case, the stud connectors in both sides carry higher loads until they start yielding.

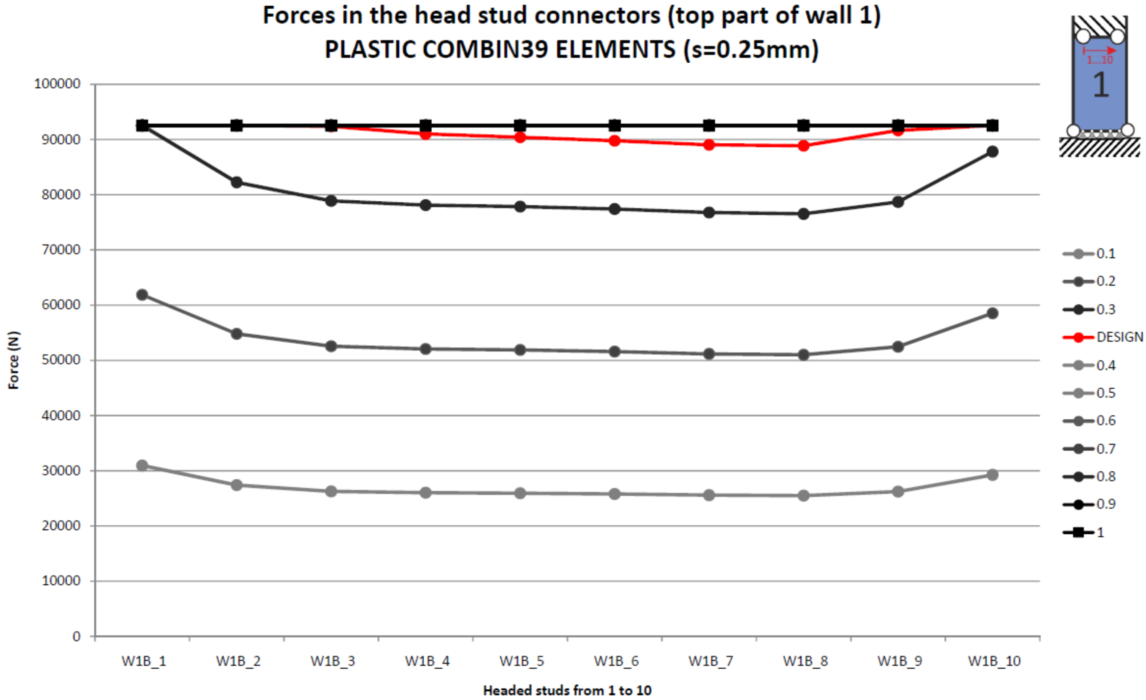


FIGURE 4-34: FORCES IN THE DIFFERENT SHEAR CONNECTORS AT THE TOP OF WALL 1

Regarding to vertical shear, its distribution is less homogenous than the one in the top and bottom of the walls. The 17 connectors are plotted from top to bottom.

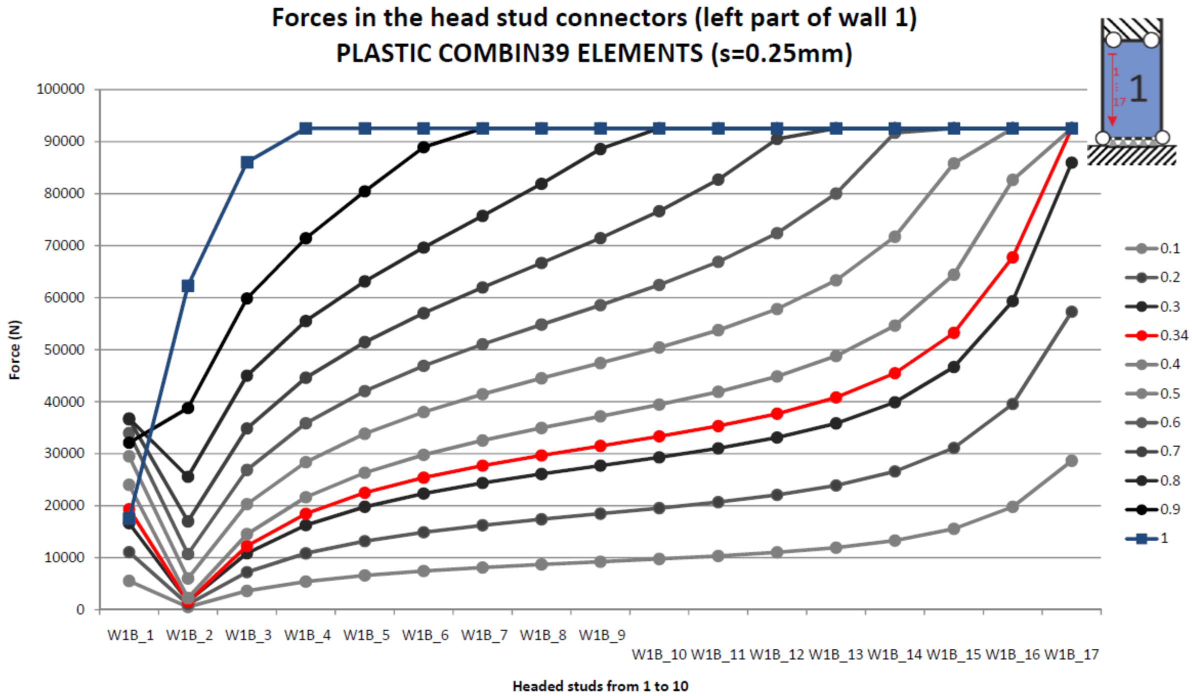


FIGURE 4-35: FORCES IN THE DIFFERENT SHEAR CONNECTORS AT THE LEFT PART OF WALL 1

One anomaly has been found in one of the vertical shear connectors. The one at the top starts decreasing its carried load after TIME=0.75 approximately. This is because of the rotation of the model, as no other connection exists between the wall and the beam over this point. Because of that, it works as a hinge and starts transferring more load in the transversal direction than in the vertical shear direction. The same happens in the right side of the wall.

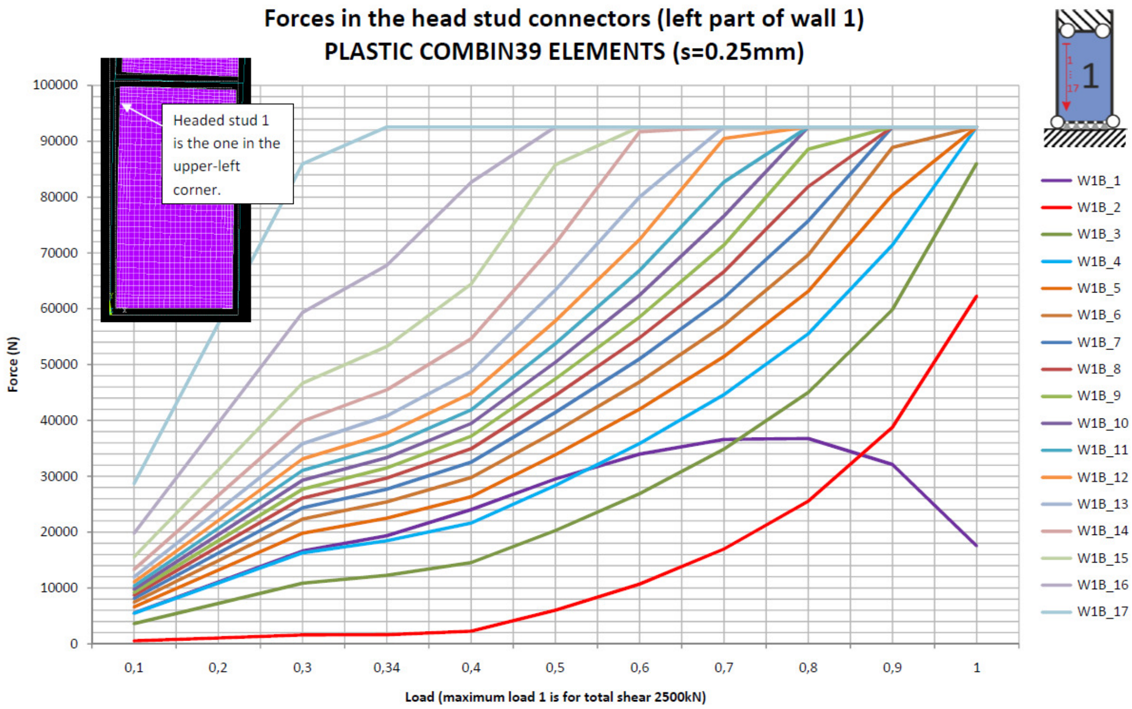


FIGURE 4-36: FORCES IN THE DIFFERENT SHEAR CONNECTORS AT THE LEFT PART OF WALL 1

The other curves for s_e=0.5mm and s_e=1mm are available in the annexes. Finally, it is also interesting to plot the slip of one stud connector (located at the central-bottom part of the first wall). It has two different slopes (elastic and plastic), because there is at the moment only one modelled nonlinearity. Its is possible to check that the design still fits in the elastic region. The curve, however, does not make sense after slip=6mm, as the connector is considered to fail.

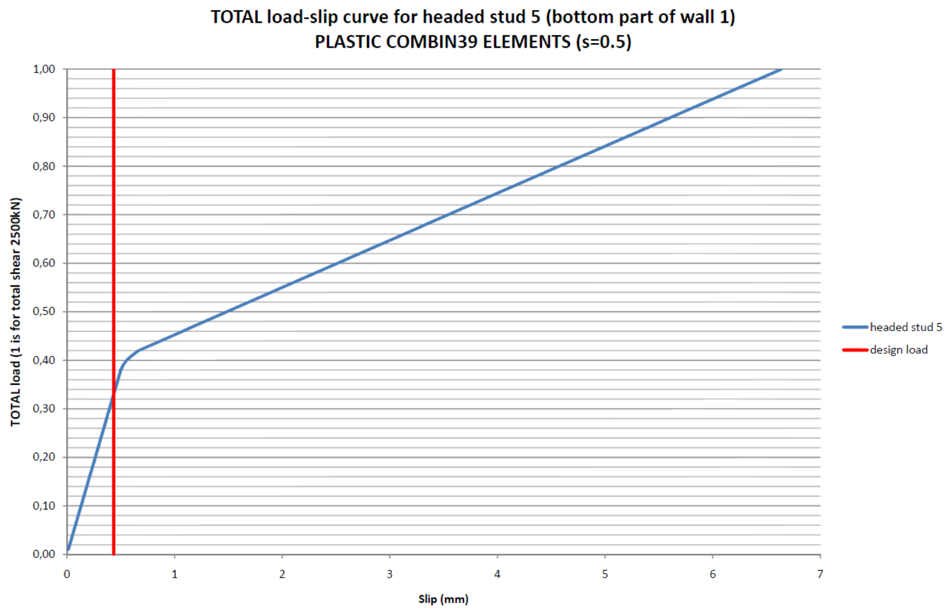


FIGURE 4-37: LOAD-SLIP CURVE FOR HEADED STUD 5

On the basis of these pushover analyses of the four different models (COUPL4, LIN4, PLAST4 and BILIN4) is is concluded that the most interesting model to simulate the shear connection is the one in PLAST4, because:

- It simulates the two working regions of the head stud connector: elastic and plastic.
- It represents with enough accurancy the general behaviour of the system with loads near and just after the design loads.
- It is has the worst behaviour in terms of performance, which will provide results on the side of safety.
- It does not produce singularities as, for example, the COUPL model.

