Software for Architecture,
Engineering and Construction

# CYPECAD

## Calculations manual

*Analysis and design of reinforced concrete, steel, timber and aluminium structures made up of: columns, walls and shear walls; concrete, steel and composite beams; joist floor slabs (generic, reinforced, prestressed, in situ, steel, and open- web), hollow core plates, composite slabs, waffle slabs and flat slabs; mat foundations or pad footings, pile caps or foundation beams; Integrated CYPE 3D jobs (steel, aluminium and timber sections) with 6 degrees of freedom per node, including the design and optimisation of sections.*

# Contents

# 1 Calculations

## 1.1 Description of problems to resolve

**CYPECAD** has been designed to carry out the analysis and design of reinforced concrete and steel structures supporting joist floor slabs (generic, reinforced, prestressed, in-situ, steel and open web joists), hollow core plate slabs, composite slabs, waffle slabs and flat slabs in buildings exposed to vertical and horizontal loads. The beams supporting the floor slabs can be reinforced concrete, steel or composite (steel and concrete) beams. The supports can consist of reinforced concrete or steel columns, reinforced concrete shear walls, reinforced concrete walls, with or without lateral earth pressure or masonry walls (generic or concrete block walls). The foundation can be fixed (by footings or pile caps) or floating (with beams and mat foundations). Just the foundation can be analysed if only column starts are entered. The stairs are made of reinforced concrete supported on the floor slabs.

The program can be used to obtain the dimensional and reinforcement drawings of the floors, beams, columns, walls and shear walls, foundations and stairs by plotter, printer and DXF/DWG and PDF files, as well as the lists of data and analysis results. If you have CYPE 3D, you can enter integrated 3D structures with steel, timber and aluminium bars.

## 1.2 Description of the analysis carried out

The analysis of the forces is carried out by means of a spatial 3D analysis using stiffness matrices made up of all the elements defining the structure: columns, RC shear walls, walls, beams and floor slabs.

The deformation compatibility is established in all the nodes, considering 6 degrees of freedom and the non-deformability loadcase of the plane on each floor is created, to simulate the rigid behaviour of the floor slab, preventing relative displacements between nodes of this floor slab (rigid diaphragm). Therefore, each floor can only rotate and move as a whole (3 degrees of freedom).

The rigid diaphragm state for each independent zone is maintained even if only beams and no floor slabs have been entered in the floor, except for unconnected beams that are disconnected from the rigid diaphragm by the user and except for walls that are not in contact with floor slabs (from v.2012.a) (see section ***12. Rigid Diaphragm*** in this manual).

If there are separate areas on the same floor, each of these shall be considered as a separate part with regard to the non-deformability of that area and will not be considered as a whole. Therefore, the floors shall act as independent non-deformable planes. A non-connected column is considered as an independent zone.

A static analysis is carried out for all load states (except when dynamic loading is considered due to seismic loading, in which case a modal spectral analysis is used) and it assumes a linear behaviour of the materials and, therefore, a first order analysis, in order to obtain displacements and forces.

In **Integrated 3D structures**, there will always be 6 degrees of freedom per node.

Stairs also have 6 degrees of freedom; they are resolved independently, and their reactions are transmitted.

## 1.3 Discretisation of the structure

The structure is discretised into bar elements, bar and node meshes, and triangular finite elements as follows:

- **Columns**

  These are vertical bars between floors, with a node at the foundation start or on another element such as a beam or floor slab, and at the intersection of each floor, its axis being that of the cross section. Eccentricities due to the variation in height dimensions on each floor are considered.

  The length of the bar is the height or free distance to the surface of other elements on the initial and final floors.

- **Beams**

  Beams are defined in plan by fixing nodes at the intersection with the faces of supports (columns, walls or shear walls), as well as at the points where they intersect with floor elements or with other beams. This creates nodes along the axis and at the ends, and at the edge of overhangs or free ends, or in contact with other elements of the floor slabs. Thus, a beam between two columns is formed by several consecutive bars, whose nodes are the intersections with the floor slab bars. By default, they have three degrees of freedom, maintaining the load of a rigid diaphragm between all the elements in the floor. For example, a continuous beam supported by several columns maintains the rigid diaphragm load, even if it has no floor slab. Unconnected beams can be disconnected from the rigid diaphragm. See section ***12. Rigid Diaphragm***.

They can be reinforced concrete, steel, composite and timber beams, in sections selected from the library. Beams are discretised as bars whose axis coincides with the midplane passing through the centre of the vertical web, and at the height of its centre of gravity.

**Wall support simulation.** Three types of beams are defined by simulating the wall support, which is discretised as a series of supports coinciding with the discretisation nodes along the wall support, whose stiffness is increased considerably (×100). This is like a very stiff continuous beam on supports with short span lengths. The types of supports are as follows:

- **Fixed.** Displacements and rotations prevented in all directions
- **Pinned.** Prevented displacements with free rotation.
- **Pinned with free horizontal displacement.** Coarticulated vertical displacement, with horizontal displacement and free rotation

It is worth noting the effect that these types of supports can have on other elements in the structure, as vertical movement is prevented, and all the structural elements supported on them or linked to them will encounter a vertical constraint that prevents such movement. This is particularly important for columns which, if defined with external fixity, are in contact with this kind of supports, so that their load is suspended from them, and is not transmitted to the foundation, which may even produce negative values of the reactions, representing the weight of the suspended column or part of the suspended load of the wall support.

In the particular case of a pinned connection with displacement, when a beam is in continuity or prolongation of the axis of the wall support, there is a continuous fixity effect at the crown beam of the wall support, which can be noted by obtaining the loading and verifying that there are negative moments at the edge. This can be observed by obtaining the moment laws and verifying that there are negative moments at the edge. However, in practice, users must verify whether the actual conditions of the job reflect or can be met for these fixity conditions, which must be guaranteed during the execution of the job.

If the beam is not a straight extension of the wall support but deviated from its axis, a hinged effect is produced.

If the beam is a straight extension of the wall support, but the user does not want the connection to behave as a fixed connection, the connection should be pinned. The reactions on this type of support cannot be obtained.

**Foundation beams.** These are floating beams supported by elastic soil. They are discretised into nodes and bars, where the nodes are assigned a spring constant which, in turn, is defined by the subgrade modulus (see ***2. Mat foundations and foundation beams*** chapter).

- **Sloped beams**

  These are bars spanning between two points, which can be at the same elevation or floor, or at different floors and which create two nodes at these intersections. When a sloped beam joins two independent zones the non-deformable effect of the plane with rigid behaviour is not produced, as they possess six degrees of freedom without restrictions.

- **Corbels**

  Please consult section ***5. Corbels***.

- **Joist floor slabs**

  Joists are bars that are defined in the hollow floor slabs between beams or walls, that create nodes at the corresponding edge and axis intersections of the beam they cross. Double and triple joist floor slabs can be defined, which are represented by a single bar with a wider web. The geometry to which each joist floor slab is assimilated (T-section for concrete slabs, steel section for steel joist floor slabs and rectangular section for timber joist floor slabs) is defined in the corresponding floor slab data sheet. See section ***6. Joist floor slabs***.

- **Hollow core slabs**

  These are joist floor slabs discretised by bars every 40 cm. The geometrical and resistance properties are defined in a floor slab properties sheet, which may be entered by the user, creating a library of hollow core slabs.

  These may be analysed based on their construction process in an approximate manner by modifying their edge fixities using a simplified method.

- **Flat slabs**

  Flat slabs are discretised as meshes of bar type elements possessing a maximum size of 25 cm. A static condensation is carried out (exact method) of all the degrees of freedom. Deformation due to shear is considered and the rigid diaphragm hypothesis is maintained. The torsional stiffness of the elements is considered.

- **Composite slabs**

  These are joist floor slabs discretised by bars every 40 cm. They are made up of a flat slab and a ribbed deck which acts as formwork for the slab. The deck can be used so it behaves in the following ways: as lost formwork and contributing deck (Composite deck). For more information, please consult the ***9. Composite slabs*** chapter.

- **Mat foundations**

  These are “floating” solid slabs whose discretisation is identical to that of flat slabs used at floor levels and whose spring constant is defined based on the assigned subgrade modulus. Each slab can have different coefficients. (see ***2. Mat foundations and foundation beams*** chapter).

- **Waffle slabs**

  The discretisation of the waffle slab sections is carried out in meshes of bar-type elements whose size is one third of the interaxis defined between ribs in the lightened zone, and whose bending inertia is (in both the solid and lightened zones) half that of the solid zone, and the torsional inertia is twice that of the bending inertia.

  The dimensions of the mesh are kept constant for both solid and lightweight zones, adopting the previously indicated mean inertias at each zone. Deformation due to shear is considered and the rigid diaphragm hypothesis is maintained. The torsional stiffness of the elements is considered.

- **Reinforced concrete shear walls**

  These are vertical elements with any transverse section composed of multiple rectangles between each floor and defined by an initial and final floor. The dimensions of each side are constant in height; however, their thicknesses may be modified.

  For walls (or shear walls), one of the transverse dimensions of each side must be at least five times greater than the other dimension. If this condition is not verified, its discretisation as a finite element is not adequate and can be considered as being a column and treated as a linear element.

  Both beams and slabs can be connected to the sides of the walls at any position or direction, by means of a beam with a width of the thickness of the span and a constant depth of 25 cm. The nodes do not coincide with the nodes of the beam.

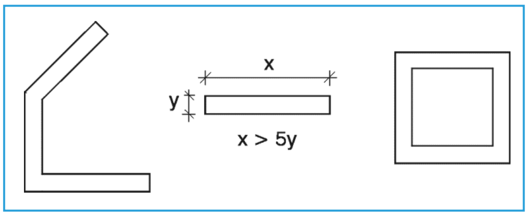



Figure 1. Shear wall examples

- **Reinforced concrete walls, masonry walls and reinforced concrete block walls**

  These are vertical elements with any transverse section composed of multiple rectangles between each floor and defined by an initial and final floor. The dimensions of each side can be different at each floor and their thickness can be reduced at each floor.

  For walls, one of the transverse dimensions of each side must be at least five times greater than the other dimension. If this condition is not verified, their discretisation as finite elements is not adequate and can be considered as being columns and treated as linear elements. Both beams and slabs can be connected to the sides of the walls at any position or direction. All nodes that are generated correspond to one of the nodes of the triangular finite element mesh of the wall.

  The discretisation is carried out by 3D thick-sheet finite elements which consider shear deformation. They are composed of six nodes, at the vertices and at the mid-points of the sides, each with six degrees of freedom. The wall is discretised into a triangular mesh adjusted to the dimensions of the wall, its geometry, openings and with a more refined mesh in critical zones, which reduces the size of the elements at angles, edges and singularities.

  Walls which are not in contact with any floor slabs are not considered as rigid diaphragms at that floor level. Please consult section ***12. Rigid Diaphragm***.

- **Stairs**

  Stairs are discretised using triangular thick shell finite elements, for both sloping and horizontal spans. The start and end supports are discretised as a floor slab simulation by means of a beam with elevated stiffness, and the intermediate supports by means of elastic supports simulating real masonry walls or ties. The loadcases considered are only those corresponding to gravitational loads, dead and live loads.

  Once the reactions have been established after a design process (carried out independently), they are integrated at the supports and represented as line loads, which are applied to the structure at its connection points and then included in the analysis of the complete structure. Their design has not been integrated in the analysis procedure due to the great impact of horizontal loads on them.

## 1.3.1 Node sizing

A group of general nodes with finite dimensions is created at column axes and at intersections of slab elements with beam axes. Each general node has one or several associated nodes. The associated nodes are formed at intersections of slab elements with beam faces and column faces, and at intersections of beam axes with column faces.

Given that they are related due to their deformation compatibility with the assumed flat deformation, the general stiffness matrix and associated matrices can be resolved and hence obtain the displacements and forces in all the elements.

As an example, the discretisation would be carried out as shown in the diagram below. Each finite dimension node can have several or no associated nodes, but there will always be one general node.

Given that the program does consider the size of the column, and supposing a linear behaviour within the support, with flat deformation and infinite stiffness, the deformation compatibility is considered.

The bars defined between the axis of column ① and its faces are considered as being infinitely rigid.

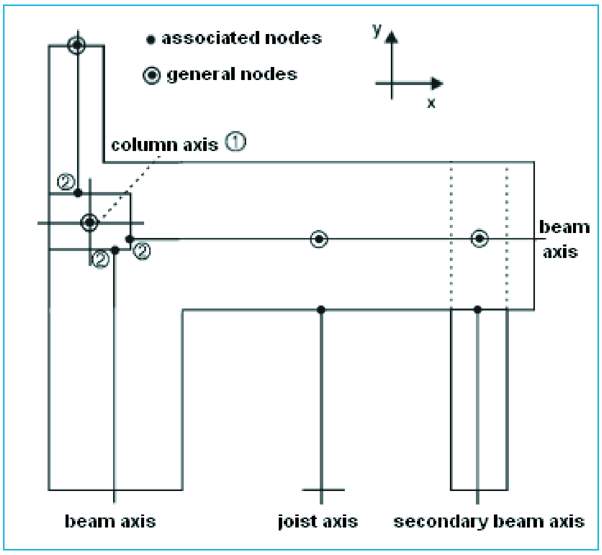



Figure 2. Discretisation of the structure

$\delta_{z1}$, $\theta_{x1}$, $\theta_{y1}$ are considered as being the displacements of column ① and $\delta_{z2}$, $\theta_{x2}$, $\theta_{y2}$ are considered as being the displacements of point ②, which represents the intersection of the beam axis with the face of the column. $A_x$ and $A_y$ are the coordinates of point ② relative to point ①.

The following conditions are met:

$$\delta_{z2} = \delta_{z1} - A_x \cdot \theta_{y1} + A_y \cdot \theta_{x1}$$

$$\theta_{x2} = \theta_{x1}$$

$$\theta_{y2} = \theta_{y1}$$

The size of the beams is considered in the same way, considering their deformation to be flat.

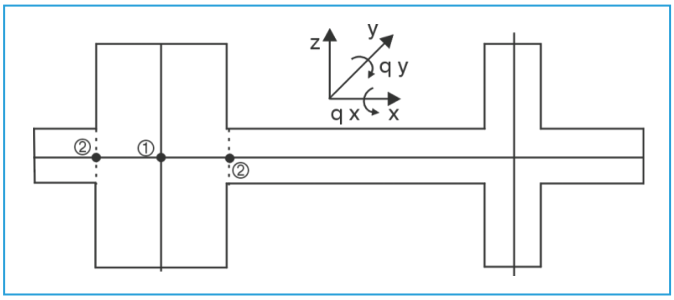



Figure 3

*The structural model defined by the program responds to the data entered by the user, so special attention must be paid to ensure that the geometry entered matches the type of element chosen and its adaptation to reality.*

*Particular attention is brought to those elements considered as linear in the program (columns, beams, joists) when they may not be in real life, resulting in elements with two-dimensional or three-dimensional behaviour whereby the analysis criteria and reinforcement provided by the program will not be adjusted to the real design of these elements.*

*Examples of where this may occur include corbels, wall-beams and plates, situations which may occur in beams, slabs which are really beams, or columns or short shear walls not complying with the geometric limits between their transverse and longitudinal dimensions. For these cases, users must carry out the necessary manual corrections after the analysis, so the results of the theoretical model are adapted to their physical reality.*

### 1.3.2 *Force envelopes at supports*

The *1990 CEB-FIP Model Code*, which inspired the *Eurocodes*, is considered. When quoting the design effective span, *article 5.2.3.2.* indicates the following:

*"Usually, the span l has to be entered as the distance between adjacent support axes. When reactions are located significantly away from the axis of support, the effective span has to be calculated considering the real position of the support section.*

*In the overall analysis of frames, when $l_{ef}$ is less than the distance between axes of columns the dimensions of the joints should be taken into account by entering rigid elements between the centroidal axis of the column and the end section of the beam."*

Generally, the reaction at the support is eccentric due to the presence of an axial force and a moment being transmitted to the support, the size of the nodes is considered when rigid elements have been entered between the axis of the support and the end of the beam. This is reflected in the considerations detailed below.

A linear reply is assumed within the support that represents the reaction of the loads transmitted by the lintel and those applied at the node which have been transmitted by the rest of the structure.

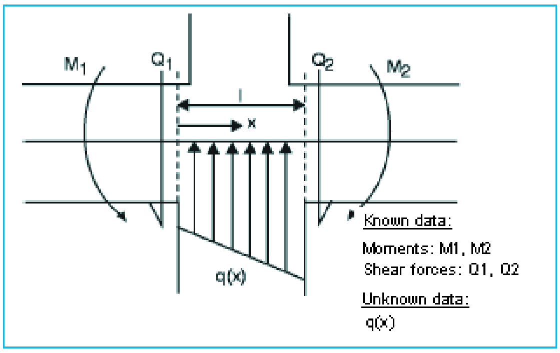



Figure 4

The following statement exists:

$$Q = \frac{dM}{dx} \qquad q = \frac{dQ}{dx}$$

The moment equations reflect, generally, a cubic parabolic curve with the format:

$$M = ax^3 + bx^2 + cx + d$$

The shear force is the corresponding differential equation:

$$Q = 3ax^2 + 2bx + c$$

By assuming the following conditions:

$x = 0 \qquad Q = Q_1 = c$

$x = 0 \qquad M = M_1 = d$

$x = 1 \qquad Q = Q_2 = 3al^2 + 2bl + c$

$x = 1 \qquad M = M_2 = al^3 + bl^2 + cl + d$

a system of four equations with four unknown variables, which can easily be resolved, is obtained.

The force diagrams are as follows:

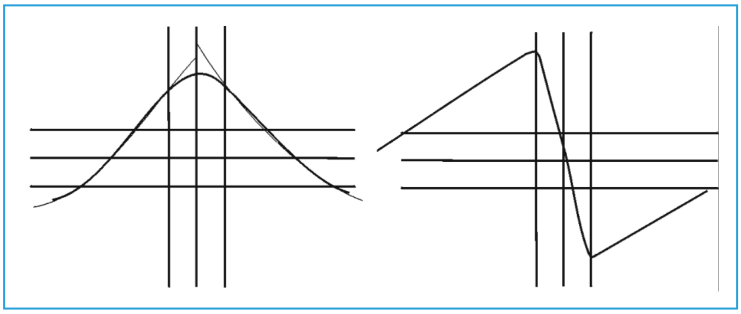


Figure 5. Bending moment diagram   Shear force diagram

These assumptions were taken on by several authors (Branson, 1977) and are related with the debate on the design span and free span and the way in which it is contemplated in various codes, as well as whether the design moment is calculated taking the distance between support axes or support faces.

The structure is being idealised into linear elements, with a length to be determined by the real geometry of the structure. In this sense, the size of the columns should be considered.

It is important to remember that, to consider an element as being linear, the beam or column is to have a span or element length greater than a third of its mean depth and greater than four times its mean width.

*Eurocode 2* allows for moments to be reduced at the supports; the reduction depending on the support reaction and width:

$$\Delta M = \frac{\text{reaction} \cdot \text{support width}}{8}$$

If it is to be executed as a single element between supports, the design moment can be taken as that of the support face and no less than 65% of the support moment, assuming a perfect fixed connection at the faces of the rigid supports.

In this sense, it is worth mentioning the *Argentinean codes*: *C.I.R.S.O.C.* which are based on the *German D.I.N.* codes which allow for the parabolic rounding of the force diagrams depending on the size of the supports.

Within the support, the depth of the beams is considered to increase in a linear manner, in accordance with a 1:3 slope, up to the axis of the support. By considering the size of the nodes, the parabolic rounding of the force diagrams and the increase of depth within the support, a more economic longitudinal reinforcement solution is obtained for bending in beams; as the maximum steel area occurs between the face and axis of the support, the most usual case being at the face depending on the geometry that has been entered.

In the case of a beam supported by a long element such as a wall or shear wall, the moment diagrams will be extended at the support as of the face of the support for a length equivalent to that of the depth of the beam, designing the reinforcement up to this length and not extending it past the point where it is not required. Even if the beam has a greater width than the support, the beam and its reinforcement are interrupted once it has penetrated a distance of one depth into the shear wall or wall.

## 1.4 Analysis options

An ample series of structural parameters of great importance for obtaining forces and for the design of elements can be defined.

Given the vast number of options available, it is recommended the help explanations of the options be consulted.

The general options for beams, floor slabs, joints are located in **Beam Definition > Job**. The reinforcement tables and specific options for each element can be found in **General data > By position** icon (located to the right of Steel: Bars). The most significant of these are mentioned below.

### *1.4.1 Redistributions considered by the program*

#### *Negative moment redistribution coefficients*

A negative moment redistribution of up to 30% is accepted in beams in joists. This parameter can be optionally established by the user; however, it is recommended it be set at 15% for beams and 25% for joists (default values). This redistribution is carried out after the analysis.

By considering a moment redistribution, the resulting reinforcement is more expensive but is safer and easier to execute on site. However, it must be noted that an excessive redistribution produces deflections and cracking which are incompatible with the internal partitions.

A redistribution of 15% in beams produces generally acceptable results and can be considered as the optimum value. In slabs it is recommended a redistribution factor of 25% be used, which is equivalent to approximately equalising the positive and negative moments.

The moment redistribution is carried out using the moments at support surfaces, which in the case of columns, is at their faces, i.e. the free span is affected. The new moment values within the support are calculated using the moments redistributed at the face and taking into account the rounding of the force diagrams indicated in the previous section.

Additionally for beams and floor slabs, besides the moment redistributions, users can define the minimum positive and negative moments specified by the code in use.

## *Fixity coefficient at the last floor*

Optionally, the negative moments can be redistributed at the top of the last span of a column at the end of the beam; this value will be between 0 (pinned) and 1 (fixed), although 0.3 is recommended as an intermediate value (default value).

The program undertakes a linear interpolation between the stiffness of bars fixed at either end and those with one end fixed and the other pinned, which affect the EI/L terms of the matrices of the last span of the column:

$$K\ \text{final} = \alpha \cdot K\ \text{fixed-fixed} + (1 - \alpha) \cdot K\ \text{pinned-fixed}$$

Where $\alpha$ is the value of the entered coefficient.

## *Fixity coefficient at the top and bottom of columns, at slab, beam and wall surfaces; pinning of beam ends*

A fixity coefficient can also be defined for each column span at its start and/or end (0 = pinned; 1 = fixed) (default value). The coefficients corresponding to the top of the column at its final span are multiplied by these. This plastic joint is physically considered at the connection point of the top or bottom of the column with the beam or flat/waffle slab reaching the node.

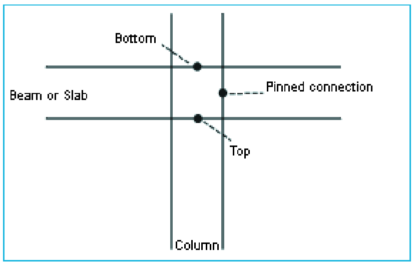



Figure 6

It may occur that absurd results, even mechanisms, be obtained at beam ends and at tops of columns when very small coefficients are used due to the presence of two hinges being connected by rigid spans.

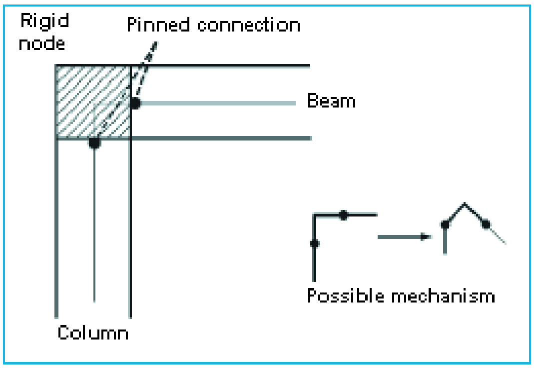



Figure 7

A fixity coefficient can be defined when using flat slabs, joist floor slabs and waffle slabs at their supported edges. The value may oscillate between 0 and 1 (default value).

A fixity coefficient whose value may vary between 0 and 1 (default value) may be defined at beam edges, in a similar way as for slabs only here for one or several edges, due to it being specified using a beam.

When fixity coefficients are defined simultaneously at beam and slab edges, these are multiplied by one another to obtain the resulting coefficient to apply at each edge.

The defined plastic joint is created at the edges of slabs and at the edge of beams and walls, and is not effective at column and shear wall contact edges where the connection is always taken as fixed. A rigid bar is defined between the support and the axis, and so there is always a moment present at the axis produced by the shear at the edge due to its distance to the axis. This moment becomes a torsional moment if the reinforcement is not continuous with that of adjacent panels. This option should be used with caution, because if the edge of a web is hinged on a beam, and the beam has reduced torsional stiffness to a very small value, without being a mechanism, it can give absurd results of the displacements of the panel at the edge, and therefore, of the design forces.

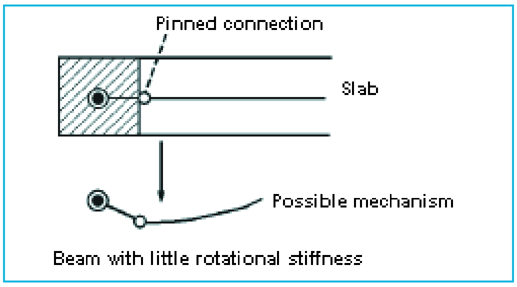



Figure 8

Joints and partial embedment coefficients can also be defined at beam ends, materialising physically on the face of the support, whether it is a column, wall, shear wall or wall support.

These redistributions are taken into account in the analysis and therefore affect the final displacements and forces that are obtained.

### *1.4.2 Considered stiffnesses*

To obtain the terms of the stiffness matrix, the gross section of all concrete elements is considered.

When calculating the terms of the stiffness matrix of the elements, the following values have been distinguished:

**EI/L**: Bending stiffness
**GJ/L**: Torsional stiffness
**EA/L**: Axial stiffness

And the coefficients indicated in the following table have been applied:

| **Element** | **($EI_y$)** | **($EI_z$)** | **(GJ)** | **(EA)** |
|---|---|---|---|---|
| Columns | G.S. | G.S. | G.S. · x | G.S. · axial stiffness coeff. |
| Sloped beams | G.S. | G.S. | G.S. · x | G.S. |
| Steel or concrete beams | G.S. | ∞ | G.S. · x | ∞ |
| Joists | G.S. | ∞ | G.S. · x | ∞ |
| Limit beam | G.S. · $10^{-15}$ | ∞ | G.S. · x | ∞ |
| Wall support (external fixity beam) | G.S. · $10^{2}$ | ∞ | G.S. · x | ∞ |
| Shear walls and walls | G.S. | G.S. | F.E. | G.S. · axial stiffness coeff. |
| Flat and waffle slabs | G.S. | ∞ | G.S. · x | ∞ |
| Hollow core plates and composite slabs | G.S. | ∞ | G.S. · x | ∞ |

G.S.: Concrete gross section
∞: Not considered due to the relative non-deformable shape of the floor
x: Torsional stiffness reduction coefficient
F.E.: Flat finite element

### 1.4.3 Torsional stiffness coefficients

There is an option for defining a reduction coefficient for the torsional stiffness (x) of the different elements (see table above). This option is not applicable to steel sections. When the dimension of the element is less than or equal to the value defined for short bars, the coefficient defined in the options will be taken. The gross section (G.S.) will be considered for torsion GJ, and for when it is required to achieve equilibrium of the structure.

Consult the default values in **Job** > **General options** > **Torsional stiffness reduction coefficients**.

### 1.4.4 Axial stiffness coefficient

The axial shortening in columns, walls and reinforced concrete walls affected by an axial stiffness coefficient varying between 1 and 99.99 is considered in order to simulate the effect of the construction process of the structure and its influence on the final forces and displacement. The recommended value is between 2 and 3, with 2 being the default value.

### 1.4.5 Minimum moments

A minimum moment that is a fraction of the assumed isostatic $pl^2/8$ can also be covered for beams. This minimum moment can be defined for both negative and positive moments with a $pl^2/x$ format, where x is a whole number greater than 8. The default value is 0, i.e. they are not applied.

At least one reinforcement capable of resisting a $pl^2/32$ moment in negative and a $pl^2/20$ moment in positive is recommended. These minimum moment considerations can be made for the whole structure or only for part of it and may be different for each beam. Each code usually indicates the minimum values.

Similarly, minimum moments can be defined for joist floor slabs and hollow core plates. They can be defined for the entire job or for individual panels and/or different values. A value of ½ of the static moment ($=wl^2/16$ for a uniform load) is reasonable for positive and negative moments. We advise users to consult the **Options**.

The bending moment envelopes will be displaced so that these minimum moments are met, followed by the negative moment redistribution that is to be applied.

The equivalent value of the applied line load is:

$$w = \frac{V_l + V_r}{l}$$

If a minimum moment (+) = has been considered, the following must be verified:

$$M_V \geq \frac{wl^2}{8}$$

If the applied minimum moment is less than the design moment, the greatest of the two is taken.

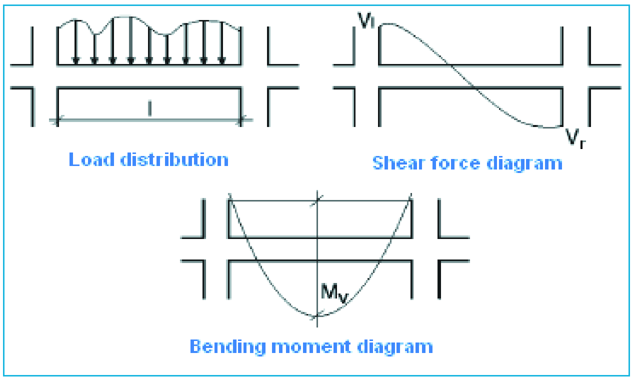



Figure 9

Note that these considerations work correctly with line loads and approximately with point loads.

## 1.4.6 *Other options*

Listed below are the unquoted options of the application which, of course, influence and customise the analyses.

### *Columns*

- **Vertical reinforcement layout** (maximum lengths, connection of short spans, intermediate overlaps). The maximum length of a bar (8 m by default) requires overlaps to be made if any section exceeds this value.

  The maximum connection length for short spans (default value of 4 m) is activated when the elevation difference between floors is small. The spans are joined and intermediate overlaps at floor level eliminated until the indicated length is reached but not exceeded. The process is applied from the top to the bottom of the column, provided that the reinforcement is identical.

The overlap at each floor level, in the case of disconnected columns, can be avoided by maintaining the reinforcement without overlapping until the next floor, or by overlapping at all floors, even if no beam reaches the column on that floor, provided that the reinforcement is identical.

- **Cutting starter bars at the last span (top of column)**. This option cuts the starter bars at the top of the column's last span. Only drawings and reinforcement take-off values are affected by this option. The process is not analysed and so care must be taken. It is recommended the fixity coefficient at the top floor be reduced to a minimum, together with the activation of the reduction of the anchorage lengths on the top floor. However, it may still be necessary to bend the ends with large diameters by calculation, but this option will cancel it out.

- **Reducing anchorage lengths in columns.** The reduction of anchorage lengths at start level at intermediate floors (deactivated by default) and at the last floor (activated by default) can be activated or deactivated, reducing, being reduced according to the ratio of the real stress in the reinforcement to the maximum stress. In this case, the columns containing reinforcement of the same diameter will result in having starter bars of a different length as a result of the analysis and therefore cannot be matched. If this is not desired, deactivate the option, even though slightly larger bends will be obtained at the last floor.

- **Reinforcement symmetry criteria in faces.** In the reinforcement tables, the reinforcement can be defined as different or equal in the X and Y faces. The result of the analysis is to check and obtain the first reinforcement sequence in the table that meets all the analysis combinations, selecting also the first one that has symmetrical reinforcement on all four faces. If the amounts are calculated in both cases and compared in percentage difference, the one that meets the criteria marked in % difference of the option (0% by default, i.e. not symmetrical) will be selected. If symmetry is desired, set a high value, e.g. 300.

- **Bar continuity criteria.** Columns are analysed by span starting at the top and moving downwards whereby, in usual circumstances, their reinforcement increases in diameter as the column descends. However this does not always occur due to the results provided by the program being those obtained from analysing the forces acting on the column and its dimensions. Using this option, one can force the program to maintain the reinforcement; the bar diameter at the corners and faces, as well as the number of bars and apply it from the last or penultimate floor to the first span. This provides results with less discontinuity in the reinforcement.

  The program applies continuity criteria applicable to corner bar diameters as of the penultimate floor.

- **Geometric cover.** Distance between the external surface to the first reinforcement element; the stirrups (the default value depends on the selected code).
- **Steel column layout.** The user can opt to reduce the size of the entered section, if possible, or maintain and check it. It is important to bear in mind that the force analysis is undertaken using the section that has been entered. If this is modified and the change in dimensions is large, it is recommended the structure be reanalysed to take into account this change. During the analysis, the program locates the most economic section.
- **Transitions due to dimension variation.** When the change in section of a column from one floor to the next is large, the vertical reinforcement must be bent. The angle at which the reinforcement is bent is limited. If these geometric conditions are exceeded, the reinforcement must be cut and anchored in the bottom span and new starter bars placed corresponding to the reinforcement of the span above. The bending angle depends on the depth of the slab or beam the column reaches.
- **Rounding of bar lengths.** Bars are commonly cut so their lengths are a multiple of a value (default value 5 cm) so to ease work on site and fabrication.
- **Hatching of columns and shear walls.** These are symbols which graphically represent if a column starts, continues to another floor or ends. The user may choose which to apply.
- **Reinforcement splices at central span zones.** For seismic zones, the reinforcement overlaps are moved to the central part of the span, far from the areas submitted to the maximum forces. The default setting of this option is set as deactivated, however it is recommended it be activated where high seismic activity is present.
- **Overlaps in walls and shear walls.** Verifies that the overlap is in compression or in tension, applying an amplification coefficient to the splice length which depends on the bar separation.
- **Required compliance factor for walls and shear walls.** The reinforcement of a wall or shear wall span may contain peak stresses which penalise the reinforcement if the user intends for the wall to pass with a 100% compliance factor. Using this option, a smaller % compliance can be checked or alternatively, the reinforcement may be modified and check its compliance factor. During the analysis, the reinforcement is designed in such a way that a compliance factor is at least the default value of 90%. It is convenient that this value be checked and if less than 100% see at which points the reinforcement fails and if local reinforcement is necessary.
- **Stirrup layout.** It is convenient stirrups be placed at intersections with slabs or beams (the default setting of this option is set as activated), also at the top and bottom of

columns at an established height and with a smaller separation than in the rest of the column (default setting = deactivated). It is recommended it be activated for structures in seismic zones.

- **Column start options**
- **Minimum ratios**
- **Corbel options**

### *Beams*

Below are options relative to beams.

- Symmetrical top reinforcement in single span beams
- Percentage difference for symmetrical top reinforcement
- Hook layout criteria
- Hooks at ends of alignment
- Minimum stirrup length
- Symmetry in stirrup reinforcement
- Stirrups of different diameters in a beam
- Anchorage length in stirrup closure
- Bend hooks in U
- Multiple stirrup layout
- Prefabricated beam reinforcement
- Prestressed beam stirrups
- Beam detailing for seismic design
- Concrete cover in beams (top, bottom and lateral)
- Concrete cover in foundation beams (top, bottom and lateral)
- Properties of precast beams*
- Prestressed beam library
- Error evaluation
- Frame ordering criteria
- Beam numbering criteria
- Assembly reinforcement
- Joining of assembly reinforcement in overhangs

- Shear envelopes (continuous or discontinuous)*
- Shear reinforcement (provision of skin reinforcement, section for shear check)*
- Stirrup selection*
- Cracking*
- Minimum foundation beam ratios
- Beam reinforcement within walls and crown beams

### *Flat slabs, composite slabs and waffle slabs*

- Flat slab and waffle slab reinforcement
- Minimum steel areas
- Mechanical steel area reduction
- Torsion reinforcement
- Minimum reinforcement bar lengths
- Flat slab cover
- Waffle slab cover
- Detail base reinforcement in drawings (default option = deactivated). It is not drawn or measured if deactivated
- Rounding of bar lengths
- Hooks in flat slabs
- Order and numbering criteria in flat slabs
- Rectangular flat slab reinforcement
- Cover in foundations
- Cover in joist floor slabs, hollow core slabs and composite slabs

### *Stairs*

- Match reinforcements
- Starter bars at start and dowel bars at end
- Reinforcement layout
- Geometric cover
- Anchorage length in slab
- Depth of foundation

### *Beam and slab general options*

- General drawing options*
- Maximum bar cut off length*
- Minimum steel areas for top reinforcement in joist floor slabs
- Minimum steel areas for top reinforcement in hollow core plates
- Reinforcement in usual slabs
- Reinforcement in hollow core plate slabs
- Minimum moments to cover with reinforcement in slabs and beams
- Girder (beam) reinforcement
- Torsional stiffness reduction coefficient in joist floor slabs
- Consideration of torsional reinforcement in beams
- Options for steel beams and joists
- Deflection limits in beams, joists, hollow core plates and composite slabs
- Active deflection and total long term deflection–Constructive process
- Shear in in-situ joist floor slabs
- Buckling coefficients for sloped beams
- Buckling coefficients for diagonal braces

### *Footings and pile caps*

- General and specific options

### *Strap and tie beams*

- General and specific options

### *Drawing*

- Layer, text size and pen thickness configuration for drawings*

There are options which are saved and conserved with the job. Others (*) are general options and may therefore vary from job to job.

To recover the default options, an “empty” installation must be completed, without the existence of the USR directory. This way all the default options and tables will be installed.

Some options contain a button offering to install the default settings which allows for them to be recovered directly without having to execute the “empty” installation.

# 1.5 Loads to consider

## *1.5.1 Vertical loads*

### *Permanent loads*

Self weight of reinforced concrete elements, calculated using its volume based on its gross section and multiplied by 2.5 (specific weight of concrete) for columns, shear walls, walls, beams and slabs.

The self weight of a slab is defined by the user upon choosing the type of slab, which can be different for each floor or panel, depending on the selected type. For flat slabs, it will be calculated by multiplying its depth by 2.5, the same is applicable to drop panels of waffle slabs. For lightweight zones of waffle slabs and joists floor slabs, the value indicated by the user in the data sheet of the selected slab will be taken. In the case of joist floor slabs, the value of the weight per square metre is multiplied by the rib spacing resulting in a line load applied to each joist. In flat and waffle slabs, the product of the weight by the tributary area of each node is applied to each node.

### *Dead loads*

These are estimated as being uniformly distributed on the floor. These are elements such as flooring and partitions (even though this last element could be considered as being a variable load if its position or presence varies with time).

The self weight of the structural elements plus the dead loads make up the **Permanent loads**. These are automatically applied to the bars of the structure.

### *Variable loads (live load)*

The applied live load is considered as being uniformly distributed on the floor. It is applied automatically to the bars of the structure making up the floors of each floor.

### *Additional loadcases (special loads)*

As indicated, **CYPECAD** generates automatic loadcases, such as **Permanent loads** (composed of the self weight of the construction elements and the dead loads entered on each group on all floors), the **Live load** (defined for each group for all the surface of the floor), the **Wind load** (generated automatically for each X, Y direction depending on the selected code and the defined structure dimensions), and the **Seismic load** which depends on the selected code.

Additional loadcases may be added to those generated automatically, which in previous version were referred to as **Special loads**, and may be established as **Dead loads** or **Live loads** regardless of whether they are point, line or surface loads.

So, additional loadcases of a different nature may be created (dead load, live load, wind, earthquake and snow) and combine them with those that have been previously created automatically and amongst themselves (this is not compatible with wind and seismic loads).

Additional loadcases associated to **Lateral soil pressures** and **Accidental loads** can also be defined.

Different load distributions can be created for each loadcase, creating groups which can be combined establishing whether they can act simultaneously by assigning them as compatible, incompatible or simultaneous.

When additional loadcases are created, the user can define whether or not they are compatible with each other.

Once all the loadcases have been defined, along with the load distribution, simultaneity and combination modes (in accordance with the selected codes, materials used and use category of the building), the combinations for all the **Limit States** are generated automatically, from material failure, bearing pressures to node displacements. A fire check can also be carried out.

All this is configured in the *Loads* section of the *General Data* dialogue box. The same can be applied to the Integrated 3D structures.

## *Vertical loads on columns*

Loads (N, Mx, My, Qx, Qy, T) can be defined acting at the top of any column. These are in reference to the general axes and can be defined for any loadcase, additional to those obtained from the analysis. The diagram below indicates the positive direction of the loads:

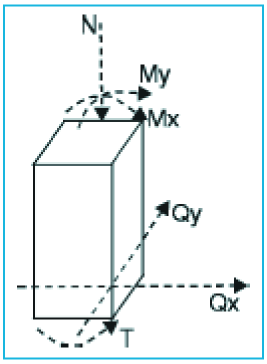



Figure 10

Columns or starts with the applied loads can be entered on mat foundations or foundation beams so to simplify an analysis.

### *Horizontal loads on columns*

Point loads and uniform loads along the whole height of the column can be applied. They can be applied in reference to the local axes of the column or to the general axes of the structure.

## 1.5.2 Horizontal loads

### *Wind loads*

The program automatically generates the horizontal loads to be applied at each floor, in accordance with the selected code, in two orthogonal directions X, Y, or in a single direction, and for the two cases (+X, –X, +Y, –Y). A load coefficient can be defined for each wind direction and case. If a building is isolated the pressure will act on the windward face of the building and the suction on the leeward side. It is usually estimated that the pressure is 2/3 = 0.66 and the suction 1/3 = 0.33 of the total pressure. Therefore for an isolated building, the load coefficient is 1 (2/3 + 1/3 = 1) for each direction. If the building is protected on its left side because of an adjacent building in the X direction, the wind coefficients may be modified to reflect the situation. In this case +X = 0.33 as there is only suction in the leeward direction and –X = 0.66 as there is only pressure in the windward direction:

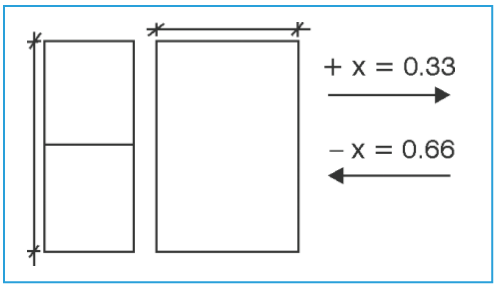



Figure 11

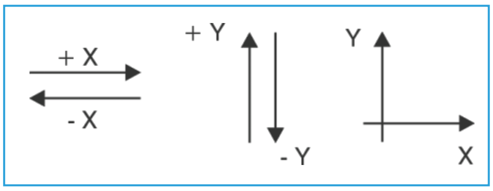



Figure 12

The tributary width is defined as the façade length perpendicular to the direction of the wind. A different value can be entered for each floor. When the wind acts in the X direction, the y length of the building is to be provided and when it acts in the Y direction, the x length of the building is to be provided.

When there are independent areas on the same floor, the load is distributed proportionally to the width of each zone with respect to the total width B defined for that floor.