4.4.2.3 Pushover analysis (PLAST4BKIN model)

The material nonlinearity of steel has been included in the previous model, resulting in a new model called PLAST4BKIN[9]. The properties of the steel are set as:

Steel	E	ν	σ_y	Tangent modulus
S355	210000	0.3	355	21
S235	210000	0.3	235	21
units	MPa	-	MPa	MPa

TABLE 4-32: MATERIAL PROPERTIES OF THE TWO TYPES OF STEEL USED IN THE BOUNDARY ELEMENTS

And the bilinear curves have the following shapes (even if ANSYS shows them as finite, they are extended infitely):

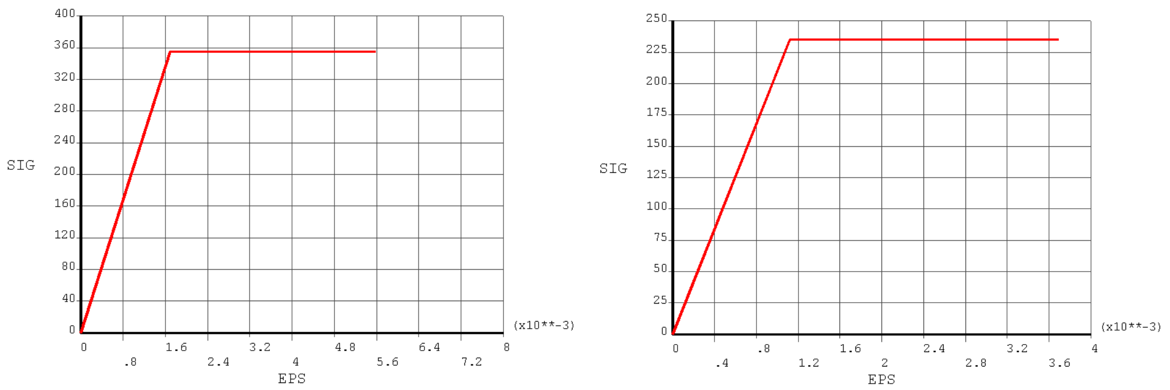


FIGURE 4-38: MATERIAL MODELS (BILINEAR) USED IN THE NONLINEAR MODEL

The S355 steel is used for the first two columns, and all other beams are made of S235. Again, a pushover analysis is performed to compare this model with the one with linear steel.

[9] BKIN comes from the internal code of ANSYS for a *Bilinear Kinematic* material, which is one of the material model able to simulate plastic steel.

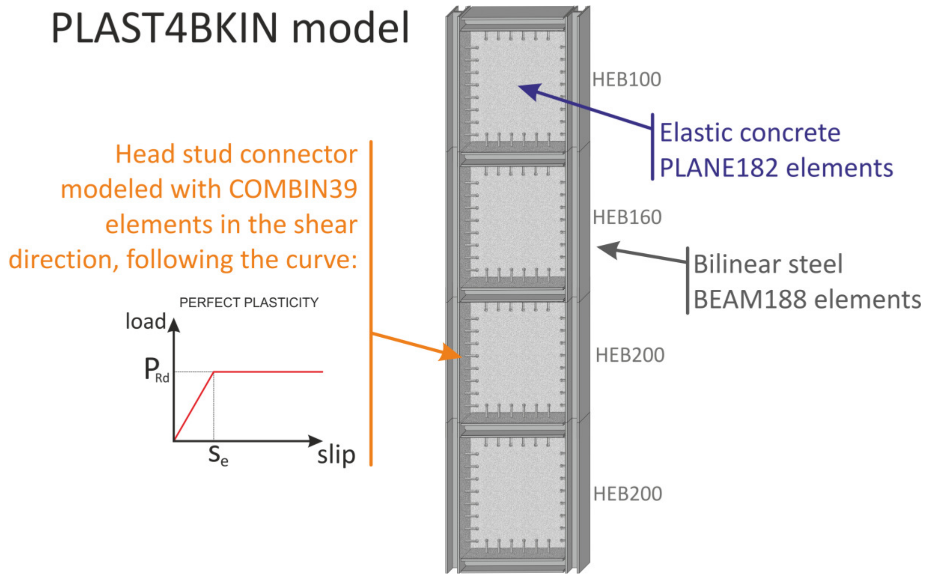


FIGURE 4-39: SCHEME OF THE PLAST4BKIN MODEL

A first look at the displacements at the top of the fourth wall gives an idea of the general response of the system. This model has not been calculated until the TIME=1 value for different reasons:

- The calculation time is longer than the previous models.
- It does not make sense to calculate with loads higher than the ones which make the head stud connectors fail (slip>6mm). This happens approximately after TIME=0.5.

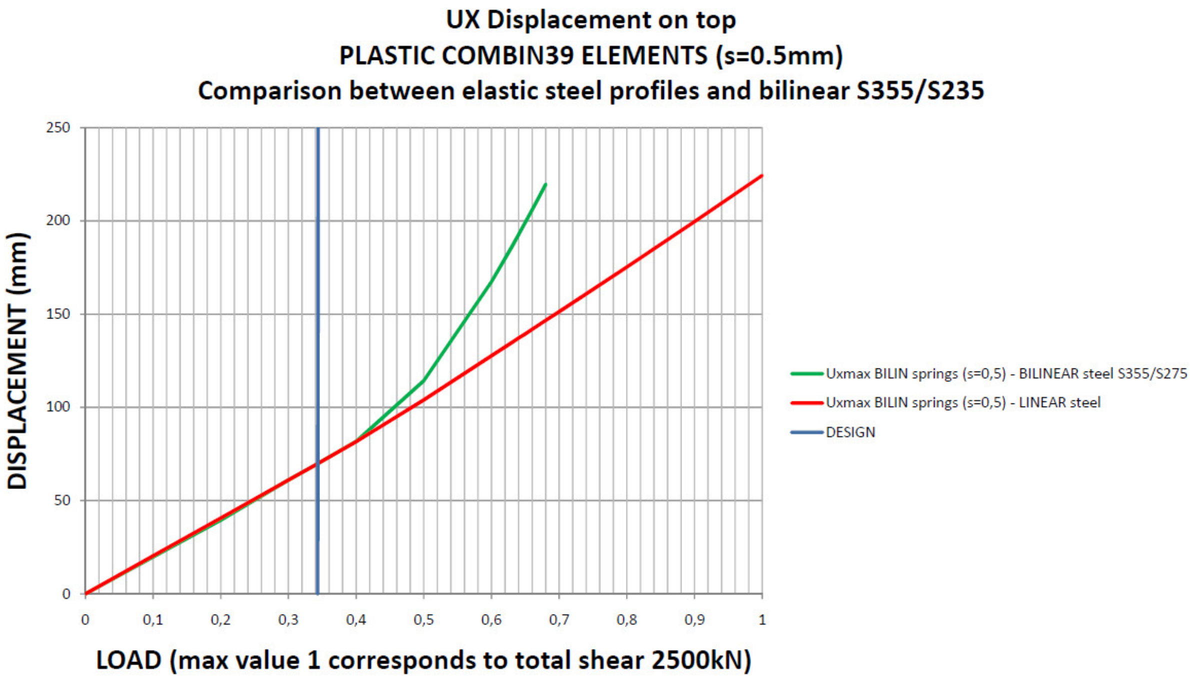


FIGURE 4-40: DISPLACEMENTS ON TOP OF THE PLAST4 AND THE PLAST4BKIN COMPARED

Now two different nonlinearities are present, and therefore the result is a curve.

It is interesting to notice that this model loses its stiffness faster than the previous ones, because of the yielding of the steel columns. Plastic hinges are formed at the basement, and rapidly propagate in other sensitive points. The following graphic showing von Mises stresses is only illustrative, in order to show the formation of the different plastic hinges (TIME=0.6):

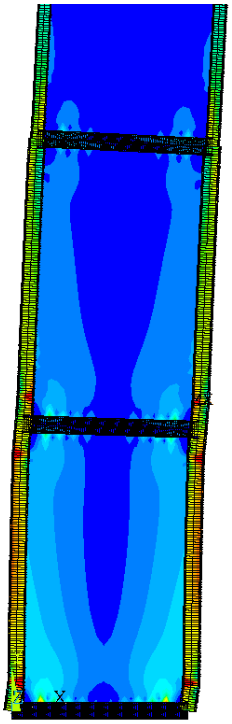


FIGURE 4-41: DETAIL ON THE FORMATION OF PLASTIC HINGES

The horizontal reactions at the basement do not change with this model, as the nodes are equally coupled and have the same behaviour ruled by the COMBIN39 nonlinear springs.

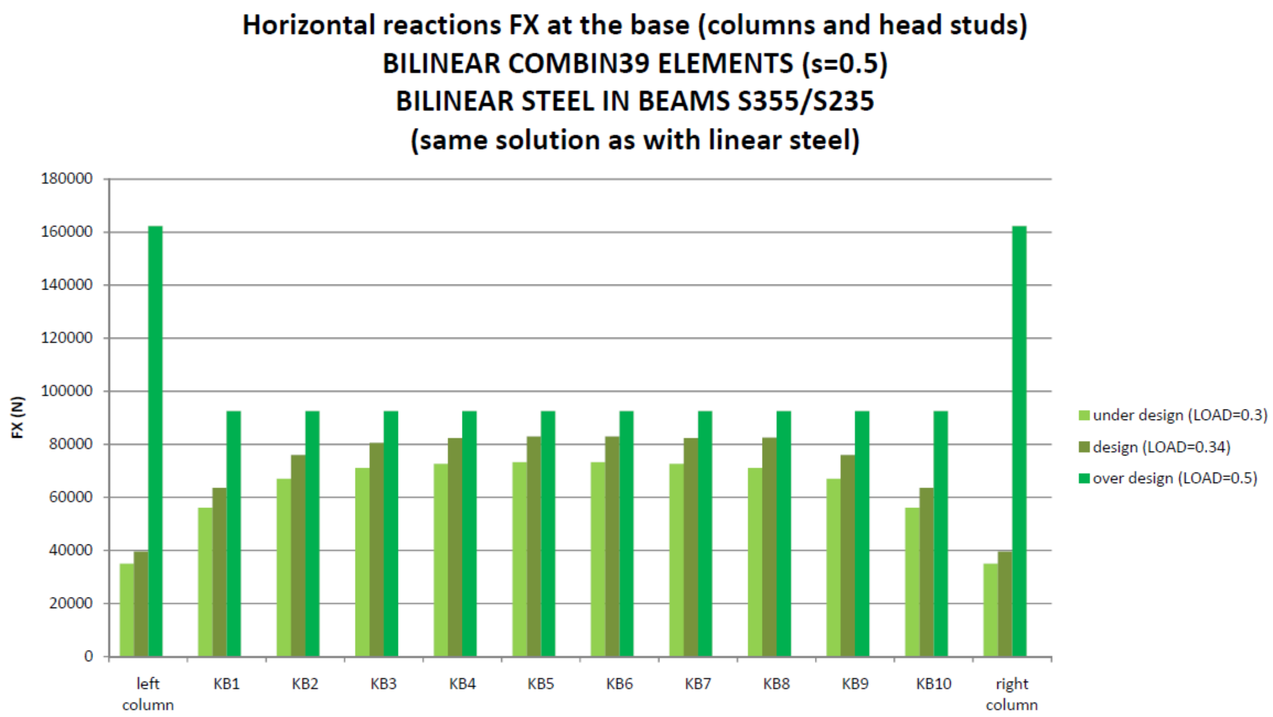


FIGURE 4-42: HORIZONTAL REACTIONS IN THE PLAST4BKIN MODEL

However, the vertical reactions do change. As a result of the yielding of the steel columns, the coupled nodes have to carry a higher amount of the vertical loads.