B is the tributary width defined when the wind acts in the Y direction. $b_1$ and $b_2$ are calculated geometrically by CYPECAD in accordance with the coordinates of the edge columns of each zone. Therefore the tributary widths applied to each zone will be:

$$B_1 = \frac{b_1}{b_1 + b_2} \cdot B \qquad B_2 = \frac{b_2}{b_1 + b_2} \cdot B$$

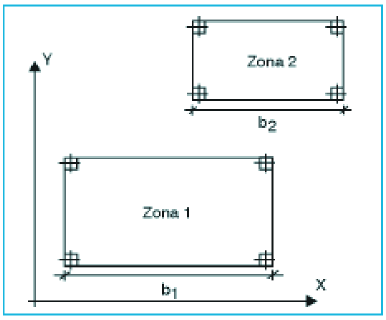



Figure 13

Once the tributary width of one floor is known, and the heights to the floor above and the floor below, if the half sum of these two heights is multiplied by the tributary width, the surface exposed to wind on that floor is obtained. If this is then multiplied by the total pressure calculated at that height and by the load coefficient, the wind load for that floor and in that direction will be obtained.

If there are guardrails (or a solid perimeter wall) on the roof, it can be taken into account by proportionally modifying the band width b and using b'.

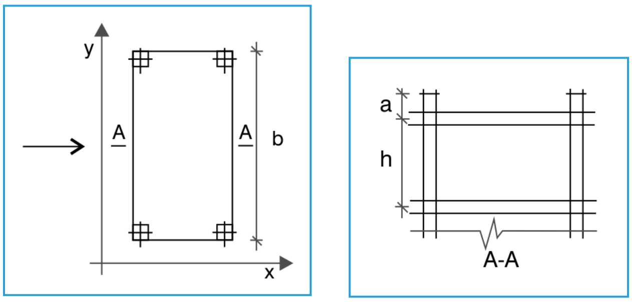



Figure 14

$$b' = b \cdot \frac{a + h/2}{h/2} = b \cdot \frac{2a + h}{h}$$

As a generic method for obtaining the wind loads in an automatic manner, select the **Generic code** option.

Having defined the directions in which the wind acts, the load coefficients and tributary widths per floor, the pressure curve must be selected. A library of pressure curves exists which allows for existing pressure curves to be selected and create new ones. These curves display the total pressure as a function of the building's height. These values are interpolated for intermediate heights, which is necessary to calculate the pressure at the height of each floor of the building.

The **Shape factor** is a coefficient which is applied to the building to correct the wind load depending on the shape of the building. This may be due to the shape of its floors being rectangular, circular, etc. and because of its slenderness.

A **Gust factor** can also be defined. This is a coefficient which amplifies the wind load so to take into account the geographical position of the building in exposed locations such as valleys, hill-sides etc. These situations produce greater wind speeds and so should be taken into account.

The total wind load applied to each floor is obtained by multiplying the pressure at its height by the exposed surface, shape and gust factors. The application point of the load at each floor is at the geometric centre of the floor, determined by the perimeter of the floor. The value of the wind load applied at each floor can be consulted and displayed in a report.

For each defined code, the pressure is calculated automatically, once the initial data has been entered which can be consulted within the code to be used.

In the case of **Integrated 3D structures**, wind loads are not generated automatically. They must be entered manually on the nodes and bars. If additional loadcases are defined, a combination can be created with the automatic loads.

*It is important the loadcase combinations be checked as well as their compatibilities when a CYPE 3D job is imported as an integrated structure, especially, if the wind loads of the job had already been generated previously using the Portal Frame Generator.*

## *Seismic loads*

Two types of general analysis methods can be used for seismic loads: static analysis and dynamic analysis.

It is possible to apply both general methods or the specific methods indicated in the code or the application regulations depending on the location of the building.

**Static analysis. Seismic loads using coefficients.** Seismic loads can be entered as a system of static forces equivalent to the dynamic loads, generating horizontal loads in two orthogonal directions X and Y, applied at each floor level, at their centre of gravity.

As a general method, the seismic loads can be applied by floor coefficients.

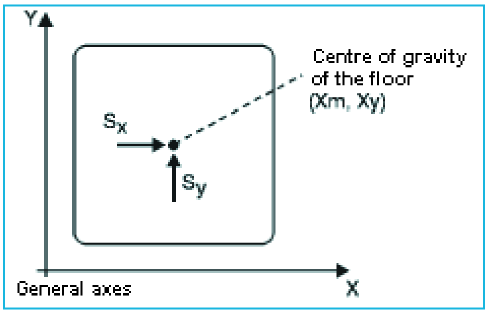



Figure 15

The static forces to apply in each direction, per floor, are:

$$S_x = (G_i + A \cdot Q_i) \cdot C_{xi}$$

$$S_y = (G_i + A \cdot Q_i) \cdot C_{yi}$$

Where:

$G_i$: The permanent loads of floor i

$Q_i$: The variable loads of floor i

A: Live load, snow load or quasi-permanent load simultaneity coefficient

$C_{xi}$, $C_{yi}$: Seismic coefficient for each direction at floor i, also known as "Seismic action in X or Y" in the data entry part of the dialogue box. The seismic mass of each floor is multiplied by this coefficient to obtain the static force applied at each floor.

The displacements of the floor with respect to the general axis are:

$$\bar{\delta}\begin{cases}\delta_{xp}\text{ :displacement in X of the floor}\\ \delta_{yp}\text{ :displacement in Y of the floor}\\ \delta_{zp}\text{ :rotation about Z of the floor}\end{cases}$$

and the applied forces:

$$\bar{F}\begin{cases}F_x = S_x\\ F_y = S_y\\ M_z = -S_x \cdot Y_m + S_y \cdot X_m\end{cases}$$

$$\bar{F} = K \cdot \bar{\delta}$$

The second order effects may be considered if the user wishes for them to be so.

Within the integrated 3D structures, if the static earthquake loadcase is activated as loads on nodes and bars, it cannot be combined with seismic loads by coefficients or with dynamic seismic loading.

Similarly, if a static analysis using floor coefficients is to be carried out in CYPECAD, it cannot be carried out for the integrated 3D structures, hence, the structure cannot be analysed. It may be analysed if a joint dynamic analysis is undertaken. An additional static seismic force loadcase could be activated, but the automatic loadcases would have to be deactivated.

**Dynamic analysis. Modal spectral analysis.** The dynamic analysis method which is generally considered by the program is the modal spectral analysis, for which the following parameters must be indicated:

- Design acceleration with respect to g (acceleration due to gravity) = $a_c$
- Ductility of the structure = $\mu$
- Number of modes to analyse
- Live load quasi-permanent coefficient = A
- Design acceleration spectrum

These are to be completed and the corresponding spectrum selected from the default library provided with the program or using another created by the user. Each spectrum is defined by coordinates (X: period T; Y: spectral ordinate a (T)) therefore allowing to view the generated graph. For the definition of the normalised elastic response spectrum, the user must know the factors that influence it (type of earthquake, type soil, damping, etc.). These factors must be included in the spectral ordinate, also known as the amplification factor and referred to the period, T.

When any type of dynamic earthquake loadcase is specified for a building, the program carries out, as well as the normal static design for gravitational loads and wind, a modal spectral analysis of the structure. The design spectrums depend on the earthquake code and the parameters of the code that have been chosen. In the case of the modal spectral analysis, the user directly indicated the design spectrum.

To carry out the dynamic analysis, the program creates the mass and stiffness matrices for each element of the structure. The mass matrix is created based on the self weight loadcase and the corresponding live loads multiplied by the quasi-permanent coefficient. CYPECAD works with concentrated mass matrices, resulting to be diagonal.

The following step consists in condensing (simultaneous with the element assembly) of the complete stiffness and mass matrices of the structure, to obtain other reduced matrices which only contain the dynamic degrees of freedom, with which the modal decomposition shall be undertaken. The program carries out a static and dynamic condensation, whereby the dynamic condensation is attained by means of the simplified classic method, and only by means of the dynamic degrees of freedom will the forces of inertia appear.

The dynamic degrees of freedom that are worked with consist of three per floor of the building: two displacements in the horizontal plane and the corresponding rotation of the plane. This simplified model corresponds to that recommended by the vast majority of earthquake codes. Hence when defining the number of modes, the user is recommended to define three for each floor of the building.

At this point in the analysis the stiffness and mass matrices, both reduced and with the same number or rows/columns, have been obtained. Each of them represents one of the previously described dynamic degrees of freedom. The next step consists of the modal decomposition which the program resolves by means of an iterative method and whose results are the auto values and autovectors corresponding to the diagonalising of the stiffness matrix with the mass matrix.

The system equations to be resolved are as follows:

$$\left[K - \omega_2 \cdot M\right] = 0.0 \text{ (null determinant)}$$

K: Stiffness matrix
M: Mass matrix

$$\left[K - \omega_2 \cdot M\right] \cdot \left[\phi\right] = \left[0.0\right] \text{ (undetermined homogenous systems)}$$

$\omega^2$: Auto values of the system
$\omega$: Natural frequencies of the dynamic system
$\phi$: Autovectors of the system or condensed vibration modes

From the first equation, a maximum number of solutions can be obtained (values of $\omega$) equal to the assumed number of dynamic degrees of freedom. For each of these solutions (auto values), the corresponding autovector (vibration mode) is obtained. Nonetheless, it is rare that the maximum number of solutions of the system be required and only the most representative of the number indicated by the user as vibration modes intervene in the analysis. Upon indicating this number, the program selects the most representative solutions of the system, which are those displaced by the mass and correspond to the natural frequencies of the greater vibrations.

The condensed vibration modes that are obtained (also called shape coefficient vectors) are the result of a homogenous (the vector of independent terms is null) and undetermined ($\omega^2$ has been calculated so the determinant of the coefficient matrix be null).

Therefore, this vector represents a direction or deformation mode, and not specific values of the solutions.

Based on the vibration modes, the program obtains the participation coefficient for each direction ($\tau_i$) in the following way:

$$\tau_i = \left[\phi_i\right]^T \cdot \left[M\right] \cdot \frac{\left[J\right]}{\left[\phi_i\right]^T} \cdot \left[M\right] \cdot \left[\phi_i\right],\ i = 1,\ldots,n \ \text{ calculated modes}$$

Where [J] is a vector which indicates the direction of the earthquake load. For example, for an earthquake load acting in the X direction:

$$[J] = [100100100\ldots100]$$

Once the natural vibration frequencies have been found, these are entered in the selected design spectrum, along with the ductility, damping, etc. parameters, and the design acceleration for each vibration mode and dynamic degree of freedom is obtained.

These values are calculated in the following way:

$$a_{ij} = \phi_{ij} \cdot \tau_i \cdot a_{Ci}$$

i: Each vibration mode
j: Each dynamic degree of freedom
$a_{ci}$: Design acceleration for vibration mode i

The maximum displacements of the structure, for each i vibration mode and j degree of freedom in accordance with the equivalent linear model, are obtained as follows:

$$u_{ij} = \frac{a_{ij}}{\omega_i^2}$$

Therefore, for each dynamic degree of freedom, a maximum displacement value is obtained for each vibration mode. This is equivalent to an imposed displacement problem, which is resolved for the remaining degrees of freedom (not dynamic), by means of modal expansion or “backward” substitution of the previously condensed degrees of freedom.

Finally, a displacement and force distribution over the whole structure is obtained for each vibration mode and for each dynamic loadcase, with which the modal spectral analysis is concluded.

The program uses de CQC method (complete quadratic combination) to attain the maximum values of a force, displacement etc, whereby a modal coefficient dependent on the ratio between the vibration periods to combine is calculated. The formula of the method is as follows:

$$x = \sqrt{\sum_i \sum_j \rho_{ij} x_i x_j}$$

$$\rho_{ij} = \frac{8\zeta^2 r^{3/2}}{(1+r)(1-r)^2 + 4\zeta^2 r(1+r)}$$

where:

$r : \frac{T_i}{T_j}$

$\zeta$ : Buckling coefficient, uniform for all vibration modes and with a value of 0.05

x: Resultant force or displacement

$x_i$, $x_j$: Forces or displacements corresponding to the modes to combine

For those cases in which the evaluation of the concomitant forces is required, **CYPECAD** undertakes a linear superposition of the various vibration modes in such a way that for a given dynamic loadcase, n groups of forces are really obtained, where n is the number of concomitant forces that are required. For example, during the design of concrete columns, three forces are being dealt with simultaneously: axial force, bending in the xy plane and bending in the xz plane. In this case, upon requesting the combination with a dynamic loadcase, the program will provide three different combination versions: one for maximum axial force, another for maximum bending in the xy plane and the other for maximum bending in the xz plane. Additionally, the created combinations are multiplied by ±1, as the earthquake loads can act in either of the two directions.

The second order effects can be considered, if the user wishes for them to be so, by activating the option, as the program does not consider them automatically.

Upon carrying out the analysis, the user may consult, for each mode, the period, the participation coefficient in each design direction X, Y and what is referred to as the seismic coefficient, which is the displacement spectrum obtained as $S_d$:

$$S_d = \frac{\alpha(T)}{\omega^2 \mu}$$

$\alpha$ (T): Spectral ordinate

$\omega$: Angular frequency = $2\pi/T$

$\mu$ : Ductility

## *Effects of torsion*

When a dynamic analysis is undertaken, the total moment and shear force is obtained due to the earthquake loads on the building. By dividing them, the eccentricity with respect to the centre of gravity is obtained. Depending on the selected earthquake code of each country, it is compared with the minimum eccentricity specified in each code and if less, the rotational mode is amplified, in such a way that at least the minimum eccentricity is obtained.

If the earthquake loads on the structure are analysed in a generic manner (*Modal spectral analysis*), the minimum eccentricity the program takes into account is 0.05.

This is important especially in the case of symmetrical structures.

## *Base shear*

When the base shear due to dynamic seismic loading is less than 80% of the static base shear, the proportion will be increased so not to be smaller than it.

## *Consideration of 2nd order effects (PΔ)*

The user may optionally choose to consider, when defining wind or seismic loadcases, to amplify the forces due to the presence of these horizontal loads. It is recommended this option be activated in the analysis.

The method is based on the P-delta effect of the displacements produced by the horizontal loads, taking into account, in a simple manner, the second order effects based on a first order analysis and a linear behaviour of the materials with mechanical properties bases on the gross sections of the materials and secant elastic modulus.

A horizontal load $H_i$ acts at each floor i. The structure deforms and displacements, $\Delta_{ij}$, occur at each column level. At each column j, and at floor level, a load with value $P_{ij}$ acts for each gravitational loadcase, transmitted by the slab to column j at floor i.

An overturning moment $M_H$ due to the horizontal forces is defined, acting at elevation $z_i$ with respect to elevation 0.00 or elevation without horizontal displacements, for each direction of action:

$$M_H = \sum H_i \cdot z_j$$

i: Number of floors
j: Number of columns

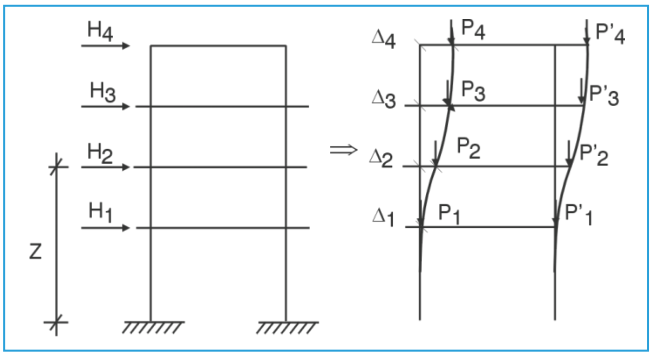


Figure 16. Wind action

In the same way a moment due to P-delta effects, $M_{P\Delta}$ is defined, due to the loads transmitted by the slabs to the columns $P_{ij}$ for each of the defined gravitational loadcases (k), because of the displacements due to the horizontal load $\Delta_i$.

$$M_{P\Delta k} = \sum_i \sum_j P_{ij}\,\Delta_i$$

where:
k: For each gravitational loadcase (self weight, live load…)

The following coefficient:

$$C_k = \frac{M_{P\Delta k}}{M_{HK}}$$

represents the stability index for each gravitational loadcase and for each direction of the horizontal force. If it is calculated, an amplification coefficient to be applied to applied loadcase safety factor can be obtained for all the combinations in which horizontal forces act. This value is referred to as $\gamma_z$ and is calculated using the following formula:

$$\gamma_z = \frac{1}{1-\left(\sum \gamma_{gi}\cdot C_i + \sum \gamma_{fqi}\cdot C_j\right)}$$

where:
$\gamma_{fgi}$: Safety coefficient for permanent loads of loadcase i
$\gamma_{fqi}$: Safety coefficient for variable loads of loadcase j
$\gamma_z$: Global stability coefficient

It must be recalled when calculating the displacements due to each horizontal load loadcase, that a first order analysis was carried out using the gross sections of the elements. If forces for ultimate limit state design are being calculated, it is logical that if a thorough calculation of the displacements is to be obtained, it be done using the fissured and homogenised sections of the elements which may result to be very laborious, as that implies linear simplification of the materials, geometry and load states. This causes it to be an unpractical solution using the analysis methods available. Therefore a simplified method must be established consisting of supposing a reduction in the stiffness of the sections, which implies an increase in the displacements, as they are inversely proportional. The program requires the increase or "multiplication factor of the displacements" to take this stiffness reduction into account.

At this point there is no single criteria, therefore it is for the user to decide which value to enter depending on the type of structure, estimated fissure grade, other stiffening elements, nuclei, stairs etc., which in reality may reduce the calculated displacements.

In Brazil, it is common to consider a reduction coefficient of the longitudinal elasticity modulus of 0.90 and assume a reduction coefficient of the fissured inertia with respect to the gross inertia of 0.70. Therefore, the stiffness is reduced by these factors:

$$\text{Reduced stiffness} = 0.90 \cdot 0.70 \cdot \text{Gross stiffness} = 0.63 \cdot \text{Gross stiffness}$$

Due to displacements being the inverse of the stiffness, the multiplying factor of the displacements will be 1 / 0.63= 1.59. This value will be entered into the program. As a general rule, if $\gamma_z > 1.20$, the structure's stiffness should be increased in that direction, as the structure can deform easily and has little stability in that direction. If $\gamma_z < 1.1$, its effect is small and practically negligible.

In the new *NB-1/2000 code*, in a simplified manner, it is recommended the displacements be amplified by 1/0.7 = 1.43 and limit the value of $\gamma_z$ to 1.3.

In the 1990 *CEB-FIP model code*, a moment amplification method is applied which recommends, unless a more precise analysis is undertaken, the stiffnesses be reduced to 50% or similarly, apply a displacement amplification coefficient equal to 1/0.50=2.00. For this assumption, it may be considered that if $\gamma_z > 1.50$, the structure's stiffness in that direction must be increased, as the structure can deform easily and has little stability in that direction. $2^{nd}$ order effects do not have to be considered if $\gamma_z < 1.1$, however it is recommended they always be activated.

The *ACI-318-95 code* refers to a stability index Q per floor, not for the global stability of the building, even though a ratio can be established for the global stability if the floor are very similar:

$$\gamma_z\text{: Global stability coefficient} = 1 / (1\text{-}Q)$$

Regarding the limits establishing whether the floor corresponds to a non-sway fame system, or what in this case would be the limit as to whether it is to be considered or not, it may be taken as Q = 0.05, i.e. 1 / 0.95 = 1.05.

For this case, it is to be calculated and always taken into account if the value is exceeded, which results in always having to consider the calculation and amplify the forces using this method.

Regarding the displacement multiplication coefficient, it is indicated that given that the horizontal loads are temporary and act during a short period, a reduction of approximately 70% of the inertia can be considered, and as the elasticity modulus is smaller (15100 / 19000 = 0.8), in other words, a displacement amplification coefficient of 1 / (0.7 · 0.8) = 1.78, which according to the global stability coefficient, does not exceed 1.35 would be a reasonable value.

It may be appreciated that the model code criteria would be recommendable and easy to remember for all its application cases:

Displacement multiplication coefficient = 2
Limit for global stability coefficient = 1.5

It is true, on the other hand, that stiffening elements are always present in buildings: façades, stairs, load bearing walls, etc., which assure that a smaller displacement be present against horizontal loads than those calculated. Because of this, the program leaves the value of the displacement multiplication coefficient at 1.00. It is left to the user's criteria as to what the value should be modified to, given that not all the elements can be discretised in the structure.

Once the analysis has concluded, the calculated values for each of the combinations can be checked at the following option: **General data > Wind and Earthquake options > Second order effects button > Amplification factors** button. Here, the maximum value of the global stability coefficient can be seen in each direction. A report can also be printed within the *Job reports* section.

The case may arise where the structure is unstable, in which case an error message is emitted before the analysis has concluded warning a global instability phenomenon is present. This will occur when the value of $\gamma_z$ tends to infinity or, which is the same in the formula, becomes negative or zero because:

$$\Sigma\left(\gamma_{fgi} \cdot C_i + \gamma_{fqi} \cdot C_j\right) \geq 1$$

It may be studied for wind and/or earthquake loads and is always recommended it be calculated, as an alternative calculation method to second order effects, especially for sway-frame structures or presenting some sway, as occurs in most buildings.

Recall that all of the live load loadcase is considered and given that the program does not carry out any automatic live load reduction, it may be convenient to repeat the analysis previously reducing the live load, which would only provide valid results for the columns.

Regarding the *ACI 318 code*, once the stability of the building has been studied, the reduction of the column stiffness for their design is carried out by applying a formula indicated in the code appendix of the program.

In that case and given the difficulty of calculating the buckling coefficients by determining the bar stiffnesses at each column end, it is sufficiently safe to assume buckling coefficients with a value = 1, with which the fictitious or additional second order eccentricity of the isolated bar will always be calculated together with the P-delta amplification effect of the considered method. This way, reasonable results are obtained within the slenderness field established by each code.

It is left to the user's choice, given that it is an alternative method. In this case, the user can opt for the rigorous application of the corresponding code.

# 1.6 Materials used

All the materials are selected from lists within the program. The material's properties are defined within a file. The data to be specified for each case is:

## *1.6.1 Concrete for foundations, slabs, columns and walls*

A file exists containing a list of concrete types defined by their design resistance, reduction coefficient, secant elastic modulus and *Poisson coefficient* $\nu$ = 0.2 defined in the code.

The concrete used can be different for each element. Additionally, in the case of columns, can be different for each floor. These values correspond to those most frequently admitted in the code.

## *1.6.2 Bar steel*

A file exists which contains a list of the steel types defined by their elastic limit, reduction coefficient and elasticity modulus defined in accordance with the code.

It is always considered due to its **position and type of element**.

The steel may be different depending on whether it be for:

### *Columns, walls, shear walls and corbels*

- Bars (vertical and horizontal)
- Stirrups

### *Floor beams and foundation beams*

- Bottom reinforcement
- Top reinforcement
- Assembly reinforcement
- Skin reinforcement
- Stirrups

### *Floor slabs and mat foundations*

- Punching shear
- Mat foundation top reinforcement
- Mat foundation bottom reinforcement
- Waffle and joist floor slab top reinforcement
- Waffle slab bottom reinforcement
- Footings and Pile caps

## 1.6.3 Steel for steel columns, beams and baseplates

CYPECAD allows for steel beams and columns to be used, in which case the type of steel to be used must be indicated. A library containing steel types which can be selected by the user is available. This library is saved as a file and cannot be modified by the user. The file contains information such as the elastic modulus, elastic limit, Poisson coefficient and all the parameters required for the analysis. Rolled, welded and cold formed steel sections are available. In the case of baseplates at steel column starts, the type of steel is defined for the plates and stiffeners, as well as the type of steel to use for the anchorage bolts. The available steel types and diameters are predefined within the program and may not be modified.

The materials to be used in Integrated 3D structures are to be defined per bar be it timber or steel.

### 1.6.4 *Integrated 3D structures materials*

The materials to be used in Integrated 3D structures (steel, timber, aluminium, concrete or generic) are to be defined in **Job > General data**.

## 1.7 Weighting factors

The weighting factors are established in accordance with the properties of the materials to use, the loads acting on the structure and the analysis method to be used which specified in the selected design code.

### 1.7.1 *Analysis method*

To calculate the weighting factors the Limit states method is used or the corresponding method to apply for each selected code.

### 1.7.2 *Materials*

The reduction coefficients to be applied to the materials used are defined for each code. The corresponding articles of the code may be consulted.

Upon selecting the material, the execution control level on site (if it exists for the selected code) must be indicated and therefore, the predefined weighting coefficient, which is predefined in a file associated with the selected design code.

### 1.7.3 *Loads*

The weighting coefficients will be applied depending on the execution control level and on the foreseeable damage that may occur within the project and on site, as well as its construction method.

It should be taken into account whether the effect of the loads is favourable or unfavourable, as well as the origin of the load. The values may vary.

These values will have to be established for each combination. To do so, the weighting and simultaneity factors defined in the corresponding combination file will be read, depending on the number of loadcases of each of the simple loadcases in accordance with its origin. This file cannot be edited or modified by the user, although he/she can define his/her own combinations.

# 1.8 Combinations

Once the basic simple loadcases that intervene in the analysis have been defined, and in accordance with the code to apply, a group of element states must be checked which may require an equilibrium, tensile, fracture, fissure, deformation, etc check to be undertaken. All this is summarised in the limit states analysis, which may, additionally, be obtained depending on the material to be used. A group of combinations is defined for each of these limit states, with their corresponding weighting coefficients, which the program generates automatically and has to be selected for the analysis. The following states are checked:

- U.L.S. for failure. Concrete. Section design.
- U.L.S. for failure. Foundation concrete. Section design.
- Soil pressures. Check for pressures acting on soil.
- Displacements. To obtain the maximum displacements of the structure.
- U.L.S. for failure. Rolled and welded steel. Section design.
- U.L.S. for failure. Cold formed steel. Section design.
- U.L.S. for failure. Timber. Section design.

Therefore, combination groups can be defined and activate the limit states which are to be checked for the selected code, and the weighting coefficients to be used. It is common practice for each country to establish the states detailed below.

## *1.8.1 Ultimate limit states*

These are defined to check and design the sections. The previously mentioned combination groups for concrete, rolled steel, welded steel, cold-formed steel, timber and aluminium are usually indicated. They are not contemplated by codes which use allowable stresses.

## *1.8.2 Project situations*

The load combinations will be defined in accordance with the following criteria for the various project situations:

### *Non-seismic situations*

- With combination coefficients

$$\sum_{j\geq 1} \gamma_{Gj} G_{kj} + \gamma_{Q1} \psi_{p1} Q_{k1} + \sum_{i>1} \gamma_{Qi} \psi_{ai} Q_{ki}$$

- Without combination coefficients

$$\sum_{j\geq 1} \gamma_{Gj} G_{kj} + \sum_{i\geq 1} \gamma_{Qi} Q_{ki}$$

### *Seismic situations*

- With combination coefficients

$$\sum_{j\geq 1} \gamma_{Gj} G_{kj} + \gamma_A A_E + \sum_{i\geq 1} \gamma_{Qi} \psi_{ai} Q_{ki}$$

- Without combination coefficients

$$\sum_{j\geq 1} \gamma_{Gj} G_{kj} + \gamma_A A_E + \sum_{i\geq 1} \gamma_{Qi} Q_{ki}$$

where:

$G_k$: Permanent load

$Q_k$: Variable load

$A_E$: Seismic load

$\gamma_G$: Partial safety factor for permanent loads

$\gamma_{Q1}$: Partial safety factor for main variable loads

$\gamma_{Qi}$: Partial safety factor for accompanying permanent loads

(i > 1) for non-seismic situations
(i ≥ 1) for seismic situations

$\gamma_A$: Partial safety factor for seismic loads

$\psi_{p1}$: Combination coefficient of the main variable load

$\psi_{a1}$: Combination coefficient of the accompanying variable loads

(i>1) for non-seismic situations
(i ≥ 1) for seismic situations

# 1.9 Data entry

The data to be entered for the analysis of a job includes:

## *1.9.1 General data of the job*

The data included in points 3 to 6 is selected from a list of materials.

1. Design codes for concrete, steel (cold-formed and rolled), aluminium, timber, block walls and composite slabs.
2. Job description (2 lines).
3. Concrete for floor slabs
4. Concrete for foundations, foundation data
5. Concrete for columns and shear walls. May be different for each floor.
6. Concrete for walls. May be different for each floor.

    6.1. Generic masonry wall properties:

    Modulus of elasticity E
    Shear modulus G
    Unit weight
    Design compressive strength
    Design tensile stress
    Consider shear stiffness
    In the case of block walls, the mortar and block resistance is selected and horizontal joint reinforcement steel.

7. Steel for concrete reinforcement

    7.1. For columns, shear walls and walls:
    Vertical bars
    Stirrups
    The program contains two tabs; one for floor slabs and the other for foundations

    7.2. For beams:
    Top reinforcement (bottom additional reinf. for foundation beams)
    Bottom reinforcement (top reinf. for foundation beams)
    Assembly reinforcement (bottom reinf. for foundation beams)
    Skin reinforcement
    Stirrups

7.3. For floor slabs:
Shear and punching shear reinforcement
Flat slab and mat foundation top reinforcement
Flat slab and mat foundation bottom reinforcement
Waffle slab, drop panel and joist floor slab top reinforcement
Waffle slab, drop panel and in-situ joist floor slab bottom reinforcement

8. Steel for steel beam and column sections
   8.1. Cold formed steel
   8.2. Hot rolled steel
9. Wind loads
10. Seismic loads
11. Fire resistance check
12. Additional loadcases (special loads)
13. Limit states (combinations)
    13.1. Concrete
    13.2. Foundations
    13.3. Cold formed steel
    13.4. Rolled steel
    13.5. Timber
    13.6. Aluminium
    13.7. Ground bearing pressures
    13.8. Displacements
14. Buckling coefficients for each floor in each direction
    14.1. Concrete columns
    14.2. Steel columns

These coefficients can be defined per floor or for each independent column. The program assumes a default value of $\alpha$ = 1 (also called $\beta$), whereby the user has to vary this value if he/she considers it to be so due to the type of structure and connections of the column with beams and slabs in both directions.

Observe the following case (Figure 17), whereby the buckling coefficients of a column which is not braced in various floors are analysed. Here, it can be seen that it may buckle along the whole of its height:

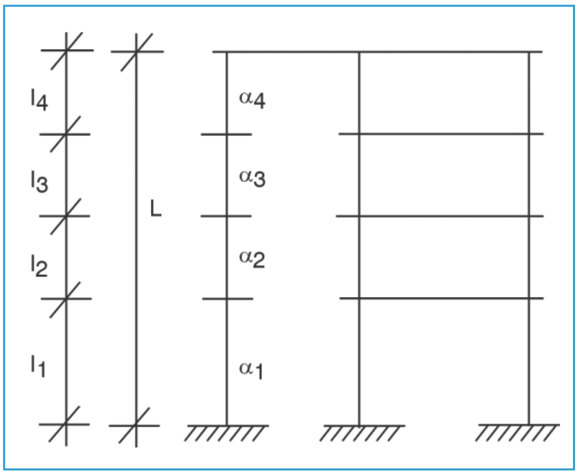



Figure 17

When a column is disconnected in both directions for several consecutive floors, the column is designed for each span or floor, therefore when it comes to calculating the slenderness and effective length $l_0$, the program takes the maximum value amongst all the consecutive disconnected spans, multiplying it by its total length = sum of all the lengths.

$$\alpha = MAX(\alpha_1, \alpha_2, \alpha_3, \alpha_4, \ldots)$$
$$l = \sum l_i = (l_1 + l_2 + l_3 + l_4 \ldots)$$

Now, $l_0 = \alpha \cdot l$ (for both X and Y local directions of the column, with its corresponding value).

When a column is disconnected in a single direction for several consecutive floors, the program will take, for each floor i, $l_0 = \alpha_1 \cdot l_i$, not acknowledging it not being connected.

Therefore, if it is to be effective, in the direction where it is not connected, the value of each $\alpha_i$ must be obtained, in such a way that:

This i the value corresponding to completely exempt span, l.

The value of each span, i, will be:

$$\alpha_i = \frac{\sum_{j=1}^{n} l_j}{l_i} \cdot \alpha$$

In the example for $\alpha_3 = \dfrac{l_1 + l_2 + l_3 + l_4}{l_3} \cdot \alpha$

Therefore, when the program calculates the buckling length of floor 3, it will calculate:

$$l_{03} = \alpha_3 \cdot l_3 = \frac{l_1 + l_2 + l_3 + l_4}{l_3} \cdot \alpha \cdot l_3 = (l_1 + l_2 + l_3 + l_4) \cdot \alpha = \alpha \cdot l$$

which coincides with that indicated for the complete unconnected span, even though the calculation is carried out for each floor, which is correct, but it will always take the length as $\alpha \cdot l$.

The height which is considered for buckling design is the free height of the column, i.e. the height of the floor minus the height of the beam or slab, whichever has the greatest depth) which reaches the column.

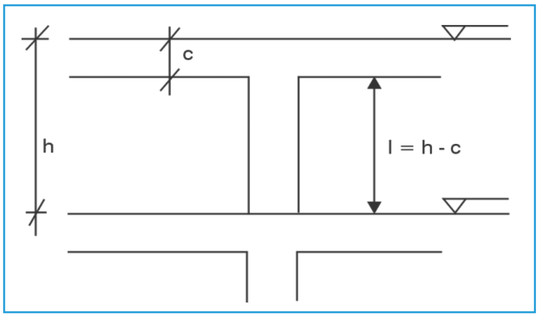



Figure 18

The final value of $\alpha$ of a column is the product of the $\alpha$ of the floor by the $\alpha$ of the span.

It is left to the user's opinion as to the values to be entered in each direction of the local axes of the columns, as the various codes only indicate how these values are exactly determined for frames, and given that the spatial behaviour of the structure does not correspond to the buckling of a frame, it is preferable not to provide the values in an inexact manner.

## 1.9.2 Loads. Groups

In this section, the user indicates whether or not to consider horizontal, wind and/or seismic loads and the code which is to be applied for each case. The program chooses the combination for each limit state internally.

Likewise, the weighting factors are validated depending on the materials used and the intervening loads. The additional loads are also selected and are assigned to each loadcase.

The user can also modify the global dead and live loads for each floor group. The self weight of the floor slab is indicated in the file containing its description.

### 1.9.3 *Wind loading*

The code to apply is to be selected. Please consult the section of the corresponding code.

### 1.9.4 *Seismic loading*

If earthquake loads are present, the data to enter will be in accordance with the selected code. Please consult the section of the corresponding code.

> ***Note:*** *Loads associated to wind and/or earthquake loadcases can be defined in the additional loadcases section, if they are not automatically previously generated.*

### 1.9.5 *Fire resistance*

The coating (if present) of each structural element and group is defined, as is the required resistance and whether the floor slab fulfils its compartmentation duty.

### 1.9.6 *Additional loadcases (special loads)*

Additional loadcases (special loads) can be defined automatically, and are different to general loads:

- Permanent loads (self weight of floor slabs + dead loads) = (permanent loads)
- Live load defined within group data (live load)
- Wind in accordance with the selected code (Wind)
- Seismic load in accordance with the selected code (Earthquake)

If the user wishes to define loads (point, line or surface loads) which are to belong to these loadcases, these must first be created. The permanent loads and live loads are always defined by default.

If alternating live loads are to be created, i.e. loads that do not act simultaneously for a combination, additional loadcases must be defined; the number of which is to correspond with the number of independent loads that are to be considered. The load disposition option within each loadcase can also be used.

The combinations are generated automatically based on the defined loadcases and their combinability.

Upon entering these special loads, be they line, point or surface loads, the loadcase associated to the load i.e. the loadcase to which it belongs, must be selected.

## 1.9.7 Limit states (combinations)

The group corresponding to each state to calculate is selected.

- Concrete
- Foundations
- Cold formed steel
- Rolled steel
- Timber
- Aluminium
- Ground bearing pressures
- Displacements

## 1.9.8 General data of floor/groups, columns, column starts and shear walls (Column Definition tab)

### 1.9.8.1 Floors/groups

Here, the data to enter includes:

- Name of the groups, live and dead loads
- Elevation of the foundation plane, name of the floor and height between floors.

Upon indicating the heights (h) of the floors, the difference between the top floor slab surface is defined (or mean top reference plane). The elevations are calculated by the program based on the indicated data.

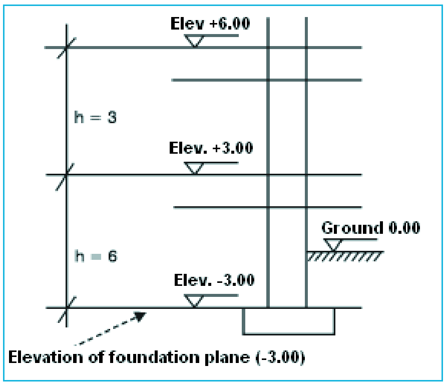



Figure 19

### 1.9.8.2 Columns

Their cross sectional and elevation geometry is to be defined, indicating:

1. The type of column (concrete or steel)
2. Cross section at each floor level
3. Reference
4. Rotation angle
5. Start at foundation (with external fixity) or whether it starts elsewhere (without external fixity) and up to which floor it reaches. If the column starts on a beam or mat foundation, it must be defined without external fixity.
6. Fixity coefficients at top and bottom of the column
7. Buckling coefficients at each floor and in both local directions; x and y (see text for Job general data).
8. If the column is a steel column, the type and series of the steel section selected from the library is indicated.
9. Support elevation change and depth, if present, so to include the details in the job drawings.

### 1.9.8.3 Column starts

It is possible to only define the start of a column (i.e. column with zero height), so that foundation elements may be designed by simply defining the applied loads as loads acting at the top of the column.

### 1.9.8.4 Reinforced concrete shear walls

First of all, a shear wall series is defined, indicating:

- Name
- Initial and final group
- Sides and vertices
- Thicknesses at each floor to the left and right of the axis of the side

The first defined vertex is the insertion fixed point even though its position may be varied. When defining a shear wall, the following parameters are selected:

- Shear wall type
- Reference
- Angle

The shear walls possess the same geometry on plan, whereby only their thickness can be varied. They may not start on columns, and columns cannot start on them. They are of constant geometry and brought about to brace the building horizontally.

### 1.9.8.5 Horizontal loads on columns

The type of load, loadcase and application point are defined.

Horizontal loads on columns can be defined with the following properties:

- Type of load: point, uniform load or strip load
- Loadcase: those defined in the job (self weight, live load, wind earthquake)
- Application point: at any column elevation
- Direction: in local or general axes, in X or Y.

### 1.9.8.6 Vertical loads on columns

Loads (N, Mx, My, Qx, Qy, T) can be defined acting at the top of the last span (where it ends) of any column. These are in reference to the general axes, for any loadcase, added to those obtained from the analysis and in accordance with the following sign criteria:

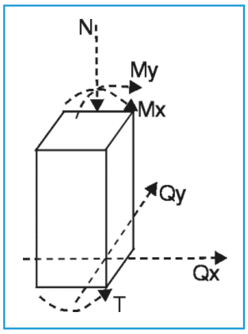



Figure 20

There is other data that can be consulted and modified such as the support conditions, fixity and buckling coefficients.

## 1.9.9 Floor slab data (Column Definition tab)

The geometry of the beams is defined graphically on the floor plan for each group of the structure. The columns and shear walls are visible for each group. The logical order in which the data is to be entered is as follows:

### 1.9.9.1 Beams, external wall supports and foundation beams

The type of beam is selected and its dimensions defined.

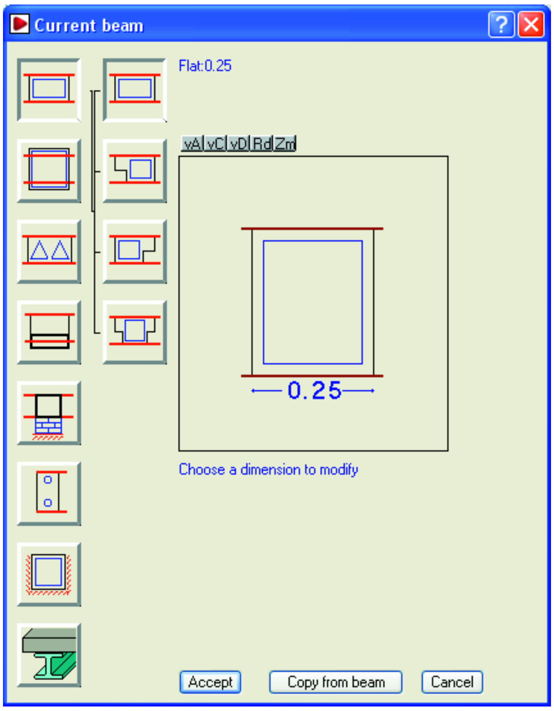



Figure 21

A Fixity coefficient can be defined at beam edges. Its value varies between 0 (pinned) and 1 (fixed). Any slab panel joining that edge of the beam will be affected by that coefficient.

Hinges can be entered at the ends of any beam span, at connection points with columns, shear walls or other beams.

If the beam entered is a foundation beam, the subgrade modulus and allowable bearing pressure are required.

If a composite beam is going to be entered, please consult section ***8. Composite beams***.

### 1.9.9.2 Walls

Two types of walls can be defined:

**Reinforced concrete walls.** These are reinforced concrete walls, which may or may not receive lateral pressures from the soil.

**Masonry walls.** These are brick or concrete block walls, which receive and transmit loads but no lateral pressures.

The following data is to be indicated:

- Start floor
- End floor
- Mean thicknesses at each floor (to the left and to the right)
- Soil lateral pressures (only for reinforced concrete walls) indicating:
  - Loadcase of the pressure
  - Apparent unit weight
  - Submerged unit weight
  - Slope angle
  - Internal friction angle
  - Loads on fill (if present)
- Beam or foundation support
  - Foundation beam (without external fixity)
  - Strip footing (without external fixity)
  - With external fixity (with or without a footing)
  - Beam without external fixity

When required, the data of the footing is to be indicated:

- Allowable bearing pressure of the soil
- Soil subgrade modulus. A high default value is provided = 100 000 kN/m$^3$, because if there are columns present, differing settlements on the ground can occur, which will not really occur if an analysis is carried out later on with pad footings below the columns. If all the foundation plane consisted of elements without external fixity (floating), a subgrade modulus corresponding to the type of soil and dimensions of the foundations would be entered. Generally speaking, it is not recommended elements

with external fixity and elements without external fixity be used simultaneously. The program will emit a warning if this occurs.

### 1.9.9.3 Type of floor slab

The floor slab is defined by giving it a name and entering a series of data:

#### *Joist floor slabs*

Several types of joist floor slabs are available:

- Concrete joists (generic geometry)
- Reinforced joists
- Prestressed joists
- In-situ joists
- Steel joists
- Timber joists
- Open web joists
- The reinforced and prestressed joists can be of the following types:
  - Manufacturer: data provided by a manufacturer. It cannot be edited.
  - Library: contains joists defined by manufacturers and by users using a specific program "The floor slab file editor" provided by CYPE, which any user can use.
  - Geometrical properties, provided by the user whereby all the data can be edited. Regarding the deflection calculation, it may be designed as a reinforced or prestressed beam.
- By joist type: This depends on the resisted moment of the joist whereby the type of joist may be visualised instead of the bending moments. The value is indicated in dNm, per metre width, per joist with the applied safety factor.

Each panel may consist of a different type of floor slab, and its position may be perpendicular to beams, parallel to beams or defined by two pass-through points.

The existence or non-existence of continuity between joists of adjacent panels can be achieved (except in the case of steel and open web joists, which are always simply supported). By copying panels, continuity is obtained from one to the next. By varying the insertion point, so joists in adjacent panels are not aligned, the continuity is eliminated. This is achieved as long as the distance between the ends of the joists of adjacent panels is greater than the **short bar** length (default value of 0.20m, which can be varied using the option **Reduction coefficient of flexural stiffness of joist floor slab**. The same continuity effect is produced if a beam is located at the end of a joist and works as an extension of the joist, where the separation of their axes is less than the short bar length.

Having defined a group, one of the previously defined groups can be copied and then carry out the required modifications.

Elevation changes can be defined between panels. These changes are reflected in the drawings and reinforcement layout of slabs and beams affecting the heights of the supports which intersect the elevation change beam. This must be used with caution, as the program does not calculate the transverse bending of the beam. It is, therefore, recommended that the construction details be consulted and the stirrups and anchorage lengths of the transverse reinforcement of the beam be checked manually.

The minimum negative and positive moments for joists can be consulted and modified. It is important they be consulted and assigned correctly.

Double, triple… joists can be entered. In this case the program will take the defined weight, which is limited to a triple joist. In this situation, a bar or joist parallel is entered at a distance equal to the width of the joist defined in the properties sheet of the slab.

A fixity coefficient can be defined at joist edges or ends (0 = pinned, 1 = fixed which is the default value) for each panel.

### *Composite slabs*

Please consult section ***9. Composite slabs*** of this *Calculations manual*.

### *Hollow core slabs*

To define a hollow core plate, its geometric data and mechanical properties must be entered.

The data contained in the properties sheet can be taken from the use authorisations of the manufacturers, or enter the values of a specific plate or slab to be built in-situ. There is some data which is required and should be clarified:

- **Reference.** To identify the properties sheet.
- **Description.** The name of the plate.
- **Total slab depth.** The total depth of the plate and the compression layer thickness (if present)
- **Plate width.** The width of the plate.
- **Compression layer thickness.** The compression layer thickness if present.

- **Minimum plate width.** Is the smallest value that may be obtained from a longitudinal cut of a typical plate, as a consequence of the panel dimensions upon reaching the edge of the panel, where a special plate with a smaller width is placed instead. The width of the last plate varies between the typical width of the plate and this minimum value.
- **Maximum and minimum bearing.** When the plate is slanted with respect to the normal of the support, the bearing at each edge of the plate is different, varying between the maximum and minimum values. If the maximum value is exceeded, the plate is bevelled.
- **Lateral bearing.** This is the value the plate may overlap laterally with a parallel or slightly sloped support with respect to the longitudinal direction of the plate.
- **Self weight.** This is the weight per square metre of the complete slab.
- **Concrete volume.** This is the concrete volume in opening infills, joints between plates and compression layer, if present. The program adopts the value corresponding to the compression layer by default.
- **Plate concrete.** This is information data to display which materials were used to calculate the resistance data of the section.
- **Layer and joint concrete.** Same as above.
- **Top reinforcement steel.** Same as above.

The resistance data of the section are described below:

1. **Slab positive flexure.** The data of the plate with joint infill concrete and compression layer, if present.
    - Ultimate moment. The maximum resisted moment (ultimate)
    - Cracking moment. To calculate the deflection using the **Branson** method.
    - Total stiffness, of the concrete-plate composite section, used to generate the stiffness matrix of the bars into which the slab is discretised.
    - Cracked stiffness. To calculate the deflection using the **Branson** method.
    - Service moment. Resisted moment depending on the type of prestressed concrete, which is not the same as the environment. The equivalence is:
      Environment I = Class III (Structures in building interiors or exterior areas of low humidity)
      Environment II = Class II (Structures in normal or non-aggressive exteriors, or in contact with normal waters or ordinary soil)

Environment III = Class I (Structures in aggressive industrial or marine environment, or in contact with aggressive soils, salt or slightly acidic waters)
The program compares, according to the environment defined for the plate, the service moment from the analysis with that from the sheet and, if it is smaller, the plate is acceptable. Otherwise, the program looks in the table for a plate that does not fail, and if one is not found, a message appears after the analysis.

- Ultimate shear. Ultimate shear resisted by the total section. It is displayed in two columns depending on whether it is greater than the decompression moment ($M_g$)
The decompression moment is that corresponding to class II, and so the positive service moment is compared with that in the table, choosing the corresponding column.

2. **Slab negative flexure**

- Diameter / Diameter / Spacing: Two columns of diameters are indicated, which allow the combination of two different diameters at a given spacing. With such reinforcement distributed in the zone of negative moments, the mechanical properties of the section are indicated in each row.
- Ultimate moment of the typical section. This is the negative moment resisted by the section for a given reinforcement.
- Cracking moment. To calculate the deflection using the *Branson* method.
- Total stiffness. To calculate the deflection using the *Branson* method.
- Cracked stiffness. To calculate the deflection using the *Branson* method.
- Ultimate shear. Shear resisted by the section for a given reinforcement.

Based on the calculated reinforcement, the value of the shear resisted by the plate is known, which is compared with the design shear.

If it fails, the program issues a message at the end of the analysis, and “Insuf.” is indicated on the plate, on screen and in the drawing. If there are no values in the sheet, shear is not checked.

## *Design process used*

Once the maximum design positive moment $M_d$ is known, the program searches in the positive slab flexure column, ULT. M., for a value greater than that obtained in the analysis. At the same time, and depending on the environment defined for the panel, the program looks in the SER. M. (1, 2, or 3) column for a value that meets the analysis service moment (obtained with the deflection combinations). A plate is chosen that meets both conditions. If this is not possible, a message is emitted warning that it is outside table range.

In the same way, and for the plate selected for flexure and environment, the program checks in the shear column, of positive and negative slab flexure, whether the design shear is less than that resisted by the slab. If it fails, a warning is emitted.

The lengths of the bars are established depending on the bending moment envelope, and the minimum lengths defined in the options.

The envelopes are obtained according to the acting forces, considered redistribution and minimum applied moments.

This check is not carried out when data has not been defined for deflection, environment or shear analysis.

Within the **Panel manager** option, the environment can be selected, as well as the fixity coefficient at edges and minimum moments for each type of span: end, intermediate, isolated or overhanging.

## *Construction process*

The analysis can be carried out as a shored or unshored construction.

**A. Shored**

The analysis that is carried out by the program when continuity is considered, with a fixity coefficient value =1, is an elastic analysis subject to the total load = permanent load + live load. This is equivalent to building the floor on shoring, and upon their removal, the floor is subject to this total load.

In this analysis, the negative moments are generally greater than the positive moments.

**B. Unshored**

The precast hollow core plate floors are generally constructed without shoring, such that the final load state is comprised of two states:

1. The plate is subject to the self-weight of the slab w, which follows the typical moment equation for a simply supported element ($M=wl^2/8$).
2. The floor in its continuity is subject to an additional load, after the erection of the slab, comprised of the dead loads and live loads.

The superposition of the two stages leads to a load state which produces greater positive moments than negative moments, in the majority of cases.

In the current version of the program, the analysis is not carried out in two stages. If the floor is constructed without shoring (case B), reasonably approximate results that concur with what is expected are obtained. This is done by modifying the fixity coefficients in the continuous panels.

As a guide, the value of the fixity coefficient to assign to panels depends on the relationship between the self-weight of the floor and the total load, assuming a state of uniform loads. The value of the fixity coefficient would be:

fix.coeff. = 1 – (floor slf.wt. / total load)

For example, consider the case of a floor weighing

400 kg/m$^2$, with paving weighing 100 kg/m$^2$ and a live load of 500 kg/m$^2$:

floor slab self-weight = 400

total load = 400+100+500 = 1000

fix. coeff = 1– (400/1000) = 1–0.4 = 0.6

A fixity coefficient of 0.6 would be assigned to continuous slabs. The program assigns this value automatically to each panel of hollow core slabs when the unshored option has been activated.

In any case, it is recommended that the user consult with the manufacturer on the construction process and ask for advice for the design, verifying that the plate in its first phase, subject to its self-weight and a construction live load (generally 100 kg/m$^2$) can withstand the construction phase.

As for obtaining the deflection, it is calculated based on the mechanical properties indicated in the slab properties sheet or those defined by the user, and with the moment envelopes for the final state. These values can be consulted as a function of the established deflection limits in the hollow core plate options.

## *Flat slabs*

The depth of the panel and reinforcement direction are defined. Each panel may have a different depth. A fixity coefficient may be applied for any type of flat slab at its edges when connecting with the beams supporting it. The value may vary between 0 (pinned) and 1 (fixed), as well as any other value between these limits. Elevation changes may be defined between panels, with the same applied observations as indicated with joist floor slabs.

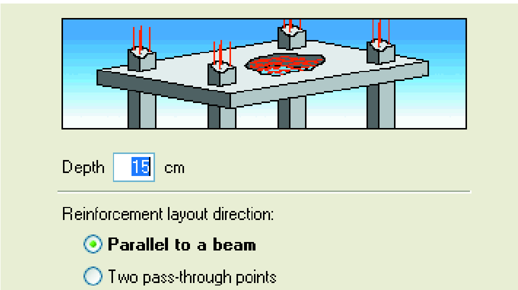



Figure 22

A base reinforcement mesh, spanning in both directions, top and/or bottom can be defined, which is considered in the analysis and has its diameter designed during the process.

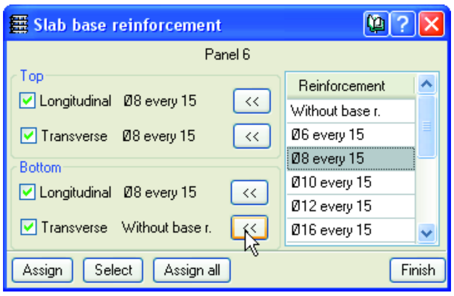



Figure 23

If this option is activated, the base reinforcement can be seen as any other reinforcement and can be edited and modified. The bottom base reinforcement is always continuous and overlaps in areas with maximum negative bending. The top reinforcement is not continuous and is only placed where necessary. In the case of mat foundations, the positions are inverted. The reinforcement is measured in the take-off reports and is drawn on drawings as additional reinforcement.

The direction in which the reinforcement is placed can also be indicated.

Mat foundations can also be used. In this case, the depth of the slab is to be indicated, as well as the subgrade modulus and allowable bearing pressure. The base reinforcement in mat foundations is determined automatically depending on the minimum steel ratio defined in the slab options.

## *Waffle slabs*

Waffle slabs are formed by panels in which two zones are present: a lightweight zone and a solid zone.

The lightweight zone is the first to be defined and is done so by selecting it from an editable program library or using the user's own definition. The lightweight zone can consist of removable or lost forms. Once the type of form is selected, the corresponding data must be filled in (Figure 24 and Figure 25):

- Reference
- Total depth
- Compression layer thickness
- Number of elements making up the lightweight form (for lost form only)
- Geometry of the transverse section: rib spacing, which can be the same or different in X and Y and the rib width (which can be variable in the case of removable forms).
- Concrete volume/m$^2$

- Self-weight of the slab (approximate, depending on the material making up the lightweight zone)

***Remember:*** *When entering data, an estimate of the total weight is carried out, as the program initially displays the volume and approximate weight of the infilled zones, compression layer and the selected default material (concrete). If this is not the case, the value may be modified. In the case of removable forms, only the value of the infilled concrete is estimated.*

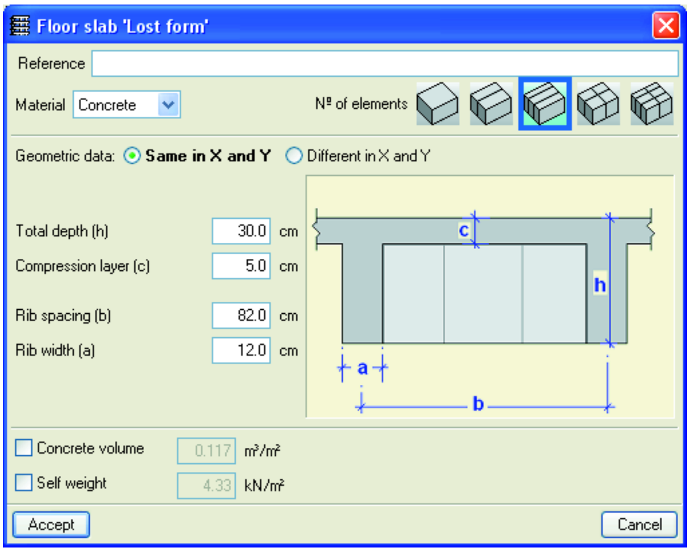



Figure 24

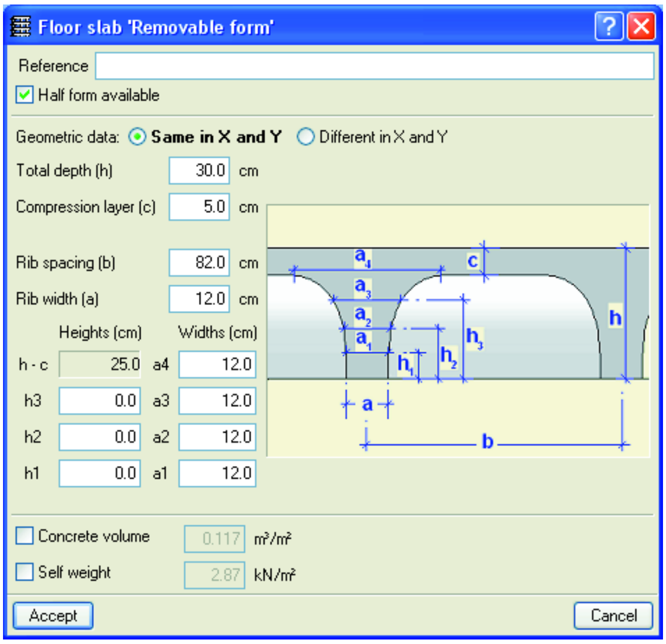



Figure 25

Having defined this data, the pass-through point, which may vary, of the rib mesh is indicated on the panel. The ribs can span in any direction. Elevation changes between panels can be defined, obeying the same rules as for joist floor slabs.

A base reinforcement mesh, spanning in both directions, top and/or bottom can be defined, which is considered in the design of the reinforcement.

***Very important:*** *If a base reinforcement is considered, the option* ***Job > General data > By position > Floor slabs tab > Options for flat, waffle and one-ways slabs > Detail base reinforcement in drawings*** *must be activated, otherwise it will not be visible on screen. Therefore, it will not be measured in the take-off reports or bar schedules in drawings. Special attention must also be paid when printing drawings so to ensure that it does exist, has been considered in the analysis and therefore must be placed on site. The drawings should be checked, and the necessary details added to indicate the overlap lengths and areas where these can occur.*

If this option is activated, the base reinforcement can be seen as any other reinforcement and can be edited and modified. The bottom base reinforcement is always continuous and overlaps in areas with maximum negative bending. The top reinforcement is not continuous and is only placed where necessary. In the case of mat foundations, the positions are inverted. The reinforcement is measured in the take-off reports and is drawn on drawings as additional reinforcement.

The floor slab can be different for each panel. If the beams separating each panel are flat, the beam will adopt the greatest depth of the two slabs on either of its sides. In the case of dropped beams, the amount by which it drops is measured as of the greatest depth. A fixity coefficient can be applied to the edges of the panel whose value may vary between 0 (pinned) and 1 (fixed).

The solid zones or drop panels can be generated automatically over columns or at any zone or the panel. These adopt the same depth as the lightweight panel in which they are situated. Their depth can be modified so their bottom surface drops below that of the lightweight section of the slab.

When drop panels are generated automatically, the dimensions in each direction are adjusted to 1/6 of the distance between the column in question and the one closest to it, with a 40º angle of vision. If no other column is “seen” (for example in the case of edge columns) the same value as was taken in the opposite sense of the same direction is used. The drop panel limits are set at 2.5-5 times the depth. An option is included in the program

(**Slabs > Drop panels > Configuration of drop panel generation in the Beam Definition** tab) which allows for these design parameters to be modified.

If these are generated manually, solid zones may be entered, always adjusting them to the lightweight zone. These are not to be used to simulate beams. In that case enter beams. Also, always do so at free edges. Drop panels always have a base reinforcement between ribs which is considered and deducts additional rib reinforcement in the analysis. It is not measured and cannot be indicated; therefore drawings must be checked, its presence indicated and construction details provided for construction on site.

When printing out the floor plan layout drawings, the program always places an information box indicating the base reinforcement of the ribs and drop panels, even though the bars are not displayed or detailed.

The forms provided in the lightweight zone of the slab can optionally be drawn.

### *Predefined reinforcement*

Reinforcement can be defined in any position and direction which is deducted from the necessary additional reinforcement in its zone of action.

### *Openings*

Panels in which no floor slabs are entered remain empty, and are represented by a question mark; therefore, users must delete the panel which is the same as entering an opening, which is represented by two intersecting discontinuous lines.

Additionally, rectangular openings may be entered in existing flat or waffle slabs.

Beams located between two openings or between an opening and the external edge, if they have been defined as flat and do not have a lateral slab, do not have their depths defined and therefore must be changed to dropped beams, indicating their dimensions.

If the type of beam considered contains a reduced form at the side at which the opening is located, it will not be taken into account and the program will warn that the data is incorrect.

*If, at any floor, there is an independent zone whose perimeter is defined by beams contained within an internal opening, even though there is no slab, the stiffness or non-deformable hypothesis of the floor is maintained.*

*Therefore, if horizontal loads are present, incorrect results will not be obtained. This situation is recommended when using sloped or exempt disconnected beams defined in the same group, which are elements that possess 6 degrees of freedom and do not consider the non-deformable hypothesis of the floor.*

*If reinforced concrete walls have been defined with acting lateral pressures, and there are joist floor slabs running parallel to the wall, these should have sufficient stiffness to behave as a rigid diaphragm, and so the infills and corresponding details, which the program does not carry out automatically will have to be provided.*

*For versions earlier than the 2012.a version, if there are empty panels next to the wall and perpendicular beams exempt from the wall, these must be placed as sloped beams so they may be designed for compressive bending, as normal beams and slabs are only designed for simple bending. Please refer to chapter* ***12****.* ***Rigid Diaphragm*** *if the program version is greater than or equal to the 2012.a version.*

*Integrated 3D structures can also be created between independent zones.*

### *Foundations*

Caps can be defined at starts of columns and shear walls “with external fixity”. Spanning between these elements, strap and/or tie beams can be inserted which can also connect to strip footings below walls.

Rectangular footings are design as rigid solids and may support several columns and/or shear walls. The same is applicable to pile caps, in accordance with a defined type of resolved cases.

Strap beams are defined to absorb the moment transmitted to the footing or pile cap they reach. Several beams may act simultaneously to absorb the moment in a given direction, in which case it will be distributed proportionally to their respective stiffnesses.

## *1.9.10 Loads. Sloped beams. Diagonal bracing*

Apart from the general surface loads, point, line and surface loads may also be entered. All these are entered graphically on screen and may be visualised to be consulted or to be modified at any moment.

Each type of load has an easily identified graphic diagram, as well as being displayed in different colours, if they belong to different loadcases.

In the case of **sloped beams**, their dimensions must be indicated, as well as the loads that may act on them (point, line, strip, triangular...), and where they span from and to (initial and final groups). They always have 6 degrees of freedom. They can consist of rectangular concrete sections or steel sections. Their ends can be fixed or pinned. The diagonal bracing consists of sloped beams, crossing each other between two supports and between two floors; these are always steel sections.

### *1.9.11 Stairs*

Please consult chapter ***10. Stairs*** of this *Calculations manual*.

## 1.10 Analysis of the structure

Once all the data has been entered, the structure can be analysed. During the process, information messages will appear regarding the analysis phase the program is running through. Error messages are also emitted if data exists which is incompatible with the analysis.

The first phase of the program consists in generating the geometric structure of all the elements forming a stiffness matrix of the structure. If the program detects incorrect data, error messages will be emitted, and the process will stop. This phase can be executed independently for one group or for all the job.

The second phase consists in inverting the stiffness matrix using frontal methods. If it is singular, a message will be emitted indicating it is a mechanism, if this situation is detected for an element or part of the structure. In this case the process stops.

In the third phase, the displacements are obtained for all the defined loadcases. A message will be emitted indicating there are excessive displacements at those points where the structure exceeds a value, whether it be due to an incorrect design of the structure or to the torsion stiffnesses defined for an element.

If global stability problems exist, the structure should be revised, when second order effects have been considered.

The fourth phase consists in obtaining the combinations and envelopes of all the defined combinations for each element: beams, slabs, columns, etc., and for each limit state.

In the fifth phase the program designs and reinforces all the defined elements in accordance with the combinations and envelopes, geometry, materials and existing reinforcement tables. A message is emitted if any of the limits indicated in the code are exceeded. The program continues until the end, emitting a report once it has concluded. It is convenient that the error messages of all the elements be checked.

If one or several Integrated 3D structures have been created, bear in mind that these can be processed individually and in an independent manner to the floor groups. It is convenient this be done, and the entered sections designed. This way, when the complete structure is processed, including the defined Integrated 3D structures, the results will be closer to the final section sizes that will be required.

It is possible that in many cases, especially when concerning steel columns and beams and Integrated 3D structures containing steel elements, that the section sizes have to be modified before the analysis, and that, due to their inertias varying significantly, make it obligatory to launch a new analysis.

This occurs frequently in steel structures and should not worry the user. Sometimes the analysis process must be repeated several times until all the steel sections are those to be used as the final section sizes. In the case of reinforced concrete, due to the program working with the gross section of the element, if the sections do not vary, or if they vary by a small amount, they are usually accepted as they remain.

Regarding an analysis carried out with **Stairs**, the user should bear in mind that the stairs are designed independently, obtaining the reactions at the start, end and intermediate supports, transforming these reactions into line loads which are applied on the structure as live and dead loads. With these loads applied to the structure, the program analyses the complete CYPECAD job. An integrated analysis has not been undertaken because their contribution and influence on the structure when exposed to horizontal loads is large and could provide results which do not usually arise in common practice, especially when what traditionally is done is apply their reactions and not integrate them.

Upon concluding the analysis, the errors and problems that have arisen during the analysis of the different elements can be consulted. They may be consulted on screen or by printing out a file, depending on the type of error. Other errors must be consulted per element, column, beam, slab, etc.

## 1.11 Results

Once the analysis has finished, the results can be consulted on screen, obtain reports from text files or via printer and copy the job in any drive.

Elements defined with “exterior fixity”: footings, pile caps, strap and tie beams can be designed simultaneously or later on. All these foundation elements can be edited, modified, redesigned or checked in an isolated manner to the rest of the structure.

## 1.11.1 Consulting on screen

The following data can be consulted at any time.

### 1.11.1.1 Job general data

It is convenient that the data entered be revised: columns, groups (live loads, dead loads), floor heights, wind and seismic loading, materials used, options, reinforcement tables, etc. The options contained in this section are saved with the job, as well as those reinforcement tables that have been converted into Special tables. It is convenient these be saved separately because if options or tables have been modified and the job is reanalysed after some time, different results may be obtained.

If this data is modified, the job must be reanalysed. If they have been validated, the result consultation may continue. Options and tables may be modified and then redesign the structure to obtain a new result.

### 1.11.1.2 Results of regular beams and foundation beams

All beam data may be consulted:

- Active deflection and other deflection, deflection/span ratio, consideration of minimum moments.
- Beam envelopes with or without seismic loading, with bending moments, shear forces and torsion moments. All this can be measured numerically or graphically.
- Beam reinforcement, considering the number of bars, their diameter, their lengths and the stirrups with their lengths. These results may be modified. The design and necessary top and bottom reinforcement areas can be consulted, for longitudinal and transverse reinforcement.
- Beam errors: excessive deflection, bar spacing, anchor lengths, compressed reinforcement, and oblique compression due to shear and/or torsion and all the other inadequate design or reinforcement data. Colour codes can be assigned to evaluate their importance.
- Fixity coefficient at beam edges
- Sections designed using steel beam option and sections which verifies all the checks of the section series. In the case of composite steel beams, the shear studs are designed.

Beam sections may be modified. If the dimensions of the beams have varied, the **Redesign** option is available to obtain new reinforcement with same forces of the initial analysis. In this case the errors should be rechecked.

It is possible to redesign only those frames that have undergone dimension changes, conserving those where only the reinforcement has been modified, or redesign all, in which case the reinforcement of all the beams which have been modified is redesigned.

The reinforcement may also be blocked, then checked after another analysis has been run.

*If the dimensions of the beams have varied greatly, it is **highly recommended** the job be reanalysed.*

*The beam reinforcement may be modified, if the user wishes to do so and he/she is responsible for this. The program displays a colour code to verify the beam contains no errors. If beam dimensions have been modified in **Errors**, study whether it is more convenient to redesign to obtain new reinforcement.*

### 1.11.1.3 Loads

The values of all the loads entered: point, line and surface, can be visualised graphically. Each group of loads associated to different loadcases has a different colour code. This way, it is possible to check whether the data is correct. If any load modifications are undertaken, the job should be reanalysed.

### 1.11.1.4 Joist floor slab results

The following data can be consulted in regard to joist floor slabs:

- Bending moment and shear envelopes in joist alignments (with applied safety factor and per beam)
- Top reinforcement for joists. Their number, diameter and length are taken into account.
- Bending moments and shears at ends with applied safety factor and per metre width for joists.

The joist bending moments and shear can be made uniform in reference to mean values, percentage differences or maximum values. All the previous values can be modified for drawings, to those values considered by the user. Please consult chapter ***6. Joist floor slabs*** for more information on data and results.

### 1.11.1.5 Composite slab results

Please consult chapter ***9. Composite slabs***.

### 1.11.1.6 Hollow core slab results

The following may be consulted:

- Bending moment and shear envelopes for the selected panel strip and with a mean value per metre width.
- Type of selected hollow core plate,
- Top reinforcement at supports, indicating, depending on the activated views, the number, diameter, spacing and length of bars.
- Deflection information,
- Analysis errors, be they due to moments, shear, deflection or environment.

The type of hollow core plate can be modified, as well as the top reinforcement.

### 1.11.1.7 Results of flat slabs, waffle slabs and mat foundations

Data of entered slabs.

- Defined base reinforcement, and when required, that modified in the design.
- Discretised element mesh (see 3D model).
- Required reinforcement area envelopes displayed per metre width, in the direction of the defined reinforcement, top and bottom.
- Displacements in mm. For any loadcase of any node.
- Forces per loadcase of any node and required design steel area in each reinforcement direction. The method used to obtain the design forces is the *Wood* method, known internationally, required for the correct consideration of moments possessing both signs and torsion forces.
- Maximum displacements of panels for each loadcase. Not to be confused with deflection. In the case of mat foundations this indicates the settlement. If the values are positive, uplift is present, and the analysis would not be correct using the applied theory.
- Consultation of the reinforcement obtained in any longitudinal or transverse direction, top or bottom of the defined base reinforcement, if present.

- Check and design of punching shear reinforcement, if required, of solid areas and ribs of the lightweight zone.
- Matching of reinforcement in any direction to obtain maximum reinforcement areas and lengths.
- Excessive bearing pressures in mat foundations.
- Force, displacement and steel area, contour lines and contour maps.

If flexure lines have been entered before the analysis, minimum reinforcement lengths and bottom reinforcement overlaps must be provided, if required, in accordance with that indicated in the option for minimum lengths for flat and waffle slabs. It is **recommended** this introduction be done before the analysis, as if it is carried out later, the overlaps will be constructive (30 cm) and will not be redesigned.

All these modifications are carried out on screen and according to the user's criteria.

Flat and waffle slabs may be redesigned after the first analysis. By executing the **Redesign** option (entering a matching line) to obtain new reinforcement using the forces of the initial analysis.

#### 1.11.1.8 Column results

It is possible to consult column reinforcement and modify the columns' dimensions so to obtain a new reinforcement arrangement. Their reinforcement can also be modified. The forces (axial, moments, shear and torsion) in columns by loadcase may also be consulted at any point along the column, at any floor, as well as being able to visualise the force diagrams.

Likewise, the worst-case forces (with applied safety factor) which determine the reinforcement to be placed, for any span can be consulted (please recall that various worst-case combinations can exist for a specific reinforcement i.e. reinforcement may be valid for those forces but the reinforcement checked immediately before is not). Deformation and stress diagrams of the concrete and steel for a section perpendicular to the neutral axis are also provided. The resultant moments due to the amplification because of accidental and second order (buckling) eccentricity, which appear, in red, below the worst-case forces table.

If the column does not verify a check, an abbreviation appears to indicate why the column fails e.g. **SAe: Excessive ratio**: due to it exceeding the limits stated in the code, even though in this case, the program does provide column reinforcement.

More messages can appear which should be consulted.

If the reinforcement or the column dimensions are modified and the column still fails, a sign appears to the left indicating the maximum steel ratios have been exceeded.

If important modifications have been carried out it is highly recommended the job be reanalysed, as the stiffnesses will have varied.

Once the data has been consulted, the next phase consists in obtaining graphical results. If a column is present with insufficient section, it will not be drawn or measured.

Using the **Column schedule** option, columns can be grouped amongst each other. Those that fail are displayed in **red**.

A column's reinforcement can be blocked and maintained; hence after running through another analysis, it may be checked to see if it does not fail.

#### 1.11.1.9 Results of shear walls, reinforced concrete walls and masonry walls

The normal and tangential stress diagrams may be consulted for the whole height of the shear wall for each calculated combination, as well as displacement diagrams for the defined loadcases. The force distribution diagrams are displayed in colour scales, indicating the maximum and minimum values.

The reinforcement can be consulted and modified to the user's judgement, as well as the thickness. The wall is displayed in red if it fails. It may be redesigned.

A coded information text exists with messages explaining the state of the design.

The **Compliance factor of the reinforcement** provided may also be consulted, given as a %, and the areas where additional reinforcement is required.

A report of the worst-case forces of the span is also available as well as the forces per loadcase of the resultant.

#### 1.11.1.10 Results of the analysis with 2$^{nd}$ order effects

If 2$^{nd}$ order effects have been considered in the analysis, be it because of wind or seismic loading, the results of the analysis can be consulted and the amplification values of the applied forces and resultant factor due to the horizontal load of each intervening combination viewed on screen. All is explained when entering the data and a report of the results may also be printed out.

#### 1.11.1.11 Wind results

The values of the wind load in X and the wind load in Y at each floor level can be consulted and the results printed out.

#### 1.11.1.12 Earthquake results

The values of the vibration period for each mode considered can be consulted, the participation coefficient of the mobilised masses in each direction and the seismic coefficient corresponding to the resultant displacement spectrum.

#### 1.11.1.13 Contour maps and contour lines in flat and waffle slabs, and mat foundations

In this section of the program, the displacements, forces and steel areas in $cm^2/m$ may be consulted for all the panels of any group.

#### 1.11.1.14 Deformed shape

A 3D model can be observed displaying the deformed shape for each loadcase and combination, as well as its animated deformation.

### 1.11.2 Printed reports

The data entered and the analysis results are displayed in a report which can be printed out or saved as a text file. The following data can be printed out:

- General reports. These include the name of the job, groups, floors, heights, coordinates, column dimensions and their fixity, shear walls, gravitational load data, wind data, seismic data, materials used, control levels, slabs used in the job, geometry and self-weight.
- Report of combinations used in the analysis.
- Beam reinforcement report. Contains the necessary mechanical capacity envelopes, moments, shear forces, torsion, provided reinforcement and active deflection.
- Envelopes report, with bending moment, shear and torsion envelope drawings.
- Beam takeoff report.
- Beam fabrication tag list.
- Beam interchange file. This is a text file which includes information on beam reinforcement.
- Floor slab and beam, surface and volume report.

- Joist takeoff report for each type of joist and length.
- Form takeoff report.
- Joist floor slab reinforcement takeoff.
- Report on the reinforcement per square metre of the job.
- Flat and waffle slab reinforcement report.
- Report on forces in sloped beams, with moment, axial force and shear moments and provided reinforcement.
- Report on columns and shear walls, which includes the reinforcement report, forces at column starts, forces by loadcase and worst-case forces in columns and shear walls.
- Report on the displacements per loadcase for each column and at each floor.
- Second order effects report.
- Wind load report.
- Report on earthquake participation coefficients, which include mode periods, mobilised mass participation coefficient, and resultant seismic coefficient in each direction (dynamic analysis).
- Report on maximum displacements of columns, at each floor for all the columns, for the most unfavourable combination for each direction.
- Maximum column distortion report.
- Foundation report. Provides a report on the material data, loads and pad footing, pile cap and strap and tie beam geometry as well as their takeoff.

  A report on the design checks of the foundation elements is also obtained.
- Corbel report.
- Integrated 3D structures report.
- Column and beam ultimate limit state checks.

The reports are a complement to the graphical information obtained on screen, just as the drawings define the geometry and reinforcement of the project.

### 1.11.3 Drawings

The project drawings can be configured in different formats, be it a standard format or one defined by the user. Similarly for the paper sizes to be used. They can also be drawn using different peripherals: printer, plotter or DXF/ DWG and PDF files. They will have to be configured in Windows for them to work correctly and have the corresponding drivers installed.

Any type of construction detail or drawing in DXF or DWG format can be included, as well as using the edition resources the program allows: texts, lines, arcs, DXF. Any scale, line thickness, text size, title block, etc. can be applied, in such a way that the drawing can be completely personalised, including the active DXF or DWG template.

All the elements are defined by layers and the elements to be drawn in each drawing can be selected. Basically, the following drawings can be drawn:

1. **Layout plan.** Draws and dimensions all the elements per floor and in reference to the layout axes. Includes, as an option, the surfaces and volumes of slabs, as well as the reinforcement areas in the information block.
2. **Floor plans.** Geometry of all the elements on the floor: beams, columns, shear walls, walls, joist floor slabs (indicating positive moments and shear forces at joist ends, lengths and bars of additional top reinforcement), flat and waffle slab reinforcement, with a block detailing the base reinforcement in flat slabs, and drop panels and ribs of waffle slabs, punching shear reinforcement in solid and lightweight zones. A summary block can be detailed with their corresponding takeoff and total takeoffs. Drawings of the foundation elements can also be obtained.
3. **Beam drawings.** Drawing of the beam alignments, including their name, scales, dimension, reinforcement number, diameters, lengths and spacing, as well as their position, stirrup type, diameter and spacing. The detailing can be displayed in a summary block together with the total takeoff.
4. **Column and baseplate schedule.** Diagram of the column sections, indicating their references, position, stirrups, type, diameter, lengths, steel sections and is grouped by equal types. A block is provided containing the baseplates at steel column starts, with their dimensions, anchorage bolts and geometry. They can be drawn or selected by floor, as well as including a summary of their takeoff.
5. **Detailing of columns.** Drawing providing details on the columns and shear walls, including an elevation of the lengths and a block with the lengths of all the bars.
6. **Foundation loads.** Drawing of all the foundation starts with the loads at the starts (by loadcase), expressed in general axes. Columns and shear walls are included.
7. **Reinforced concrete and load bearing wall elevations.** Elevation of each wall span, with a reinforcement table for each span and floor, including an approximate takeoff.
8. **Load distribution.** Special applied loads are drawn by loadcase for each group.
9. **Corbels drawing.** The geometry and reinforcement are drawn.
10. **Contour lines.** The contour lines and contours are drawn for flat and waffle slabs.
11. **3D Structure.** This is drawn if the user possesses CYPE 3D and Integrated 3D structures have been created.

# 1.12 Design and check of elements

The parabola-rectangle and rectangular diagram methods are used for the design of reinforced concrete sections at ultimate limit state, together with stress-deformation diagrams of the concrete and for each type of steel, in accordance with the current code (see ***13. Code implementation*** chapter).

The limits required for both geometric and mechanical, minimum and maximum steel areas indicated in the codes are used. The required bar layout, regarding the number of steel bars to provide, minimum diameter and minimum and maximum spacing between bars, is also taken into account. These limits can be consulted and modified on screen in **Options**. Other parameters are saved in internal files.

## *1.12.1 Horizontal and inclined panel beams*

### *1.12.1.1 Longitudinal reinforcement due to bending*

The reinforcement is determined by carrying out a simple bending calculation at, at least, 14 points of each beam span, limited by the elements with which it contacts, be it joists, flat or waffle slabs. At each point, and based on the bending moment envelopes, the required top and bottom reinforcement (tension and compression reinforcement depending on the sign of the moments) is determined. This reinforcement is checked with the minimum geometric and mechanical values provided in the code, adopting the largest value. It is determined for seismic and non-seismic envelopes and the largest steel area obtain for both is placed.

#### *Bottom reinforcement*

Once the required design area is known for all the calculated points, the reinforcement sequence immediately after the required area is located in the bottom reinforcement table. The reinforcement tables are defined for the specified width and depth.

The total reinforcement in the reinforcement tables is divided into 3 types. Each one may have a different diameter. The first type consists of the reinforcement spanning between supports but goes beyond them and anchored in a constructive manner. In other words, the support axis passes up to the opposite face by at least 3 centimetres, except if (because the bottom reinforcement is close to or reaches the support or because compression reinforcement is required at the supports) it is necessary to anchor the reduced anchorage length as of the axis. The default reinforcement tables provide this first type of reinforcement whose steel are is always greater than a third or a quarter of the total reinforcement in the default reinforcement tables of the program. If the tables are modified, the user must ensure this proportion is maintained.

The second and third types may be of a smaller length, but are always symmetrical, complying with minimum percentage lengths (d and e in the figure) of the span specified in **Options**.

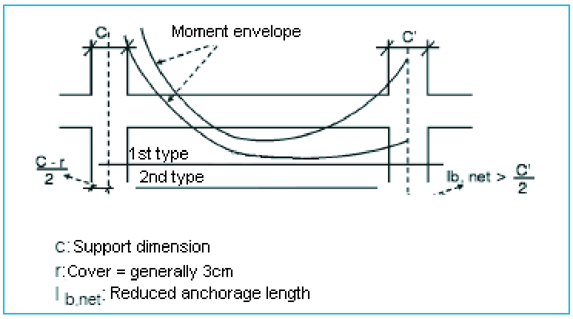



Figure 26

***Note:*** *The first type always passes beyond the support by 10 diameters measured from the face of the support.*

When a reinforcement combination cannot be found within the reinforcements table that covers the required steel area for the dimensions of the beam, $\phi$25 will be provided. The program will indicate: Bottom reinforcement outside table.

## *Top reinforcement*

There are two types of top reinforcement:

**Top reinforcement (in normal beams, bottom in foundation beams).** Once the required design area is known for the calculated points, the reinforcement sequence immediately preceding the necessary reinforcement in the top reinforcement table is placed. Reinforcement with up to three different cut off lengths can be placed; in Beam reinforcement options, a minimum span % can be defined for each group. The reinforcement tables are defined for the widths and depths specified in the table. The total reinforcement is divided into three types. Each type can be of a different diameter.

**Assembly reinforcement: Continuous or stirrup-holder.** Continuous assembly reinforcement is used when the steel of beams spanning from one support to the next is constructed in the workshop, including the top reinforcement and stirrups, and having to place the additional top reinforcement (or bottom in the case of foundation beams) at the supports on site. The user can choose whether or not to consider the assembly reinforcement as collaborating in regard to top reinforcement. When top compression reinforcement is required, it always collaborates. The anchorage of this assembly reinforcement is optional, with a hook or straight, as of its end or the axis, and is displayed clearly in the options dialogue box.

- For T sections, additional reinforcement is placed to hold the ends of the stirrups at the top of the T.
- The assembly stirrup holder reinforcement is used for in-situ assembly, by placing it between the ends of the top reinforcement, using small diameter bars and a construction splice with the reinforcement. This is required so there is reinforcement holding the stirrups. It may also be used in seismic zones where the splices of the nodes are to be placed further away. It is convenient it be selected and choose what is usually used.

When one which does not fail cannot be found in the reinforcement tables, the required number of 25 mm diameter bars will be placed. The program will emit the message: "outside table" be it for assembly reinforcement or regular reinforcement.

When the top reinforcement lengths at either side of a span are joined (consult **Options**) automatically, it becomes collaborating assembly reinforcement.

## *Other longitudinal reinforcement considerations*

Within the support zone of the column, a linear variation of the depth of the beam will be considered (1/3), which leads to a reduction of the required reinforcement. This will be the greatest obtained between the faces of the edges of the support, where the most usual case will consist of it being close to or at the edge of the support.

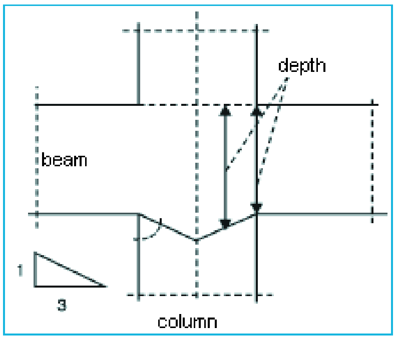



Figure 27

Regarding shear walls and walls, depending on the width of the side to which the beam reaches, a length or design span equal to the smaller of the following two values is calculated:

- The distance between shear wall axes (or mid-point of the axis of the cut beam)
- The free span (between faces) plus two times the depth

With these criteria, the envelopes within the shear wall are obtained and the cut-off length of the reinforcement, which do not exceed the design span by a value of two depths, is also obtained.

If skin reinforcement is required, due to the depth of the beam, which is defined in **Options**, it will be placed in the lateral faces with the defined diameter and minimum separation, in accordance with the code and that indicated in the options.

## *Longitudinal reinforcement due to torsion*

Once the longitudinal reinforcement due to bending is known, the required reinforcement due to torsion is calculated in accordance with the code, for each section. If the real reinforcement, placed at the corners, is able to absorb this increment with respect to that required due to bending, it will be sufficient. Otherwise, the longitudinal reinforcement will have to be increased and add reinforcement in the lateral faces, as if it consisted of skin reinforcement. The check for oblique compression due to torsion and shear is carried out at a distance of the effective depth from the edge of the support in accordance with the formulae of each code.

## *Longitudinal reinforcement cut-off*

Once the envelope of required top and bottom reinforcement area is known, an equivalent envelope is established whereby each point on the envelope has been displaced a distance equal to the effective depth plus the reduced net length (= anchorage length · required/real area) depending on its position (II = bad adherence, I = good adherence). Hence, a maximum length is established in its zone for each of the reinforcement groups in the unfavourable or decreasing force direction. These lengths are adjusted to defined minimum values which correspond to a percentage of the span and multiples of 5 cm. The reinforcement is anchored at the ends by means of hooks, calculating the necessary vertical leg, and providing a minimum if it is indicated in the options. At intermediate supports, the bottom reinforcement is anchored at either side of the support as of its axis, as well as of a distance equal to ten diameters measured from the face of the support (Figure 26).

When the maximum lengths of the bars are exceeded, the bars are cut and overlapped by a value equal to double the anchorage length. When earthquake loads are present, an option exists where the reinforcement is anchored and overlapped outside the confined zone next to the supports.

## *Transverse reinforcement (Stirrups)*

When designing for shear forces, a check for oblique compression is carried out at the edge of the direct support. The stirrups are designed as of the aforementioned edge or, optionally, at a distance equivalent to a percentage of the effective depth from the edge of the support (Figure 28). Minimum diameters and separations can be selected for the stirrups, or shear reinforcement, depending on the dimensions of the beam. Their layout can also be defined, whether they be laid out symmetrically and whether different calibres be used depending on their position along the beam. Simple (section perimeter), double, triple and vertical legs can be defined. Stirrups and legs can also be used jointly, up to two or three in the same section.