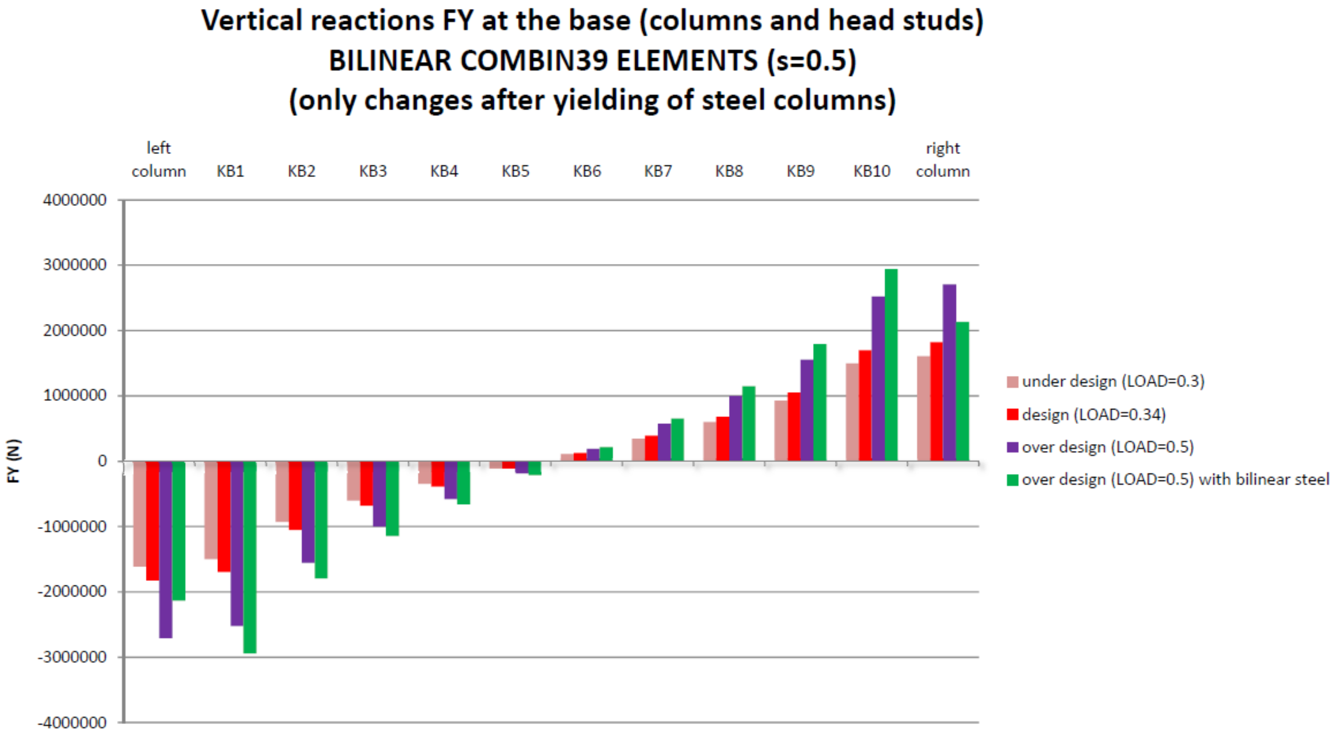


FIGURE 4-43: VERTICAL REACTIONS

This model for the vertical connection at the basement is not supposed to work after yielding of the steel profiles. In fact, it does not make sense to consider that the vertical connection resists more (specially under tension) than the columns. In this model the axial forces, shear forces and moments in the boundary elements of the first wall have been extracted and plotted for three different load intensities: one before the design load, then the design load, and one after the design load (TIME=0.3; TIME=0.34 and TIME=0.5, respectively).

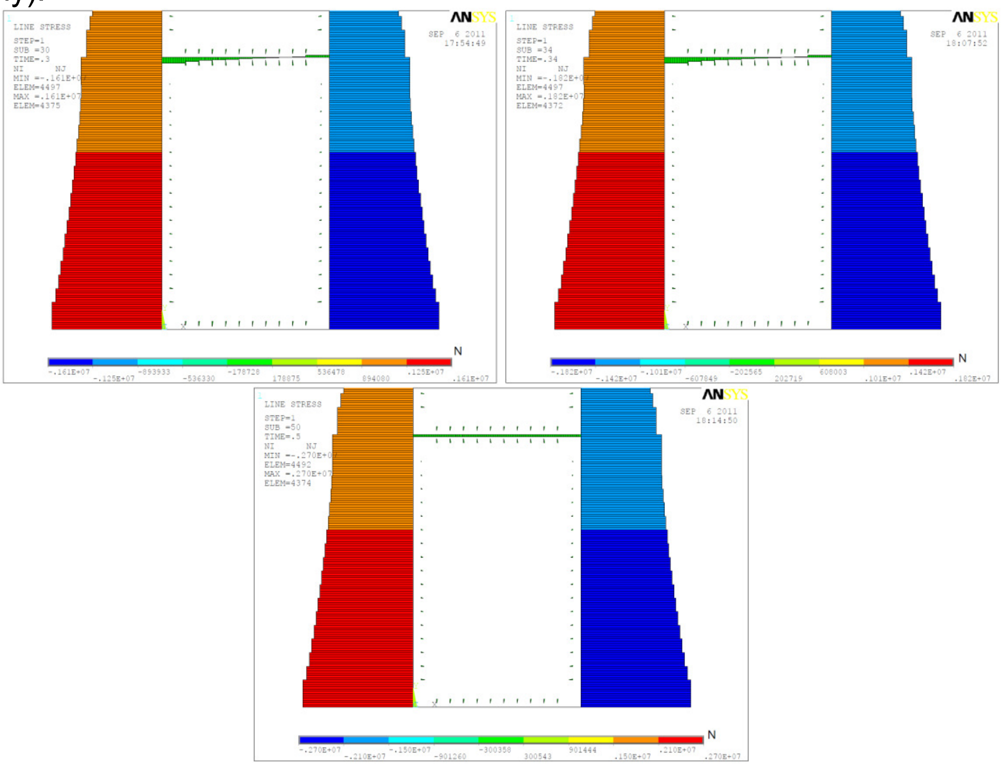


FIGURE 4-44: AXIAL FORCES

It can be seen that the distribution pattern of the axial loads does not change with different loads (larger versions of these images are available in the annexes).

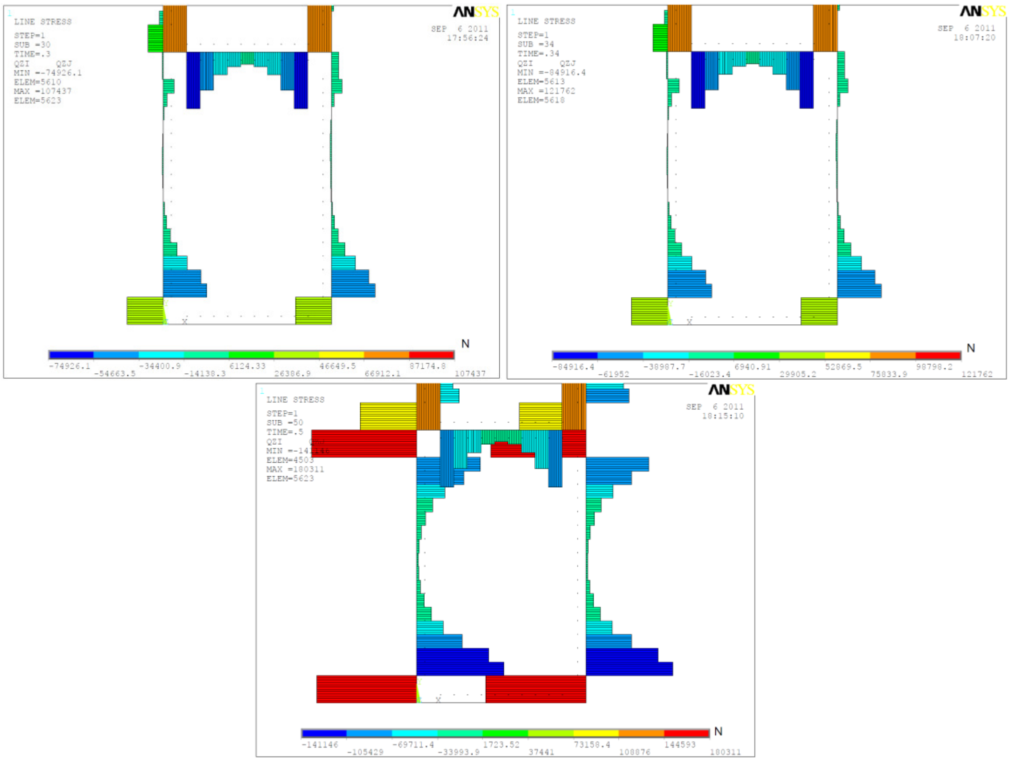


FIGURE 4-45: SHEAR FORCES

Shear forces do change severely after the yielding of the steel. That is because, after that point, the head stud connectors carry higher loads, and therefore they transfer this perfectly to beams (as they are coupled). It is also interesting to appreciate how the shear forces are segmented, and do change after each position of a shear connector (17 times in the columns and 10 times in horizontal beams).

Moments will not be plotted here (only in the annexes) as they are nearly zero. This is an important conclusion, as this has been an initial assumption. Also it makes sense, as all BEAM188 elements are connected with joints, and they do not transfer moment loads.

Even if the material used for the modelling of the concrete wall is elastic, it is also interesting to analyse the stress distribution along the wall. Principal stresses are plotted for the first wall when subjected to the design horizontal load. ANSYS represents the principal stresses with vectors of different colours: black for σ_1, green for σ_2 and blue for σ_3. As the wall is modelled with plain stress, only two of those three stresses are showed at each node, and this can be plotted in a 2D diagram:

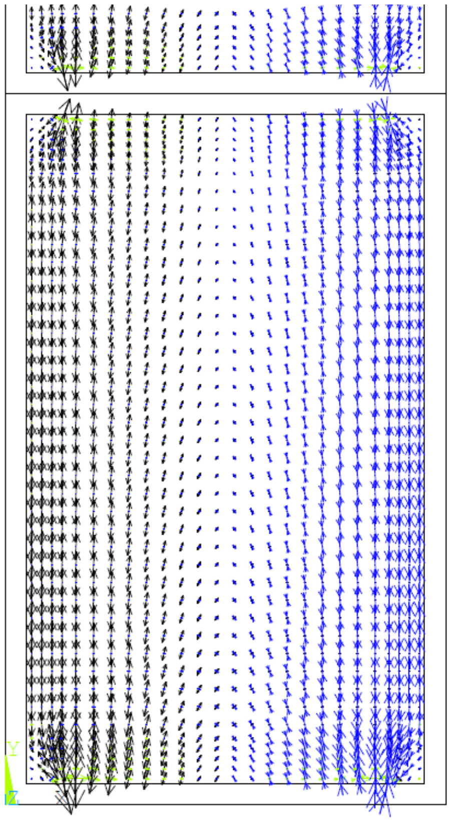


FIGURE 4-46: PRINCIPAL STRESSES ON THE WALL

It can be seen that, as it is expected, in the left side tension dominates, while in the right side compression does (as the load is applied from the left side). Higher stresses concentrate mainly at the corners, where the wall is being “confined” in two perpendicular directions.