There are tables which can be defined by the user in which stirrups and legs can be used, as mentioned.

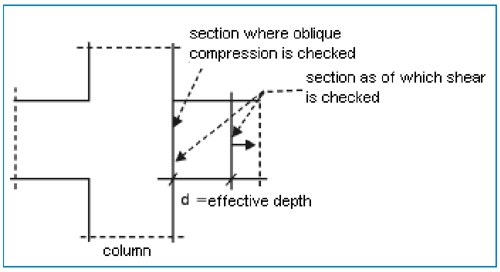



Figure 28

The minimum stirrups are established in accordance with the selected code, the section of the beam and the reinforcement table, checking the length it can cover of the shear force envelope in its central zone.

At lateral zones, to the left and right, the required stirrups up to the supports are established and are placed at their required length plus half an effective depth. It is checked that these lengths are greater than the minimum indicated lengths in the **Options**.

Finally, if torsion is present, the required transverse reinforcement due to torsion is calculated, whereby the minimum values are those stated in the corresponding code (minimum spacing, closed stirrups). This is added to that obtained for shear, resulting in a stirrup layout whose diameters, spacing and layout length covers the sum of both effects. For this last case, a joint check (oblique compression) of the tangential shear and torsion stresses is carried out.

*Columns starting on beams. Loads close to supports. Beams of great depth and width. In the special case of columns starting on beams (without external fixity), the vertical stirrups are designed using the value of the shear between the edge and the support of that span. It is important to remember that, in the special case of columns starting on beams or point loads close to supports, i.e. at a distance less than or equal to the effective depth, that a transmission of loads is produced by means of inclined struts in compression and tension requiring horizontal reinforcement, in the same way as in the case of a corbel, whose design criteria is not contemplated in the program. In this case, a manual design and check should be carried out of the span or spans at which this occurs, in accordance with that indicated in the code for these cases, as well as complementing the beam drawings with the corresponding additional details. It may also be resolved using inclined bars.*

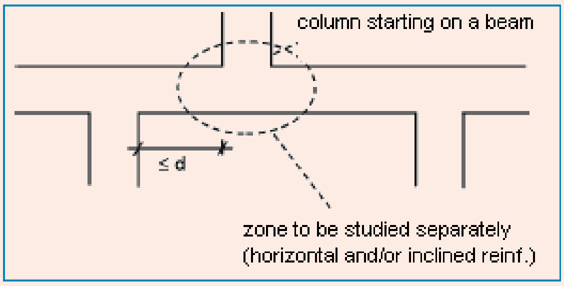



Figure 29

*Due to the importance this type of support possess and how fragile it is, it is fundamental that its control be extreme, in regards to its design and construction on site.*

It is checked that the spacing of the stirrups corresponds to that specified in the code when the longitudinal reinforcement is in compression, which affects the diameter and the maximum separation, depending on the longitudinal reinforcement in compression.

*The starts of columns without external fixity, i.e. columns starting on a beam, should be checked. It is recommended the fixity coefficient be reduced as much as possible at the base of the column in its first span, to avoid large diameters which result in large anchorage lengths at the starts.*

*When short spans of beams of great depth are present, the case may arise whereby the span is less than twice the depth, in which case, a manual check and design of the span where this occurs should be carried out.*

*It may also occur that the width is greater than twice its span. In this case, this wide beam is not really a beam or flat two-dimensional element or slab, whereby it is recommended the discretisation be checked and it be entered as a slab instead of a beam, as the design criteria are different.*

*Finally, recall that in the case of flat beams, where, due to their width, the width of the support is exceeded by more than a distance equivalent to the depth, a manual check for punching shear should be carried out as well as a verification of the stirrups at the support, reinforcing it with transverse reinforcement, if necessary.*

*If hanging loads applied below the neutral axis of the section are present, or point loads due to beams supported by other beams, the required reinforcement should be added to suspend these loads, as the program does not carry this out.*

## *Check for cracking in beams*

A limiting crack width can be defined optionally. The formulae used corresponds to the *CEB-FIP Model Code*. The characteristic width is calculated as:

$$W_k = 1.7 \cdot S_m \cdot E_{sm}$$

$$S_m = 2c + 0.2s + K_1 K_2 \frac{\phi A_{c,effective}}{A_s}$$

$$E_{sm} = \frac{\sigma_s}{E_s}\left[1 - \frac{K_3}{2.5\,K_1}\left(\frac{\sigma_{sr}}{\sigma_s}\right)^2\right] \le 0.4\frac{\sigma_s}{E_s}$$

where:

c: Cover of the reinforcement in tension

s: Spacing between bars. If s > 15 d, s = 15$\phi$

$K_1$: 0.4 (corrugated bars)

$K_2$: 0.125 (simple bending)

$A_S$: Total area of the bars within the effective area

$A_{c,effective}$: Effective area surrounding the reinforcement at a height of ¼ of the depth of the beam.

$\sigma_S$: Stress of the reinforcement

$\sigma_{Sr}$: Stress of the reinforcement at fissure point

$E_S$: Elasticity modulus of the steel

$K_3$: 0.5

This formula is generally applied, except for the *NB-1 code* and *Eurocode 2* which have their own specific formulas.

If the check is activated and is not verified, the bars are made longer, or the steel area is increased so it does comply. A warning message (not an error message) is emitted in the beam error list.

## 1.12.2 Sloped beams

They may be made of reinforcement concrete or steel. These elements are designed for bi-axial bending and compression, based on the bending moment and axial force envelopes and in the case of the stirrups, based on the shear force envelope. The design process consists in designing the reinforcement for the two planes parallel to the sides of the beam, i.e. for both, the horizontal and vertical planes.

The indicated top and bottom longitudinal reinforcement is the maximum or envelope of all the calculated sections along the length of the sloped beam. Reinforcement is drawn on drawings for this type of beam and can only be consulted on screen. The user must provide a separate detailed layout of its reinforcement for its end node reinforcement connections.

The envelope of these forces can be displayed in a job report if the user wishes for it to be so, as well as a description of the connection nodes.

If the beam is a steel beam, the same criteria that is applied for the design of steel columns is applied here.

## 1.12.3 Steel beams

These are designed in accordance with the corresponding code and steel type. The program proposes the optimum section within the series. They are designed for simple bending, as the axial force is not considered. Lateral buckling of the bottom flange, in the case of beams placed below the floor slab, can be optionally checked.

The corresponding design code and deflection limits and are applied as design criteria. The usage coefficient is expressed as a percentage with respect to the design code and deflection limits.

The design procedure for composite beams can be consulted in chapter ***8. Composite beams***.

Castellated beams are modelled as Vierendeel beams and are designed as rolled steel using the corresponding code.

The ultimate limit state check reports result to be very illustrative of the checks run by the program.

### 1.12.4 Columns, shear walls and reinforced concrete walls

Concrete column design is carried out by means of bi-axial compression. Based on the reinforcement table selected for the job, the reinforcement areas are checked sequentially. These may be arranged symmetrically at two sides, at four sides or with a percentage difference. The program also checks whether the reinforcement provided can withstand all the possible load combinations. Compatibility between forces and deformation is established as well as checking that the admissible stresses of the concrete and steel are not overcome or their deformation limits.

The minimum or accidental eccentricity is considered, as well as any additional eccentricity due to buckling in accordance with what is indicated in the code. Due to the application field of the applied formulas being limited because of the slenderness, if this is overcome, the section is insufficient (even though the user may manually enter reinforcement bars) resulting in the emission of a message indicating excessive slenderness.

The limits or minimum and maximum, geometric and mechanical, steel areas are defined in a hidden file within the program, so that these are complied with when designing the reinforcement. If the reinforcement fails and the maximum limits are overcome, this will be indicated in the report and on screen as excessive ratio SAe.

In this case the concrete section must be increased. If reinforcement cannot be found within the table which can cope with the design forces, the program will search for reinforcement, until the bars on one side do not fit in a single layer, in which case the message "*MANUAL REINFORCEMENT*" will be displayed. The type of reinforcement within the reinforcement tables should be increased and the column redesigned. In this case only those columns which require to be redesigned can be re-analysed without having to reanalyse the complete job. Alternatively, the section of the column can be increased, and the section is automatically redesigned.

Please recall that if the section dimension modifications vary greatly from how they were, it is convenient the job be completely re-analysed due to the stiffness variations. The diameters and spacing of the stirrups are designed in accordance with the default application code, with predefined arrangements defined in the reinforcement tables. These can be modified by the user as well as the spacing and diameters which depend on the longitudinal reinforcement.

There are reinforcement tables for which, depending on the vertical reinforcement, different stirrup and leg configurations can be defined as a function of the transverse dimensions, whereby different tables can be selected depending on the job. If a section does not have its stirrup arrangement defined in the table, only the perimeter stirrup will be provided.

The shear check that is carried out is that due to oblique compression and due to tensile failure of the web, depending on the provided reinforcement. A warning message is emitted if the reinforcement fails and is displayed in the column next to the stirrups.

Splice lengths are calculated as anchorage lengths in position 1 (good adherence), depending on the type of steel, concrete and consideration of dynamic loads. A reduction can be applied to the indicated anchorage length depending on the required and real reinforcement, without reducing that which has been reduced. These lengths can be edited and modified.

It is assumed that a column works predominantly in compression, and so if any columns are working in tension (as ties), the anchorage lengths must be increased manually and provide the appropriate complementary details.

Regarding the vertical reinforcement of a column, the reinforcement in its last and penultimate spans is designed in accordance with its forces from that point downwards, span by span, in such a way that the reinforcement of the span below will never be smaller than that provided in the span above, in case the corresponding continuity bar criteria be adopted within the **Options**.

The sections that are checked to obtain the reinforcement of a column at a floor are those indicated in Figure 30; the top and bottom of the current span, and bottom of the span above. If horizontal loads have been applied to the columns, intermediate sections will be checked as the force envelopes could increase.

When elevation changes are present, the same method is applied for each span in which the column is divided due to the elevation change.

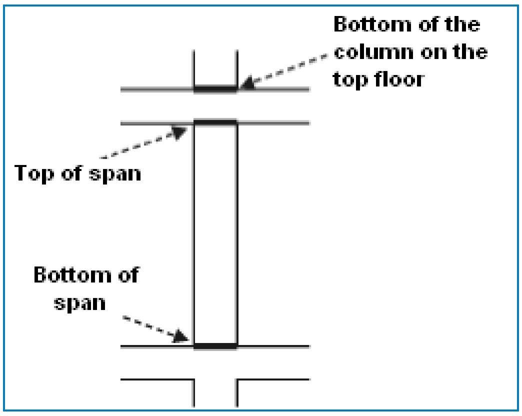



Figure 30

*If the reinforcement tables are modified, the stirrup arrangement must be checked. If no stirrups or reinforcement have been defined for the column section, complete the table providing the required stirrups and legs. Check the shear forces. The provided stirrups are checked and if these fail, a message is emitted: "Qe". In this case, the following steps must be taken until the stirrups no longer fail:*

- *Reduce the stirrup spacing.*
- *Increase the diameter.*
- *Propose other stirrup and/or leg arrangements. In this case the reinforcement tables must be modified and the columns redesigned.*
- *In some cases, increase the vertical reinforcement.*

The user can choose whether or not the reinforcement is to be continuous or not, as well as whether the diameter of the reinforcement in the corners or the number and diameter of the bars at the sides is to be kept.

Finally, the section may be modified, in which case the reinforcement is redesigned, as well as being able to modify the vertical reinforcement and the stirrup type.

#### 1.12.4.1 Steel columns

If steel columns have been defined, they are designed in accordance with the selected code for the type of steel, be it rolled or welded. The previously mentioned buckling coefficients must be entered by the user. If the criteria to maintain the current section is adopted, the column must be checked after the analysis so to ensure it does not fail in any way.

If, on the contrary, the user allows for the program to place the required section, the user must bear in mind that the design forces were obtained with the initial section and that if this varies considerably, it is recommended the job be re-analysed, as the forces can vary substantially.

Finally, the baseplates are designed at the start of the steel columns. In this case, the program verifies the general and local stresses of the steel, concrete, bolts, and checks for punching shear and tearing.

If the column starts on an element, other than a foundation element, the length of the anchorage bolts and anchorage conditions should be checked manually.

The span starts at each floor should also be checked whether it be at a beam or floor slab, as this is an important construction detail which is not contemplated in the analysis.

#### *1.12.4.1 Shear walls and reinforced concrete walls*

Having calculated the forces and for each combination, the stresses and deformations of the concrete and steel for the reinforcement provided in the tables is checked for each side of reinforcement. The program runs through the table in a sequential manner until it finds a reinforcement combination which does not fail for all the combinations. The program runs through a similar process in the transverse direction, calculating the additional reinforcement that must be provided if necessary. This process is repeated for each of the sides of the shear wall or wall.

The checks for minimum and maximum steel areas, maximum and minimum spacing, dimensions of the sides (the width of one sides must be greater than five times its thickness; the program emits a warning if this is not verified and the limits applicable to columns are used) are carried out in accordance with the selected application code. Slenderness limits for shear walls and walls are checked for each side, and if exceeded, the program emits a warning.

Finally, the reinforcement obtained can be checked on screen as well as any design errors that have been incurred. If the reinforcement and/or thickness is varied, a check is carried out. The program will emit any errors that arise. If the cross section is varied, the program can redesign the wall and hence obtain new reinforcement and undertake a new check.

Within the *Wall edition* dialogue box, a value is displayed indicating the **Compliance factor** of the wall. The default value is set at 90%. If a smaller value is displayed (for example 80%) and is redesigned, less reinforcement is obtained and red dots appear on the wall elevation. These **red** dots represent the 20% of the total surface of the wall which fails with that reinforcement.

By clicking on the **Show additional reinforcement** button, the additional reinforcement required at each red dot can be consulted. It will be the user’s decision whether or not to include this additional reinforcement, which will have to be added manually to the edited results in the drawings.

It is also possible to modify the reinforcement directly and calculate the compliance factor for the new reinforcement.

When a reinforcement arrangement fails, as well as the program displaying a warning message, the text is displayed in **red**.

Splices at each floor can be edited, and are calculated with a different length depending on whether it is tension or compression reinforcement.

#### 1.12.4.2 Masonry walls

The compressive stress and tensile stress (10% of the compressive) limits are checked with a compliance factor equal to 80%.

If it fails, a warning is emitted in the final analysis report.

### 1.12.5 Joist floor slabs

Joist floor slab analysis is carried out in an individualised manner for each joist for simple bending. The maximum value for the maximum positive bending moment, displayed in kNm per metre width and with the corresponding applied safety factor is provided. The bending moments can be matched for panels to the maximum or minimum values or mean values as a function of the percentage difference between adjacent joists, hence obtaining more uniform results.

The values of the moments can be displayed by type, expressed as a name, if the resisted moment values for each type have been indicated for that type of floor slab. If the values of the indicated table are exceeded, INSUF is displayed. In this case the table should be modified to include a broader range of values.

Negative moments are calculated for simple bending. The top reinforcement provided is obtained from the corresponding reinforcement table. Its length complies with the minimum length specified within the **Options**, as well as the minimum required area. This top reinforcement can be modified and matched as a function of the percentage difference in length.

When compression reinforcement is required in the negative moment area, forms will be removed up to the point where it is no longer required. This will be indicated on the floor by means of an infill line of the joists.

The ultimate moment and shear envelopes (with the applied safety factors) can be consulted on screen. At joist ends, even if the negative moment is null, reinforcement is provided, designed for a moment equal to the maximum positive moment of the span (see **Options**).

Minimum positive and negative moments can be defined for the whole job or one specific panel.

As the value of the positive moments is consulted, the check of whether compression reinforcement is required in the span is not carried out. Finally, the user is reminded that the expressed shear value at the ends of the joists is the ultimate shear force i.e. has the applied safety factors and per metre width.

Regarding the deflection, this is checked in accordance with the code depending on whether it is a prestressed or reinforced beam.

More information can be found in chapter ***6. Joist floor slabs*** of this *Calculations manual*.

### *1.12.6* Composite slabs

Please consult chapter ***9. Composite slabs*** of this *Calculations manual*.

### *1.12.7* Hollow core slabs

The analysis process has been explained in the ***1.9. Data entry*** section of this manual.

### *1.12.8* Flat slabs

#### *1.12.8.1* Base reinforcement

Top and bottom, longitudinal and transverse base reinforcement may be optionally defined. These may be different and can be defined and modified in a reinforcement table. This reinforcement always collaborates with the additional reinforcement as long as it has been defined. It may be increased, if having concluded the analysis the user observes more reinforcement due to bending is required, due to the reinforcement being in compression or because of minimum steel area requirements specified in the **Options**.

The user can choose whether or not to show the base reinforcement in the drawings. This is an important option as it has repercussions in the way in which the reinforcement is displayed as well as how it is measured for the take-off reports. If the base reinforcement is shown, it will be displayed with the additional reinforcement, cutting off and overlapping where necessary as if it were one more reinforcing element. Its take-off and cut-off lengths can be obtained. If the base reinforcement is not shown, it will not be drawn or measured; only its diameter and spacing will be indicated. Therefore, in this case, it should be complemented with the opportune details, for floor layout drawings and in the take-off schedule.

#### *1.12.8.2* Additional longitudinal reinforcement

The bending moments in the two directions and the torsional moment are known for each node of the mesh. Generally, the main directions of the slab do not coincide with the reinforcement directions. By applying the Wood method, internationally known and which considers the effect of torsion to obtain the reinforcement moment in each specified direction, a transverse distribution is carried out at each node with its adjacent nodes to the left and to the right, in a metre wide strip. The forces of the node plus those of the distribution are obtained at each node, and it is with these forces that the required top and bottom reinforcement areas are obtained in each direction. These reinforcement areas are specified per metre width which are then divided by the distance between the nodes or the size of the mesh, to obtain a homogenous value at all the nodes.

It is checked that the minimum required steel areas are complied with in each direction and per metre width, and then the additional longitudinal reinforcement is calculated in accordance with the defined reinforcement tables. The cut-off point of the bars is carried out by increasing this length by the net reduced anchorage length, depending on its position (I or II) and the shifting of the envelope depending on its effective depth and in accordance with the code.

Compliance of the maximum diameters and spacing is carried out by means of the reinforcement table, in which the diameters and spacing values are specified for varying depths. The program offers the option on whether to consider torsion, however it is recommended it always be considered.

#### *1.12.8.3* Pre-established reinforcement

This reinforcement corresponds to the reinforcement, be it top or bottom reinforcement or in any direction, with any diameter or length which has been pre-established by the user. This reinforcement will be deducted, in its influence zone, from the additional reinforcement that is to be placed. This is useful in zones where the forces are already known, such as the top zone of supports, allowing for the rest of the reinforcement to be more uniform.

Mat foundations are treated in exactly the same way as normal flat slabs in regards to their reinforcement design.

### 1.12.8.4 *Transverse reinforcement*

#### ***Punching shear***

The punching shear is checked, in accordance with the code, at all surfaces parallel to support edges, such as columns, shear walls, walls, beams and external fixity supports, within a distance of half the effective depth of the slab (0.5d). It must not be forgotten that the punching shear check is a check of the tangential stresses. This is what the program carries out, and hence obtains the value of the tangential stresses based on the shear forces of the closest nodes, and linearly interpolating at the points of the punching shear perimeter.

This approach is correct, looking at it from a theoretical point of view: a check of the tangential stresses, generally resolves the problem. This does not coincide with the formulae provided by some codes which tend to apply formulae which depend on the acting axial force and moment, with simplified formulae which only resolve specific cases.
If failure occurs due to the punching shear, a red line will appear indicating the maximum punching shear stress has been overcome. In this case, the depth or concrete resistance should be increased.

If the limiting stress without transverse reinforcement is exceeded, additional transverse reinforcement must be provided. The program indicates the number of reinforcement bars, with their corresponding diameter, which are to be provided at the required intervals, which depend on the number of bars placed in a specific length.

The user should, in this case, place the bars vertically in the most convenient structural arrangement for the job (Figure 31), in such a way that their spacing does not exceed 0.75 times the effective depth or the equivalent section, and placed between the top and bottom reinforcement.

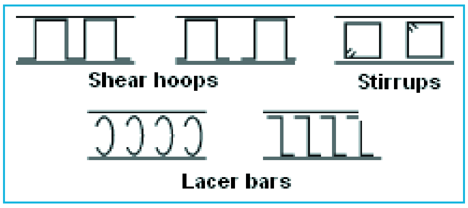



Figure 31

In areas where beams are present, be they flat or dropped beams, the tangential forces will be resisted by the stirrups of the beam. Therefore, the tangential stresses are only calculated for the slab and for surfaces parallel to the sides of the beams.

### *Shear*

A shear check is carried out on all the surface of the slab for the same section used for the punching shear check (0.5d) and for parallel surfaces at a distance of 0.75d, until all the radiated surfaces are found as of the edges of the support. If reinforcement is required, the number and diameter of the reinforcement to be provided is displayed in the same way as is done for the punching shear reinforcement.

Similarly, if the check fails, a red line will appear indicating the maximum punching shear stress has been exceeded. In this case, the depth of the slab or the size of the support or the resistance of the concrete should be increased.

Mat foundations are treated in exactly the same way as normal flat slabs in regards to their reinforcement design.

### *Reinforcement matching*

Lines or rectangles may be defined in any direction, for the top or bottom reinforcement, before or after the analysis which allow for the reinforcement to be matched to the maximum reinforcement (in steel area and length) in that zone. An option exists for the automatic matching of the top reinforcement over columns in the areas adjacent to the indicated columns.

Flexure lines can be defined, which should be used before the analysis and entered in the directions of the supports.

These lines are taken as being maximum negative moment points, and therefore the ideal place for bottom reinforcement overlaps, if required. The additional top reinforcement lengths are calculated in accordance with minimum percentage values of the distance between the lines (span) and overlapping the bottom reinforcement, if possible at these lines.

Finally, the diameter and separation of the additional reinforcement can always be modified by the user (up to his/her criteria) and also modify and place the top and bottom anchorage hooks.

### *Reinforcement anchorage at beams or supports*

The anchorage lengths are measured as of the edge of the support adjacent to the slab. Check the lengths when the edges are wide, as they may possibly not completely cross the beam and remain partially anchored. This is important, and the reinforcement must be extended when wide beams are used.

Mat foundations are treated in exactly the same way as normal flat slabs in regards to their reinforcement design.

An option is available whereby in the case of rectangular flat slabs, supported by beams, the program provides uniformly distributed mean reinforcement in both directions.

## 1.12.9 Waffle slabs

The criteria used for the design of waffle slabs is the same as that used for flat slab design, with the following exceptions.

### 1.12.9.1 Base reinforcement

The user can choose whether or not to define a base reinforcement, with the further option of providing a different base reinforcement for lightweight and drop panel zones.

#### *Drop panel base reinforcement*

The program automatically places a default reinforcement mesh composed of 2 bars, in accordance with the tables, which is extended between the two edges of the drop panel and distributed between the axes of the ribs. This reinforcement always collaborates if it is considered.

This reinforcement is not measured or drawn in the current CYPECAD version. Therefore, the user must provide a detail indicating the base reinforcement that is to be provided to complement the information indicated in the drawings.

#### *Rib base reinforcement*

The default setting of the program is set so it does not automatically provide this reinforcement. Therefore, it must be chosen and established for each direction. Reinforcement tables are available so it may be defined, as well as how it can be combined with additional reinforcement bars to be placed at the ribs. Within the **Options** section of the program, the user is to indicate if it is to be displayed, drawn and/or measured. Otherwise, a general label will be provided but will not be measured or drawn.

### 1.12.9.2 Additional longitudinal reinforcement

The same criteria is applied as in the case of flat slabs, only the reinforcement is concentrated in the ribs. The envelopes of the adjacent elements must previously be grouped for the concentrated calculation of the reinforcement at the position of the rib.

### 1.12.9.1 Transverse reinforcement

An identical analysis to that of flat slabs for punching shear is undertaken for drop panel zones.

For the ribs of the lightweight zone, a shear check is carried out every 0.75d. If additional reinforcement is required, vertical bars with the required diameter and spacing are placed, which are then drawn on the drawings and are visible on screen.

*Please recall that local additional reinforcement is provided for punching shear, which should be checked and modified where appropriate to provide more uniform results with the aim to facilitate work on site. It is recommended the construction details of the CYPE library be consulted.*

### 1.12.9.2 Reinforcement matching

The reinforcement may be matched in the same way as flat slab reinforcement, concentrating the reinforcement in the designated ribs.

## 1.13 Beam deflections

The following deflection limits may be optionally defined:

- Instantaneous deflection:
  Self weight
  Live load
  Total
- Total long term deflection
- Active deflection

The relative L/xxx or L/xxx + xx cm; or the absolute deflection in centimetres can be limited for each of these deflections.

Each code can establish different limits, and the user can fix these values to what he/she deems appropriate for each analysis.

The most usual is the active and long term deflection.

To establish the active deflection and total long term deflection, the coefficients to apply, depending on the respective construction process, have been indicated in the options, for permanent loads and live loads. These multiply the instantaneous deflections to obtain the differed deflections.

The total deflection is the sum of the instantaneous deflections plus the differed deflections for each case.

The maximum active and total deflections are established for beams using the double integration method. Upon analysing a series of points, the gross, homogenised, fissured, inertia is found and the rotation for each loadcase, obtained from the curvature variation diagram.

The program calculates the forces and displacements for each loadcase, based on the secant longitudinal elasticity module of the concrete. If this module is to be reduced due to environmental factors, the curing process, etc., it must be modified by the user by changing the corresponding construction process option coefficients to be applied to the instantaneous and differed deflections.

The first deflection that is obtained, the active deflection, is the instantaneous deflection plus the differed deflection due to the permanent loads and the variable loads (after building the partitions). The coefficients that depend on the construction process to be used (or factors of the instantaneous deflection) to calculate the beam deflections may be consulted in the general options section of the program, as well as the default values.

The deflection is calculated using the indicated method due to the permanent loads ($f_g$) and the variable loads ($f_q$). The total active deflection will be:

$$f_A = \alpha_g \cdot f_g + \alpha_q \cdot f_q$$

Where:

$\alpha_g$: Permanent load coefficient

$\alpha_q$: Variable load coefficient

These values may be varied as a percentage of the loads defined as permanent and variable loads within the **Beam options**, as well as the coefficients used to define its instantaneous or differed effect.

The total long term deflection is equal to the active deflection plus that which occurs until the damaging element is built (usually a partition).

It is recommended the respective code and project control companies be consulted so these coefficients may be defined correctly as there are many factors such as the construction process to be carried out, the level of humidity, the curing of the concrete, period of formwork removal, age at which the element becomes completely load bearing, affect the value of the deflection. Therefore the values indicated in the program are for guidance purposes and can be used for usual favourable construction conditions.

# 1.14 Slab deflections

## 1.14.1 *One-way spanning slabs*

The same principles can be applied to joist floor slabs and hollow core slabs as to beams even though their options can be accessed independently within the floor slab options. The stiffnesses considered for precast elements are obtained from their properties sheet. For the remaining cases, the equivalent inertias are calculated.

## 1.14.2 *Composite slabs*

Please consult chapter ***9. Composite slabs*** of this *Calculations manual* as well as that indicated in the previous point.

## 1.14.3 *Flat and waffle slabs*

The deflection values for each simple loadcase (those that have been defined in the job: permanent or self weight; variable, which include live loads; wind and seismic loads) are obtained for all the floors at any node of the mesh. The maximum displacement for each loadcase can be obtained for each slab.

It is left to the user's judgement to estimate the active deflection, using the creep coefficients he/she deems adequate, and based on a manual calculation of the known instantaneous deflections, deduced from the vertical displacements for each loadcase provided by the program.

Please recall that the vertical displacements of a flat slab are absolute values, i.e. a node next to a column or support is consulted, it will also have an associated displacement (in the z-axis). To then establish the deflection between supports, the displacements of the supports are to be subtracted, as the deflection is a descent relative to the end supports, or inflexion points in a given deflection direction. This effect is greater for higher floors because of the elastic shortening of the concrete columns.

If the column displacements are very small, the deflection can be taken as being the sum of the displacements due to gravitational loads (dead loads + live loads) multiplied by a value between 1.5 and 4, depending on the construction process and type of deflection to establish. This way, approximate values can be obtained for when designing a building.

Once the absolute deflection has been found, the relative deflection (L/xxx) can be determined, bearing in mind the supports of the zones adjacent to the point with maximum absolute deflection and taking the smallest span amongst all those possible.

## 1.14.4 Deflection between 2 points

### 1.14.4.1 Introduction

Upon marking an initial and final point on the plan view of a flat or waffle slab, a continuous yellow line appears joining the two points displaying the vertical displacements of all the points creating an approximately sinuous curve below the line. The blue curve displays the deformed shape of the slab, for a simple loadcase, combination or worst case combination (G+Q) with amplified displacements.

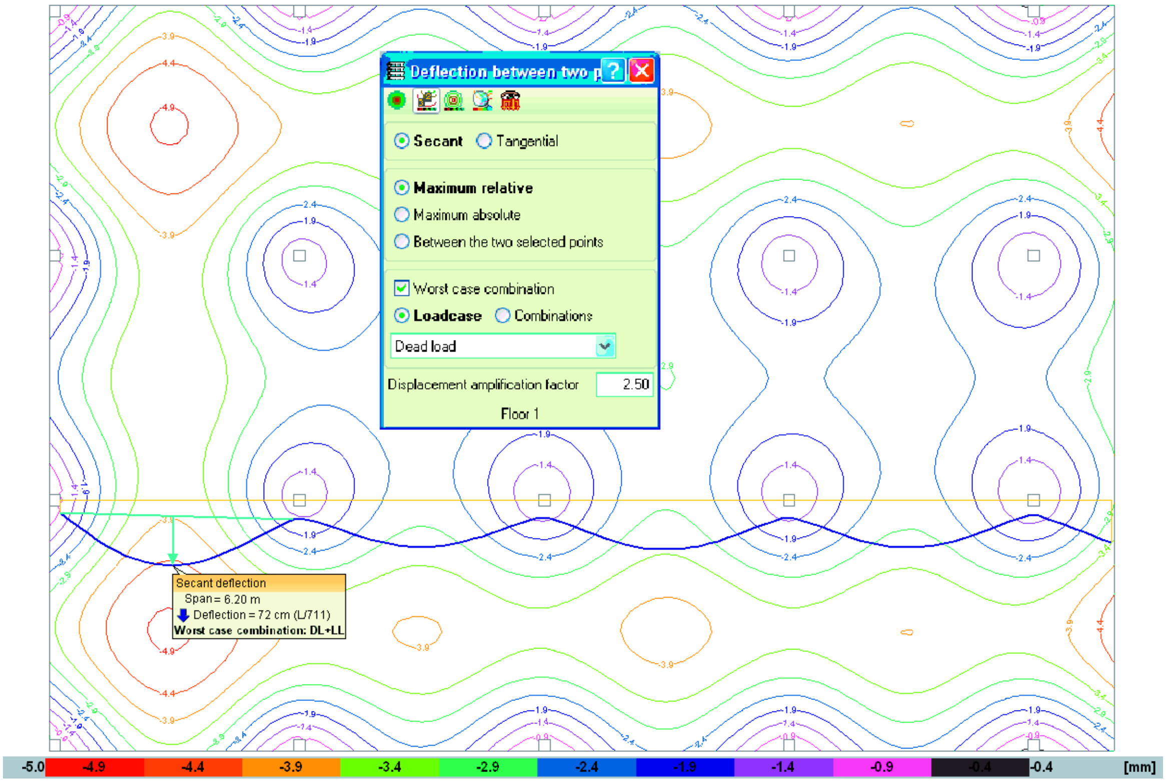



Figure 32

When the floor layout of the supports is visible on screen and the two points to calculate the deflection are marked, the user can observe that convex distributions are present in the support zones whereas at span centres concave distributions are visible.

Let us study what occurs in the case of displacements in the Z direction (vertical) corresponding to the combination G+Q of the permanent load G loadcase (which usually represents the greatest percentage) and live load Q loadcase to observe where the maximum values are produced. At a first glance, looking at the displacement contour lines, the maximum values coincide with the areas with greatest concave distributions.

In the case of a two-way spanning slab, we do not know beforehand whether the area with greatest concave distribution corresponds to that of the maximum absolute deflection. It is logical to think it should be, as the descent of the supports is generally small, and we should concentrate on those areas where these maximum values can be seen. There is also the uncertainty as to in which direction (X, Y, diagonal) the points should be marked to obtain the maximum relative deflection. However, one could say, from observing the layout of a floor, that it should be in the direction of the smallest distance between two points of the concave distribution perimeter.

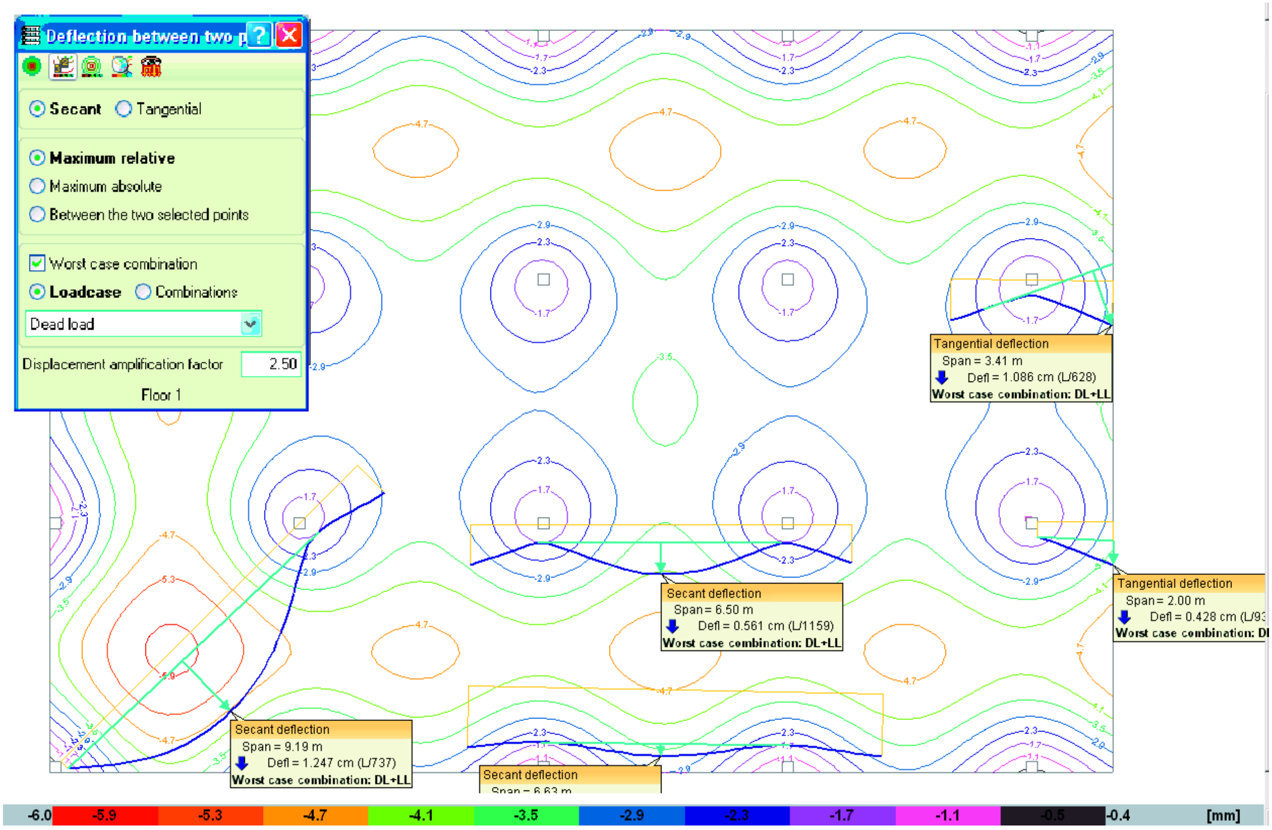



Figure 33

It is recommended it generally be done in this manner.

To find the edges of the concave distribution perimeters, it may be useful to previously select points which lie further away so to find their position.

Let us carry out a few simple examples so to clarify what has been explained. The data is as follows:

- Mean span = 6.50 m
- External span

- Approximate moment distribution:

  Negative moment internal support $M^- = \frac{pl^2}{10}$

  Positive moment for span $M^+ = \frac{pl^2}{17}$

  Negative moment external support $M^- = \frac{pl^2}{30}$

- Partitions = 1 kN/m$^2$ built after 60 days (2 months)

  - Screed = 1 kN/m$^2$ laid after 120 days (4 months)
  - Live load = 2 kN/m$^2$
  - Materials = HA-25 ($f_{ck}$=25 MPa) y B500 ($f_{yk}$=500 MPa), mean temperature and humidity conditions during building process and use.

### 1.14.4.2 1$^{st}$ case: Flat slab

Let us estimate a depth of 26 cm, whose slenderness is L/h = 650/26 = 25. Its deflection has to be checked. According to table 50.2.2.1.a of the *Spanish EHE-08 code*, the value for the slenderness is 23 (for weakly reinforced elements) and so its verification must be justified.

Let us carry out an approximate analysis on a metre wide section, with the aforementioned properties and to obtain the forces acting on it. Even though the support distribution is that of a flat slab with isolated supports, we shall assume there are uniform supports so to simplify the analysis.

The gross inertia of the section of the flat slab is $I_b = bh^3/12$ therefore $I_b = 100 \cdot 26^3/12 = 146\,466.67\ \text{cm}^4$ in our case. The stiffness of the model can be calculated using this value and the instantaneous elastic displacements for each selected loadcase and combination are obtained.

To establish the deflection between two points based on the displacements, the displacements of the initial and final points must be deducted.

The graph shows a line which passes through those two points (secant) or a line at a tangent to the initial point (tangential), depending on the section carried out. The deflection is measured with respect to that line.

The values displayed are:

- Span L

- Absolute deflection
- Relative deflection with respect to: L (secant); 2L (tangent)
- Worst case combination G+Q obtained from the selected **Displacements combinations** in the general data section of the job. Only gravitational loadcases G and Q are considered (activated by default). It may also be selected by loadcase or combination loadcase.

A "Displacement amplification factor" = 2.50 (default value) is displayed. This does exactly as it says; multiplies the displacements due to that loadcase by that value and displays the result. This amplification factor should be incorporated and all the parameters relative to the following be taken into account:

- Period of formwork removal
- Construction process
- Thermal phenomenon
- Rheological phenomenon (creep, retraction)

Once the instantaneous elastic deflection is known, the total deflection or active deflection (after the damageable elements have been built, for example partitions or rigid paving without joints) can be found, by estimating the aforementioned parameters. By placing the adequate value, the deflection sought will be obtained and so can be compared with the limits stated in the code.
Assuming a usual construction process, with the corresponding building loads:

- Self weight of the slab (h=26 cm) = 6.5 kN/m$^2$ unpropped after 28 days (1 month)

  Using the laws applicable to an external span (as if virtual frames were to be applied), the bending moment diagram and the required reinforcement per metre width can be found.

$$Md^-_{left} = [1.35 \times 8.5 + 1.5 \times 2] \times \frac{6.5^2 \times 2 \times 0.76}{10} = 14.475 \times \frac{42.25 \times 2 \times 0.76}{10} = \frac{930.2}{10} = 93.02\ \mathrm{kN \cdot m/m}$$

  Hence an providing an approximate area, $A^-_{snec} = 10.2\ \mathrm{cm^2/m}$

$$Md^+_{centre} = \frac{14.475 \times 42.25 \times 2 \times 0.6}{17} = 43.16\ \mathrm{kN \cdot m} \rightarrow A^+_{snec} = 4.8\ \mathrm{cm^2/m}$$

$$Md^-_{right} = \frac{930.2}{30} = 31\ \mathrm{kN \cdot m} \rightarrow A_{snec} = 3.4\ \mathrm{cm^2/m}$$

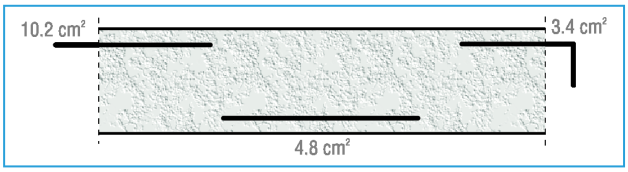



Figure 34

Now to calculate the cracking moment:

$$M_{crack} = f_{ct,m,fl} \cdot W_b$$

$$f_{ct,m,fl} = \max\left\{\left(1.6 - \frac{h}{1000}\right) \cdot f_{ct,m}; f_{ct,m}\right\}$$

Where $f_{ct,m,fl}$: Mean tensile bending resistance of the concrete

$f_{ct,m}$: Mean tensile resistance of the concrete

$$f_{ct,m} = 0.3\, f_{ck}^{2/3} = 0.3 \times 25^{2/3} = 2.565\ \mathrm{N/mm^2}$$

$$1.6 - \frac{260}{1000} = 1.6 - 0.26 = 1.34$$

$$f_{ct,m,fl} = 1.34 \times 2.565 = 3.437\ \mathrm{N/mm^2}$$

$$W_b = \frac{bh^2}{6} = 1000 \times \frac{260^2}{6} = 11\,266\,667\ \mathrm{mm^3}$$

$$M_{crack} = 3.437 \times 11\,266\,667 \times 10^{-6}\ \mathrm{kN \cdot m} = 38.72\ \mathrm{kN \cdot m}$$

An added safety factor is applied by considering the service moments due to the total load:

$$M^{-}_{left} = \frac{10.5 \times 42.25 \times 2 \times 0.76}{10} = 67.4 > 38.72\ \mathrm{kN \cdot m}\ \ \text{CRACKS}$$

$$M^{+}_{centre} = \frac{10.5 \times 42.25 \times 2 \times 0.6}{17} = 31.3 < 38.72\ \mathrm{kN \cdot m}\ \ \text{DOES NOT CRACK}$$

$$M^{-}_{right} = \frac{10.5 \times 42.25 \times 2 \times 0.76}{30} = 22.48 < 38.72\ \mathrm{kN \cdot m}\ \ \text{DOES NOT CRACK}$$

To determine the cracked inertia $I_{crack}$ and Branson's equivalent inertia $I_e$:

$$I_e = \left(\frac{M_{crack}}{M_a}\right)^3 \cdot I_g + \left[1 - \left(\frac{M_{crack}}{M_a}\right)^3\right] \cdot I_{crack}$$

where:

$I_e$: Equivalent inertia

$M_{crack}$: Cracking moment

$M_a$: Moment acting on the section

$I_g$: Gross inertia

$I_{crack}$: Cracked inertia

This shall be carried out on the 3 sections to be studied:

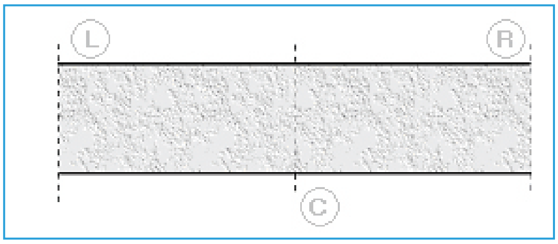



Figure 35

And the mean equivalent inertia of the span will be determined:

$$I_e = \frac{\left(I_e^L + I_e^R\right)/2 + I_e^C}{2} = 122\ 768\ cm^4/m$$

Where:

$$I_e^L = \left(\frac{38.72}{67.4}\right)^3 \times 146\ 466 + \left[1 - \left(\frac{38.72}{67.4}\right)^3\right] \times 29\ 584 = 51\ 744\ cm^4/m$$

$$I_e^R = I_e^C = I_b = 146\ 466\ cm^4/m$$

$$I_e = 0.84\ I_g$$

To be on the safe side and by reducing the equivalent inertia to $I_e \approx 0.66 \cdot I_g$, a correction coefficient due to the cracking of the sections is to be applied to the elastic deflection provided by the program. This coefficient $K_e$, has a value of:

$$K_e = \frac{I_g}{I_e} = \frac{I_g}{0.66 I_g} = \frac{1}{0.66} \approx 1.50$$

If the method described in the *Spanish EHE-08 code* (similar to *ACI-08*), where the differed deflection is assumed to be proportional to the instantaneous deflection by a factor $\lambda$ whose value is:

$$\lambda = \frac{\xi}{1 + 50 \cdot \rho}$$

$\xi$: Coefficient which depends on the time or duration of the load
$\rho$: Mean area of reinforcement in compression

If a top and bottom mesh is provided which will cover, for example 1‰ ($\rho$ = 0.001), $\xi$ will take the following values:

| t | $\xi$ |
|---|---|
| $\infty \geq$ 5 years | 2.0 |
| 1 year | 1.4 |
| 6 months | 1.2 |
| 1 month | 0.7 |
| 2 weeks | 0.5 |

$$\lambda = \frac{1}{1+50\times 0.001}\cdot \xi = 0.95\ \xi$$

Let us now look at the construction process and the deflections that are produced.

First of all, assume the total deflection:

| **Permanent load** | **Value** | **t** | $\xi$ | $\xi = \xi(\infty) - \xi$ (t) |
|---|---|---|---|---|
| | **kN/m$^2$** | **days** | | |
| Self weight | 6.5 | 28 | 2–0.7 = 1.3 | |
| Partitions | 1.0 | 60 | 2–0.8 = 1.2 | t: when load begins to act |
| Screed | 1.0 | 120 | 2–1 = 1 | |
| Live loads | 1.0 | 365 | 2–1.4 = 0.6 | |

With the assumed live load, only the quasi-permanent part produces differed deflection (for dwellings $\Psi_2$=0.3). By applying it to the total:

$$\xi_{mean} = \frac{1.3\times 6.5 + 1.2\times 1 + 1\times 1 + 0.6\left(0.3\times 2\right)}{6.5+1.0+1.0+2.0} = \frac{11.01}{10.5} \approx 1.05$$

Now, $\lambda = 1.05\times 0.95 \simeq 1.00$

To obtain the instantaneous deflection corrected by the reduction of the inertia due to cracking, the value of the instantaneous deflection $\delta_i$ is multiplied by $K_e$:

$$\delta_{inst} = K_e \cdot \delta_i = 1.50\ \delta_i$$

On the other hand, for the total differed deflection a coefficient of $\lambda$ = 1.00 has been obtained, hence

$$\delta_{dif} = \lambda \cdot \delta_{inst} = 1.00 \cdot \delta_{inst}$$

and the total deflection is:

$$\delta_{total} = \delta_{inst} + \delta_{dif} = 1.5\ \delta_i + 1.00 \times 1.5\ \delta_i = 3.00\ \delta_i$$

This value of 3.00 is precisely that coefficient which is presented as **Displacement amplification factor** (whose default value is set at 2.5).

*CYPECAD's user's manual* mentions that this value can lie between 2.5 and 3.0, depending on the construction process.

For this example the value obtained has been 3.00, amount which is to be entered in the dialogue box to consult the total deflection and view the selected results between two points.

Similarly, to determine the active deflection, in this case, that which arises after the damageable element has been built, the deflection which occurs before it is built must be subtracted. In this example, the instantaneous deflection due to the self weight of the slab and the differed deflection since removing the props until the partitions were built after 60 days have to be removed. Hence, $\xi_{s.weight}$ = (2–0.8) = 1.2 and so:

$$\xi_{mean} = \frac{1.2 \times 6.5 + 1.2 \times 1 + 1 \times 1 + 0.6\left(0.3 \times 2\right)}{6.5 + 1.0 + 1.0 + 2.0} = \frac{10.36}{10.5} = 0.987$$

$$\lambda = \xi_{mean} \times 0.95 = 0.987 \times 0.95 = 0.93$$

Therefore the differed deflection will be $\delta_{dif}$ = 0.93 $\delta_{inst}$

Regarding the instantaneous deflection to be considered, the corresponding part due to the self weight must be removed. Hence by the proportion of the loads without the self weight to the total loads:

$$\%\delta_i = \frac{1.0 + 1.0 + 2.0}{6.5 + 1.0 + 1.0 + 2.0} = \frac{4}{10.5} = 0.38$$

Therefore:

$$\delta_{inst} = 0.38 \times 1.5 \times \delta_i$$

$$\delta_{active} = 0.38 \times 1.5\ \delta_i + 0.93 \times 1.5\ \delta_i = \left(0.38 \times 1.5 + 0.93 \times 1.5\right)\delta_i = 1.965\ \delta_i$$

This would be the value to enter in the *Displacement amplification factor* box to obtain the active deflection.

Obviously, it can be seen that for the foreseen loads and construction periods, by rounding off to whole values, a flat slab has an active deflection of approximately 2.0 times the instantaneous elastic deflection established using the gross section and a total deflection of approximately 3.0 times that instantaneous deflection.

### 1.14.4.3 2$^{nd}$ case: Waffle slab with lost forms

A similar estimate will now be undertaken for a waffle slab entered in CYPECAD.

The lost form waffle slab for this example will have the following geometry:

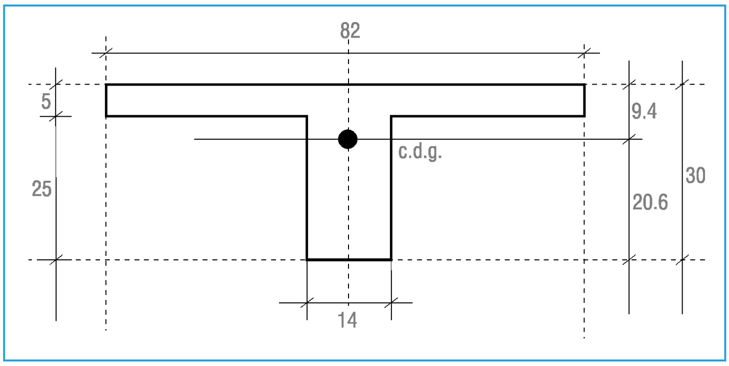



Figure 36

Having found the centre of gravity of the section, the inertia of the gross section with respect to this centre of gravity can be calculated:

$$I_g^{lightweight} = 61\ 567\ cm^4$$

By repeating the same process with the drop panel zone, it can be seen that it resembles a flat slab with a depth of 30 cm and a width of 82 cm:

$$I_g^{d.panel} = 184\ 500\ cm^4$$

The mean inertia of the section along its span could be estimated to be:

$$I_g^{mean} = \frac{61\ 567 + 184\ 500}{2} = 123\ 033.5\ cm^4$$

In the CYPECAD model, the inertia applied to all the mesh of a panel of any waffle slab, be it in the lightweight or solid zone, is the same and is taken as half of that of the solid zone, which in this case is:

$$I_g^{model} = \frac{I_b^{d.panel}}{2} = \frac{184\ 500}{2} = 92\ 250\ cm^4$$

and it is with this inertia that the elastic displacements for each loadcase are found.

The next step would be to verify whether the sections crack or not due to the forces acting on it and establish the equivalent inertia.

For the solid zone and taking a width = 0.82 m, the cracking moment is:

$$M^-_{crack} = 40.96\ kN{\cdot}m$$

and for the lightweight zone:

$$M^+_{span\ crack} = 10.27\ kN{\cdot}m$$

Let us estimate the moments acting on the structure:

| | |
|---|---|
| Slab self weight: | 4.6 kN/m$^2$ (*) |
| Partitions: | 1.0 kN/m$^2$ |
| Screed: | 1.0 kN/m$^2$ |
| Live load: | 2.0 kN/m$^2$ |
| | 8.6 kN/m$^2$ |

(*) The lightweight zone has been considered, as the solid zone has little influence on the acting moments.

The moments for hypothetically supported rib would be:

$$M^-_{left} = 8.6 \times \frac{6.50^2}{10} \times 0.82 \times (2 \times 0.76) = 45.2\ kN{\cdot}m \geq M_{crack}\,(40.96\ kN{\cdot}m)$$

$$M^-_{right} = 8.6 \times \frac{6.50^2}{30} \times 0.82 \times (2 \times 0.76) = 15.1\ kN{\cdot}m < M_{crack}\,(40.96\ kN{\cdot}m)$$

$$M^+_{span} = 8.6 \times \frac{6.50^2}{17} \times 0.82 \times (2 \times 0.6) = 20.1\ kN{\cdot}m > M_{crack}\,(10.27\ kN{\cdot}m)$$

It can be seen that there is very little cracking in the negative moment zones, however there is in the positive moment zone of the span.

The equivalent inertia of the span can be estimated as:

$$I_e = \left(\frac{M_{crack}}{M_a}\right)^3 I_g + \left[1 - \left(\frac{M_{crack}}{M_a}\right)^3\right] I_{crack}$$

$$I_e^C = \left(\frac{10.27}{20.1}\right)^3 \times 61\,567 + \left[1 - \left(\frac{10.27}{20.1}\right)^3\right] \times 27\,156 = 31746\ cm^4$$

For the right support, it shall be taken equal to the gross section:

$$I_e^R \approx I_g^{solid} = 184\,500$$

and for the left support:

$$I_e^L = 145\,530$$

By taking the average:

$$I_e = \frac{\left(I_e^L + I_e^R\right)/2 + I_e^C}{2} = 98\,380\ cm^4$$

As the inertia considered in the analysis $I_g^{model}\left(92\,250\right) < I_e\left(98\,380\right)$ no correction will be applied, however for greater safety the equivalent inertia $I_e = 0.8\, I_g^{model}$ will be taken and so amplifying by $1/0.8 = 1.25$.

Regarding $\lambda = \dfrac{\xi}{1+50\rho}$ we will assume (to be on the safe side) that $\rho$= 0, hence $\lambda$=$\xi$, with and the same construction process and durations will be present. Therefore:

$$\xi_{mean} = \frac{1.3\times4.6 + 1.2\times1 + 1\times1 + 0.6\left(0.3\times2\right)}{4.6+1.0+1.0+2.0} = \frac{8.54}{8.6} \approx 1$$

It can be said that the differed deflection is approximately equal to the instantaneous deflection ($\delta_{dif} \approx \delta_{inst}$ = 1.25 $\delta_i$); and the total deflection is:

$$\delta_{total} = \delta_{inst} + \delta_{dif} = 2.5\,\delta_i$$

As $\delta_{inst}$ = 1.25 $\delta_i$

Now to obtain the differed deflection after the partitions have been built:

$$\xi_{mean} = \frac{1.2\times4.6+1.2\times1+1\times1+0.6\left(0.3\times2\right)}{4.6+1.0+1.0+2.0} = \frac{8.1}{8.6} = 0.94$$

and the instantaneous deflection must be left without the self weight of the slab:

$$\%\delta_i = \frac{1.0+1.0+2.0}{8.6} = \frac{4}{8.6} \approx 0.47$$

hence the active deflection will be:

$$\delta_{active} = 0.47\times1.25\,\delta_i + 0.94\times1.25\,\delta_i \approx 1.75\,\delta_i$$

It can be said that the total long-term deflection for a waffle slab must be obtained with a displacement amplification factor of the order of 2.5 times the instantaneous elastic deflection of the program, and for the active deflection, the user would have to multiply the instantaneous elastic deflection by 1.75.

It must be taken into account that the indicated coefficients are those applied to usual building cases such as those mentioned regarding, span lengths, depths, loads, construction procedure, normal environment conditions, unpropping periods, and entering into force of the loads in time.

If these conditions are varied, then logically the pertinent correction coefficients have to be applied. If the deflection to be verified is that for comfort, it will be sufficient if the deflection due to the live loads in loadcase LL are analysed.

<table>
<tr><th>Guarantee</th><th>Deflection</th><th>Amplification</th><th>SLAB</th><th>COMBINATION</th></tr>
<tr><td rowspan="2">INTEGRITY</td><td rowspan="2">ACTIVE</td><td>2.00</td><td>FLAT SLAB</td><td rowspan="4">DL + LL<br>Worst case</td></tr>
<tr><td>1.75</td><td>WAFFLE SLAB</td></tr>
<tr><td rowspan="2">APPEARANCE</td><td rowspan="2">TOTAL LONG TERM</td><td>3.00</td><td>FLAT SLAB</td></tr>
<tr><td>2.50</td><td>WAFFLE SLAB</td></tr>
<tr><td rowspan="2">CONFORT</td><td rowspan="2">LIVE LOAD INSTANTANEOUS</td><td>1.50</td><td>FLAT SLAB</td><td rowspan="2">Loadcase LL</td></tr>
<tr><td>1.25</td><td>WAFFLE SLAB</td></tr>
</table>

## 1.14.5 Foundation elements

Please consult the chapter corresponding to foundation pads, pile caps, strap and tie beams.

# 2 Mat foundations and foundation beams

## 2.1 Discretisation

The discretisation carried out for mat foundations and foundation beams is the same as for slabs:

**Mat foundations.** Bar element mesh with a size of 0.25×0.25 m (mesh with springs at nodes).

**Footings and foundation beams.** Linear bar elements, with nodes at intersections with other elements, divided into 14 spans with nodes, if no intersections are present with other elements. Springs are present at the nodes.

The foundations are taken as bearing on elastic soil (subgrade modulus method), in accordance with the Winkler model, based on a proportionality constant between forces and displacements, whose value is the subgrade modulus. The user is reminded that this method cannot study the interaction between foundation elements in close proximity.

$$p = K \cdot y$$

Where:
p: Bearing pressure ($kN/m^2$)
K: Subgrade modulus ($kN/m^3$)
y: Vertical displacement (m)

This hypothesis is valid when applied to homogenous soils. It is a fact that the amount by which a small foundation settles compared to a large foundation is different for the same bearing pressure transmitted to the soil and therefore should be applied with caution. It is also known that the behaviour of granular soils is different to that of cohesive soils.

Usually, laboratory results are obtained, which together with the soil study report and an estimate of the size of the foundation or foundation beam widths which are to be provided, the subgrade modulus to be applied can be established.

An edometric modulus $E_o$, is provided, established in the laboratory, and if the width of the beam, mat foundation or load test plate is known, the subgrade modulus K can be found, assuming the compressible layer of the soil is infinite and homogenous:

$$K = \frac{2E_o}{b}$$

Where:

$E_o$: Edometric modulus
b: Foundation dimension

In some cases, the subgrade modulus of a soil will be established by means of a load test plate of specific dimensions.

## 2.2 Subgrade modulus for mat foundations and foundation beams

The subgrade modulus is an element of data to be entered in the program. It is determined using empirical methods using a load test plate.

A soil study report is usually available for a job and it is here where the exact value of the subgrade modulus should be displayed for the dimensions the mat foundation, footing or foundation beam is to have.

If the soil study has been carried out but the subgrade modulus value that is provided is that corresponding to a 30×30 cm load test plate (or other test plate size), and not for the total size of the mat foundation, this will have to be adjusted. Bear in mind that:

$$K_1 \cdot d_1 = K_2 \cdot d_2$$

In other words, the subgrade modules $K_1$ and $K_2$ established with plates with diameters $d_1$ and $d_2$ comply with the previous ratio.

Therefore, in an approximate manner, it may be taken for sandy soils that:

$$K_1 = \frac{K_p \cdot (b+30)^2}{(2 \cdot b)^2}$$

Where:
$K_1$: Subgrade modulus of the mat foundation or foundation beam
$K_p$: Subgrade modulus of the 30×30 plate
b: Smaller side (width) of the foundation (in cm)

For rectangular footings, the following formula can be applied:

$$K' = \frac{2}{3} \cdot K_1 \cdot \left(1 + \frac{b}{2l}\right)$$

For clay soils:

$$K_1 = \frac{K_p (n + 0.5) \cdot 30}{(1.5 \cdot n \cdot b)}$$

Where:

$K_1$: Subgrade modulus of the mat foundation or foundation beam

$K_p$: Subgrade modulus of the 30×30 plate

b: Smaller side (width) of the mat foundation, foundation beam or footing (in cm)

n: Ratio of the length to the width of the foundation element

For mat foundations, it is recommended by **Professor Rodríguez Ortiz**, that **b** be taken as the width of the mean equivalent tributary area of the columns, which is approximately equal to **0.70L**, where **L** is the mean quadratic span of the distances between columns in both directions of the mat foundation.

The following formula can be used for footings and beams bearing on clay soils:

$$K_1 = \frac{K_p \cdot 30}{b}$$

where each variable has the same meaning as in the previous formulas.

If a soil study report is not available, the user may choose between the following indicative subgrade modules:

5 000 kN/m$^3$ for bad soil
40 000 kN/m$^3$ for medium soil
120 000 kN/m$^3$ for good soil

taking these values as those obtained from a 30×30 cm load test plate trial.

*An example of bad soil is peat. Medium soil can be taken as being similar to humid clay. Good soil can be understood as natural gravel.*

Example: A medium soil is present composed of sandy-clay, whose subgrade modulus is known = 40 000 kN/m$^3$ from a load test plate trial. The mat foundation measures 2.00 m by 8.00 m. The subgrade modulus to be used in the analysis is obtained as follows:

Sandy-clay soil is present, therefore two subgrade modulus values will be obtained and then an average value will be found.

- **Sandy soil**:

$$K_s = K_p \cdot \frac{(b+30)^2}{(2 \cdot b)^2}$$

Where:
$K_p$: Subgrade modulus of the 30×30 plate
b: Smaller side (width) of the mat foundation (in cm)

$$K_s = 40\,000 \cdot \frac{(200+30)^2}{(2 \cdot 200)^2} = 40\,000 \cdot 0.33 = 13\,225 \text{ kN/m}^3$$

- **Clay soil**:

$$K_c = \frac{K_p (n+0.5) \cdot 30}{(1.5 \cdot n \cdot b)}$$

Where:
$K_p$: Subgrade modulus of the 30×30 plate
b: smaller side (width) of the mat foundation (in cm)
n: Ratio of the length to the width of the mat foundation = 4

$$K_c = 40\,000 \cdot \frac{(4+0.5) \cdot 30}{1.5 \cdot 4 \cdot 200} = 40\,000 \cdot 0.1125 = 4500 \text{ kN/m}^3$$

Logically, settling will be greater when clay soil is present than in the case of sandy soil. The subgrade modulus is inversely proportional to the amount by which it settles.

In this case the proportion is not known and so a mean value will be taken:

$$K_{sc} = \frac{(13\,225 + 4500)}{2} \approx 8900 \text{ kN/m}^3$$

Below is a list providing guide values for the subgrade modulus depending on the type of soil for a square 0.30×0.30 m plate:

| **Soil types** | **Subgrade modulus × $10^4$ (kN/m$^3$)** |
|---|---|
| Light peat and boggy soil | 0.5 - 1.0 |
| Heavy peat and boggy soil | 1.0 - 1.5 |
| Fine sand | 1.0 - 1.5 |
| Peat, sand and gravel layers | 1.0 - 2.0 |
| Wet clay soil | 2.0 - 3.0 |
| Humid clay soil | 4.0 - 5.0 |
| Dry clay soil | 6.0 - 8.0 |
| Hard dray clay soil | 10.0 – |
| Finely layered humus with sand and few stones | 8.0 - 10.0 |
| Finely layered humus with sand and many stones | 10.0 - 12.0 |
| Fine gravel with a lot of fine sand | 8.0 - 10.0 |
| Medium gravel with fine sand | 10.0 - 12.0 |
| Medium gravel with coarse sand | 12.0 - 15.0 |
| Coarse gravel with coarse sand | 15.0 - 20.0 |
| Coarse gravel with little sand | 15.0 - 20.0 |
| Coarse gravel with little sand, very fine layers | 20.0 - 25.0 |

If the subgrade modulus, K and the width of the mat foundation are known, the differential equation for an applied system of loads can be solved as follows:

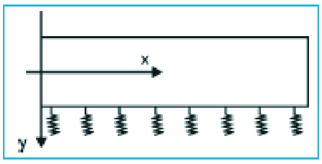



Figure 37

$$\frac{d^2 M}{dx^2} = -b\left(q(x) - p(x)\right)$$

$$Q = -\frac{dM}{dx}$$

obtained by differentiating the equation y(x) is the deformed shape of the element.

Additionally,

$$M = -EI\frac{d^2 y}{dx^2}$$

By substitution:

$$EI = \frac{d^4 y}{dx^4} + bK \cdot y(x) = b \cdot q(x)$$

which is the general solution without shear deformation, and when resolved the solution of the system is obtained.

The shear deformation factor is established:

$$\phi = \frac{24\, I(1+\nu)}{A_{shear} \cdot L^2}$$

I: Inertia of the element
$\nu$: Poisson coefficient
$A_{shear}$: Shear area
L: Length of the element

If this factor, $\phi$, is less than 0.1, shear deformation is not considered and the general solution is valid, which, is also exact. If it is greater than 0.1, an approximate solution is obtained by decomposing the stiffness matrix into a stiffness matrix of the bar and another of the soil.

To obtain a more approximate solution, third degree polynomials are taken as shape functions to provide an approximate solution of the integration, and establishing a final stiffness matrix by combining both matrices.

Generally speaking, slabs are decomposed into short elements measuring 0.25 m in length, where $\phi > 0.1$, and so the approximation due shear deformation is applied. The same occurs with foundation beams which support slabs. Intermediate nodes are generated, and therefore, short bars. In the case of long foundation beams where $\phi < 0.1$, the exact formula will be applied.

Once the deformed shape has been obtained, the displacements at the nodes are known, and so the forces for each loadcase can be found.

## 2.3 Design options

All the design options, definable parameters, redistribution, minimum moments, steel areas, reinforcement tables, etc., which can be defined for beams and slabs are also applicable to floating foundations.

## 2.4 Loads to consider

Regarding foundation beams and mat foundations, it must be said that they form part of the complete structure and hence interact with the rest of the structure as they too are included in the global stiffness matrix of the structure. Therefore loads can be applied on these elements, as can be done on any beam or slab of the structure.

## 2.5 Materials

The materials to be used: concrete and steel, are defined separately from the rest of the materials of the job because they are ground bearing materials.

## 2.6 Checks and combinations

The limit states to check are those corresponding to the design of reinforced concrete elements (ultimate limit states), and to the bearing pressure, equilibrium and uplift checks (serviceability limit states).

**Uplift.** When there is upward vertical displacement in a mat foundation or foundation beam, the program indicates uplift is present. This may occur in one or several displacement combinations. It may occur and it sometimes occurs in projects where strong horizontal actions are present. If this arises, the structure should be checked and have its base stiffened, if possible, and increase the on plan size and/or thickness of the foundations.

**Equilibrium.** This is checked in the case of foundation beams. If, for the transverse section of the beam, the resultant of the stresses lies outside the width of the beam, there is no equilibrium and an error message is emitted, included within the beam errors. It is a message inherent to the method, as tensile forces are not permitted along the width of the beam.

**Bearing pressures.** Once the displacements of the nodes for each combination is known, the stresses are calculated by multiplying by the subgrade modulus:

$$p = K \cdot y$$

In the case of foundation beams, the bearing pressure at the edges is calculated using the vertical displacement, plus the product of the rotation of the section and the distance from the axis entered to each edge. The points and the bearing pressures of all the nodes exceeding the allowable bearing pressure defined for the soil and those at the edges that exceed the bearing pressure in 25% are included in a text file.

## 2.7 Design of mat foundations and foundation beams

As was mentioned earlier, mat foundations and foundation beams are designed as elements forming part of the complete structure, and therefore, an integrated analysis of the foundation with the structure is carried out.

If columns with external fixity have been defined and hence their displacements are prevented or if external fixity beams have been defined, which also have their displacement prevented, the user must be careful when these are used together with mat foundations and foundation beams.

This is similar to when piles and pad footings are used, or simply pad footings or pile caps which are designed with supports with external fixity and coexist with mat foundations and foundation beams in the same foundation.

Example of the foundations of a small building:

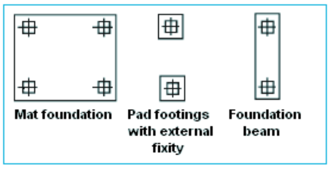



Figure 38

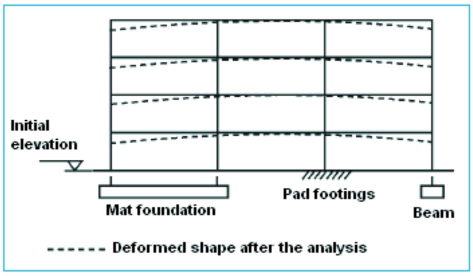



Figure 39

Observe how the columns with external fixity (pad footings) have no vertical displacement, whilst the mat foundation and the foundation beam do have vertical displacement, the amount of the displacement depends on the loads acting on them, their dimensions, the geometry of the structure and the subgrade modulus. All this results in a deformed shape of the structure which does not reflect what really occurs.

If good soil is present, with a high subgrade modulus value, these differential displacements are not problematic as these are very small. But when bad soil is present and the number of floors is greater (and therefore more loads are present), other preventive measures must be taken.

First of all, the dimensions of the pad footings should be established. Once known, enter them as small mat foundations around the columns, having changed the fixity of the columns from **With external fixity** to **Without external fixity**.

This way, all the foundation elements are designed on an elastic platform and there is compatibility amongst the displacements as all the elements are **Without external fixity** and hence preventing these problematic displacement differences from arising.

Tie beams between the pad footings (small mat foundations) have not been included. If the user really wishes to include them in the design, two options are available:

Beam acting as a tie beam: in which case it does not contribute or transmit pressures to the soil. This element simply ties, and minimum steel areas for the bars are provided in the beam.

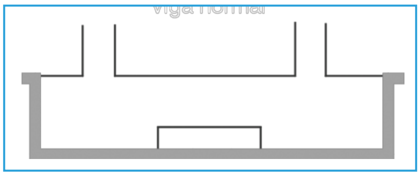


Figure 40. Normal beam

Beam acting as a foundation beam: in which case collaborates and transmits pressures to the soil.

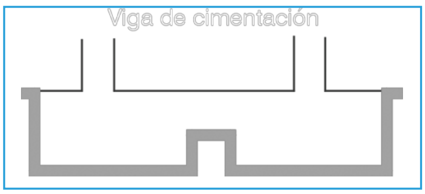



Figure 41. Foundation beam

Different results are obtained in each case.

Having completed this procedure, reanalyse the job. An integrated analysis of the foundation with the whole structure is obtained for the second case. For the first case, as the tie beam does not do anything, at least the task of representing it in the drawings is made easier.

The reinforcement to be provided and the foreseen displacements (with the considered subgrade modulus) for each loadcase for the mat foundation, can be consulted by clicking on **Envelopes > Maximum displacements...** as can be done with any flat or waffle slab, as well as clicking on the *Contour Maps* tab.

In the same way as the user is warned to be careful when using columns and shear walls with starts with or without external fixity, a similar problem arises when using external fixity beams to represent basement walls or similar elements.

The care that has to be taken when using external fixity beams has been mentioned in other sections. This can be illustrated with an example: In a building, the lift machinery roof is supported around its perimeter by a brick wall or concrete wall.

The error that can arise if the external fixity beam is used with a pinned support or roller support is important especially when horizontal loads are present.

In the case of vertical displacements, errors would arise if the building is very tall (> 15 floors), where the elastic shortening of the concrete in the columns is significant, and those areas of the structure supported by the external fixity beam, logically do not shorten (vertical displacements = 0), and hence creating an unreal differential displacement effect.

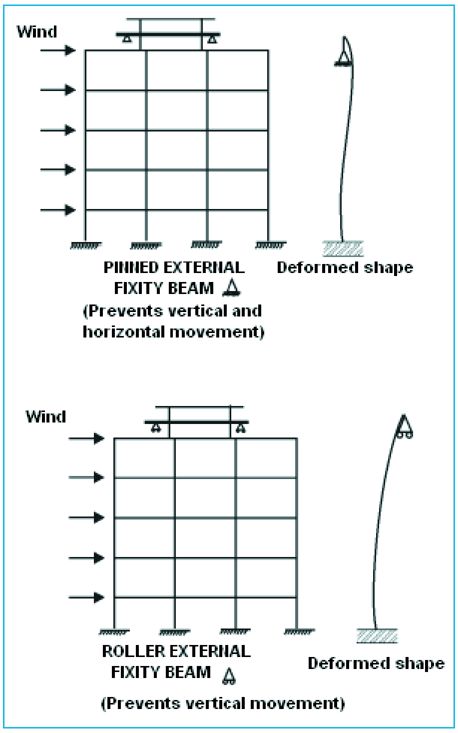



Figure 42

If this is used together with external beam supports at lower floors to simulate the presence of basement walls with mat foundations and foundation beams, the following preventative measures should be taken. There are two cases:

**1. The external fixity beam intersects with columns**

- If the external fixity beam has not been disconnected from the columns, the external fixity beam fixes the columns and hence these cannot move vertically. Additionally, the load from the columns is transmitted to the external fixity beam, which as its name indicates behaves as an external fixity . Therefore the loads are absorbed and hence, none are transmitted to lower levels of the structure.

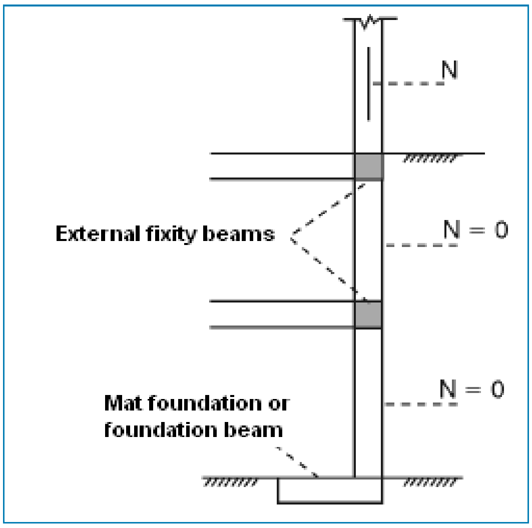



Figure 43. Section

Figure 44. Plan view

In this case, the program will not admit that a mat foundation or foundation beam exists below the columns intersecting with an external fixity beam at a lower level, and so will emit a message.

If foundation elements with external fixity have been used, the program does not emit a warning. However, the final result will be incorrect as the axial force below the external fixity beam will be null (N = 0).

- If the external fixity beam has been disconnected from all the columns with which it intersects and no connection exists with the floor slab, for example:

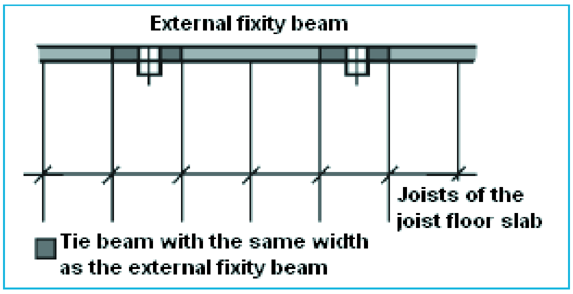



Figure 45

The loads in the column descend to lower levels and a mat foundation or foundation beam can be entered.

Additionally, the floor slab which is in contact with the external fixity beam should be assigned a fixity coefficient of zero i.e. pinned, so no force transmission can occur or so the floor slab does not remain suspended from the beam.

- In this case, as in the previous case, if the floor slab is a waffle or flat slab, even if the external fixity beam is disconnected from the columns, the load from the column can still be suspended from the slab and reach the beam:

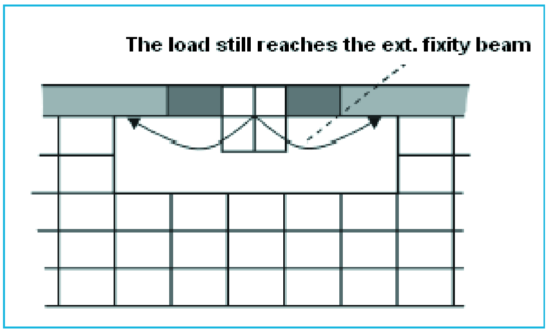



Figure 46

It can be seen more clearly when the column is greater than the thickness of the wall.

In this case, part of the load can descend to lower levels and part of it can be absorbed by the external fixity beam via the slab. In any case, the analysis would be incorrect if foundation beams or mat foundations are entered at the base of the columns. This also occurs with foundation elements with external fixity.

2. **The external fixity beam does not intersect with columns or shear walls.**

   Generally speaking, this case does not present any problems, however the following issues should be taken into account:

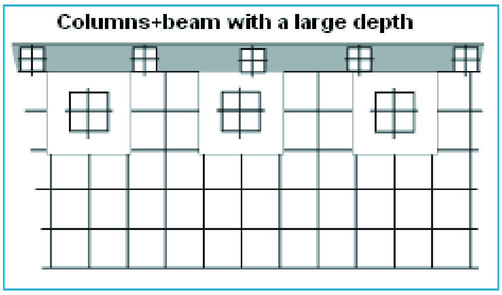



Figure 47. Plan view

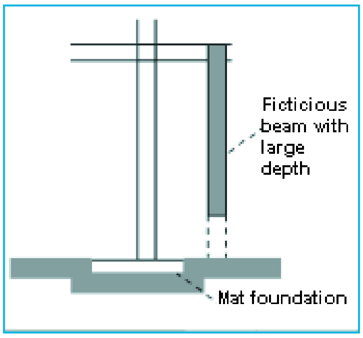



Figure 48. Section

- The external fixity beam is very close to the columns.

  In this case it is possible that part of the loads from the columns are absorbed by the external fixity beam and not descend to the mat foundation or foundation beam. The shear force diagram can be consulted for the nodes between the column and the external fixity beam to verify that there is no sudden change in sign of the shear force, or a sudden high value appearing, both being clear signs of loads being absorbed by the external fixity beam.

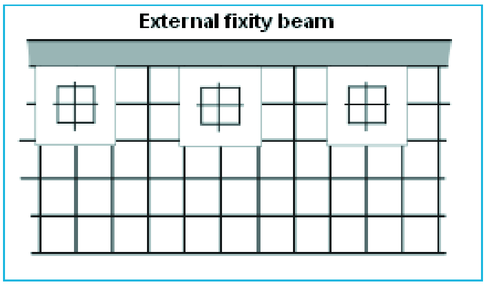



Figure 49

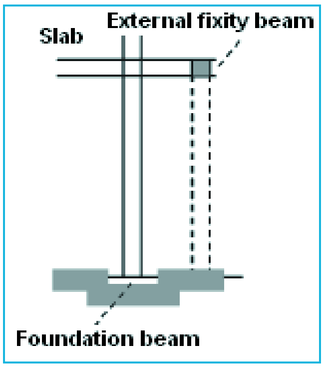



Figure 50

If this problem is encountered, it is recommended the external fixity beam be eliminated and to represent it by means of columns and a beam with a great depth between them. Provide the same type of foundation for those columns as is provided for the parallel row of columns of the structure and, if they really are very close, provide a combined foundation system for both rows. Later on carry out the required corrections due to the stiffness of the wall not having been considered.

- The external fixity beam is at a distance equal to approximately a span of the building.

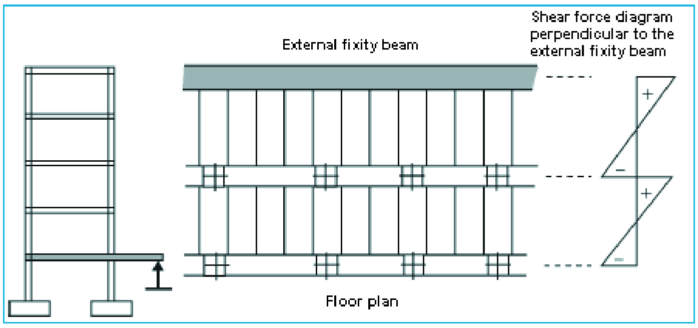



Figure 51

If the shear force diagrams, as in this example, change sign for the joists perpendicular to the wall, no special precautions have to be taken, and hence mat foundations or foundation beams can be provided below the columns.

Bear in mind all the explanations and indications provided in this *Calculations manual* when using foundation systems on elastic soil, columns with external fixity and external fixity beams, as well as the program carrying out an integrated analysis of the foundations.

# 2.8 Analysis results

These are consulted in the same way as normal beams and slabs, and can be modified and obtain drawings in the same manner.

# 2.9 Element design and check

## *2.9.1 Foundation beams*

The design is carried out in the same way as for a normal beam, bearing in mind the parameters, steel areas and tables defined in the foundation beam options.

In the specific case of ⊥ or L beams, the transverse bending of the flanges is calculated, obtaining a reinforcement area due to bending $A_{sb}$. The pass-through reinforcement $A_{sp}$ is obtained, as is the thread reinforcement of the flange-web joint $A_{sa}$. The greater of the two plus the area due to bending is placed.

$$A_{S\ TOTAL} = MAX\ (A_{sp}, A_{sa}) + A_{sb}$$

This reinforcement is compared with that obtained due to shear in the web and the greater of the two is provided, for both the web and the flanges, with the same diameter and spacing.

A shear and punching shear check can be carried out optionally, for a section situated at a distance of half the effective depth from the edge of the column, with a width equal to the width of the column plus the effective depth.

The program checks the tangential stress for that section does not overcome the limiting stress without the need of punching shear reinforcement. If this stress is overcome, a warning message is emitted. In this case the depth must be increased to that proposed by the program so no punching reinforcement has to be provided.

## *2.9.2 Mat foundations*

Mat foundations are designed in the same way as flat slabs, and have the same criteria applied to them. Special options are available for foundation elements such as steel areas, reinforcement arrangements, tables, etc.

# 2.10 General recommendations

## 2.10.1 *Mat foundations*

The election of the minimum depth to provide is important, and should not be less than a tenth of the design span between supports plus 20 cm. It is best if a perimeter overhang can be provided as smaller bearing pressures will result at the edge, as well as helping to avoid any punching shear problems.

## 2.10.2 *Foundation beams*

Choose transverse sections which possess minimum stiffness, especially in the case of T or L sections and limit the span/depth ratio to 2, so the flat deformation hypothesis is valid.

*Consult the beam errors upon concluding the analysis. Also consult the uplift and bearing pressures of the points which fail.*

# 3 Walls

### *Masonry and concrete block walls, Reinforced concrete walls, Reinforced concrete walls with lateral pressures*

Two types of load bearing walls are available:

- Reinforced concrete wall
- Masonry wall

Discretisation: Six node triangular finite elements making up a thick sheet are used in both cases.

The foundations can consist of elements with external fixity or without external fixity.
The footing or beam, in regards to longitudinal and torsional effects is considered to lie on an elastic layer (Winkler), when it is defined as not having external fixity.

When external fixity is present, the foundation may consist of a strip footing. The wall may rest on a foundation beam or mat foundation when it is defined as without external fixity or as consisting of another structural support.

The specific cases of each type are detailed below.

## 3.1 Masonry walls

These are walls built following traditional methods such as brick walls or concrete block walls.

The behaviour of these walls is not linear, and so the discretisation that is carried out and the modelling of the wall as a linear element are not adequate, but these are the only methods available with the program. As long as the forces and stresses are not high, the analysis results are acceptable. Bear in mind that the tensile forces that may arise are not real, and so these values should be consulted in the **Envelopes > Forces in walls** option and verify they are null or very small.

This can be clarified using an example:

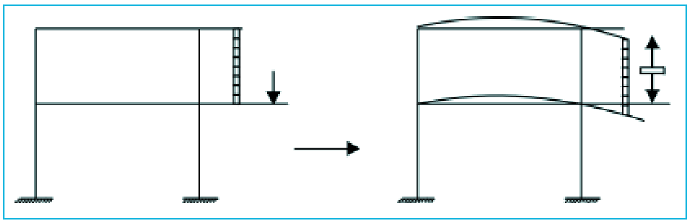


Figure 52

The masonry wall is in tension, as if it were a bracing rod. This is not real, but it can be analysed and the user obtain results. Therefore special attention must be paid to these results.

### 3.1.1 Masonry wall properties

The values required to define the mechanical properties of the masonry walls are the following:

- Modulus of elasticity E = 1 GPa (default value). The value of E is usually estimated by:

$$E = \frac{\sigma_C}{\varepsilon}$$

where:

$\sigma$: Design compressive stress of the masonry wall

$\varepsilon$: Unit deformation of the material

The default value provided with the program can be estimated assuming a design stress of 1MPa (1,000,000 Pa) and a unit deformation of 0.1%:

$$E = \frac{1000\,000}{0.001} = 1000\,000\,000\,\text{Pa} = 1\,\text{GPa}$$

- Shear modulus: 0.4 GPa
- Unit weight: default value of 15 kN/m$^3$
- Design compressive strength: 2 MPa
- Design tensile stress: 0.2 MPa

The transverse shear stiffness is taken as being null, however, an option is available so it may be considered.

The program checks the stress state of the masonry wall, for all the defined concrete combinations. Assuming that the design tensile stress is 10% of the design compressive strength, if these values are exceeded in over 10% of the area of the wall, a message will be emitted at the end of the analysis warning there are excessive tensile and compression forces.

In the case of concrete block walls, the values taken are those provided by the user or manufacturer. **Design options** are available to provide vertical or horizontal reinforcement.

## 3.1.2 Masonry wall introduction

The following data must be indicated:

1. Initial and final group
2. Wall thickness (thicknesses to the left and right of the axis)
3. Lateral pressures
4. Type of support
   - Foundation beam
   - Strip footing
   - With external fixity (with or without a footing)
   - Without external fixity (foundation beam)

The overhangs and depths of the beams and footings are to be defined and if these bear on soil, the subgrade modulus and allowable bearing pressure are required.

It is not recommended lateral pressures be applied, even though the program allows for them to be defined. If the user opts to apply the lateral pressures, he/she must ensure the masonry wall can resist these pressures.

## 3.1.3 Correct use of masonry walls

Care should be taken when masonry walls are entered and the user should try and adjust the entered model to what will be physically present.

It is recommended the instructions below be followed as these are applicable for the most common cases that may be encountered.

### 3.1.3.1 Raised floor slab

These are built at a small distance above the foundation plane (< 1 metre), leaving an air chamber which serves as insulation.

Usually a strip footing or foundation beam is built to support the small masonry wall, which in turn supports the joists of the raised floor slab. Usually the joist floor slab will be a self supported slab, as it is not usually possible to provide shoring due to the lack of space. Various different cases are described below.

The direction of the masonry walls coincides with the structure’s column alignments.