4.4.2.4 Analysis of the alternatives to model concrete

A prior analysis on the principal stresses gives an idea on the importance of modelling concrete with more accuracy. Focusing on compressive stresses (as, theoretically, all the tension shall be carried by the reinforcement steel), and even though reinforcement steel also helps in compression, helps having a general idea of the behaviour of the concrete wall. In the following Figure compressive σ_3 is plotted in three different colours.

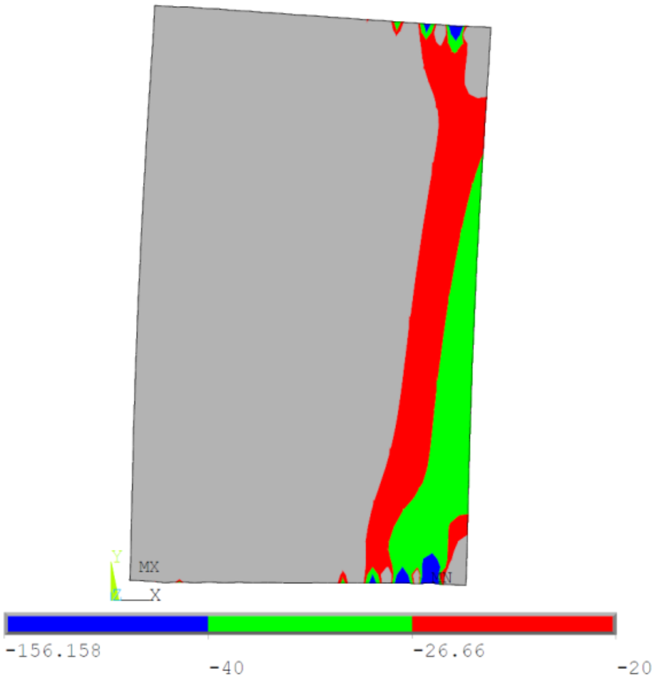


FIGURE 4-47: PRINCIPAL STRESS 3 DETAILED

In the red areas the compressive stress is expected to be smaller than 26.66MPa, this is the design value for the uniaxial compressive strength of the C40/50 concrete, which is being used in this case study. In green areas the stress is between 26.66MPa and the characteristic value of f_{ck}=40MPa, and in blue areas it is higher, so we could consider this areas to fail because of crushing of concrete, under a too high compressive load.

Concrete is a complicated anisotropic material, which fails quickly under a tensile load and resists well compressive loads. Because of that, when modelling concrete, it is necessary to model the steel reinforcement also, as they work together. A comparison between the elastic model of concrete used in the previous models and the dedicated concrete element of ANSYS (SOLID65 element), is carried out.

Two models of one wall are created, without any boundary elements. The first one is extracted from the previous models, so it is a wall modelled with PLANE182 elements with thickness, and the elastic properties of concrete (E_{cm}=35000MPa and ν=0.2).

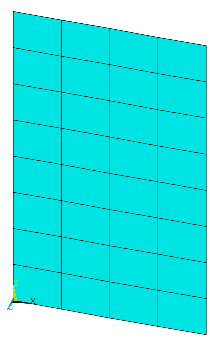


FIGURE 4-48: SIMPLIFIED MODEL OF A WALL WITH PLANE182 ELEMENTS

The second model is called CONCR, and it is a 3D model formed by SOLID65 elements.

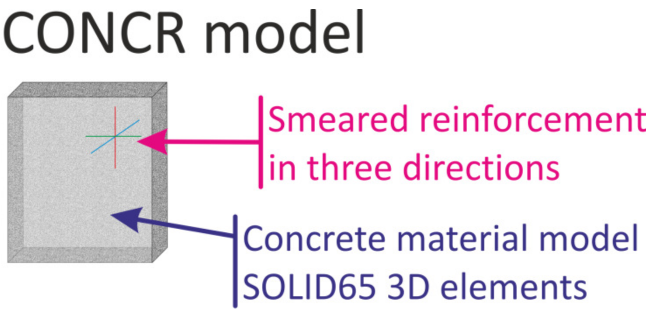


FIGURE 4-49: SCHEME OF THE CONCR MODEL

As explained when describing the elements, it is possible to define a smeared reinforcement (as volumetric ratio) in three different directions. ANSYS considers the steel reinforcement and the concrete to be homogenous and with perfect bonding between them. ANSYS shows for checking purposes the three different directions where reinforcement has been set as three lines. Red line is for the greatest value of volumetric ratio of reinforcement, green for the next, and blue for the smallest ratio.

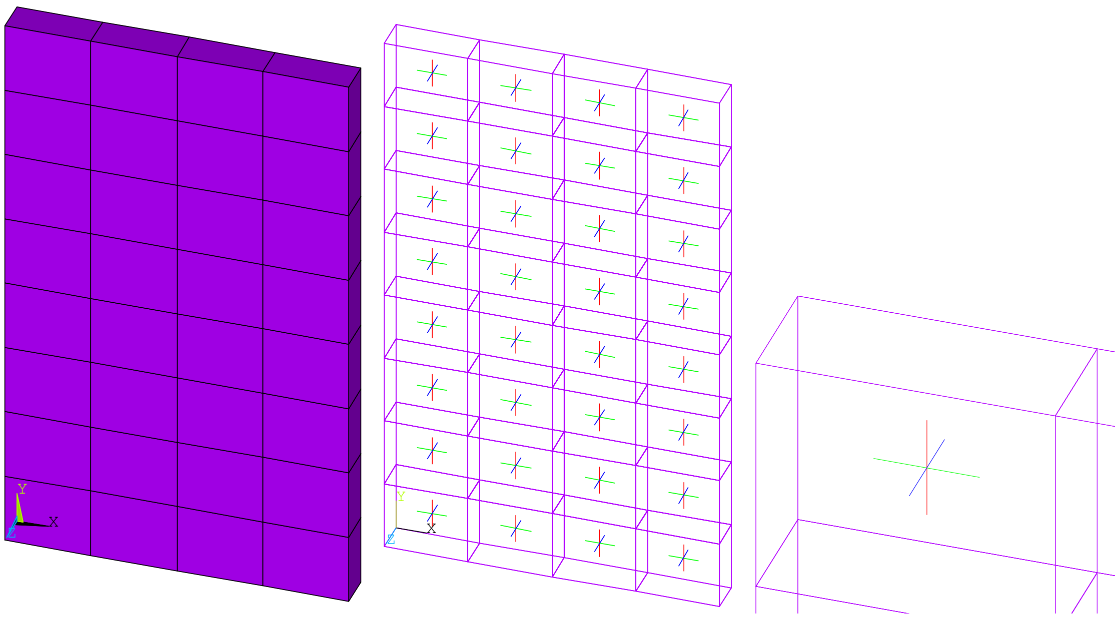


FIGURE 4-50: DETAIL OF THE REINFORCEMENT CAPABILITIES OF THE SOLID65 ANSYS ELEMENTS

For this testing purpose, an average reinforcement ratio has been calculated for the whole wall in each direction taking into account the diameter of each bar, by following:

Direction	Reinforcement taken into account	Volumetric ratio
UX	A_{sh}, A_{swh}	0.003926
UY	$A_{sv,b}$, $A_{sv,i}$, A_{swv}	0.0182
UZ	A_{sh}, $A_{swh,}$ $A_{sv,b}$, $A_{sv,i}$, A_{swv}	0.00066

TABLE 4-33: VOLUMETRIC RATIO OF REINFORCEMENT USED IN THE SIMPLIFIED MODEL FOR ONE WALL

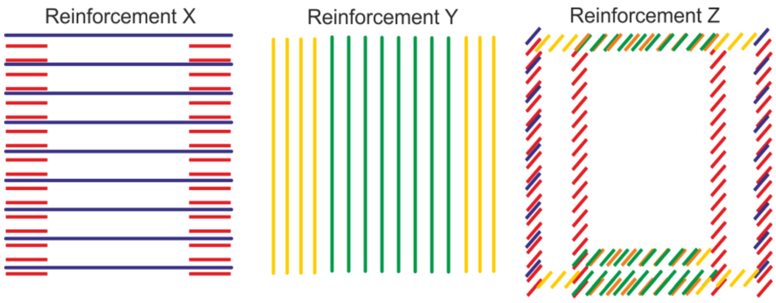


FIGURE 4-51: DETAILS OF THE REINFORCEMENT IN EACH DIRECTION

However, only the part of the reinforcement in each direction has been taken into account.

Displacement in the horizontal direction of a wall subjected to the design load will be plotted. For the PLANE182 version (elastic concrete):

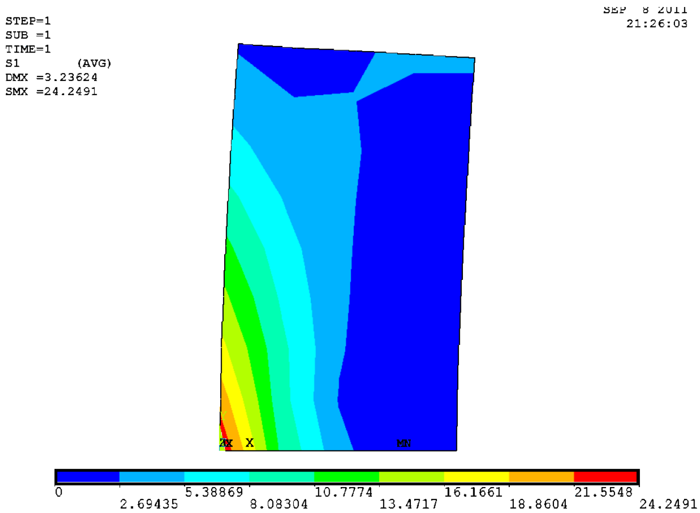


FIGURE 4-52: PRINCIPAL STRESS 1

And with the SOLID65 elements, this is the CONCR model:

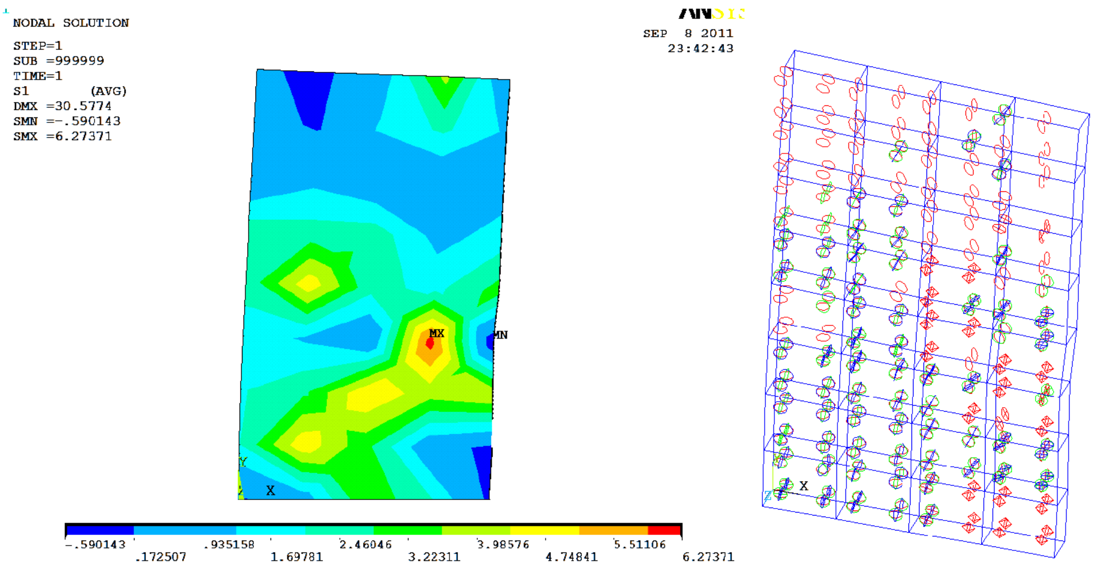


FIGURE 4-53: PRINCIPAL STRESS 1 AND CRACKING/CRUSHING PLOTTED

The top displacement of the elastic model is 3.23mm, while for the CONCR model it is 30.58mm. The two values are strongly different, which makes sense as concrete has a nonlinear behaviour. It is also interesting to check at the crushing and cracking capabilities of the SOLID65 elements. ANSYS shows crushes with a red tetrahedron, first cracking with a red circle, second with a green circle and third with a blue circle. Cracking can be produced in three different planes, after that the concrete loses its complete stiffness, but it still can carry a little amount of shear stress. Cracks can also close, but they recover only a part of its shear resistance. These parameters are set with the material properties (shear transfer coefficients). Closed cracks are shown with an X inside the circle:

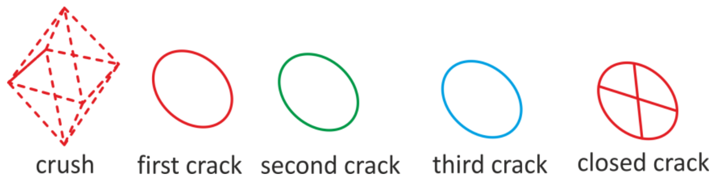


FIGURE 4-54: SYMBOLS USED BY ANSYS TO REPRESENT CRACKING AND CRUSHING

It can be seen that the parts of the wall subjected to compression have already crushed (near the bottom-right corner). Several cracks have been produced all over the wall. However, it is important to notice that the wall conserves its integrity because of the contribution of the steel reinforcement, and also that no boundary elements have been modelled in this tests, even though they also contribute to the stiffness of the system.

4.4.3 NUMERICAL MODELS FOR DYNAMIC TRANSIENT ANALYSES

Static analyses use static loads (forces, displacements, moments...) applied to the model. However, if a real seismic analysis is going to be performed, it is necessary to have an earthquake register. Which is more important about these registers (accelerograms) is that they should fit the design spectrums of Eurocode 8. Artificial accelerograms are generated with different software solutions. For example, SYNTH is a program developed by Meskouris in the Chair of Structural Statics and Dynamics of the RWTH Aachen, and SIMQKE is another solution developed by Gasparini and Vanmarcke, which has also been improved by other authors. These programs usually work by iterating wave forms and superposing them until the target design sprectrum if fitted with a particular level of accuracy.

A response spectrum is a graphic showing the peak response of a series of oscillators with different natural periods covering a certain range of frequencies, and for a particular value of damping. Those oscillators are subjected to a dynamic load and its peak responses are computed (acceleration, velocity and displacement). The response spectrums in the Eurocodes are abstractions made from different seismic registers, but they are only an idealisation.

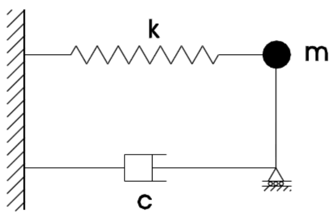


FIGURE 4-55: ONE DEGREE OF FREEDOM OSCILATOR

Seven accelerograms for acceleration, velocity and displacement have been provided by the Institute of Steel Structures, as they are being used within the INNO-HYCO project. They have been generated for the input data of the case studies, this is for for type spectra 1, ground type C and a value a_g=0.25g (α=0.25), and standard damping η=1. Each artificially generated earthquake lasts 20 seconds.

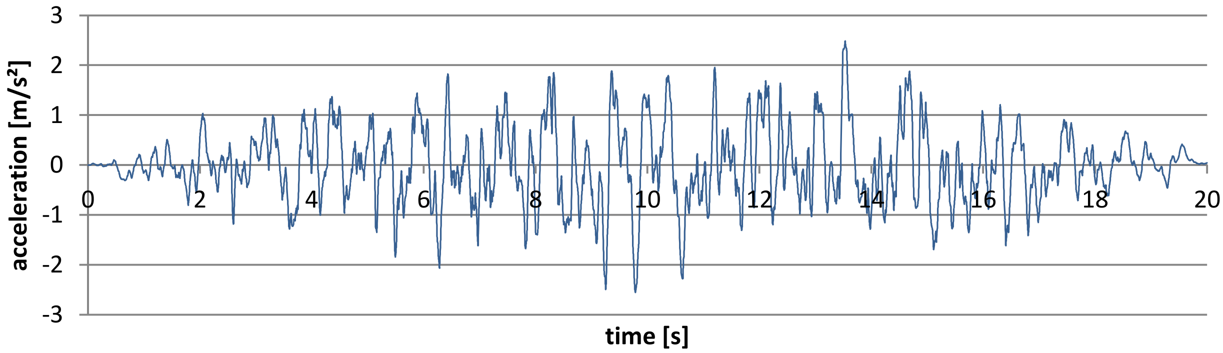


FIGURE 4-56: ARTIFIALLY GENERATED ACCELERATION RECORD

The acceleration register is normally the reference, but it has to be integrated twice in order to apply the loads in the dynamic transient analyses in ANSYS. Velocity and displacement are obtained from integration, and those registers were also provided by the Institute. The register for velocity, obtained from the previous accelerogram.

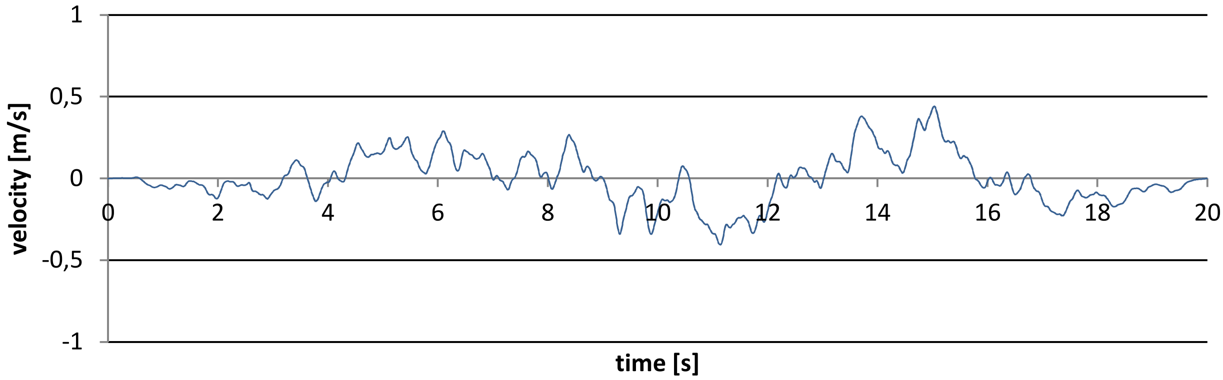


FIGURE 4-57: ARTIFIALLY GENERATED VELOCITY RECORD

And displacement, which will be the register used for testing the models:

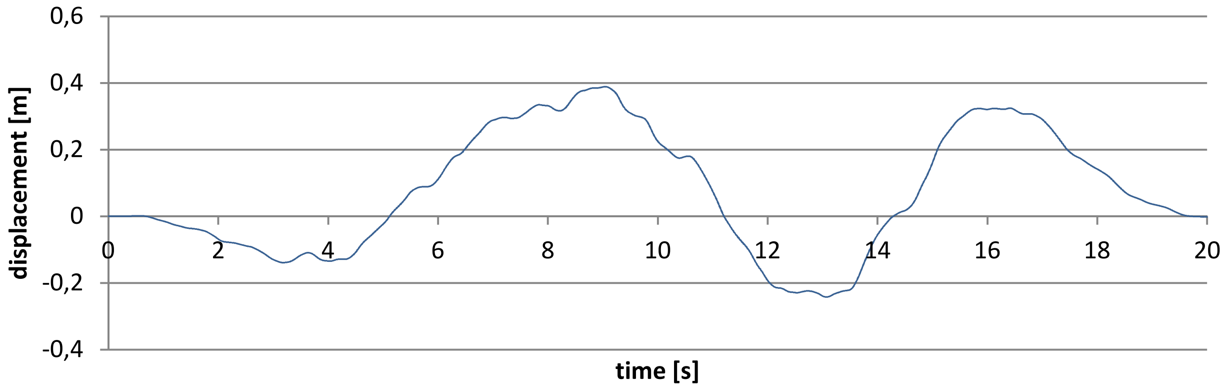


FIGURE 4-58: ARTIFIALLY GENERATED DISPLACEMENT RECORD

Seven different accelerograms should be used to reduce variability in the results, according to Eurocode 8, as explained in the corresponding chapter.

4.4.3.1 One-wall model for dynamic transient analyses

A first simplified model has been made to analyse the behaviour of the first wall, as it is the most critical part of the system. In this model it is necessary to set an equivalent mass at the top of the wall, which will be also distributed along the beam with 93 MASS21 elements. However, as only one wall has been used, four times the equivalent mass used in the FREQ model has been set. This is one simplified way to make the first wall carry the whole horizontal load due to mass. The vertical loads are also multiplied by four, but they are set as forces. An equivalent load of 4·236.8kN=947.2kN has been distributed also along the beam, at the same locations of the MASS21 elements.

The stud connectors are still modelled with the previously adopted solution, that is with COMBIN39 elements following a perfectly plastic curve. Concrete is still also idealised as an elastic material. This model is called DYN1, as it will be used for dynamic analysis, and has one of the four walls, but with equivalent loads. It is however important no notice that this is not really an equivalent system, but works as an approximate equivalent system in order to check the general behaviour of the whole system and, even more important, check if the meshing is correct and prevent possible failures when calculations with higher models start.

The scheme of the model is the following:

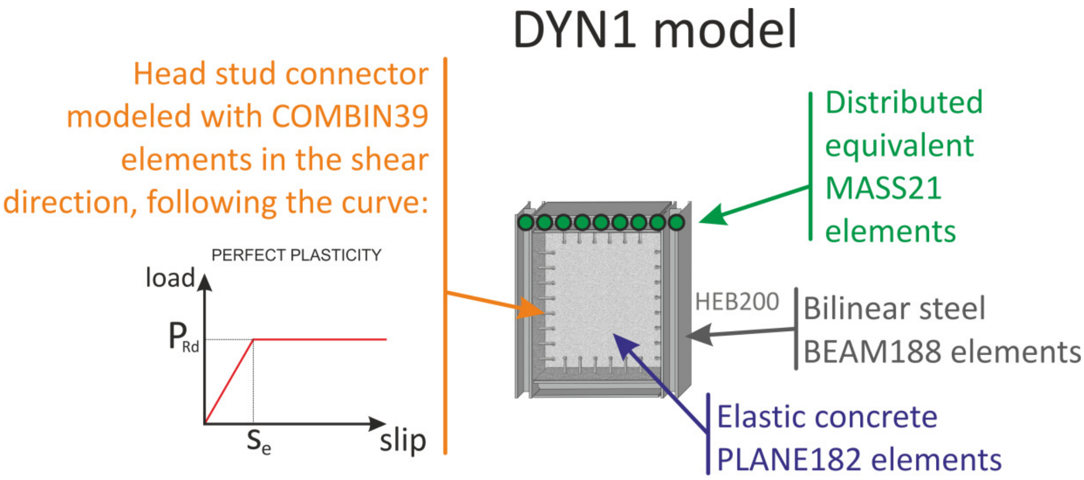


FIGURE 4-59: SCHEME OF THE DYN1 MODEL

This model has also a difference with all the models used previously. Instead of constraining the degrees of freedom at the bottom, at each time step the value of displacement at the base will be the applied load. This movement at the base will generate accelerations, and thus forces (as the system has mass). The accelerograms have a value of displacement for every 0.005 seconds, but an automatic time stepping has been configured, so that ANSYS can decide if it applies the load in more substeps. A good way to understand this is, for example, when suddenly the direction of movement changes. The whole mass of the building was moving in one direction, and then must move in the opposite way. This could happen in only 0.005s, but ANSYS maybe needs to divide this “load” into smaller substeps (that is, apply it more slowly) to reach convergence. ANSYS has its own mechanisms to check for convergence, and they are based on a tolerance of the differences between the applied loads and the resulting reactions. If the differences are too high (higher than a customizable value of tolerance), it considers that the solution has not converged. With the automatic time stepping option, it creates a bisection (divides the load into smaller loads, by creating more substeps) and starts calculating again from the previously converged solution. Systems with strong nonlinearities require often from several substeps.