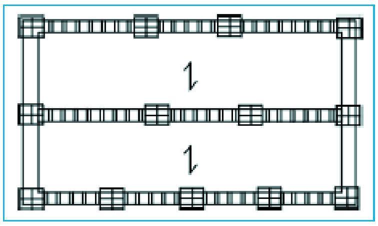


Figure 53

1. **If pad footings are provided below the columns they will be designed jointly by CYPECAD**. In this case, the columns have been defined as having external fixity and so, for compatibility reasons, the support of the masonry wall must also be defined as having external fixity by means of a strip footing.

   This way the load from the columns does not spread out to the masonry wall. This can be checked by consulting the **Envelopes > Forces in columns and shear walls** and ensure that the axial force in the first span is greater than or equal to the axial force in the span directly above it.

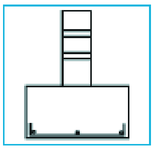


Figure 54. Strip footing

2. **A mat foundation is provided**

   The same is recommended as in the previous case, i.e. it is to be entered only this case "without external fixity" as the masonry wall bears on the mat foundation. However a foundation beam should be entered below the wall.

   The foundation beam will really remain contained within the mat foundation. It is recommended it be entered with its overhangs equal to zero (without overhangs) and with a depth equal to the depth of the mat foundation.

   In this case, a stiffening effect of the wall with the mat foundation may arise, especially if a high wall is used or if the modulus of elasticity is increased. If this effect does occur, try reducing the modulus of elasticity to the minimum allowable value (E = 0.1 GPa) and reanalyse the job to verify the results. To be able to detect this effect, which is possible by observing the reinforcement that has been provided and with some experience of having used the program, the job can be copied and edited. In this copy, delete the raised floor slab and substitute the walls with line loads on the mat foundation. Analyse this copied job and compare the results with the original job.

3. **The foundations of the building consist of foundation beams, coinciding with the masonry walls**

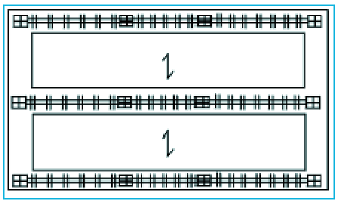


Figure 55

The reactions of the raised floor should be calculated so to obtain line loads, and then enter these line loads on the foundation beams of the building.

Create a new job and only enter the raised floor slab (and the columns), defining it as "with external fixity" as has been indicated in point 1.

*A masonry wall should never be defined with a foundation beam or footing when using it jointly with columns defined "without external fixity". Even though the stress analysis may be acceptable, the beam or footing reinforcement will be incorrect and reinforcement areas less than what are really required will be provided. This is due to the coupling of the foundation beam with the masonry wall and raised floor slab, which produces a "Vierendeel" effect of the combination. This causes part of the load from the column to be transmitted to the masonry wall and relieving the column of the load at its base, obtaining results which are not adapted to what really physically occurs.*

4. **A joint foundation is to be used composed of a mix of elements: pad footings, mat foundations and foundation beams.**

In this case the problem is more complex. Please check that indicated in sections 1 and 3, and the section on ***2. Mat foundations and foundation beams***.

In any case, please recall that columns with external fixity and without external fixity should not be used together due to the different settlements that can arise. This provides an inadequate model and hence provides unsuitable results.

*For these cases, an initial analysis can be carried out with all the columns with external fixity and then enter mat foundations and foundation beams in accordance with the magnitude of the forces reaching the foundations and the allowable bearing pressure of the soil. Finally, check that the bearing pressures and displacements are compatible and have reasonable values.*

5. **The direction of the masonry walls does not coincide with the columns. In this case, there a no connection problems between the masonry walls and the columns.**

   What was mentioned in the previous case is still valid and insisting on the importance of not mixing elements with external fixity and elements without external fixity.

### 3.1.3.2 Masonry walls between floor slabs

If masonry walls are used to support part of a floor slab bearing at the top of the wall on a floor slab below, which is the usual case, the user must ensure that, due to the structural design of the building and the applied loads, that the wall is in compression and is not behaving as a tie (which would therefore make it unsuitable, as indicated in Figure 52 of this section).

To ensure this is so, the force results should be checked and the user should observe that the wall is indeed acting in compression. Some tensile forces, due to imperfections of the analysed model, may appear but these should be negligible.

In this case, the wall is always defined as not having external fixity, and whereby the beam on which it rests is assigned the appropriate depth and overhangs.

Usually, the overhangs will be null and the depth will be that of the floor slab on which it rests. When a masonry wall is supported by a joist floor slab, perpendicular to the joists, theoretically, the wall will distribute the load proportionally amongst the joists.

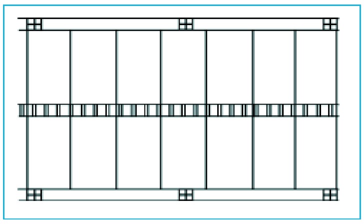


Figure 56

If the loads are correctly transmitted, the bending moment diagram will be as shown below:

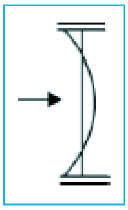


Figure 57

If, on the contrary, the joists supporting the wall appear to be supported by the wall, tensile forces are present in the masonry wall and hence bending moment diagrams such as those below will be obtained:

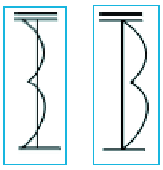


Figure 58

In this case check what could be causing the wall to behave in such a way.

It may also occur that the wall is acting as a “wall-beam”. If beams cross it perpendicularly, the wall may be supported by the beams and suspending the floor slab below:

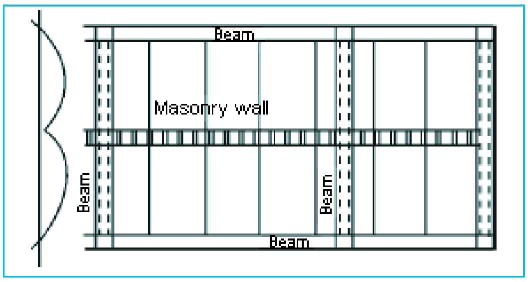



Figure 59

If this is the case, the forces will not be transmitted correctly, as the longitudinal stiffness of the wall is not real.

Therefore, care must be taken when using walls between floor slabs, and once the job has been analysed, check the results of the supporting and supported elements.

Due to the complexity of the structures and the diversity of cases that may be entered in the program, it is recommended masonry walls be used when necessary and always controlling the results.

Masonry walls can be applied in a large variety of cases; as load bearing walls in low rise buildings, as supports for lift machinery slabs, load bearing walls due to the façade being set back a distance at attic level, and any other type of load bearing wall within the building structure.

**Please recall that, when a masonry wall bears on the foundations, it will also contribute and absorb horizontal forces.** This is inevitable as they are stiff elements. This must be taken into account, if the user does not wish for this to occur, such as in the case of raised floor slabs and masonry walls located within the first floors of the building.

In regards to raised floor slabs, when the wind is blowing in the direction of the masonry walls, the columns remain practically fixed at the level of the raised floor slab.

A different case would be that shown in the figure below:

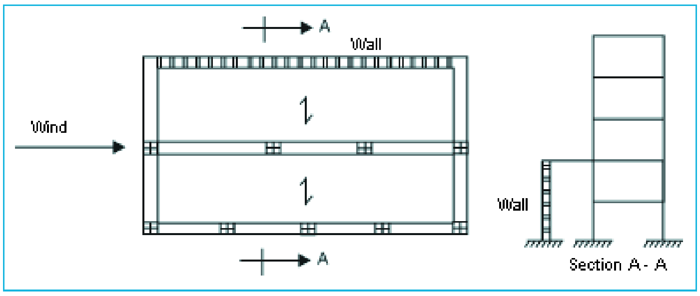



Figure 60

When the wind is blowing, the masonry wall has great stiffness in the direction of the wind and will absorb nearly all the horizontal force that is incident on it.

If the user does not wish for this to occur, repeat the analysis, only this time eliminating the wall and the overhanging floor slab, and instead enter their reactions. In a separate copy of the job, the wall can be added with the overhanging slab so to then be able to design the slab.

The presence of the stiffness of the walls must always be considered by the user, as they produce bracing effects which should not be accounted for.

## 3.2 Reinforced concrete walls

There are two types of reinforced concrete walls, although, for the program they are identical and only one type exists and its design varies depending on the data entered by the user. The two types are:

- Reinforced concrete basement walls (retaining)
- Reinforced concrete load bearing walls

### *3.2.1 Reinforced concrete basement walls*

These are usually used to define a perimeter basement wall and have two main functions: to resist any lateral soil pressures and support the loads transmitted by the structure to the foundations.

### 3.2.1.1 Data to enter

The data that has to be entered is identical to that indicated for masonry walls. In this case the program establishes the mechanical properties internally by taking into account the properties of the reinforced concrete to be used for the job.

For this type of walls, the user has to define the lateral pressures acting on it due to the soil.

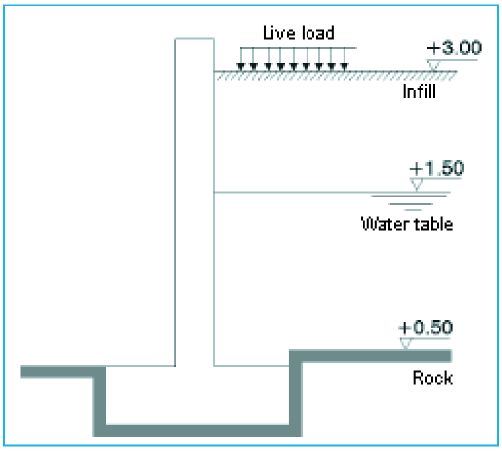



Figure 61

The program allows for lateral pressures to be defined on both sides of the wall and for these to belong to different loadcases, which will later be used in the various load combinations of the complete structure. It is recommended they be treated as a live load and even better, as a separate type of live load, as these lateral pressures may or may not be present regardless of the rest of the building.

A generic definition of a section of a reinforced concrete wall can be that shown in Figure 61. The following aspects are considered:

- Soil-wall friction is neglected, therefore the pressures act horizontally.
- The lateral pressures are calculated considering the pressures at rest: $\lambda_h=1-\sin\phi$, ($\phi$: internal friction angle)
- Below the elevation of the rock layer, the lateral pressures are cancelled, except any hydrostatic pressures that may be present.
- Evacuation due to drainage is taken into account at infill level, either due to saturation or infiltration. Its effect is combined with the hydrostatic pressures acting on the wall at the same elevation as the infill; the lateral pressure is multiplied by the inverse of the evacuation percentage due to drainage.

$$\text{Lateral pressure coefficient} = \frac{100 - \%\ \text{evacuation}}{100}$$

In other words, when there is 100% evacuation due to drainage, no additional lateral pressures are present, as (100–100)/100 = 0, and when 0% evacuation: (100–0)/100 =1, the water table is at the same elevation as the infill.

- Below the water table, the submerged density of the infill is used for lateral pressure calculations as well as any hydrostatic pressure that may be acting.
- The weight of the soil over the overhangs of the footing is not considered when calculating the soil bearing pressures or when designing the footing.
- An infill possessing a slope angle can be defined.
- Loads acting on the infill can be defined. These can consist of:

  a. Uniformly distributed loads

  b. Strip load parallel to the crown of the wall

  c. Line load parallel to the crown of the wall

  d. Point load or concentrated load in reduced areas (footings)

The formulas applied are indicated below:

**a. Lateral pressures due to a uniformly distributed load**

The *Coulomb* method is applied. The horizontal pressure that arises due to the uniformly distributed load with a value of q per unit slope length is equal to:

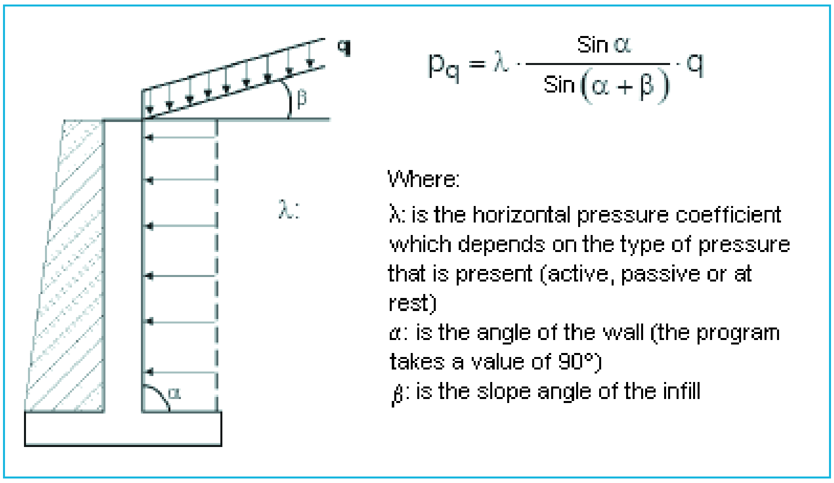



Figure 62

b. **Lateral pressures due to a strip load parallel to the crown of the wall**

The horizontal pressure due to a strip load in the case of a wall whose backfill side is vertical and the soil is levelled horizontally, and in accordance with the **Elasticity theory** is:

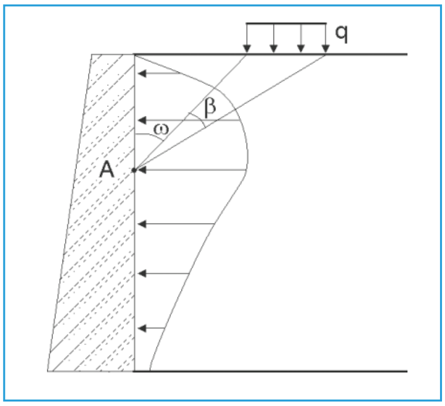



Figure 63

$$p_q = \frac{2q}{\pi}\left[\beta - \sin\beta \cos 2\omega\right]$$

Where

$p_q$ : Horizontal pressure

q: Strip load per unit area

$\beta$ and $\omega$: Angles shown in Figure 63

c. **Lateral pressures due to a line load parallel to the crown of the wall**

A method based on the *Elasticity theory* is used. The horizontal pressure due to a linear live load q acting on a wall whose backfill side is vertical and the soil is levelled horizontally is:

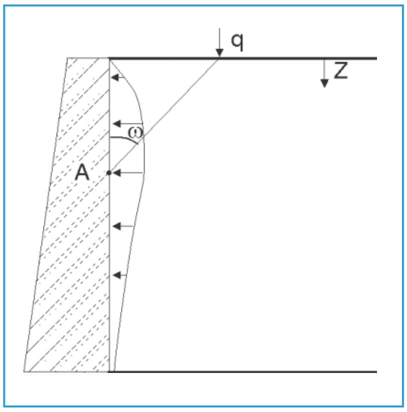



Figure 64

$$p_q = \frac{q}{\pi \cdot z} \sin^2 2\,\omega$$

Where

$p_q$ : Horizontal pressure

q: Line load per unit length

$\omega$: Angle shown in Figure 64

**d. Lateral pressures due to a point load or concentrated load in reduced areas (footings)**

A method based on the *Elasticity theory* is used. The horizontal pressure due to a point load acting on a wall whose backfill side is vertical and the soil is levelled horizontally is:

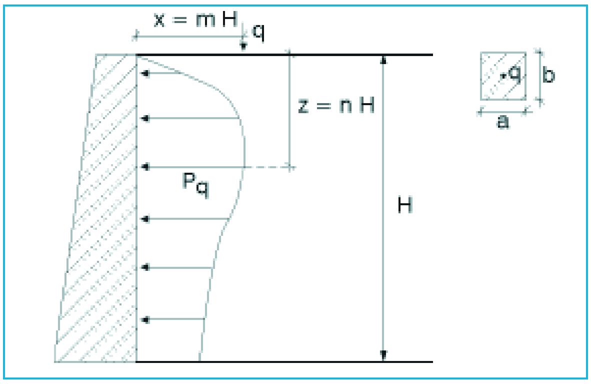



Figure 65

If (m < 0.4):

$$p_q = 0.28 \cdot \frac{q}{H^2} \cdot \frac{n^2}{\left[0.16 + n^2\right]^3}$$

If (m ≥ 0.4):

$$p_q = 1.77 \cdot \frac{q}{H^2} \cdot \frac{m^2\,n^2}{\left[m^2 + n^2\right]^3}$$

### 3.2.1.2 *Correct use of reinforced concrete walls*

Reinforced concrete walls may be classified based on their type of support:

- With external fixity: Adequate solution when the columns of the structure are defined as also having this type of fixity. A strip footing can be defined below the wall and obtain its design.

- If the foundation is to consist of a mat foundation, the wall can be entered on the mat foundation without external fixity, with a beam with no overhangs and with the same depth as that of the mat foundation. When defining the beam, the data to enter is to be the same as that defined for the mat foundation.

- If all the columns of the structure rest on foundation beams or a mat foundation, i.e. without external fixity, enter a strip footing below the wall. To do so, an initial design of the footing has to be carried out in order to estimate its width. To do so, estimate the value of the line load transmitted by the structure, add this value to the weight of the wall and divide the total by the allowable bearing pressure.

The foundation of a wall can consist of a beam or footing (recommended).

The whole structure can also be analysed "with external fixity", and with a strip footing below the wall. This way, an initial design of the footing can be obtained.

### *3.2.2 Load bearing walls*

Reinforced concrete basement walls without **lateral pressures** can also be defined, which causes it to behave as a load bearing wall when it comes to resisting vertical and horizontal loads. These may replace the shear walls defined within the **Column Definition** tab with the added advantage that they are also more versatile as they can be used simultaneously with columns; start at columns and contain columns which may start at any elevation of the wall, with dimensions greater or smaller than the thickness of the wall; walls which start and end at different levels may be joined to one another, etc.

Below are some wall examples (Figure 66):

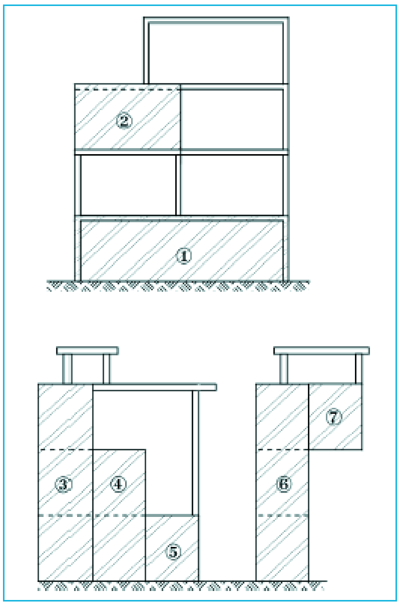



Figure 66

The user is reminded of the rigid diaphragm hypothesis at each floor level, so to bear in mind the free movement restrictions of the wall displacements. For more information please consult chapter ***12. Rigid Diaphragm***.

The program considers the floor slabs to be fixed to the walls at their intersections. The option to define a fixity coefficient at panel edges is available. Hence, the fixity coefficient of the panel may be reduced to a value smaller than 1, until a hinged connection is obtained. In these cases, a construction detail of the connection between the floor slab and wall should be provided in the drawings to ensure it is built correctly.

When a wall is defined as not having external fixity, the dimensions of the beam on which the wall rests are to be provided.

### 3.2.3 Correct use of reinforced concrete walls

Please recall:

- If the reinforced concrete wall starts at a floor slab, the beam on which it rests is to have the same depth as the floor slab. The case may arise whereby the beam continues past the edge of the wall or is an extension of an existing beam. In this case, assign the same dimensions to the beam below the wall. Using the beam reinforcement editor, control the beam reinforcement and join the required bars to provide continuity without overlaps.
- A wall can never start from an existing beam or coincide with others at other levels which it may go through, or even at where it ends. A message will appear warning the user of this circumstance and will not allow it to be entered.
- If the wall starts at foundation level, the most adequate solution as to what type of foundation to provide is that of a strip footing. It may be defined as with or without external fixity and may start from a mat foundation. In this case, please recall that the beam is to be defined as having 0 cm overhangs at either side, and its depth equal to that of the mat foundation. The allowable bearing pressure and subgrade modulus are also to be the same as that of the mat foundation. A wall cannot start on an existing foundation beam.

  In any case, define the foundation beam of the wall and then connect it to the foundation beam of other parts of the structure.

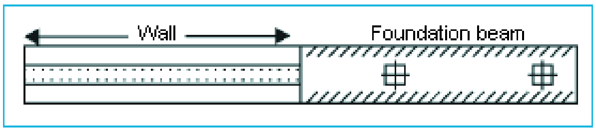



Figure 67

- Remember: do not mix elements bearing on elastic soil (without external fixity) with elements with external fixity.
- The rigid diaphragm hypothesis at floor level always exists, even if a floor slab is not entered. Therefore reinforced concrete walls with lateral pressures cannot be used as retaining walls, due to there, supposedly, always being a slab at the top of the wall and at intermediate levels.
- The lateral pressures acting on the walls are transmitted to and absorbed by the floor slabs acting as rigid diaphragms. However, neither the floor slabs nor the beams are checked for compression or tension.

  In this case, it is convenient the following be recalled:

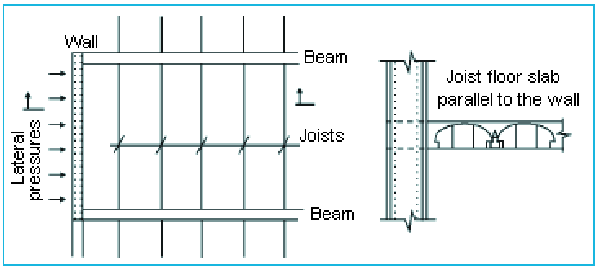



Figure 68

Joist floor slabs, for those cases where the joists are parallel to the wall, offer a weak resistance and must be absorbed by the beams which support them.

If, additionally, these beams are exempt, with openings adjacent to the wall, they will behave as an elbow joint and these must be designed for compression.

Normal beams are designed only for simple bending. Hence these beams must be deleted and entered as sloped beams which are designed for combined bending. As of the 2012.a version, exempt beams can be disconnected from the rigid diaphragm, and therefore, be designed for compression. Please consult chapter ***12. Rigid Diaphragm***.

As of the 2012.a version, if a wall is exempt at floor level or is only in contact with exempt beams or other walls, the analysis is correct, as there is a foreseen deformation of the wall in the opening in the design model, due to there being 6 degrees of freedom per node in contact with the opening. If the wall is in contact with a floor slab along any part of its length on a floor, the entire wall will b connected to the rigid diaphragm of that floor, including those spans where there is no floor slab. Nonetheless, the user can partially eliminate the rigid diaphragm hypothesis. Please consult chapter ***12. Rigid Diaphragm*** to know for each case if the exempt wall is disconnected or not from the rigid diaphragm.

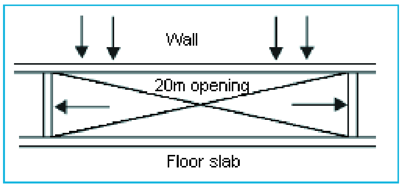



Figure 69

It is recommended all the indicated observations be taken into account.

### 3.2.4 *Wall design*

When designing the horizontal wall reinforcement, the minimum steel area depends on whether the wall has:

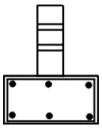
- Lateral pressures acting on it, in which case, the minimum steel ratios are those stated in the code
- No lateral pressures acting on it. These are designed in the same way as shear walls

Bearing pressure check. Upon concluding the analysis, a message appears indicating the foundation beams or footings whose bearing pressure exceeds the allowable bearing pressure of the soil or that the maximum bearing pressure at one of their edges exceeds the allowable bearing pressure by 25%.

If no checks fail, no messages are emitted.

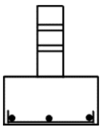
### 3.2.5 *Foundation design*

The wall foundation can be defined in two different ways.

- **As a foundation beam** 

  If it is defined as a foundation beam, it is designed in exactly the same way as other foundation beams (simple bending).

  The minimum steel areas are indicated in the **Options** for foundation beams**.**

- **As a strip footing** 

  Please consult the section ***4.4. Strip footings below walls***.

## 3.3 Practical advice for reinforced concrete wall in buildings

Read the contents of the message that appears when the option **Define wall** is selected, as in most cases, by following the advice provided in that dialogue box the wall introduction can be quicker and more effective.

Use the **Pin/Disconnect** option which allows for the "external fixity beams" to be disconnected from the columns, without the need of having to enter reinforced concrete walls. In this case remember that the floor slab loads transmitted to the external fixity beam will not be transmitted to the columns.

Remember not to enter structures whose model is incoherent with the real behaviour of the building.

The most important aspects for buildings with joist floor slabs and pad footings with a perimeter reinforced concrete wall (the most common type of building) will now be highlighted.

The first problem which the user should always be aware of:

a. If the internal columns have been entered "with external fixity" in order to calculate them with pad footings, the reinforced concrete wall must be defined "with external fixity" so the perimeter does not settle with respect to the remaining supports. Define a strip footing for the wall.

b. If the internal columns have been defined "without external fixity, these will start on a foundation beam or mat foundation., in which case the foundation of the wall can be a strip footing or beam bearing on elastic soil (without external fixity).

   If all the columns and wall are to bear on a mat foundation, a foundation beam without overhangs (left = right = 0) and with the same depth, allowable bearing pressure and subgrade modulus as the mat foundation can be used below the wall.

   Once the analysis has concluded, drawings may be obtained as follows:

   - Obtain the drawing of the wall. In the wall elevation change the "See beams drawing" to "See mat foundation drawing".
   - The starter bars are valid. If the beams drawing for Group 0 (foundation level where the mat foundation is usually located) has been drawn, delete those beams below the wall or simply omit that drawing.
   - Obtain the drawings for the mat foundation, using the usual drawing and reinforcement configuration and add the construction detail corresponding to the

start of a wall on a mat foundation (detail CCM013); as well as any others that may be considered adequate for the real conditions of the project: intersections with floor slabs, wall crowns, etc.

c. It is common practice to provide strap beams perpendicular to the reinforced concrete wall, and recommended, when all the foundations have been entered “with external fixity”. This can be applied to the footing of the wall.

   To obtain the foundation layout drawing, composed of the reinforced concrete walls, pad and strip footings and strap beams, the following drawings will have to be obtained:

   - Floor layout drawings of Group 0 in DXF format (or of whichever group the foundation has been defined at).
   - Reinforced concrete and masonry wall elevations

Complete the drawings by complementing them with the construction details available in the **Construction detail library**.

## 3.3.1 *Revision of the analysis results of the wall*

Once the analysis has finished, the footing of the wall should be checked as well as the elevation of the wall. It is possible that within the analysis errors report, that one or two messages may appear related to the bearing pressures transmitted to the soil by the footing. One may appear indicating that the allowable bearing pressure of the soil has been exceeded. Alternatively, a message may appear informing the user that the allowable bearing pressure of the soil has been exceeded by over 25% at an edge of the mat foundation. If either of (or both) these messages appear, the report will indicate the group, frame and beam at which it occurs, as well as the value of the bearing pressure transmitted to the soil.

The following steps may also be taken:

- Place the main view of the screen at the indicated group, which will usually be at Group 0, if the foundations were entered at this level.
- Click on **Beam alignments > Show alignments**, and type the number of the frame to revise. The frame will be displayed in bright red.

This foundation beam will have to be modified for it to comply with the allowable bearing pressure. For this, a simple rule can be applied:

If B is the current width of the beam, the new width B' will have to be equal to:

$$B' = B \cdot \frac{\sigma}{\sigma_{abp}}$$

E.g.: B = 0.60, $\sigma_{abp}$ = 0.2 MPa, $\sigma$ = 0.225 MPa

$$B' = 0.60 \cdot \frac{0.225}{0.2} = 0.675\,m \Rightarrow B' = 0.70\,m$$

To correct this value, the user has to decide whether to leave the depth of the beam untouched or to increase it as well:

- If the depth is not increased. Within the **Results** tab, click on **Beam errors** and select the beam in question. Within the dialogue box that appears, type the new value of the width of the beam (70). Click on **Correct**.
- If the depth is increased. Within the *Beam Definition* tab, Click on **Edit** and then select the wall. Within the dialogue box containing the corresponding data of the beam, modify the values and increase the overhang by 10 cm and the assign the beam the desired depth.

The job should now be redesigned or reanalysed. To redesign, click on **Analyse > Redesign frames with changes**. A new reinforcement arrangement will be obtained as well as new bearing pressures. If it fails, it will be indicated in the analysis report and the process will have to be repeated.

Once the bearing pressures are suitable, the reinforcement of the foundation beam can be checked. Place the main view of the screen at the foundation group and click on **Beams/Walls > Edit beams** within the *Results* tab and then on the beam in question. The reinforcement may also be edited using this option.

Check the beam. The elevation of the wall should also be checked. To do so, place the main view of the screen at a group where the wall is present (other than the foundation group). Click on **Beams/Walls > Edit walls** within the *Results* tab. The *Reinforcement edition* window will appear displaying the reinforcement of the wall.

The thicknesses of the wall may be modified. If these changes are small, the job does not have to be reanalysed, however this is not the case if the changes are noticeable.

The reinforcement may also be modified. These will be highlighted in red if they fail.

The **Compliance factor** button is located within the *Reinforcement edition* dialogue box. A window will appear upon clicking the button where the user can modify the value as well as receive useful information on this helpful tool:

*"When the reinforcement is designed, it is arranged in such a way that the % compliance value of the reinforcement does not fail for all the points (nodes of the finite element mesh) of the elevation of the wall. The forces acting on the wall are not uniform and vary at different points, be it because of the lateral pressures, loads transmitted by floor slabs and columns, etc. Therefore, it is normal to observe areas where forces act in a more concentrated manner in critical zones, such as intersection points with floor slabs (which is a very rigid connection, due to the horizontal diaphragm hypothesis), at points where walls start or intersect with other walls. If the reinforcement to provide is to cover all these 'peak forces', the general reinforcement of the wall is most likely being penalised by providing more than what is really necessary in the rest of the wall."*

For example, imagine a wall whose vertical reinforcement turns out to be $\phi$20 bars every 10 cm, when usually $\phi$12 bars every 20 cm (from previous experience) for walls of this type is sufficient. The reinforcement can be modified to the second value, which the program will automatically check once it has been entered.

It is possible that it will fail at one or more points. The program indicates this by displaying the reinforcement in **red**. Small rectangles hatched in red appear on the elevation of the wall where it fails, and in the bottom area of the dialogue box, the program will display it fails and provides the % compliance factor. This value indicates the % area of the wall that does not fail.

Imagine, for this example, the value provided is 87%. This implies that 13% is the area represented as hatched in red which fails.

The program contains an option which allows the user to pre-establish the value of the *Compliance factor* (consult **Job data > By position > Column options**). The provided default value is set at 90%, as due to the discretisation of the wall, it is normal for small peak forces to be present, and so it is reasonable that if the reinforcement provided covers at least 90% of the surface, logical and foreseeable results will be obtained.

Returning to the example, even though the value of 90% has not been reached (the value is 87%). It does not seem logical to provide $\phi$12 bars every 10 cm, when by providing $\phi$12 bars every 20 cm, 87% of the wall is covered with that reinforcement. It seems more correct for, in any case, the reason(s) for the presence of the peak forces to be analysed. It may be that a column might start at the location of the red rectangle, and so it is normal for there to be a concentration of local stresses at that point. However this extra reinforcement that is required is covered by the starter bars of the column.

The user can check to see what reinforcement is required at that point. By pressing the **Show additional reinforcement** button, the red point will turn yellow and by clicking on it, a window will appear displaying the additional reinforcement that is required at that point. If the user decides to include the additional reinforcement, a construction detail will have to be provided and indicate its location on the wall elevation drawings.

The case may arise when the additional reinforcement to provide is the minimum steel area available, i.e. $\phi$6 bars every 25 cm, in which case it is not worth placing. Each case has to be evaluated individually.

Click on **Finish showing add. reinf.** to recover control of the wall reinforcement.

It may also be possible that the user wishes for a smaller compliance factor to be used. To change this value (only for this wall elevation), click on **Compliance F.** and reduce the value to 80%. Click on **Accept**, followed by **Redesign**. Observe that an improved reinforcement distribution is obtained, e.g. $\phi$10 bars every 20 cm, and a new compliance factor is displayed (for this example 81.3%). Additionally, a new point appears where the reinforcement fails.

Carry out the procedure detailed above, checking the forces. Finally, adopt the most adequate reinforcement layout for the global elevation of the wall and, if necessary, with the required local additional reinforcement.

Our experience tells us that when the points lie on the edge of the mesh, at intersections with floor slabs or columns and at a point or very localised zone, a compliance factor of approximately 90% (± 5%) is sufficient.

Another case which also arises frequently is when transverse reinforcement has to be provided, which is not usually the case for reinforcement smaller than $\phi$12 bars. If this value were to be greater, then it would be normal to have to provide transverse reinforcement, which in reality are tie bars of the reinforcement of each face of the wall to restrict the buckling of the bars. This is convenient if the wall is strongly in compression.

If, after the analysis, the program indicates transverse reinforcement is to be provided and is not of great significance, it may be eliminated by entering a value of 0 in the number of transverse bars to be provided, then checking the compliance factor, the additional reinforcement, and at which points it is required.

As indicated before, if it occurs at the critical zones, it is not reasonable to provide transverse tie bars. Even though the analysis may not indicate it is required, our experience recommends it be provided as assembly reinforcement and for safety against local buckling of the bars.

Once a floor level has been modified, if more than one basement level is present, check all the levels and try to provide uniform diameters and bar spacing so overlaps and splices are carried out in a logical manner. Even though it is not obligatory, it does make building it on site a lot easier.

Finally, if the reinforcement is modified and the user wishes to recover the original reinforcement provided by the program, it may be restored by clicking on **Redesign**. The modifications will be lost and the reinforcement provided will be designed to meet the current **Compliance factor** for that wall.

In this section, the general aspects that have been considered for checking and design footings defined in CYPECAD that lie below the vertical supports of the building which have been defined as “with external fixity”.

# 4 Footings and pile caps

Remember they can be designed sequentially with the rest of the structure or independently. As they are elements “with external fixity”, the foundation is calculated later on based on the reactions on these supports. Given that they may be designed independently, do not forget that modifications can be carried out on the structure without them implying that these will affect the foundations.

The program may also be used as an editing tool, whereby the foundation elements can be entered without designing them and obtain drawings and material take-off quantities.

## 4.1 Footings and pile caps

In the 2011 version of the program, changes were carried out regarding the force calculation of foundations “with external fixity”, for both footings and piles and their strap and tie beams.

The design provided by this version can differ to that provided by previous versions: once the reactions at the structural supports (columns, shear walls and walls) have been obtained, a model is created with all the foundation elements “with exterior fixity” and their beams, represented by its stiffness matrix. The stiffness matrix together with the loadcases defined as loads acting on the foundation (obtained reactions), are resolved using frontal methods to obtain the displacements and forces in all the elements, hence allowing all their stiffnesses to intervene and interact amongst each other.

The design process is iterative and starts off using the initial dimensions of each element. During the first iteration, the following considerations are established to determine the stiffness and fixity of each type of element:

- **Pad footing or pile cap**

  It is taken as being a rigid solid with a central support, whose fixity in each direction can consist of a hinged support if it is reached by a strap beam or a fixed support if it is reached by a tie beam or if no beam reaches it.

- **Strip footing below wall**

  It is defined as a rigid solid with a central support which, in the transverse direction, is considered to be hinged if strap beams and/or other walls reach it, and, fixed in the longitudinal direction.

- **Tie beam**

  It is taken as being a bar with the stated dimensions and with hinged ends reaching the axis that passes through the centre of the element they are tying.

- **Strap beam**

  It is taken as being a beam with the stated dimensions. Its ends are fixed to the edge of the element it is balancing for the following cases:

  - Edge and corner footings, and for one or two-pile pile caps, in the directions that require balancing.
  - When the user marks the balancing action manually.

  Strap beam ends are hinged for the following cases:

  - Centred footings or pile caps with three or more piles.
  - Edge and corner footings, and for one or two-pile pile caps, in the directions that do not require balancing.
  - When the user deactivates the manual balancing action.

  Therefore, the end fixity can be modified by the user and, in a similar way as what occurs with strap beams, the ends reach the axis that passes through the centre of the element which it joins.

- **Perimeter walls and strap beams reaching a transverse strip footing below a wall**

  This element combination is a special case, although common. It may be seen in the figure below.

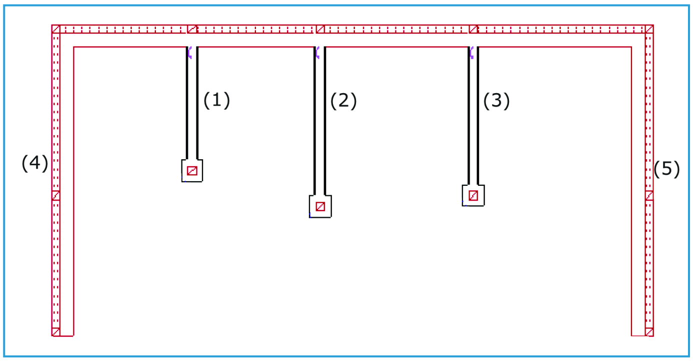



Figure 70

Transverse walls (4) and (5) have a large stiffness compared to beams (1), (2) and (3). Therefore their effect is much greater than that of the beams. Hence, a simplification has been carried out, so the balancing effect is shared equally amongst all the transverse elements that reach the footing. This has been done by using the mean stiffness of all the strap beams and assigning this stiffness to all the elements they balance, including the walls, and so all the elements contribute in a balanced and equal way.

To obtain a more precise analysis, the soil–structure interaction should be considered as well as having an adequate soil model. However, due to the complexity of this analysis, it seems reasonable to use the aforementioned simplified analysis, as the footing is also considered to be rigid and does not receive torsional forces.

The number of iterations carried out by the program depends on the option selected by the user:

- **Minimum dimensions**

  Once the first iteration has been carried out, the footings and pile caps are designed with the forces that have been obtained. A second iteration is undertaken and the elements are redesigned, including the beams. Using the geometry from this last iteration, a final design is carried out and all the elements are checked to see if any fail.

- **Complete design**

  After the first iteration, the program continues iterating and tries to provide results so that none of the elements fail, which it will do unless the permitted maximum design limits for each element are reached, in which case, some will fail.

## 4.2 Advanced design of surface foundations

Besides all that has been mentioned previously, additional features can be obtained using the **Advanced design of surface foundations** module:

- Beams can be defined spanning between other beams.
- Line, point and surface loads can be applied on the foundation elements, which are taken into account in the design of all the elements.
- Loads defined for the foundation group, are applied on the surface of the elements.
- The loads corresponding to stairs that start at foundation level, both their start loads and those transmitted by the masonry walls supporting intermediate spans are applied on the foundation elements.

# 4.3 Pad footings

CYPECAD carries out the analysis and design of reinforced concrete and mass concrete footings (consult section ***4.9. Mass concrete footings*** of this manual). The following types of footings are designed:

- Footings with constant depth
- Footings with variable depth or tapered

On plan, they are classified as:

- Square
- Centred rectangular
- Eccentric rectangular (for special cases: edge or corner footings)

An unlimited number of supports (columns, shear walls and walls) can bear on each footing and at any position.

The loads transmitted by the supports are transmitted to the centre of the footing, obtaining its resultant. The transmitted forces may include:

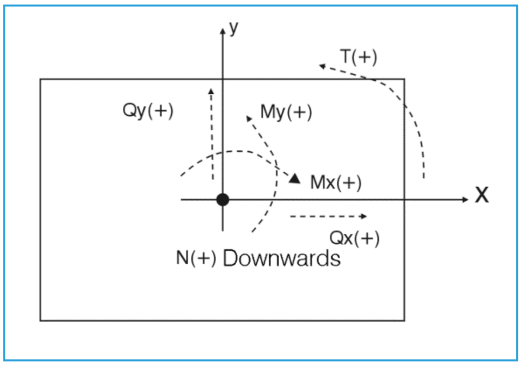



Figure 71

N: Axial
Mx: X moment
My: Y moment
Qx: X shear
Qy: Y shear
T: Torsion

The loadcases considered by the program include: Dead, live, wind, snow and seismic loads.

The states to check include:

- Bearing pressures on the soil
- Equilibrium
- Concrete (bending and shear)

A design can be carried out based on the default dimensions defined in the program option, or dimensions provided by the user.

The reinforcement can also be obtained for a specific footing geometry.

The check consists in verifying the code aspects of the geometry and reinforcement of the footings.

## 4.3.1 *Bearing pressures on the soil*

A flat deformation diagram is taken for the footing, hence depending on the forces, a trapezoidal bearing pressure diagram will be obtained. Tensile forces are not admitted, and so, when the resultant force lies outside the central nucleus, zones without pressure will be displayed.

The resultant force must remain within the footing, otherwise there is no equilibrium. The self weight of the footing is taken into account.

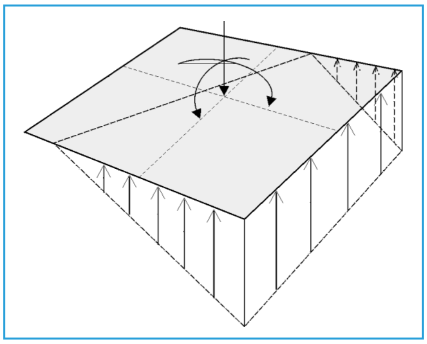


Figure 72

The program checks that:

- The mean bearing pressure does not exceed the allowable bearing pressure of the soil
- The maximum bearing pressure at the edge does not exceed the mean pressure by the following percentages for the load combinations detailed below:
    - Gravitational loads: 25%
    - With wind loads: 25%
    - With seismic loads: 25%

These values are optional and may be modified. Within *General data* dialogue box, different bearing pressures can be defined for persistent or accidental situations as well as for seismic situations.

## 4.3.2 Equilibrium states

By applying the corresponding ultimate limit state combinations, the program checks the resultant lies within the footing.

The excess with respect to the safety coefficient is expressed as a percentage safety reserve:

$$\left(\frac{0.5 \cdot \text{footing width}}{\text{resultant eccentricity}} - 1\right) \cdot 100$$

If it equals zero, the equilibrium is limited and if the value is large, the footing has a large safety factor with respect to the equilibrium.

## 4.3.3 Concrete states

Bending in the footing and the tangential forces are verified.

### 4.3.3.1 Bending moments

In the case of a single column, the check is carried out using a reference section situated at a distance of 0.15 times the dimension of the column towards the inside of the column.

If there are several supports, the program sweeps across the footing and calculates the moments at many sections. This is carried out in both x and y directions, with steel columns and baseplates, at the mid-point between the baseplate and the steel section.

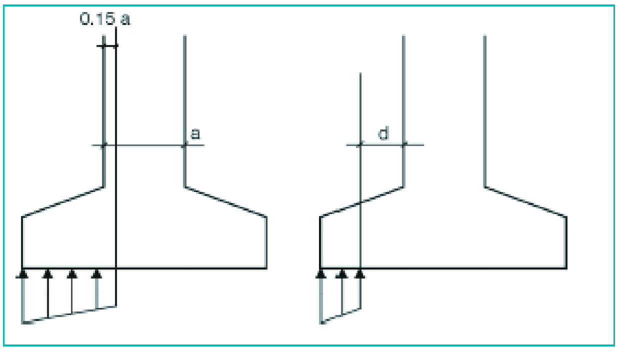



Figure 73

### 4.3.3.2 Shear forces

The reference section is situated at a distance equal to the effective depth from the edges of the support. If there are several edges, the reference sections could overlap due to them being in close proximity, in which case a warning will be emitted.

### 4.3.3.3 Reinforcement anchorage

The anchorage of the reinforcement ends is checked, providing the corresponding anchorage lengths for each case and position.

### 4.3.3.4 Minimum depths

The minimum depth specified by the selected code is checked.

### 4.3.3.5 Reinforcement spacing

The minimum spacing between reinforcement bars stated in the selected code is checked.

However when designing a minimum of 10 cm is taken.

### 4.3.3.6 Minimum and maximum steel areas

The minimum mechanical and geometric steel areas specified in the selected code are checked.

### 4.3.3.7 Minimum diameters

The program checks the reinforcement diameters are at least those indicated in the selected code.

### 4.3.3.8 Design

When designing for bending, the program provides concrete depths so compression reinforcement is not required.

Similarly, when designing for shear, the program places concrete depths so transverse reinforcement need not be applied.

### 4.3.3.9 Check for oblique compression

This check is carried out at the edge of the support. The stress in the concrete is not permitted to exceed that required for failure due to oblique compression. Depending on the type of support, the axial force of the support will be multiplied by:

- Internal supports: 1.15

- Edge supports:1.4
- Corner support: 1.5

This is done so to take into account the eccentricity of the loads.

Footings are always designed as rigid footings, however the program only emits a warning if the overhang/depth ratio is less than or equal to 2.

When designing a footing with several supports, the slenderness is limited to 8, where the slenderness is the ratio between the span between supports divided by the depth of the footing. **Design options** are available which the user may edit so the growth direction of the footing can be chosen, or fix a specific length depending on the type of footing. The results will logically vary depending on the selected options.

When the bearing pressure distribution does not occupy the whole footing, tensile forces may arise at the top surface due to the weight of the overhanging footing. The program will provide top reinforcement if required.

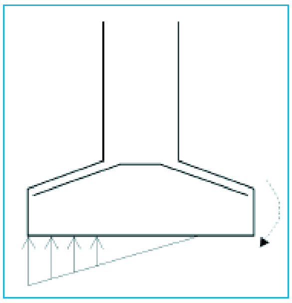


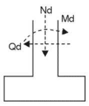
Figure 74

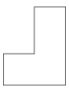
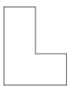
## 4.4 Strip footings below walls

The program designs reinforced concrete strip footings below walls.

This type of footing can be provided below retaining, basement or load bearing walls in buildings.

There are three types of strip footings:

- Overhang on both sides 

- Overhang on the left 
- Overhang on the right 

These footings can be used for reinforced concrete walls and masonry walls.

Their geometry is defined within the data entry dialogue box of the wall.

They are designed and checked in the same way as rectangular footings (consult the section on **Pad footings**), and therefore have the same possibilities available to them (including containing columns in close proximity to each other) and the same design conditions and limits.

The only difference consists in how the loads are applied.

In the case of a column, the loads are applied at its geometrical centre axis, be it square or rectangular. However, in the case of a wall, the loads are modelled as a distributed load along the wall in a discrete manner. It is as if a resultant force is converted into a pressure distribution along the base of the wall, and is discretised internally in steps by the program depending on its dimensions.

To summarise, using images:

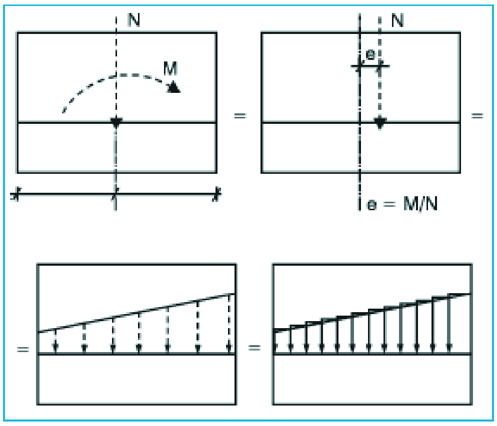
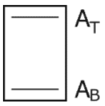
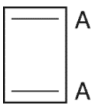



Figure 75

## 4.5 Strap beams

Strap beams are using to balance the moments in footings and pile caps. Two types are available:

- Negative moments: 

$A_T > A_B$
- Positive moments: 

Symmetrical reinforcement

Definable and editable reinforcement tables are available for each type of beam.

The forces acting on the strap beams are:

- Moments and shears required for their balancing action.
- Loads cannot be entered on the beams, nor are their self weights considered. The program assumes their self weight is transmitted to the soil without it suffering any forces.
- When several strap beams reach a footing or pile cap, the force each one receives is proportional to its stiffness.
- They may receive forces at one or both ends.

If their length is less than 25 cm, the program will warn of there being a short beam present.

A reinforcement table exists for each type. The program checks that the selected reinforcement does not fail for the loads to which the beam is submitted to.

The following checks are carried out:

- Minimum longitudinal reinforcement diameter
- Minimum transverse reinforcement diameter
- Minimum steel area in tension
- Minimum mechanical steel area (reductions are accepted)
- Maximum longitudinal reinforcement steel area
- Minimum spacing between longitudinal reinforcement
- Minimum spacing between stirrups
- Maximum spacing between longitudinal reinforcement
- Maximum spacing between stirrups
- Minimum beam width (≥1/20 span)
- Minimum beam depth (≥1/12 span)
- Check for cracking (0.3 mm)
- Top reinforcement anchorage length
- Skin reinforcement anchorage length
- Bottom reinforcement anchorage length

- Check for bending (not have compression reinforcement)
- Check for shear (concrete + stirrups resist the shear)

A certain amount of deviation is permitted at the point where the strap beam goes by the edge of the footing (15º).

An option is available whereby the user can fix the minimum tension reinforcement geometric area.

The program does follow some criteria when arranging the beam with respect to the footing, depending on the relative depth between both elements and is levelled at the top or bottom face.

The program uses the strap beam combinations, taken as a reinforced concrete element, for all the designs and checks, except for in the case of cracking, where the program uses the bearing pressures on the soil.

## 4.6 Tie beams

The program designs tie beams between reinforced concrete foundation elements.

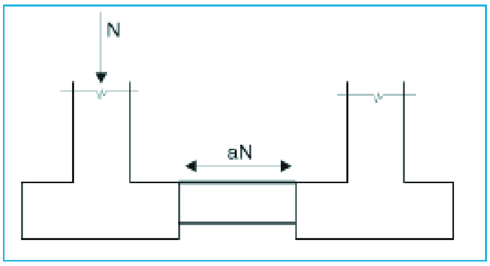



Figure 76

Tie beams are used to brace/tie footings, absorbing any horizontal forces due to seismic loads.

The maximum axial force is multiplied by the design seismic acceleration “a” (no less than 0.05) and these forces are considered as tensile and compression forces (a·N).

The user can optionally choose to design these for bending due to a uniformly distributed surcharge of w = 10 kN/m, produced by compacted soil and the screed above. They are designed to support a positive and negative moment equal to $wl^2/12$ and a shear force of wl/2, where l is the span of the beam.

For their design, the combinations used for strap beams as reinforced concrete elements are used here.

Reinforcement tables containing symmetrical reinforcement arrangements for both faces are used.

The following checks are carried out:

- Minimum longitudinal reinforcement diameter
- Minimum transverse reinforcement diameter
- Minimum tension reinforcement geometric steel area (if the compacted soil surcharge option has been activated)
- Minimum compression reinforcement geometric steel area (if the compacted soil surcharge option has been activated)
- Minimum mechanical reinforcement
- Minimum spacing between longitudinal reinforcement
- Maximum spacing between longitudinal reinforcement
- Minimum spacing between stirrups
- Maximum spacing between stirrups
- Minimum beam width (≥1/20 span)
- Minimum beam depth (≥1/12 span)
- Check for cracking (0.3 mm, not considering seismic loads)
- Top reinforcement anchorage length
- Skin reinforcement anchorage length
- Bottom reinforcement anchorage length
- Check for bending (only with compacted soil surcharge)
- Check for shear (only with compacted soil surcharge)
- Axial load check

Options are available to extend the stirrups to the surface of the footing or up to the support.

The position of the beam; whether it be levelled with the top or the bottom of the footing, depending on its relative depths.

# 4.7 Pile caps

The program designs reinforced pile caps on square or circular piles with the following arrangements:

- Pile cap with 1 pile. (A)
- Pile cap with 2 piles. (B)
- Pile cap with 3 piles. (C)
- Pile cap with 4 piles. (D)
- Linear pile cap. The user can choose the number of piles. The default value is set at 3. (B).
- Rectangular pile cap. The user can choose the number of piles. The default value is set at 9. (D)
- Rectangular pile cap on 5 piles (one pile in the centre). (D)
- Pentagonal pile cap with 5 piles. (C)
- Pentagonal pile cap with 6 piles. (C)
- Hexagonal pile cap with 6 piles. (C)
- Hexagonal pile with 7 piles (one pile in the centre). (C)

***N.B.:*** *Using CYPECAD, several supports can be defined on one pile cap, but do not forget that the resultant force is obtained and it is with this force that the program checks the pile.*

## *4.7.1 Design criteria*

Type A pile caps are designed based on a model consisting of concentrated loads. They are reinforced using vertical and horizontal stirrups with the option to include diagonal stirrups).

Type B pile caps are designed using the strut and tie method. They are reinforced as beams, with bottom, top and skin longitudinal reinforcement as well as vertical stirrups.

Type C pile caps are based on strut and tie models. They may be reinforced with lateral and diagonal beams, top and bottom meshes, and perimeter reinforcement.

Type D pile caps are based on strut and tie models. They may be reinforced with lateral and diagonal beams (except the rectangular model), top and bottom meshes.

Any pile cap can be designed or checked.

The check consists in verifying the geometric and mechanical aspects when the pile cap has been designed or entered with a set of dimensions and reinforcement. Loads may or may not be defined. The design does not require loads, and as of a set of minimum dimensions, taken by the program, (complete design) or initial dimensions provided by the user (minimum dimensions), the program provides (if possible) the geometric parameters and reinforcement in accordance with the code and the defined options.

As the *EHE Spanish code* is the code which contains the most information and analysis for pile cap design, it has been adopted as the basic design code. They are also designed to be rigid and, where possible, principles of other sources such as the *ACI-318/98*, *CIRSOC*, *NB-1*, *EH-91*, technical bibliography such as the *Marcelo da Cunha Moraes's book "Foundation structures"* and criteria of CYPE; have been applied.

## 4.7.2 Sign criteria

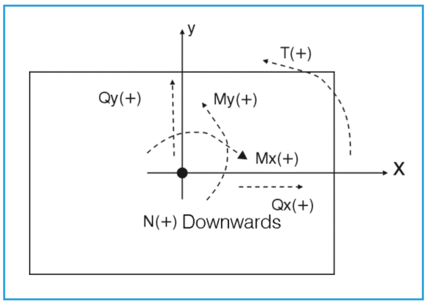



Figure 77

## 4.7.3 Design and geometry considerations

When defining a pile cap, the user also has to indicate the type of pile, how many there are and their position. The carrying capacity of the pile, i.e. the service load it can hold (without applying the safety factor) has to be defined.

The load the piles are going to receive has to be calculated first. This is obtained by considering the self weight of the pile, the external loads and applying **Navier's classic formula**:

$$P_i = \frac{N}{\text{number of piles}} + M_x \cdot \frac{x_i}{\sum x_i^2} + M_y \cdot \frac{y_i}{\sum y_i^2}$$

with the bearing pressure combinations on the soil.

The most loaded pile is compared with its carrying capacity and if it is exceeded the program emits a warning.

When a pile is defined, the minimum distance between piles is required. This data must be provided by the user (default value set at 1.00 m). The program checks this distance is greater than the minimum distance.

The checking and design of piles is based on the maximum load of the most loaded pile after applying the selected concrete combinations to the loads for the loadcases that have been defined.

If the user wishes for all the piles of the same type to be of the same size and contain the same reinforcement arrangement, an option is available within the pile cap options, **Loads per pile**, whereby the user may unify the pile caps, upon activating it, and so they can be designed for the carrying capacity of the pile.

In this case, the user has to define a load factor (safety coefficient to consider it as one more loadcase) which shall be taken as the **Usage coefficient** of the pile (default value of 1.5). If not all of the carrying capacity of the pile is to be considered, only a percentage may be considered. This value is defined as the **Fraction of load** and may vary between 1 and 0 (default value of 1). In this case, the program will determine the maximum value between the previous value, which is a function of the carrying capacity of the pile and the maximum of the piles due to the applied external loads.

This is common practice in some zones and countries as a single pile cap is obtained for each pile diameter and number of piles, hence simplifying its execution on site. The program has this option deactivated and must be activated by the user.

With respect to the forces, the following checks are carried out:

- Warning of tensile forces in the piles: maximum tension ≥10% maximum compression
- Bending moment warning: strap beams must be provided
- Excessive shear: if the shear force for any combination exceeds 3% of the axial force with wind, or in other combinations where inclined piles may be required
- Warning of torsional forces if these have been defined in the loads

If strap beams are entered, they will absorb the moments in the direction in which they act. For pile caps with a single pile, strap beams are always required in both directions. For linear pile caps with 2 piles, they are required in the direction perpendicular to the line of piles. In these cases, the strap beam is designed for an additional moment of 10% of the axial force.

Increase the moments by this value ($0.10\times N$) for the corresponding loadcases if deemed necessary and if possible -only in the case of column starts; or check the loads acting on the piles and their load reserve.

If more than one strap beam were to act in the same direction, the moment would be shared proportionally with respect to their stiffness. The following checks are undertaken:

- **General checks:**
  - Shear wall warning
  - Warning supports are too separated (in CYPECAD)
  - Warning no supports have been defined
  - Minimum overhang as of the perimeter of the pile
  - Minimum overhang as of the axis of the pile
  - Minimum overhand as of the column
  - Minimum width of the pile
  - Carrying capacity of the pile
- **Specific checks:**

  The geometric and mechanical checks specified in the selected code are carried out for each type of pile. It is recommended the user carry out an example of each type and obtain their respective reports containing the checks that have been undertaken as well as any warnings which may have been emitted, references to articles of the code or criteria used by the program.

  Reports containing the data that has been defined for the pile caps can be obtained, as well as their material take-offs, pile table and check list.

  Regarding drawings, the geometry and reinforcement arrangement of the piles is available graphically, with additional material take-off and summary tables.

*As has been mentioned previously, various supports may be defined on the same pile, be it column or shear wall. Therefore some geometric restrictions have been imposed which warn the user when the distances between the supports and the edge of the pile caps or to the piles is limited.*

*When several supports bear on a pile cap, their resultant force is applied at the centre of the pile cap, using the strut and tie method, and assuming the pile cap is rigid. Therefore the validity of this method must be assumed, however, depending on the specific case being analysed, the case may arise that this method is not valid. Hence, the user must carry out the appropriate manual corrections and complementary calculations.*

## 4.8 Baseplates

When checking a baseplate, the program considers the plate to be rigid in accordance with the Bernoulli hypothesis. This implies that the plate is assumed to remain flat when submitted to the applied forces, and so, any deformation which may arise is ignored and does not affect the load distribution. For this to be fulfilled, the baseplate must be symmetrical (which the program always guarantees) and sufficiently rigid (minimum thickness depending on its length).

The checks that are carried out to validate a baseplate can be divided into three groups, depending on the element being checked: foundation concrete, anchorage bolts and the baseplate itself, with stiffeners, if present.

1. **Concrete bearing check.** This consists in verifying that the allowable bearing pressure of the concrete is not exceeded below the most compressed point. The allowable bearing pressure method is used in this case, whereby a triangular pressure distribution is assumed to act on the concrete which can only act in compression. The concrete check is only carried out when the plate bears on it, and does not have a simple or composite tensile state. Any friction between the concrete and baseplate is ignored, i.e. shear and torsion resistance is provided exclusively by the anchorage bolts.
2. **Anchorage bolt check.** Each bolt is, generally, submitted to an axial force and a shear force. Each is evaluated independently. The program considers the bolts of plates bearing directly on foundation elements to only work in tension. If the baseplate is at a distance from the top of the foundation element, the bolts may work in compression, in which case the corresponding buckling checks will be carried out on the bolts (the model of a beam fixed at both ends is taken, with the possibility of the supports being able to slide in a direction normal to its longitudinal axis: b=1) and the forces transferred to the foundations (bending forces appear due to the shear forces acting on the section).

   The program creates three check groups for each bolt:

   **Stress in bolt stem.** This consists in checking that the stress does not exceed the design resistance of the bolt.

   Bolt anchorage in concrete. Apart from failure of the bolt stem, another reason for failure may be due to failure of the surrounding concrete. This may occur because of the following reasons:

   - Slipping of the bolt due to lack of adherence
   - Failure due to concrete fracture cone
   - Failure due to shear forces (stress concentration)

To be able to calculate the fracture cone for each bolt, the program takes a set of lines with a common apex that form an angle of 45º with the axis of rotational symmetry. The program does take into account the reduction of the effective area within the fracture cone in question, due to the presence of other bolts nearby.

The following effects are not taken into account by the program and must be verified by the user:

- Bolts which are very close to the edge of the foundation element. No bolts should be at less than their anchorage length from the edge of the foundation element, as this reduces the effective area of the fracture cone. Additionally, another failure mechanism appears: lateral failure due to shear, which is not contemplated by the program.
- Reduced thickness of the foundation element. The program does not contemplate the global fracture cone effect which appears when there are several bolts and the thickness of the concrete is small.
- The program does not contemplate the possibility of using pass-through bolts, as it does not carry out the required checks for this case (stresses on the other side of the concrete).

**Crushing of the plate.** The program also checks that, at each bolt, the shear force that would produce crushing of the plate against the bolt is not exceeded.

**3. Checks carried out on the plate.**

Calculation of global stresses. The program builds four sections at the perimeter of the section, and checks each of them against the stresses. This check is only undertaken for baseplates with overhangs (local buckling of the stiffeners is not taken into account, and the user must check that their respective thicknesses do not make them excessively slender).

Calculation of local stresses. The section and stiffeners divide the main plate into local plates which are then checked. Using the pressure distribution of the concrete and the axial forces in the bolts, the weighted worst case bending moment of each local plate is obtained. This is compared with the plastic failure bending moment. This seems reasonable, as the worst case point of each local plate is taken when it is being checked, as it is here where a local stress peak will be produced due to the appearance of plastic deformation, without reducing the safety factor of the baseplate.

## 4.9 Mass concrete footings

Mass concrete footings are those in which the ultimate limit state forces are resisted exclusively by the concrete.

The program allows for reinforcement meshes to be placed in the footings. However, the element as a whole shall be taken as a weakly reinforced element in which the mission of the reinforcement is really to control any fissures due to thermal retraction and contraction, but does not contribute in increasing its force resistance.

At this point, it is worth highlighting that, against common belief, mass concrete structures require extra care in their design and execution stages than reinforced or prestressed concrete.

The main differences between mass concrete footings and reinforced concrete footings will be dealt with in this *Calculations manual*. For options which are common to both types, the user will be referred to the section on reinforced concrete footings.

### *4.9.1 Design of footings as rigid solids*

The design of pad footings as rigid solids consists of two checks:

- Check for overturning
- Check of the allowable bearing pressures on the soil

These two checks are identical to those carried out for reinforced concrete footings and are explained in their respective chapter.

### *4.9.2 Design of footings as mass concrete structures*

It is in this section where the footings present the greatest differences with respect to reinforced concrete footings. The three checks carried out for the structural design of mass concrete footings are detailed below.

#### *4.9.2.1 Check for bending*

The reference sections of mass concrete footings used for bending design are the same as those used for reinforced concrete footings and these are explained in the corresponding section of this *Calculations manual*.

The program verifies for all the sections that the bending stresses, for the flat deformation hypothesis, produced due to the design bending moment, have to be smaller than the tensile bending resistance, given by the following formula:

$$f_{ck,min} = 1.43 \cdot \left( \frac{16.75 + h^{0.7}}{h^{0.7}} \right) \cdot f_{ctd,min}$$

$$f_{ctd,min} = \frac{0.21}{1.5} \cdot \sqrt[3]{f_{ck}^2}$$

Where $f_{ck}$ is in N/mm$^2$ and h (depth) in mm.

### 4.9.2.2 Check for shear

The reference sections used for shear design are the same as those used for reinforced concrete footings and are explained in the corresponding section of this *Calculations manual*.

All the sections have to verify that the maximum tangential stress produced by the shear force must not exceed $f_{ct,d}$ given by:

$$f_{ct,d} = \frac{0.21}{1.5} \cdot \sqrt[3]{f_{ck}^2}$$

### 4.9.2.3 Check for oblique compression

The check for concrete failure due to oblique compression is carried out at the edge of the support. The program checks the design tangential stress at the perimeter of the support is less than or equal to a maximum value.

This check shall be carried out in the same way, regardless of the concrete code being used; by applying article *46.4* of the *Spanish EHE-98 code*. This article establishes the following conditions:

$$\tau_{sd} \leq \tau_{rd}$$

$$\tau_{sd} = \frac{F_{sd,ef}}{u_0 \cdot d}$$

$$F_{sd,ef} = \beta \cdot F_{sd}$$

$$\tau_{rd} = f_{1cd} = 0.30 \cdot f_{cd}$$

Where:

- $f_{cd}$ is the design compression resistance of the concrete
- $F_{sd}$ is the axial force the support transmits to the footing
- $\beta$ is a coefficient which takes into account the eccentricity of the load. When there is no moment transmission between the support and the footing, this coefficient is equal to one. If there is moment transmission, depending on the position of the columns, the coefficient will be equal to the values indicated in the table below.

| | $\beta$ |
|---|---|
| Internal supports | 1.15 |
| Edge supports | 1.4 |
| Corner supports | 1.5 |

Load eccentricity coefficient values

- $u_0$ is the perimeter being checked. The values are as follows:
  - For internal supports, it is equal to the perimeter of the support.
  - For edge supports: $u_0 = c_1 + 3 \cdot d \leq c_1 + 2 \cdot c_2$
  - For corner supports: $u_0 = 3 \cdot d \leq c_1 + c_2$

    Where $c_1$ is the width of the side of the support parallel to the side of the footing in which the support is an edge support and $c_2$ is the width of the footing in the direction perpendicular to the edge.
- d is the effective depth of the footing.

This check is carried out for all the supports reaching the footing and for all the combinations of the concrete combination group.

The maximum tangential stress incurred after checking all the supports and for all the combinations can be seen in the analysis report.

This check is similar to that carried out for reinforced concrete footings.

## 4.9.3 Design report

In this section, the checks undertaken by the program for mass concrete footings (constant depth, variable depth and tapered) will be commented on.

#### 4.9.3.1 Minimum depth check

The program checks that the depth of the footings is greater than or equal to the minimum value indicated in the corresponding code for mass concrete footings.

In the case of tapered or variable depth footings, this check is carried out at the edge.

#### 4.9.3.2 Minimum depth check (for reinforcement anchorage lengths)

The program checks the depth of the footing is greater or equal to the minimum value required to anchor the column reinforcement or the baseplate bolts which are supported by the footing.

In the case of tapered footings, the depth that is checked is the depth at the pedestal.

#### 4.9.3.3 Maximum slope angle check

This check is similar to that carried out in the case of reinforced concrete footings.

#### 4.9.3.4 Check for overturning

The overturning check is similar to that carried out in the case of reinforced concrete footings.

#### 4.9.3.5 Soil bearing pressures check

The soil bearing pressures check is similar to that carried out in the case of reinforced concrete footings.

#### 4.9.3.6 Check for bending

The check is carried out in accordance to that indicated in section ***2.1. Discretisation.*** Displayed below is the data shown in the check report for each direction.

If all of the sections successfully pass the bending check in one direction:

- The worst case design moment acting at the section.

- Within the additional information section, the maximum usage coefficient, which is the ratio of the maximum acting moment to the resisting moment.

If a section fails, the data displayed in the check list for each direction are as follows:

- The first bending moment which has been found which the section cannot resist.
- The coordinate of the section at which this bending moment acts.

#### 4.9.3.7 Check for shear

The shear check is carried out in accordance to what has been explained in the corresponding section of this *Calculations manual.* The data displayed in the check reports is shown below.

If all of the sections successfully pass the shear check in one direction:

- The design tangential stress which produces the greatest acting tangential stress to resisted tangential stress ratio.
- The resisting tangential stress of the same section whose maximum design tangential stress is displayed.

If a section (in one direction) fails the shear check, the data shown in the check report is as follows:

- The design tangential stress of the first section found for which the shear check fails.
- The coordinate of the section which has been found to fail.

#### 4.9.3.8 Check for oblique compression

This check is similar to that carried out in the case of reinforced concrete footings and is explained in the corresponding section of this *Calculations manual*.

#### 4.9.3.9 Minimum reinforcement spacing check

This is the only check carried out on the reinforcement a user may place in a footing, as these are not taken into account in the analysis.

In this check, the program verifies the separation between the reinforcement axes is equal to or greater than 10 cm. This value is imposed regardless of the selected code as criteria of CYPE.

This check is undertaken only in the case of when the user decides to place a mesh, and what the program intends to avoid is for the bars to be placed so close to one another that pouring of the concrete of the footing may be hindered.

## 4.10 Specific checks due to the code that has been considered (footings, beams and pile caps)

All the general checks that have been indicated can or cannot be undertaken depending on the selected code. It is recommended a checklist be created of the results containing all the design code and calculated values that have been applied to each foundation element.

## 4.11 Footings with non-rectangular limits

All that has been exposed previously is valid for non-rectangular footings, bearing in mind that the same method is applied as when there is more than one support on a footing.

# 5 Corbels

CYPECAD allows for corbels to be defined at column faces. Only reinforced concrete or steel beams may be entered to rest on the corbel, and transmit the vertical load to the centre of the support at a distance “a” from the face of the column.

The corbel transmits, with its eccentricity, the forces to the column as an eccentric rigid bar. The corbels have been brought about to be used, for example, instead of a double column at a joint, or if, for whatever reason, it may not be convenient to fix the beam to the column in that particular direction.

Corbels should not be used as column start points, as the program does not allow for the column to be entered on the beam on which it starts, and the beam be supported at both ends. In other words, columns cannot start at corbel ends.

To develop the analysis and design of reinforced concrete, the methods described in each concrete code that has been selected are followed, in all aspects and with the corresponding checks. In the case of codes not containing any specifications, the criteria of other codes, which are most similar and follow the same criteria of the program, are used and these are stated in the check report.

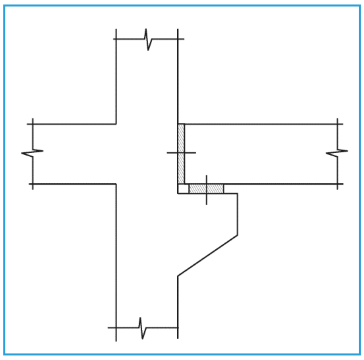


Figure 78. Corbel diagram

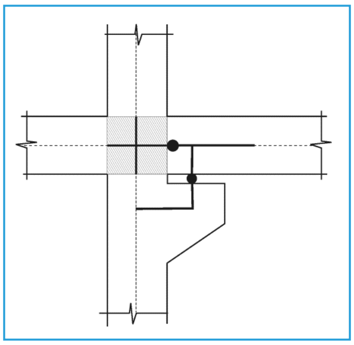


Figure 79. Simplified model of the intersection zone of the corbel

# 6 Joist floor slabs

## 6.1 Concrete joists

### 6.1.1 Geometry

This is defined in the data sheet of the floor slab.

### 6.1.2 Stiffness considered

The gross stiffness, when obtaining the stiffness matrix of the bars of the structure, is that corresponding to a T section with haunches.

The secant modulus of elasticity defined for the concrete of the floor slabs will be taken here.

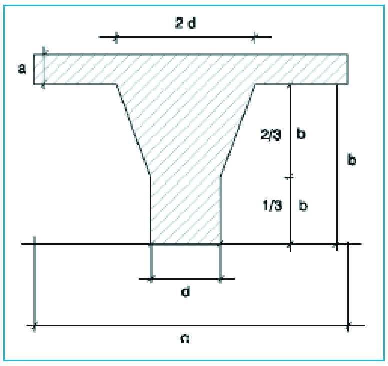



Figure 80

Where:
d: Rib width = rib width + rib width increment
a: Compression layer thickness
c: Rib spacing
b: Height of the form

### 6.1.3 Estimating the deflection

The same method used for beams is applied here (*Branson* method), whereby the equivalent stiffness is calculated along the joist at 15 points.

The gross stiffness is that estimated for the analysis and the cracked stiffness is obtained depending on how the user has defined the check for deflection in the floor slab dialogue box options:

- **As reinforced joist.** The top reinforcement is designed and known. This is not the case for the bottom reinforcement, and so the program obtains the required reinforcement based on the positive bending moment, and hence this way estimates the cracked stiffness.
- **As prestressed joist.** In this case the cracked stiffness has to be indicated as a %age of the gross stiffness. This depends on the type of joist and how much it has been prestressed. It may be convenient to consult manufacturers to enter an adequate value.
- **Check for shear.** The value of the shear force at the supports is provided. The user is then responsible for the shear check.

## 6.2 Reinforced/Prestressed joists

These joists are prefabricated joists at specific factories which are then transported onto site to be placed in the building structure.

A specific properties sheet for the different types of joists is provided displaying their moments, stiffnesses, etc. The data entered in the program is that contained in the data sheets provided by the manufacturers. These may not be edited, nor can the user create his/her own user data sheets. The user is to contact our Technical Department, and send the required documents so their technical data sheets may be included in future editions of the program, which shall be implemented as soon a the data provided has been validated.

Users can also create their own properties sheet (Library) using an independent program (Floor slab file editor) whereby a file with all the properties can be created, imported to the Library and then used in any job.

The deflection is estimated and a shear check is carried out. When designing the joist for bending, the program verifies if there are any joist types which do not fail due to positive bending and have their top reinforcement defined for negative bending within their data sheets. Please recall that the top reinforcement is defined in the data sheets designed to resist a specific moment with a specific cover, which should be taken into account to accept the validity of these manufacturer sheets.

When data is present in the sheets, the cracked limit state can be checked depending on the environment or permissible cracked width, forcing the design.

The property sheets can only be created for the *Spanish*, *Portuguese* and *Brazilian codes*. This feature is not available for other codes.

## 6.3 In-situ joist floor slabs

### 6.3.1 *Geometry*

The basic parameters are defined, entering them in the sheet and selecting the type of form.

### 6.3.2 *Stiffnesses*

These are obtained based on the gross section of the concrete variable width T rib depending on the form, compression layer and rib spacing.

### 6.3.3 *Estimating the deflection*

The *Branson* method is applied, as both the top and bottom reinforcement are known, which are designed and the deflection lengths obtained.

### 6.3.4 *Design for bending*

The same criteria are applicable for the top reinforcement as is for the previous types of concrete joists. The bottom reinforcement is designed in accordance with the general concrete code selected for the design of all the concrete elements. Joist top reinforcement tables are available which are also used for generic concrete joists, and a specific table for bottom reinforcement of in-situ joists. Its structure is similar to that of waffle slab ribs.

### 6.3.5 *Design for shear*

As the rib is known and its longitudinal reinforcement, as well as the shear forces acting on the joist, the program checks if vertical reinforcement is required. If it is, the program places vertical reinforcement legs in accordance with a diameter/spacing table.

Bottom reinforcement anchorage. In accordance with that indicated in the various codes, the anchorage lengths are obtained at the end supports (beams or supports) dimensioning the bar end lengths, and necessary hooks.

## 6.4 Steel joists

### 6.4.1 *Geometry*

The following data is defined: type of form, compression layer thickness, rib spacing, type of section (from the program's section library).

They are designed using the same criteria applied to steel beams, with the exception that, as all the spans are considered to be simply supported, i.e. pinned at their supports, lateral buckling is not considered, as the top flange is considered to be braced by the compression layer when designing the joists for positive moments. The program does not design the joists to resist negative moments. Hence if the case arises, the program will emit this as an error, such as in the case of overhanging joists. This is because the program does not detail the fixed or continuous connection with other panels made up of these joists which are submitted to negative moments, even though the program designs it in this manner when this fixity is required when calculating the forces to obtain equilibrium in these bars.

If a solution has not been provided for these connections, the user will have to design the steel joists in areas where negative moments are present.

The user is reminded that the sections are designed for simple bending, with moments and shear, ignoring axial forces and forces in the plane of the floor slab, due to the rigid diaphragm hypothesis.

## 6.5 Open-web joists

### 6.5.1 *Geometry*

These ribs are made up of trusses composed of steel sections containing a top and bottom chord and diagonals spanning between them. The chords may consist of single, double or quadruple circular or square hollow sections/ tubes, or double or quadruple angles. The diagonals will be of the same section, only simple, and of the same series.

The following data is defined: total external depth (nominal depth) of the truss, rib spacing and the thickness of the slab above the truss. The slab does not collaborate with the truss, it simply resists and supports the applied loads

### 6.5.2 *Considered stiffness*

The stiffness considered by the program is that of the steel truss made up by the two chords with the defined separation, and taking the first section defined in the sections of the job, or whichever has been assigned in a later analysis. In a similar way as with steel joists, these are designed as simply supported spans, pinned at their ends, and hence they are not designed to resist negative moments.

### 6.5.3 *Joist design*

They are designed as trusses, the moments are decomposed into a compressive force applied at the top chord, which is assumed to not buckle due to the bracing effect of the concrete slab, and a tensile force at the bottom chord. The diagonals are designed for tension or compression, assuming the loads are applied at the nodes, whereby their resultants are found depending on the geometry, height and spacing of the truss. When designing for buckling, they are considered as being bars pinned at their ends, with an effective length equal to the real length of the diagonal bar.

The deflections are obtained as a beam with the aforementioned stiffness.

## 6.6 Comments on the use of joist floor slabs

The previously mentioned joist floor slabs are discretised in the analysis of the structure as bars in the integrated analysis of the whole structure, coinciding with the axis of each rib that has been defined. The compression layer and form produce a “load distributing” effect, which in reality, occurs due to deflection compatibility. Nonetheless, it is always more approximate than the assumption of the floor slab behaving as a continuous beams supported by rigid pinned supports, which only really occurs when the beams are rigid and torsional stiffness can be ignored even when the spans vary greatly.

In practice, flat beams and dropped beams are present which, with the spans used, deflect more than they should, and present a certain amount of flexibility which the analysis detects.

The deformation compatibility which must always be complied with, unless the section fractures or becomes excessively plastic, makes it obligatory for both beams and joists to displace in a joint manner. This may cause the joist floor slabs to behave in a non-foreseeable or anomalous way, whereby structural schizophrenia phenomena can arise, usually caused by forcing the design of the structure in such a way that the joists support the beams.

This does not imply that the analysis is incorrect, simply that the design, due to it being inadequate, exhibits unusual behaviour.

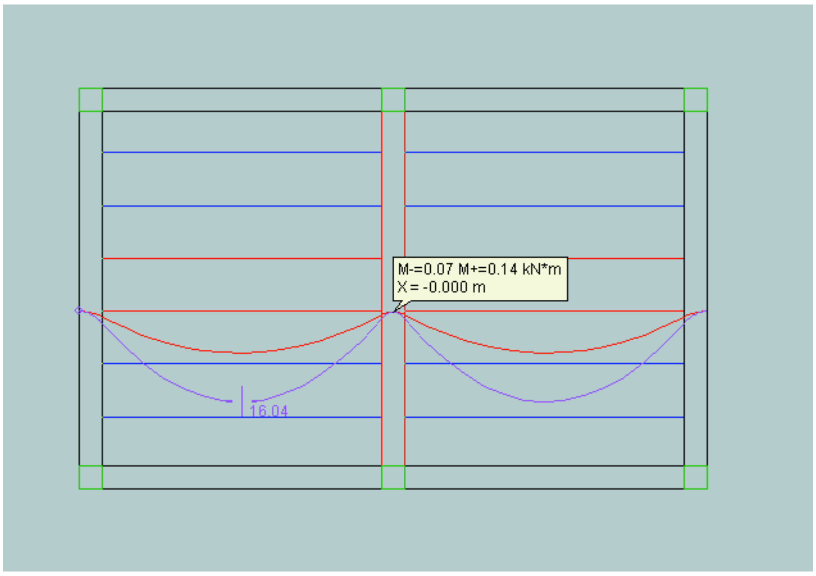



Figure 81

For this reason, as of the 2002 version, whenever positive moments appear at the support, the program shows this situation by displaying all the joists where this occurs in red.

This is particularly important in the case of generic joist floor slabs (even if the user already knows beforehand which type of joist is going to be used, prefabricated reinforced concrete joists and prestressed joists, where the user cannot assure the anchorage at the supporting edge of the beam.

When faced with these circumstances, the user's choice of action can differ depending on each case:

- Vary the structural design, shortening the span and increasing the stiffness of the beam.
- Pinning the edges of the panels, so the joist floor slabs become simply supported.
- Using in-situ joists in which the bottom reinforcement can pass through and so anchoring and overlapping.

Whichever the case, an unavoidable step consists in consulting the force envelopes of the joist alignments. The user can then opt to ignore the warning if the positive moment is negligible.

The user should also consult the shear force envelopes, as by using these, the user can deduce the force transmission of the joists to the beams, and the case may occur, when this transmission is scarce or negative, as has already been mentioned.

If the user adopts the habit of checking the force envelopes, the consequences of entering horizontal loads, i.e. wind and earthquake loads, can be “seen”.

If the structural design is based on a roughly orthogonal grid composed of beams spanning across supports, the joists will usually limit themselves to transmitting vertical loads to the beams.

If, however, within the structural design and dominant direction adopted by the joists, there are no beams present that tie the supports, a virtual beam-frame is created using the joist or joists that run in close proximity to the support. This implies that they also support the horizontal loads in the same way as the other frames of the structure in that same direction. The user will have to be careful with this model, especially when the alternating of the moments in those joists leads to the appearance of large positive moments at the supports. Therefore, a solution will have to be provided be it conveniently reinforcing the joists (in-situ, infilling and providing positive moment reinforcement, etc.), placing beams or, if the user wishes to “trick” the model and vary the position of the joists so none of them pass through the support, entering equivalent non-structural beams, which transmit the vertical loads but do not collaborate in resisting the bending moments due to frame-effect.

If the user still wishes for the joists to not collaborate, or may collaborate due to torsion of the beams, the adequate model would consist of not entering the floor slab and substitute its reactions on the beams with line loads, calculated manually. Alternatively, an analysis using continuous slabs can be carried out only for vertical loads. This can then be copied and the user modify the fixity coefficients at the support edges so they equal zero, i.e. pinned, hence the floor slabs will behave as if they were simply supported. The position of the joists can also be modified, so none coincide with the support and a very small torsional stiffness coefficient (0.001) in the case of short bars can be entered.

Having explained the matter, the user must recall that an expected solution cannot always be obtained using approximate methods. The alternative methods “work”, however in many cases, it is the absence of the service loads and the use of partial safety coefficients which allow the structure to “work”.

On the other hand, the user must also recall that the results of the analysis should be checked and analysed using the available tools.

# 7 Sloped floor slabs

Sloped floor slabs may be entered in CYPECAD, as is explained in the *User's manual*, with the indicated possibilities and limitations.

It should be known that sloped floor slabs have the same properties as horizontal floor slabs, and in the structural model that is generated upon inclining a plane, logically, the dimensions of the bars within that plane will vary and the lengths of the supports reaching the slab will differ. All this may be visualised and consulted by activating the option **Envelopes > 3D Model**, for the last job that has been analysed.

To "view" the structure, use the 3D view of the building or floor, to be able to see a particular floor.

It is very important that the following information on sloped slabs be taken into account:

- The rigid diaphragm hypothesis is maintained, this implies there is no relative displacement at 2 points of the floor, even if they are sloped slabs.

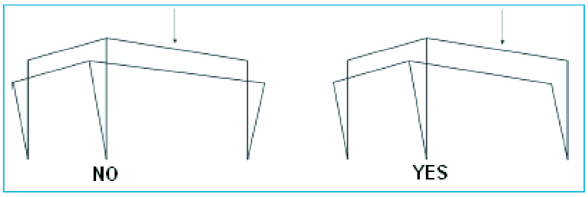



Figure 82

  In other words, the group of horizontal and sloped planes are displaced in a joint manner, in accordance with the rigid diaphragm hypothesis.

- It is recommended that at the intersecting edges of the sloped slabs, where beams have been defined, that supports be present to hold the beams (Figure 83b), (ridge and valley beams) and not defining structural systems whereby planes are supported by other planes.

  This is important as all the elements that belong to horizontal or sloped planes, beams, joists, hollow core plates, flat and waffle slabs are designed for simple bending and shear, ignoring any effects due to axial force, be it a compressive or tensile force. Hence, structural designs which inevitably produce these forces should be avoided.

  It is convenient the user recall that axial forces appear even with horizontal planes. A simple test can be carried out by analysing a basic frame or lintel spanning over two columns and axial forces will arise in the lintel, which varies depending on the stiffness of the supports. This is usually taken as a second order effect and is ignored.

Therefore, sloped floor slabs have to be used wisely, so these effects and possible non-compensated lateral forces do not arise. By carrying out normal structural designs and based on general practice, problems will rarely be incurred.

Use “sloped beams” (Figure 83d) with 6 degrees of freedom, which are designed for axial forces, when cases that do not follow the previous recommendations arise.

Exempt beams disconnected from the rigid diaphragm can also be used. Consult chapter ***12. Rigid Diaphragm***.

- When the user wishes to eliminate columns at ridges or valleys, a horizontal slab can be entered which acts as a tie. If normal inclinations and spans are employed, this floor slab, with its corresponding mesh, will be able to absorb the tensile forces (Figure 83e).

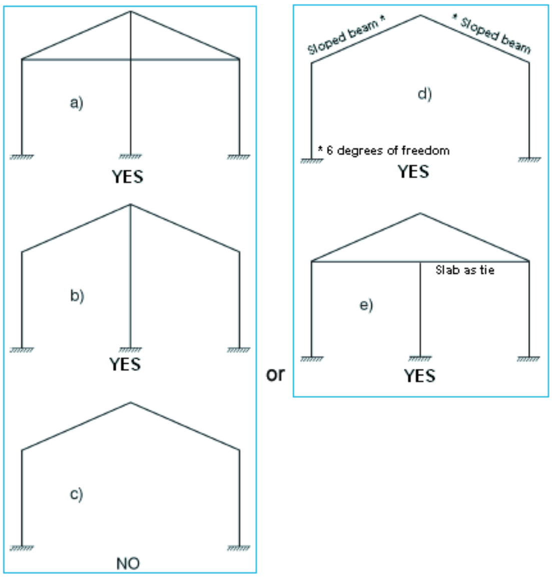



Figure 83

- Sloped slabs are not to be supported by external fixity beams, unless they are very small, as the external fixity beam will absorb the horizontal forces without transmitting them to the rest of the structure.

  They are not to be supported by masonry walls either, unless other structural elements are present which are capable of absorbing the horizontal forces.

  Masonry walls are elements which work well when confronted with vertical loads, however they do not when exposed to bending in a direction normal to their plane.

- Common beam. This option is used to define beams which belong simultaneously to two groups, where one of them is a sloped slab which reaches the beam.

In diagrams (a) and (e) of Figure 83, common beams would be used for the end beams perpendicular to the plane of the drawing.