The calculations of a complete 20 seconds artificially generated earthquake record last with this model four hours when using the Center for Computing and Communication of RWTH Aachen University. However, these computational centers usually assign a limited amount of resources to each calculation, so it lasts approximately the same as it would in a new high range computer.

4.4.3.2 Four-walls model for dynamic transient analyses (DYN4)

A four-wall model has been developed including equivalent masses in each storey. The scheme of the model is:

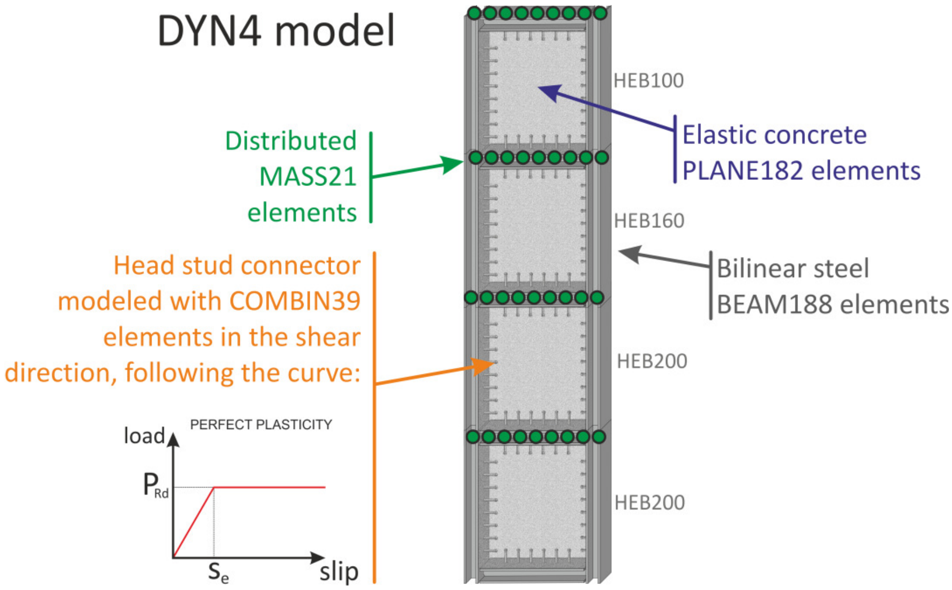


FIGURE 4-60: SCHEME OF THE DYN4 MODEL

It important to notice that the calculations with this model do not last simply four times the calculation time needed by the DYN1 model, but much more. Even if only the equivalent masses in each building are considered, the calculation time is approximately of 24-30 hours for 5 seconds of the complete earthquake record (this is ¼ of the total time). The size of the resulting file is approximately of 25GB. This means that for a complete record more than 5 complete days of calculation would be needed, and also 100GB of storage space. The post-processing of those files is also highly time-consuming. Only five seconds have been calculated for checking purposes and in order to evaluate the computational cost of the calculation of a complete record.

4.4.3.3 Four-walls model with nonlinear concrete material (DYN4CONCR)

The most complex model that has been developed within this project is DYN4CONCR. This is similar to DYN4, but it has been necessary to redesign it and substitute the PLANE182 elements in the first wall by SOLID65 concrete elements. This is not a direct process, as the first ones only exist in a two-dimensional plane, while the seconds are three-dimensional elements. Boundary conditions and constraints had also to be modified.

Provided that the reinforcement ratios in the confined zones of the wall (in yellow) are sufficiently different to the ratios in the central part (in blue), it has been necessary to model the first wall in three parts. The two different values of smeared volumetric ratio of reinforcement are set as real constants of SOLID65 (ANSYS parameters)

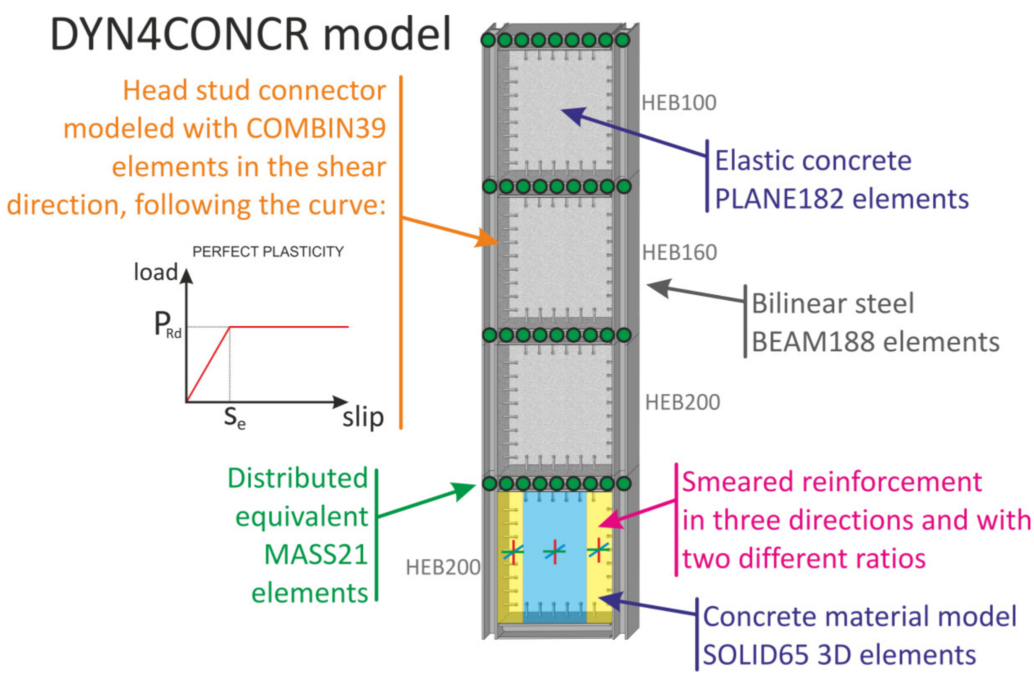


FIGURE 4-61: SCHEME OF THE DYN4CONCR MODEL

A static analysis has been performed with this model to analyse the results, with the design loads derived from the Eurocode 8 prescriptions. Vertical loads have been applied first, and then the lateral design forces in each floor. It is possible to see that several cracks have been produced all over the wall, but the whole system still conserves its integrity. The UX displacement on top (approx. 75mm) is higher than the one with elastic concrete (approx. 50mm), as is expected.

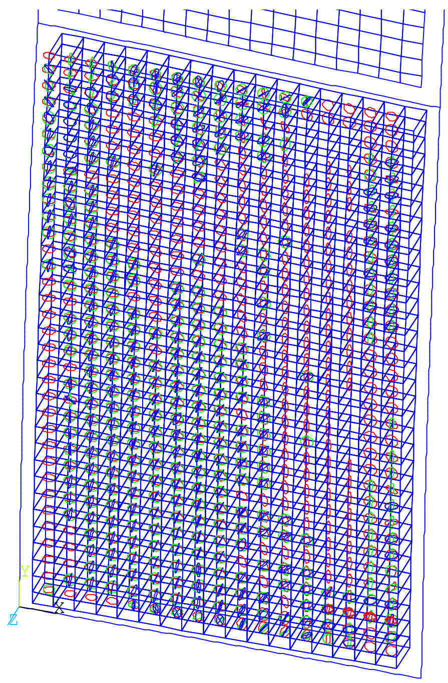


FIGURE 4-62: CRACK/CRUSH PATTERNS IN THE DYN4CONCR MODEL SUBJECTED TO DESIGN STATIC LOADS

Having a look at the principal stresses, it is possible to check that none of the principal stresses is higher than the uniaxial compressive strength and the uniaxial tensile strength of the concrete used. Principal stresses 1, 2 and 3 are plotted:

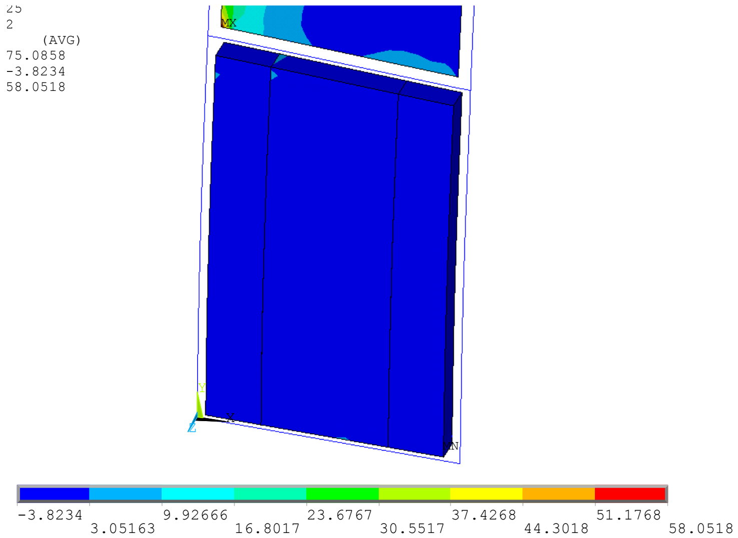


FIGURE 4-63: PRINCIPAL STRESS 1

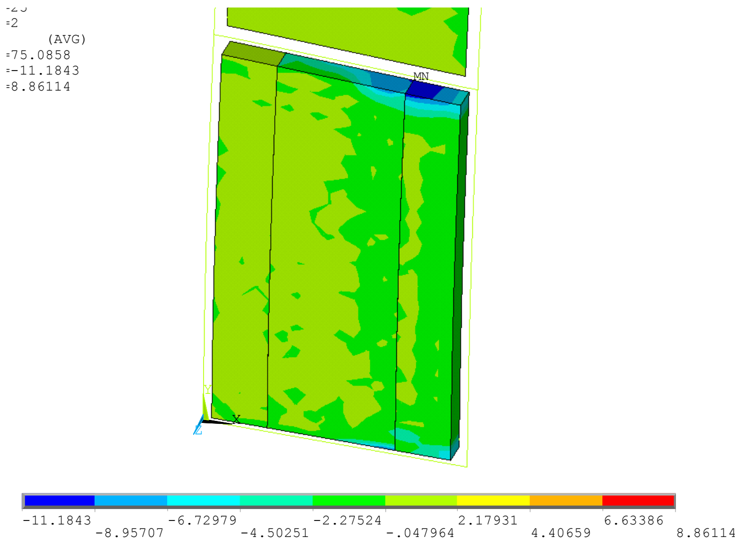


FIGURE 4-64: PRINCIPAL STRESS 2

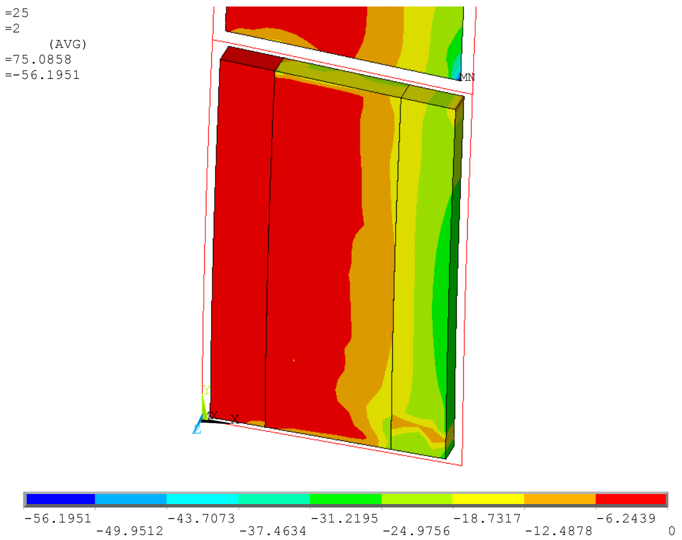


FIGURE 4-65: PRINCIPAL STRESS 3

Von Mises equivalent stresses show that in the left column the yield stress of the steel used (S355) is reached at some points:

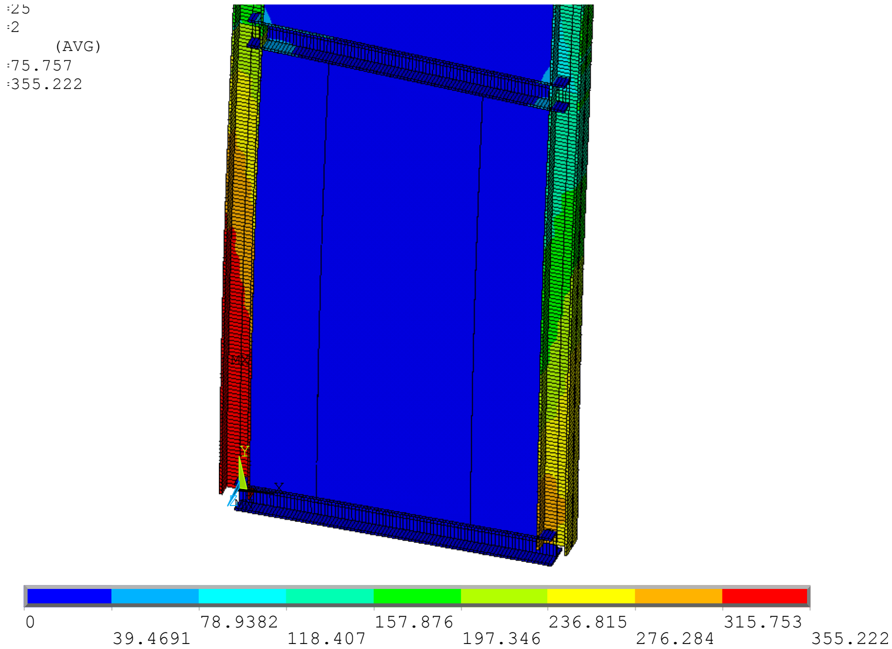


FIGURE 4-66: VON MISES EQUIVALENT STRESS IN BOUNDARY ELEMENTS AT THE DESIGN LOAD

And it is also possible to check in which exact areas the steel has already gone into the plastic region. This actually happens in small tensioned areas near the headed stud connectors of the left part of the wall:

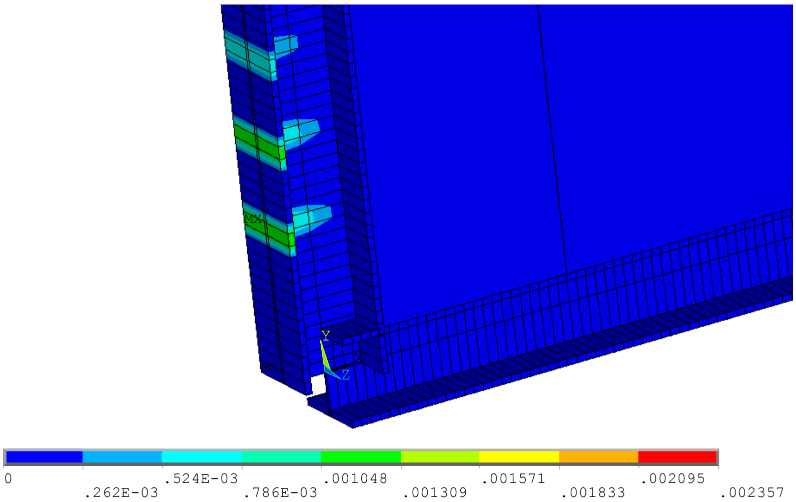


FIGURE 4-67: VON MISES PLASTIC EQUIVALENT PLASTIC STRAIN AT THE BOUNDARY ELEMENTS UNDER DESIGN LOAD

Regarding to the stresses at the reinforcement bars, it has been proved that they are far from the yielding point. This is the reason why the wall still is still carrying load. The stresses are only computed as average for each SOLID65 element, so they are easy to plot. The stresses[10] in the reinforcement present in X, Y and Z directions are shown in the following figures:

[10] Stresses for reinforcement 1, 2 and 3 are coded in ANSYS as ETABLE results: SMISC 2, 4 and 6 respectively.

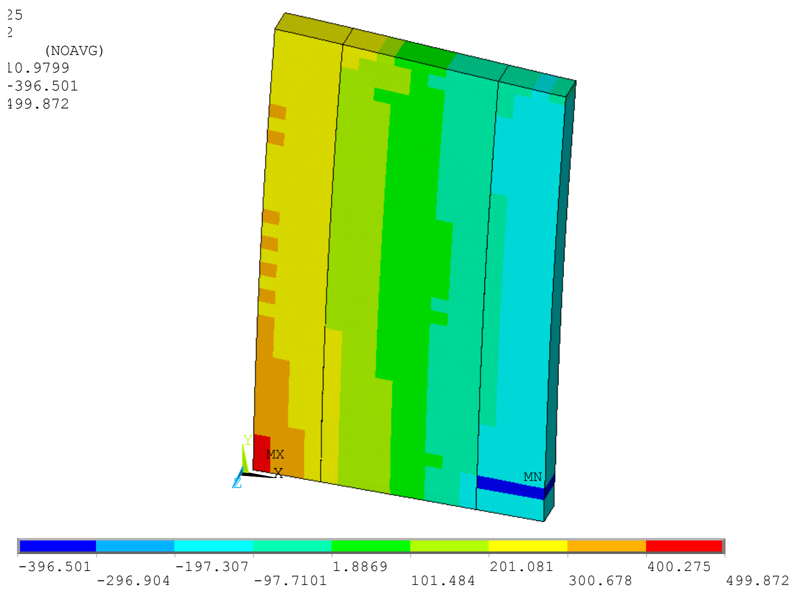


FIGURE 4-68: STRESS IN THE HORIZONTAL REINFORCEMENT (X DIRECTION)

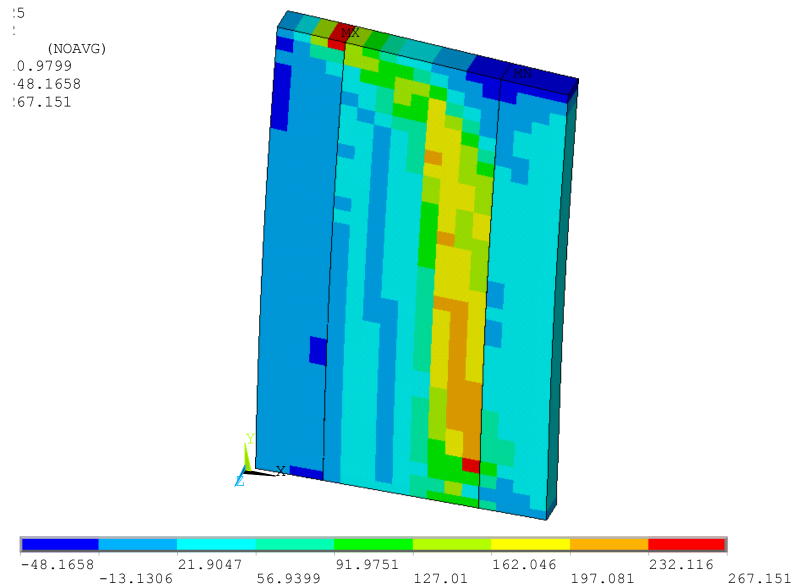


FIGURE 4-69: STRESS IN THE VERTICAL REINFORCEMENT (Y DIRECTION)

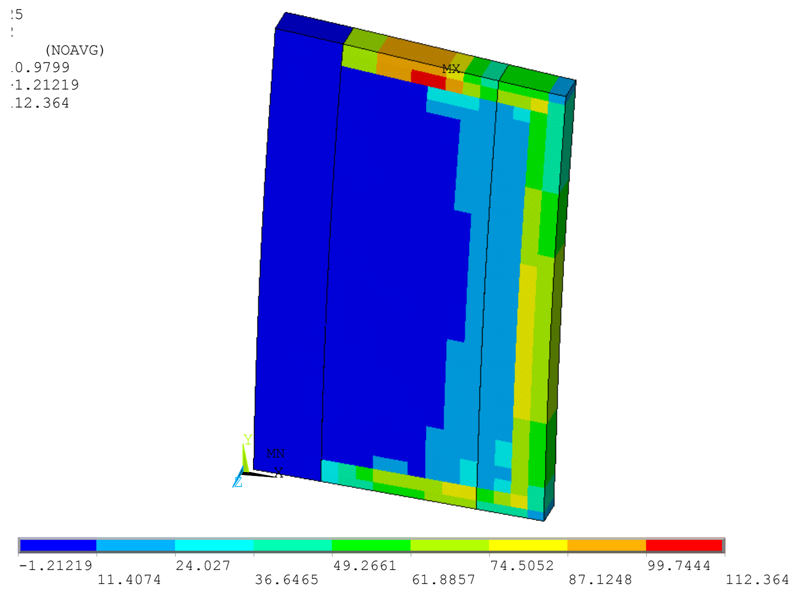


FIGURE 4-70: STRESS IN THE TRANSVERSAL REINFORCEMENT (Z DIRECTION)

Positive values of stress are for tensile stresses, and negative for compressive. The reinforcement of the SOLID65 elements does not have shear strength.

4.5 PERFORMANCE AND EVALUATION OF THE FUTURE IDA ANALYSES

Incremental dynamic analyses (IDA) consist on performing different transient analysis with an earthquake record (artificially generated, when no other data is available or is insufficient), scaled at different levels of an intensity measure (IM). A commonly used IM is the peak value of ground acceleration (also called PGA). The response of the system is obtained from transient analyses, performed by a finite elements program capable of carrying out dynamic analyses.

In this case, as displacements are being used, they can be scaled with the same factor, as they are only directly two times integrated from acceleration. The seven artificially generated earthquake records of the INNO-HYCO projects were originally scaled for a PGA of 0.25g. Dividing the displacement by 0.25 we would obtain a displacement record scaled to the reference gravity value. After that, the accelerograms could be scaled to the desired values by multiplying it by a factor, normally called α. Typically initial values of 0.1g (α=0.1) and then increasing by 0.1 are used, and the maximum load can be decided in function of the response of the system. In this case studies, for example, it would be a good criterion to scale the records until the headed stud connectors start failing (that is, have values of slip higher than 6mm) or when the concrete wall or the boundary elements start having convergence problems. This could happen when the reinforcement starts yielding and therefore the concrete wall looses rapidly its stiffness, or when high areas of the boundary elements have already yielded.

Normally variables (EDP) such as maximal interstorey drift ratio or maximal displacement at each storey are plotted against this IM, in order to obtain a general understanding of the performance of the building. It is also interesting to find the points where the structure fails or plastic hinges are formed.

Interesting variables to be plotted are the maximum interstorey drift (or also the maximum interstorey drift ratio) and the maximum displacement at the top of the building recorded during the artificial earthquake.

It is important to remember that the dynamic analyses have to be performed with, at least, seven different artificially generated seismic records.

An example to show the aspect of an IDA curve has been done with the DYN1 model, taking the maximum interstorey drift ratio for six different scales.

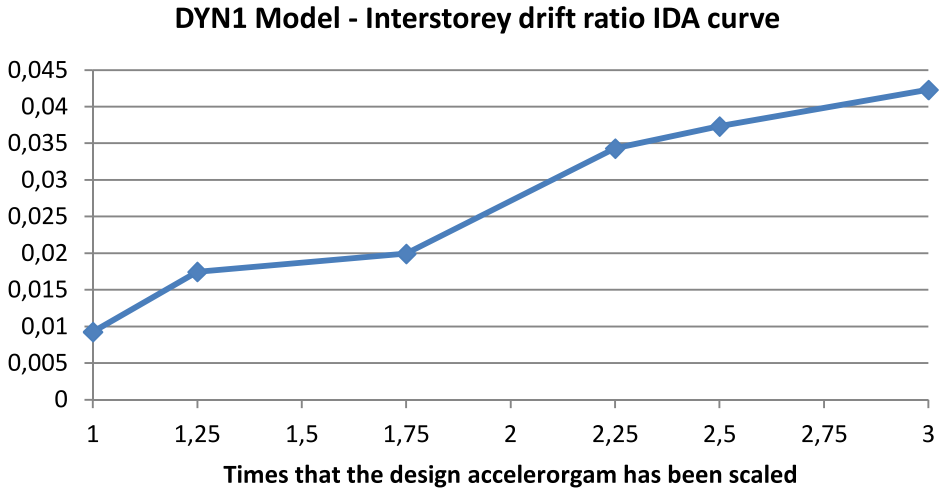


FIGURE 4-71: EXAMPLE OF AN IDA CURVE FOR THE DYN1 MODEL

5 SUMMARY AND CONCLUSIONS

In the chapter 2 of this thesis, the different methods of analysis allowed by the Eurocodes have been described and compared with the methodology proposed by the Performance-Base Earthquake Engineering. After that, a particular case study, which belongs to the INNO-HYCO project carried out by the Institute of Steel Structures of the RWTH Aachen, has been dimensioned according to the Eurocodes. Different finite element models have been developed in order to calibrate the final mathematical model, which is capable of representing the different nonlinearities of the system (related to steel, concrete and the shear connectors used as interface). A simplified model has been tested with an artificially generated accelerogram, fitting the spectrum of Eurocode 8.

The most relevant conclusions of the work done, which are interesting for the upcoming work of the INNO-HYCO project, are:

- The number of shear connectors needed and their diameter have been dimensioned according to Eurocode 4. However, these shear connectors are normally used in different applications, such as composite beams, for example. After validating a model to implement the load-slip curves of those connectors, it is possible to conclude that the dimensioning was correct. In the pushover analyses, the headed stud connectors behave elastically when subjected to the design load, and only enter their plastic region when this load is exceeded.
- On the other hand, the axial load transfer capability of those shear connectors is still not well understood. Therefore, in the models used in this case study this degree of freedom has been coupled between the steel profiles and the concrete wall. In order to obtain a more accurate model, it would be interesting to model this connection in a way that it works as a coupled connection when under compressive loads, but has a different behaviour under tensile loads. However, it would be necessary to analyse this behaviour experimentally before being able to implement it, as it has been done for the shear connection, by using a reference load-displacement curve.
- After the static analysis of the final DYN4CONCR model it has been proved that the dimensioning was sufficiently accurate. However, yielding of the boundary elements has been obtained at the design load, which should theoretically not happen. The model is not accurate enough to conclude that the yield stress of the steel used should be increased, but more analyses should be done to analyse why this happened. The problem is probably related to the coupling of the nodes between the boundary elements and the concrete wall.
- The SOLID65 elements of ANSYS are an interesting solution for modelling the concrete walls. The smeared reinforcement can be used as a good approximation of the real behaviour of the steel bars. However, these systems are highly reinforced, especially in the confining areas and near the headed stud connectors. This effect is not easy to simulate with a smeared reinforcement. Even though for the models used in the dynamic analyses it is a good solution, the model should be calibrated by comparing the response of a wall with the steel reinforcement bars modelled with, for example, LINK8 elements. However, those elements are not able to transfer shear forces, the same limitation that the smeared reinforcement has.

- The final model DYN4CONCR is able to simulate the nonlinear behaviour of the systems under analysis, including nonlinear steel, nonlinear concrete and nonlinear connection between them with the accuracy needed to perform IDA analyses with it. However, this model is computationally expensive due to the inclusion of all these nonlinearities.
- The simplified one-wall model needed about four hours to calculate the dynamic response to an artificially generated record. The DYN4 model needed more than 4 days to complete a calculation. The static analysis of the concrete model lasted three hours. The DYN4CONCR model will probably last many hours, making calculations complicated with the normally available resources. More substeps are needed when calculating with SOLID65 elements, especially when cracking and crushing start, or cracks must be opened and closed because of cyclic loads. The simplified one-wall model is interesting for checking purposes, and could be interesting to analyse the response of a real concrete SOLID65 wall to static loads, but has a different dynamic behaviour (due to a strongly different natural frequency). Simplifying the model by substituting the second, third and fourth wall by springs could be an option, but this model would not be able to transfer the vertical loads to the wall with sufficient uniformity.

6 REFERENCES

1. **European Committee for Standarisation.** *Eurocode 8: Design of structures for earthquake resistance.* 2004.

2. *A comparative study of the design spectra defined by Eurocode 8, UBC, IBC and Turkish Earthquake Code on R/C sample buildings.* **Livaoglu, Ramazan and Dogangun, Adem.** s.l. : Springer Science, 2006.

3. *A Detailed Evaluation of Eurocode and ISO Methodology on Earthquake-Resistant Geotechnical Designs.* **Wang, J. and Thusyanthan, I.** Beijing : s.n., 2008, Vol. The 14th World Conference on Earthquake Engineering.

4. **Fardis, M., et al.** *Designer's guide to EN 1998-1 and EN 1998-5.* London : Thomas Telford, 2005.

5. **Applied Technology Council.** *Next-Generation Performance-Based Seismic Design Guidelines FEMA-445.* Washington, D.C. : Federal Emergency Management Agency, 2006.

6. *A framework for performance-based earthquake engineering.* **Deierlein, G.G., Krawinkler, H. and Cornell, C.A.** 140, 2003, Vol. 2003 Pacific Conference on Earthquake Engineering.

7. *Next-generation performance based earthquake engineering.* **Whittaker, A., Huang, Y. N. and Hamburger, R. O.** s.l. : EJSE, Vol. 1st International Conference on Modern Design Construction and Maintenance of Structures.

8. *Performance-based design in earthquake engineering: state of development.* **Ghobarah, Ahmed.** s.l. : Elsevier Science Ltd., 2001.

9. *Interim Testing Protocols for Determining the Seismic Performance Characteristics of Structural and Nonstructural Components FEMA 461.* **Applied Technology Council.** Washington, D.C. : Federal Emergency Management Agency, 2007.

10. *Incremental Dynamic Analysis.* **Vamvatsikos, Dimitrios and Cornell, C. Allin.** s.l. : John Wiley & Sons, Ltd., 2002, Vol. Earthquake Engng Struct. Dyn.

11. **Meskouris, Konstantin.** *Structural Dynamics. Models, methods, examples.* Berlin : Ernst & Sohn, 2000. 3-433-01327-6.

12. *Performance Based Seismic Design.* **Priestley, M J N.** 2831, s.l. : 12 WCEE, 2000.

13. *Vulnerability / seismic performance of older RC waffle flat-plate systems in a low-to-moderate seismicity region.* **Catalán Goñi, A., Cahís i Carola, X. and Benavent-Climent, A.** Beijing : s.n., 2008, Vol. The 14th World Conference on Earthquake Engineering.

14. *Seismic hazard mitigation decisions using PBEE: Financial metrics and a case study.* **Ince, Ufuk and Meszaros, Jacqueline.** Vancouver : Pacific Earthquake Engineering Research Center, 2004, Vol. 13th World Conference on Earthquake Engineering.

15. **Lee, Won K. and Billington, Sarah L.** *Simulation and Performance-Based Earthquake Engineering Assessment of Self-Centering Post-Tensioned Concrete Bridge Systems.* Berkeley : Pacific Earthquake Engineering Research Center, 2009.

16. **ECCS.** *ECCS Manual on Design of Steel Structures in Seismic Zones.* Brussels : s.n., 1994.

17. **Rooij, Arthur Lancelot Parcifal de.** *Steel frames with precast reinforced concrete infill panels. Experimental and numerical research.* Eindhoven : Technische Universiteit Eindhoven, 2005.

18. **K. Chopra, Anil.** *Dynamics of structures.* New Jersey : Prentice Hall, 1995.

19. **European Committee for Standarisation.** *Eurocode 0: Basis on structural design.* 2002.

20. —. *Eurocode 2: Design of concrete structures.* 2004.

21. —. *Eurocode 3: Design of steel structures.* 2004.

22. —. *Eurocode 4: Design of composite steel and concrete structures.* 2004.

23. **Hamburger, Ronald O.** A Framework for Performance-Based Earthquake Resistive Design. [Online] 1997. [Cited: 1 06 2011.] http://nisee.berkeley.edu/lessons/hamburger.html.

24. *Performance-based design in earthquake engineering: state of development.* **Ghobarah, Ahmed.** 23 (878-884), Ontario : Elsevier, 2001, Vol. Engineering Structures.

25. **K. Lee, Won and L. Billington, Sarah.** *Simulation and Performance-based Earthquake Engineering Assessment of Self-Centering Post-Tensioned Concrete Bridge Systems.* Berkeley : University of California, 2009.

26. **Applied Technology Council.** *Next-Generation Performance-Based Seismic Design Guidelines.* Washington D.C. : Federal Emergency Management Agency, 2006.

27. *Composite steel and concrete structural systems for seismic engineering.* **Hajjar, Jerome F.** Minneapolis : Elsevier, 2002.

28. *INNO HYCO REFERENCE 2.*

29. **Simon, Michael.** Diplomarbeit: Untersuchungen zum Tragenverhalten von teilverdübelten Stahlverbundträgern aus hochfesten Stählen und hoch- sowie normalfesten Betonen. Aachen : Rheinisch-Westfälische Technische Hochschule Aachen (Lehrstuhl für Stahlbau), 1999.

30. **Standarisation, European Committee for.** *Eurocode 0: Basis of structural design.*

ANNEX A: PUSHOVER ANALYSES

ANNEX B: ANSYS BATCH FILE (DYN4CONCR MODEL)