The beams receive the loads from both slabs and are visible at both groups. They are drawn differently (with a discontinuous line along their axis) so they may be differentiated from other beams. They are always designed with rectangular sections even though they may be trapezium shaped due to the intersection of both planes.

Ridge and valley beams possess the same condition; they are designed as rectangular beams, even if they are flat and drawn V-shaped.

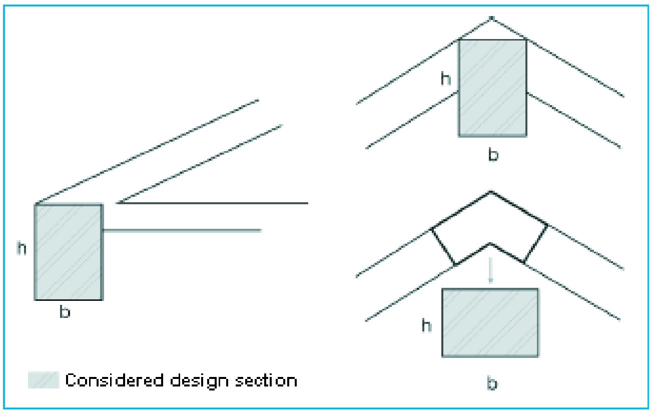
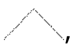
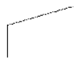



Figure 84

Sloped slab reinforcement (joist floor, flat and waffle slabs) are drawn projected on plan, however its real design length is provided.

When the beam bends, an optional symbol is indicated: , ,  , so the user may know the shape of the bars at the points.

- The design mesh and reinforcement arrangement in flat and waffle slabs is always orthogonal, in which one of the reinforcement directions is always placed in the direction of the maximum slope and the other perpendicular to it.
- Regarding applied loads:
    - The self-weight of the structural elements (beams and floor slabs within inclined planes) and the dead loads included in the floor groups (**Column Definition tab > Introduction > Floors/Groups > New floors or Edit groups**) are obtained directly and calculated automatically, as their real magnitude is known.
    - Any additional loads associated to any other permanent load loadcases (self-weight and dead loads), must be increased proportionally due to the inclination of the plane.

For example, in the case of a 100% slope (45º), which is a large slope:

$$p = \frac{q}{\cos \alpha} = \frac{q}{\cos 45^{o}} = \frac{q}{0.707} = 1.41\,q$$

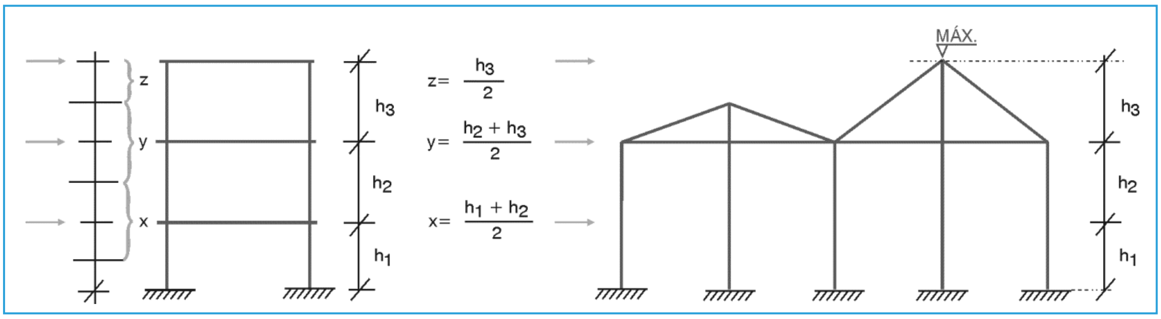



Figure 85

- There is no need to modify the live load as it is considered to be projected horizontally, therefore, if the live load is 1kN/m$^2$, 1 is the value which is to be entered as the live load of the floor, or as the surface load as a special load within a zone with a polygonal outline.

Regarding line loads, for example due to partitions on sloped panels whose vertical height is constant and known, these can be obtained by multiplying the height by the weight per square metre of the partition.

- Snow should be entered as a live load.
- Regarding horizontal loads, the following should be taken into account:
  - Wind. This load is found for each floor level, whose value is obtained by multiplying the defined band width by the sum of the half-heights of the floor, and then applying it at the geometric centre of the floor as a horizontal load. Therefore the user is to bear in mind that if the roof contains any sloped slabs, the height (h) of the floor is to be that of its highest point, which will always provide a wind load with a larger applied safety factor.

Neither components projected vertically nor components normal to the sloped slabs (perpendicular to the panels) are considered, as these are usually not determinant in a building, even though the pressures may vary between 0.1 and 1 kN/m$^2$ and if minimum values (always greater than that wind pressure) have been considered when defining the live loads, there is no need to worry. Logically, this is for the case of sloping roofs exposed to wind loads where the dead load and cover material represent approximately 80% of the total load for normal buildings. It is not the case for a warehouse with lightweight roofing. If this were the case, the load entered would have to be the envelope of the snow and wind loads.

## 7.1 Element design

As has been mentioned earlier, the program designs all sloped elements for simple bending and shear, ignoring any axial forces.

The reinforcement details of beam alignments belonging to sloped panels are presented as beam elevation drawings with their real magnitude and shape.

Top reinforcement of joist floor slabs and bottom reinforcement of in-situ joists, waffle slabs and flat slabs are drawn projected horizontally and dimensioned with their real magnitude.

# 8 Composite beams

The analysis and design of composite beams is carried out in accordance with *Eurocode 4* (Design of composite steel and concrete structures).

Steel sections (I beams) can be entered below flat slabs whereby the concrete above it collaborates by means of shear studs.

A partial fixity coefficient of 0.05 is applied at edges connected to composite beams (in the same way as the top end of the last span of columns), so to reduce the negative moment at the support by increasing the positive moment.

Composite beams are designed in such a way that the steel section resists all the forces in the area with negative moments, whilst the area of positive moments is resisted by the composite section.

When designing for bending, the width of the contributing concrete is not required; the program calculates this value automatically:

- In the case of flat slabs, the effective width defined in *Eurocode 4* is taken.
- For sloped, waffle, hollow core and joist floor slabs, the value will be the smallest amongst the effective width and the width of the flange plus 10 cm at either side (if not at an edge, if it is, the program calculates the width of the flange plus 10 cm).

To check the sections in positive moment areas, the effective width is different to that considered when designing for bending. Therefore the effective width for the negative moment area appears within the beam reinforcement editor, in case reinforcement is added at the supports.

To design the selected steel section and concrete slab, the program uses the concrete and steel codes selected in the *General data* dialogue box.

# 9 Composite slabs

Composite slabs are composed of a reinforced concrete slab and steel deck, which also serves as formwork for the concrete. The deck can be used to work in the following ways:

- As a form deck. During the construction phase, the deck alone resists its self weight, the weight of the fresh concrete and the construction live loads. During the service phase, it is only the reinforced concrete slab that has a resisting function.
- As a composite deck. During the construction phase, the deck works as lost formwork (as in the previous case). During the service phase, the deck is considered to combine structurally with the hardened concrete, acting as reinforcement in tension, resisting the positive moments in the finished floor slab. The deck is capable of transmitting shear stresses at its interface with the concrete as long as a mechanical connection is provided by the deformations in the deck (embossments).

The analysis and design of the decks is carried out in accordance with *Eurocode 4 (Design of composite steel and concrete structures)*.

Composite slabs are applicable to structural building projects when the loads imposed are predominantly static loads, including the case of industrial buildings whose floor slabs may be submitted to mobile loads.

The following parameters are limited: total depth of the composite slab, thickness of the concrete above the deck ribs and minimum height of the shear studs above the deck ribs (in the case of composite beams).

The deck may be supported by steel, composite or concrete beams, walls, etc. where a minimum support length is required.

The analysis and design process is carried out in two phases:

**a. During the construction phase**

- When analysing the resistance of the deck, the program takes into account the weight of the concrete, the steel deck and construction loads. The construction loads represent the weight of the workers and concreting equipment and any impact or vibrations which may arise during its construction.
- When calculating the deflection, the construction loads are not taken into account.
- A fixity coefficient of 0 between the panel and any perimeter beams is considered internally by the program (simply supported ribs).

- The option is available to design the deck if an ultimate limit state or deflection limit state is not complied with, or calculate the separation between shoring without designing the deck. If, in the first case, a valid result is not obtained, the program calculates the distance between shoring.

**b. During the service phase**

- For the service phase, the design continues as of the deck obtained in the previous phase.
- The program automatically assigns the panels a fixity coefficient of 0, so the load distribution to the steel beams supporting the floor slab, is carried out in accordance with the theoretical band width and avoids the appearance of positive moments at intermediate supports. This can only be achieved by assigning a fixity coefficient of 0, regardless of the stiffness of the beams, or by correctly pre-designing the beams. By carrying out an initial analysis and designing the beams, the user can change the value of the fixity coefficient (between 0 and 1) and repeat the analysis. If the user assigns a fixity coefficient other that 0, two outcomes may be obtained:

    1. In the previous phase, a slab was obtained without requiring shoring (self supported), if a suitable deck was found. In this case, the slab must only be designed with the additional load after the slab has been executed, composed of the respective dead and live loads, as the deck has supported the self weight of the slab. The way in which the program can approximately take into account these loads is by applying fixity coefficients, which the program calculates and applies internally to continuous panels. As a guide, the value of the fixity coefficient to assign to panels depends on the ratio between the self weight of the slab and the total load, assuming a uniform load distribution. The value of the fixity coefficient would be: Fixity coeff. = User's fixity coeff × (1 – (self weight slab/total load)).
    2. In the previous phase, a slab requiring shoring was obtained. In this case the program considers the total load to be in the service phase, as well as the analysis the program undertakes when continuity is considered, using the value of the fixity coefficient at the edges assigned by the user. In an elastic analysis the total load = permanent load + live load, which is equivalent to building the slab using shoring, then removing it, and so the slab is left submitted to this total load.

        The deck may be optionally designed. The bottom reinforcement may also be designed, whether the user has opted for the deck to be designed and no suitable deck has been found, or whether it has not been designed. In either case, if bottom reinforcement is provided, any contribution on behalf of the deck is neglected.

When reinforcement has to be placed within the depth of the concrete, at least one bar per rib will be placed.

The resistance of a composite slab will be sufficient to support the design loads and to ensure that no failure limit state is reached. The failure modes include:

- **Critical section I**. Bending: ultimate design bending moment value at mid-span. This section may be critical if there is complete connection at the interface between the deck and concrete.
- **Critical section II**. Longitudinal shear force: the resistance of the connection is decisive. The design value of the ultimate bending moment in Section 1 cannot be reached. This situation is defined as partial connection.
- **Critical section III**. Vertical shear and punching shear: design value of the ultimate shear force next to the support. This section will be critical only in special cases, for example in slabs with great depths and small spans with relatively large loads.

The design value of the bending moment resistance of any section is determined using the plastic moment theory of a section with complete connection.

To obtain the effective area of the steel deck, any embossments are ignored. This piece of data is to be supplied by the user.

The program calculates the design value of the positive moment resistance of the composite slab depending on whether the neutral axis is located above or within the deck.

# 10 Stairs

The stairs module analyses and designs the reinforcement of stair slabs as separate elements of the job. Depending on the geometry, type, support arrangement and applied gravitational loads, the program established the reactions on the main structure. These are transferred to the structure as line loads and surface loads (for steps built on the floor) as dead and live loads.

A staircase is a group of stair spans defining the vertical circulation of a specific zone of a building. A flight is the inclined part of a stair slab spanning between two horizontal planes and containing a series of steps.

The landing is an intermediate horizontal plane between two consecutive flights.

The program resolves staircases whose spans between floors are composed of flights (parallel or perpendicular to one another) of the following types:

- Straight flight
- Two straight flights with half turn landing
- Three straight flights with quarter turn landings
- Two straight flights with full turn landing
- Two consecutive flights with intermediate landing
- “n” straight flights with half turn landings
- “n” straight flights with quarter turn landings

Other types can be defined whereby each span belonging to the same staircase can be defined using any type of stair containing any of the following elements:

- Straight flight
- Intermediate landing
- Quarter turn landing
- Half turn landing

Straight flights can be defined as having straight landing at their start and end. The user defines the length of each slab which will have the same width as the stair.

Any initial steps that are built on the floor can be defined in the following way:

- Straight
- Intermediate landing with a turn, unsplit
- Intermediate landing with a turn, split.

The width of half turn landings can be different to the width of the staircase.

# 10.1 Common data of the staircase

The geometrical properties and loads of the staircase are defined in the *Staircase data* tab. These properties are common for all the stair spans of the same staircase. If there is a specific case whereby different values have to be applied for the same staircase, e.g. different loads, then two staircases must be defined.

## *10.1.1 Geometrical properties*

- **Width.** This is the width of the stair or length of the steps
- **Tread and riser.** The tread is the mean step width on plan and the riser is the vertical distance between two consecutive treads (step height).
- **Rotation**. The user can choose between left or right, depending on the direction followed by the person when going up the stair.
- **Last step format.** The last step of an ascending flight can be designed in two ways:
    - The intermediate landing or landing form the last step
    - Last step on the inclined span

## *10.1.2 Loads*

- **Steps.** The program allows the user to select whether the steps are poured with the concrete slab, or if these are built on using bricks, which are the two most usual ways of creating the steps of a stair.

  This data affects the load results of the analysis and the concrete take-off of the job.
- **Handrail, floor and live loads.** The weight of the handrails corresponds to the total weight of the handrails. Therefore, the user has to enter this value bearing in mind whether there is one handrail or two.

# 10.2 Staircase flight data

The stair flights are fractions of the staircase that span from one floor to another and can consist of one or several flights. The flight properties can be different for each flight (slab depth, initial steps built on floor, flight arrangement and landings, number of step of each flight, stairwell width, steps build on landings, landing supports, etc.).

The type of flight can be chosen from the range of predefined types included in the program.

The range of predefined types forms part of the job library, hence, one or more types can be used for one or several stair spans and, therefore, in one or several staircases of the job.

In the **Flight > Create** dialogue box, the following data is specified:

- **Reference.** Identifies the type of span.
- **Slab depth.** The program can propose the slab depth or it can be indicated by the user. The program will automatically design the slab depth if the *Slab depth* box is left deactivated; and so assign a value equal to 1/30 of the maximum real span between supports.

  If the **Slab depth** box is activated, the user manually specifies the depth of the stair slab.
- **Start level difference.** An elevation difference can be indicated for the start of each stair span (greater or equal to 10 cm) to represent a possible infill of the floor slab.

  This is usually applied in cases such as when stairs start at foundation level and go past a high raised floor or the screed of a basement, or when stairs start at elevated slabs with respect to the corresponding floor.
- **Initial steps built on floor.** If steps have been built on the start of a stair span, the number of steps has to be indicated. Once the stair has been designed, the program applies a surface dead load reaction corresponding to the mean weight of the steps built on the floor. The program does not generate the live loads of the zone taken up by the steps as the surface lies on a structural elements (for example a floor slab) which already has an applied live load.
- **Flight and landing arrangement.** One of the indicated types is selected.
- **Number of steps.** Depending on the selected stair type, the user has to indicate the number of steps for each flight.
- **Stairwell width.** This is only required if there are half turn landings.
- **Steps built on intermediate landing.** If there are any, the number of steps is to be indicated.
- **Landing supports.** The program allows for landings to be supported or free spanning (overhanging).

In the case of supported landings, front supports or lateral supports or both can be provided.

- **Support type and width**. The program allows for the following types of supports to be provided at landing edges:
  - Hanging bar: These are bars that hold the edge of the landing by means of a structural element (usually a beam) situated on the floor above the landing. The program requires the width to be able to consider the weight of the brick wall situated between the landing and the floor above.
  - Masonry wall (*)
  - Concrete wall (*)

    (*) Its load is applied on the structural element situated under the supported edge of the landing on the floor below. The self weight is taken as the load of a wall of a given width and of height equal to the level difference between the intermediate landing and the floor below. A specific weight of 15.70 kN/m$^3$ is taken for the case of a masonry wall and of 24.53 kN/m$^3$ for the case of a concrete wall. If there were to be a wall spanning between the intermediate landing and the floor above, or any other type of partition or external wall, the user must apply the line load on the floor below corresponding to the load of the aforementioned element.
  - With connectors: these are shear connectors joining the intermediate landings to a structural element such as a beam or wall, to which the end reaction is transmitted.

Once the staircase has been inserted, the program allows for new staircases, identical to the one before, to be entered, whose references are numbered consecutively.

Click on the right mouse button to finish entering the staircases.

If there is a geometrical problem with the staircase introduction, the program will indicate this by means of an error message on screen.

# 10.3 Results, reports and drawings of the Stairs module

## *10.3.1 View staircase reinforcement details*

Click on the button from the *Stairs* floating menu, followed by a click with the left mouse button on the staircase to view its reinforcement. If it is the first time the staircase has been selected or modifications have been carried out since the last analysis, the program will proceed to design and reinforce it. Once the staircase has been designed, a window will be appear displaying the reinforcement of each flight making up the staircase.

To view the reinforcement of other flights of the staircase, click on the drop down menu situated at the top left hand corner of the window.

### 10.3.2 *View forces and displacements using contour maps*

The program designs the stairs individually and by finite element method, generating a thick shelled triangular mesh, taking into account the usual loadcases in the design of staircases: dead and live loads.

The start and end supports are simulated by means of an elastic beam with an assigned stiffness that simulates a floor slab support, in a similar way as to what occurs with intermediate supports with their hanging bars, masonry walls or connectors. The reactions are obtained and are integrated resulting in a line load which is applied to the structure.

To view the forces and displacements of a staircase click on the button from the *Stairs* floating menu, followed by a click with the left mouse button on the staircase whose results you wish to consult. If it is the first time the staircase is selected or if changes have been carried out since the last analysis, the program will design it. Once the staircase has been designed, a window displaying a three dimensional view of the stair flight will open. The displacements and forces of the selected flight can then be consulted.

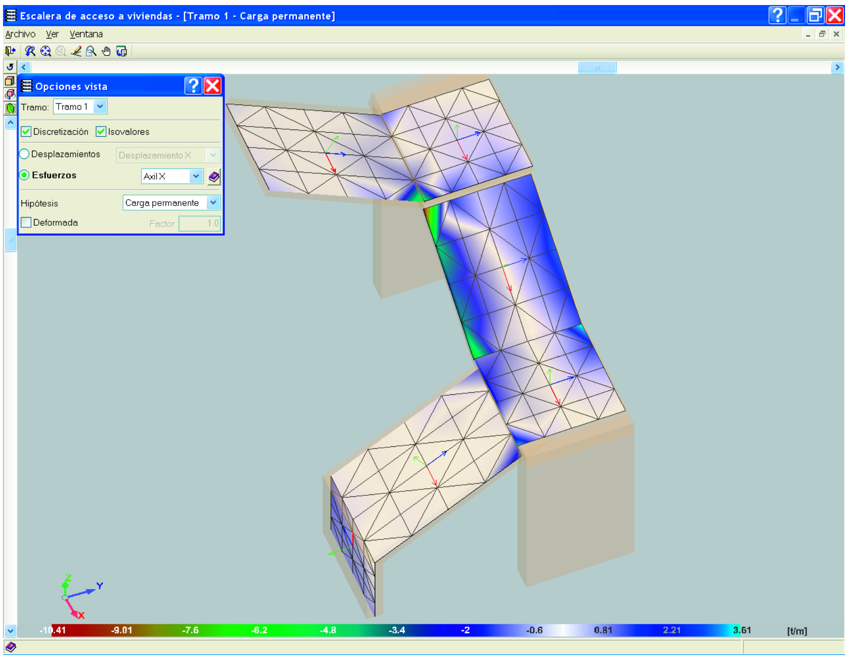



Figure 86

## 10.3.3 Staircase design

Staircases can be designed using any of the following methods: Complete analysis with CYPECAD and Individual design.

When the job is being analysed, all the staircases are also designed, so that their reactions can be applied to the main structure. Therefore, the first thing the program analyses are the stairs.

If the job has not been analysed, each staircase can be designed individually upon selecting the or buttons and clicking on the staircase for the first time or after the staircase has been modified.

If the user modifies the staircase once the job has been analysed and these changes affect the reactions of the staircase on the structure, a new analysis should be launched so that these modifications are taken into account. The program warns of this situation.

## 10.3.4 Reports

Added to the list of job reports, is the option to generate a report of all the staircases entered in the job.

The stair reports contain the general data of all the staircases of the job (materials and code used), common data (geometry, loading, etc.) and specific data (reactions on the main structure, reinforcement, resultant forces in each span section) of each flight.

## 10.3.1 Stair drawings

The drawings display all the required information to define the stair layout: longitudinal and transverse sections, property tables of each span with its geometric data, loads and materials. The reinforcement take-off tables are also included (per staircase, flight and total steel summaries).

# 11 Integrated 3D structures

As of the 2007 version of the program, and for those users who have acquired CYPE 3D with the same license as CYPECAD, a connection between both programs is available. Its presentation is almost identical to that of CYPE 3D, so users can define one or several independent zones as Integrated 3D structures and connected to the general structure, made up of groups and floors, of CYPECAD.

A possible example for this use could be: A shopping centre with flat slabs (CYPECAD) and steel roof (CYPE 3D), with a terrace partially covered by timber beams (CYPE 3D).

There are many other examples for which the Integrated 3D structures can be used. Below are their main properties:

1. Connection between structures (CYPECAD-Integrated 3D structure):
    - It has to be at existing columns or previously created starts (in the *Column Definition* tab).
    - They can also be connected to beams, walls, flat or waffle slabs.
    - The connecting bar can be adjusted to the centre, side or vertices; generally, at any point.
    - Elevation differences can be defined for the connection, so it can connect at the middle zone of any column or start.
2. The analysis method is the same as that described in the *Calculations manual* of CYPE 3D, with the same graphical interface and features.
3. All the loadcases and combinations that are generated are shared, and are defined in the *General data* dialogue box of CYPECAD.
4. Any Integrated 3D structure can be analysed, designed and checked independently. The supporting nodes are considered to have external fixity and coincide with the connections defined in CYPECAD.
5. When the main structure of CYPECAD is analysed, both models are integrated in a combined matrix which is resolved with the current sections of each Integrated 3D structure and applying the loadcases, which are common to both parts, a completely integrated analysis is provided.
6. Import CYPE 3D jobs is an option within the program which imports a CYPE 3D job and transforms it into an Integrated 3D structure. The materials and steel section series must be the same as those in CYPECAD, otherwise, they will be lost when the job is imported.

The dead loads are placed in the dead loadcase, whereas the live loads (such as wind, snow or accidental loads) are created within additional loadcases. The combination groups are those defined in the *General data* of CYPECAD. For the wind loads, if the job that has been imported from CYPE 3D has loadcases its loadcases defined, it is recommended their compatibility with the automatic loadcases generated by CYPECAD be studied.

*We recommend that the calculation manual of CYPE 3D be studied as a complement to fully understand how integrated 3D structures behave, as these are basically, the same as an isolated CYPE 3D job, with the additional advantage of being integrated in CYPECAD.*

# 12 Rigid Diaphragm

## 12.1 Rigid diaphragm in exempt beams

As of the 2012.a version, the possibility has been included of eliminating the consideration of the rigid diaphragm in exempt beams, i.e., those beams that have been entered in the floor distribution using the types of beams available in the *Current beam* dialogue box and do not have any floor slab contacting them (except the external fixity and limit beams types). The program, by default, considers all the beams to be “connected” to the rigid diaphragm and hence possess 3 degrees of freedom and the rigid diaphragm hypothesis is maintained. For example, a continuous beam supported by several columns, even though it does not have a floor slab, conserves the rigid diaphragm hypothesis. As of the 2012.a version, exempt beams can be disconnected from the rigid diaphragm using the option **Rigid diaphragm** in unconnected beams (**Beam Definition tab > Beams/Walls** menu). Beams that are disconnected using this option go on to possess six degrees of freedom at either end with the corresponding forces: axial force, moment in the vertical and transverse plane, and torsional moment; and hence their reinforcement will be designed to resist all these forces. Regarding buckling in exempt horizontal steel or concrete beams, the free length of the beam, regardless of whether it is in the vertical plane or horizontal plane, is taken as the buckling length.

## 12.2 Rigid diaphragm in reinforced concrete walls, masonry walls and reinforced concrete block walls

In previous versions to the 2012.a version, any wall in contact with a floor slab, beam or other wall was considered to comply with the rigid diaphragm hypothesis at that floor level. As of the 2012.a version, the rigid diaphragm consideration at floor level is only maintained if the wall is in contact with a floor slab. Therefore, if the wall is exempt at floor level or is only in contact with exempt beams or other walls, all the nodes of the bar that are generated at the intersection of that floor and the nodes of the triangular finite elements of the wall have 6 degrees of freedom.

If the wall is in contact with a floor slab along any part of its length on a floor, all the wall will remain connected to the rigid diaphragm of that floor, including those spans where there are no floor slabs. Nonetheless, the rigid diaphragm hypothesis can be partially eliminated using the option **Divide beam** (**Beam Definition tab > Beams/Walls** menu), so that by applying this division at the transition points between the part with the applied rigid diaphragm hypothesis and the exempt part, will leave these last parts free or exempt with 6 degrees of freedom.

If the option to reinforce wall crown beams is activated (**Job > General data > By position button > Beam reinforcement within walls and crown beams**), the crown beam spans that are located on wall spans without floor slabs (if the user has applied the **Divide beam** option to disconnect that span), and crown beams of walls without floor slabs along all their length, are designed for the six acting forces.

Similarly, beams coinciding with intermediate floor and crowns of masonry and concrete block walls are also designed in the same way if the option is activated for these types with the advantage that it also designs at intermediate floor level.

# 13 Code implementation

## 13.1 Load codes

| | Wind loading | Seismic loading | Snow loading |
|---|---|---|---|
| Algeria | RNV 99 | RPA 99 / v 2003 | |
| Argentina | CIRSOC 102 - 2005<br>CIRSOC 102 - 1984 | CIRSOC 103 - 1991<br>CIRSOC 103 - 2005<br>CIRSOC 103 - 2008 | |
| Bolivia | | NDBS 2006 | |
| Brazil | NBR 6123 | NBR 15421:2006 | |
| Bulgaria | Ordinance Nº3, 21.07.2004. Section VI: Wind loads | Decree Nº2, 23.07.2007 | Ordinance Nº3, 21.07.2004. Section V: Snow loads |
| Canada | NBC 05 | NBC 05 | NBC 05 |
| Chile | NCh - 432.Of71 | NCh - 433.Of96 | |
| China-Macau | RSAEEP | | |
| Colombia | NSR - 10 | NSR - 10<br>NSR - 98 | |
| Costa Rica | RC80 | CSCR 2010<br>CSCR 2002 | |
| Cuba | NC 285:2003 | NC 46:1999 | |
| Dominican Rep | Bulletin No. 9/80 | M-001 1979<br>R-001 2011 | |
| Ecuador | NEC - 11 | CPE INEN 5:2001<br>NEC - 11 | |
| Europe | Eurocode 1<br>Eurocode 1 (Belgium)<br>Eurocode 1 (France)<br>Eurocode 1 (Portugal)<br>Eurocode 1 (United Kingdom)<br>Eurocode 1 (Singapore) | Eurocode 8<br>Eurocode 8 (Belgium)<br>Eurocode 8 (France)<br>Eurocode 8 (Portugal) | Eurocode 1<br>Eurocode 1 (Belgium)<br>Eurocode 1 (France)<br>Eurocode 1 (Portugal) |
| France | NV 65:2009 | PS 92<br>PS 92 ( 2010 revised version) | NV 65:2009 |
| Germany | DIN 1055-4:2005 | DIN 4149:2005 - 04 | DIN 1055-4 |
| Guatemala | NSE-2 | NSE-10 | |
| Honduras | CHOC-04 | CHOC-04 | |
| India | IS:875 (Part 3) - 1987 (Reaffirmed 1997) | IS 1893 (Part 1): 2002<br>IS 13920: 1993 | IS:875 (Part 4) - 1987 (Reaffirmed 1997) |
| Italy | N. T. C. | N. T. C. | N. T. C. |
| Mexico | NTC<br>CFE 2008 | CFE 2008<br>NTC - 2004<br>NTC - 95<br>CFE93 | NTC |
| Morocco | Moroccan wind code | RPS 2011<br>RPS 2000 | |
| Nicaragua | RNC-07 | RNC-07 | |
| Panama | REP - 04 | REP - 04 | |
| Paraguay | NBR | | |
| Peru | E.020 Technical Code | E.030 Technical Code | |
| Portugal | RSA | RSA - Modal spectral | RSA |
| Puerto Rico | | 2011 PRBC | |
| Romania | | P100 - 1 / 2006 | |
| South Africa | SANS 10160-3: 2011 | SANS 10160-4: 2011 | |
| Spain | CTE DB-SE AE<br>NTE | NCSE - 02<br>NCSE - 94 | CTE DB-SE AE<br>NTE |
| United Kingdom | BS 6399 - 2: 1997 | | |
| Uruguay | UNIT 50 - 84 | | |
| USA | ASCE / SEI 7 - 05<br>ASCE / SEI 7 - 10 | ASCE 7 - 05<br>UBC<br>2009 IBC | ASCE 7 - 05 |
| Venezuela | COVENIN 2003 - 89 | COVENIN 1756-1: 2001 | |

# 13.2 Material codes

| | Concrete | Steel | Timber | Aluminium | Reinf. Conc. Block Walls | Composite Elements | Fire resistance |
|---|---|---|---|---|---|---|---|
| Argentina | CIRSOC 201-2005<br>CIRSOC 201-1982 | | | | | | |
| Bolivia | CBH 87 | | | | | | |
| Brazil | NBR 6118:2003<br>NBR 6118:2007<br>NB -1 | AISI<br>NBR 14762: 2010<br>NBR 14762: 2001<br>NBR 8800: 2008<br>NBR 8800: 1986 | NBR 7190 | | | | NBR 7190 |
| Canada | | CAN / CSA S16 - 01<br>CAN / CSA S136 - 07 | | | | | |
| Chile | ACI 318-99<br>NCh430 | NCh427 | | | | | |
| Europe | Eurocode 2<br>Eurocode 2 (Bulgaria)<br>Eurocode 2 (Portugal)<br>Eurocode 2 (Romania) | Eurocode 3<br>Eurocode 4<br>Eurocode 3 (Bulgaria)<br>Eurocode 4 (Bulgaria)<br>Eurocode 3 (France)<br>Eurocode 4 (France)<br>Eurocode 3 (Italy)<br>Eurocode 4 (Italy)<br>Eurocode 3 (Portugal)<br>Eurocode 4 (Portugal) | Eurocode 5<br>Eurocode 5 (Belgium)<br>Eurocode 5 (France) | Eurocode 9 | Eurocode 6 | Eurocode 4 | Eurocode 2 (Concrete)<br>Eurocode 3 (Steel)<br>Eurocode 5 (Timber) |
| France | BAEL - 91(R-99) | | | | | | |
| Germany | | DIN 18800:2008 - 11 | | | | | |
| India | IS 456: 2000 (ACI 318M-08) | IS 800: 2007 | | | | | |
| Italy | NTC 14-01-2008 | NTC 14-01-2008 | | | | | |
| Mexico | NTCRC<br>NTCRC: 2004 | NTCRC Metal Struct.<br>AISI/NASPEC-2007 (LRFD) | | | NTCRC | | |
| Peru | NTE E.060: 2009 | | | | | | |
| Portugal | REBAP | MV110<br>REAE | | | | | RSCI |

## 13.3 Combinations codes

| | Concrete | Steel | | Timber |
|---|---|---|---|---|
| | | Rolled and welded | Cold-formed | |
| Algeria | BAEL - 91 (R99)<br>RPA 99 | | | |
| Argentina | CIRSOC 201 - 2005<br>CIRSOC 201 -1982 (103)<br>CIRSOC 201 - 1982 (105) | | | |
| Belgium | | | | Eurocode 0 |
| Brazil | NBR 6118:2003<br>NB - 1<br>NBR - 8681/84 | NBR 8800:2008<br>NBR 8800:1986 | NBR 14762:2001<br>AISI | NBR 7190 |
| Bulgaria | Eurocode 0 | Eurocode 0 | Eurocode 0 | |
| Canada | | CAN/CSA S16 - 01 | CAN/CSA S136 - 07 | |
| Chile | ACI 318 - 99 | ASD | ASD | |
| Europe | Eurocode 0 | Eurocode 0 | Eurocode 0 | Eurocode 0 |
| France | BAEL - 91 (R99) | Eurocode 0 | Eurocode 0 | Eurocode 0 |
| Germany | | DIN 1055 - 100 | DIN 1055 - 100 | |
| India | IS 875 (Part 5): 1987 | | | |
| Italy | | Eurocode 0 | Eurocode 0 | |
| Mexico | Regulation DF | NTCRC Metallic Struct. | AISI/NASPEC2007 (LFRD) | |
| Morocco | BAEL - 91 (R99)<br>RPS 2000 | | | |
| Peru | NT E.060 | | | |
| Portugal | Eurocode 0<br>RSA | Eurocode 0<br>RSA | Eurocode 0<br>RSA | |
| Romania | Eurocode 0 | | | |
| Spain | CTE DB - SE<br>EHE - 98<br>EH - 91 | CTE DB - SE<br>EA - 95 (MV103) | CTE DB - SE<br>EA - 95 (MV110) | CTE DB - SE |
| USA | ACI 318M - 08<br>ACI 318M - 99 | ASCE/SEI 7 - 05<br>ASD<br>AISC LRFD 86 | ASCE/SEI 7 - 05<br>ASD | |

# 14 Interaction of the structure with the construction elements

The analysis of the effects earthquakes have on buildings confirms the need to encourage the development of tools to analyse structures exposed to seismic action in a more realistic way.

**Seismic Risk** comprises two fundamental concepts: seismic danger and the dynamic behaviour of structures exposed to accidental external loads. Current knowledge of these two concepts is still limited due to the uncertainty that exists with the information and methods to study them.

For example, in the case of seismic danger, there is uncertainty regarding the prediction, location and quantification of the strength of an earthquake, whilst in the case of the dynamic behaviour of the structures, the uncertainty lies in knowing the behaviour of the structure itself, the quality of the construction materials, the construction processes that have been undertaken, etc.

The work undertaken for this module has been aimed at studying the dynamic behaviour of structures and how they interact with the non-structural elements the buildings are composed of, as well as establishing the worst-case design forces obtained from combining seismic action with other static loads acting simultaneously.

Nowadays, the structural analysis of buildings can be carried out using different analysis methods with different degrees of complexity. A simple procedure consists in using a simplified static method based on obtaining equivalent static forces, which avoids having to carry out a dynamic behaviour analysis of the building. Further to the application restrictions, any possible inaccuracies of the method are counteracted with oversized elements which reduce the competiveness of the resulting designs

A more general analysis approach, contemplated by current design standards, is to carry out an analysis using maximum response spectrums. Based on a three-dimensional model of the structure, which considers the most representative degrees of freedom, and the response spectrums defined by each design standard, adapted to specific variables (seismic zone, type of soil, etc.), the maximum requirements are obtained for each vibration mode of the structure.

The modal spectral analysis is a dynamic analysis process which consists in establishing the maximum response of the structure based on its vibration modes (deformed shape-eigenvectors) and their respective natural vibration frequencies (eigenvalues). The final response of the structure is given by an adequate combination of these modal contributions. This analysis method is applied based on the following general conditions:

- The vibration modes of the system must be obtained using methods established in structural dynamics.
- All the modes of the structure which may contribute in a significant manner to its dynamic response must be considered.
- The maximum response of each mode is obtained using the defined design spectrum ordinates in accordance with the corresponding design code, for the vibration period of the mode.
- The maximum modal responses for each variable that intervenes in the analysis (displacements, distortions, forces, etc.) are combined to obtain the total maximum resultant response to the seismic action.

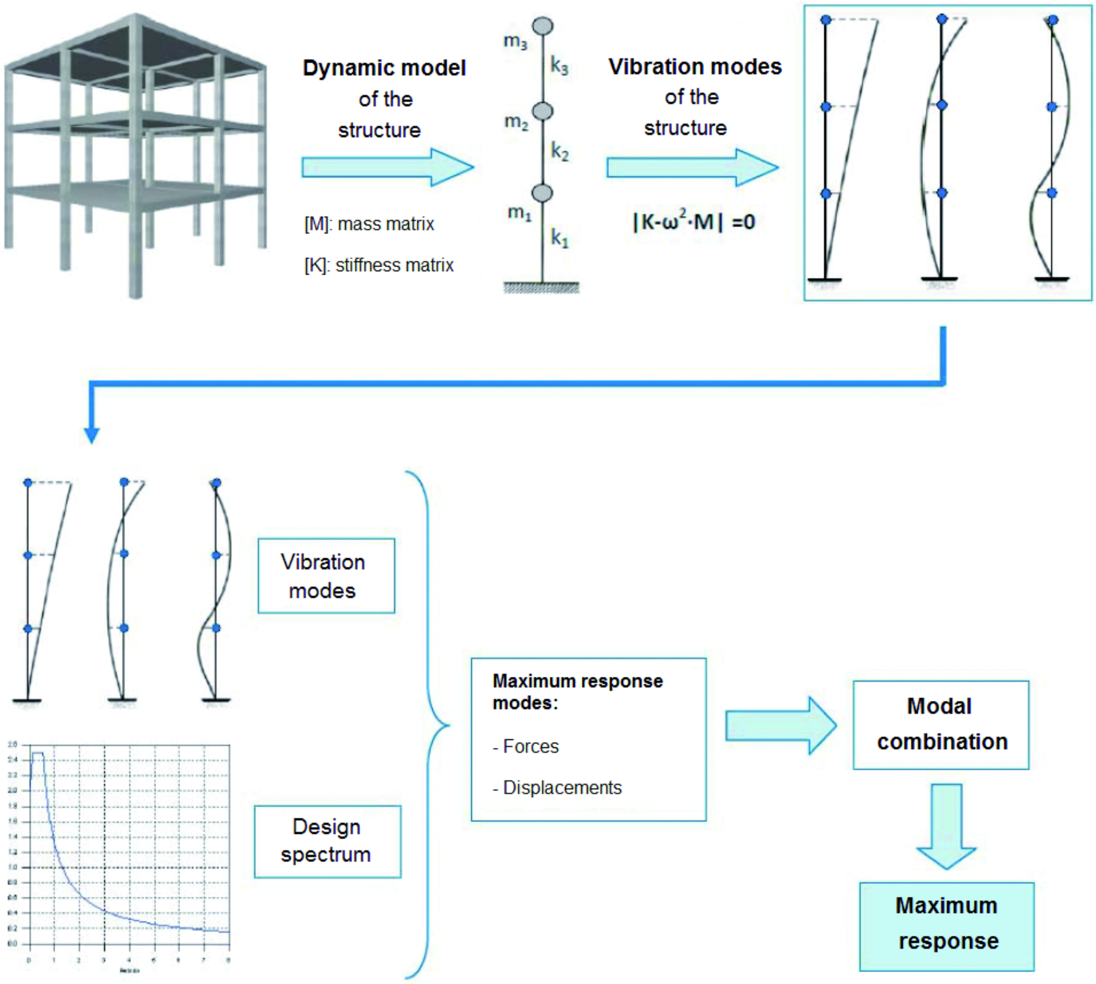



Figure 87. Summary of the dynamic modal spectra analysis process

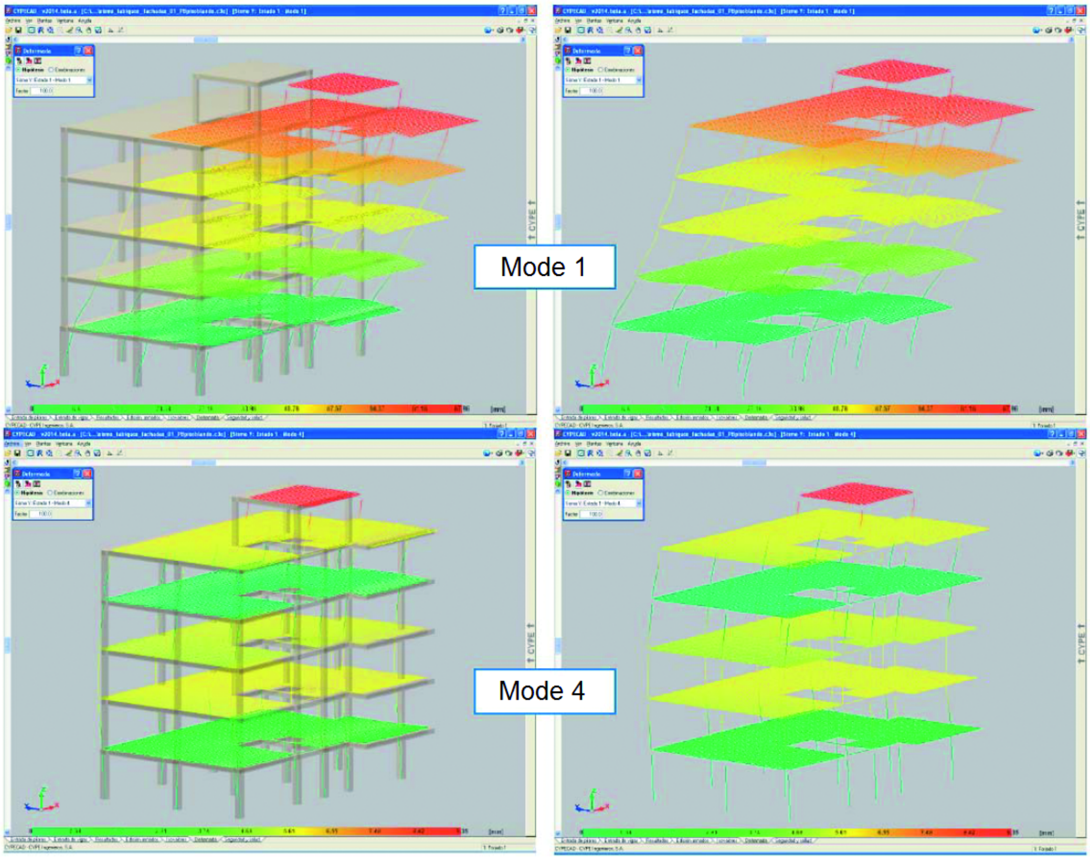



Figure 88. Deformed shapes of a structure associated to the vibration modes calculated by the program

The ground floor of many residential buildings in urban zones is destined to be used as retail precinct, whereas the upper floors contain dwellings. The floor slab is usually designed as having a flat bottom surfaces, consisting of a joist floor slab with flat beams, waffle or flat slab, although it is also possible they may have dropped beams with respect to the bottom plane of the floor slab.

When there are no additional earthquake-resistant elements, such as shear walls, the horizontal forces due to seismic action are resisted by the stiffness of the connections between the columns and floor slab (rigid node).

The façades and partitions of the building are considered to be "non-structural" elements; however, they do provide stiffness to the structure and modify the distribution and magnitude of the forces that arise due to seismic action. For example, when the stiffness associated with the partitions is not uniform on all floors, the horizontal forces have greater impact on the columns belonging to the floors with less stiffness, producing shear forces of a high magnitude in the columns. If these have not been designed accordingly, the forces can cause a fragile fracture, endangering the stability of the building, even leading to its collapse.

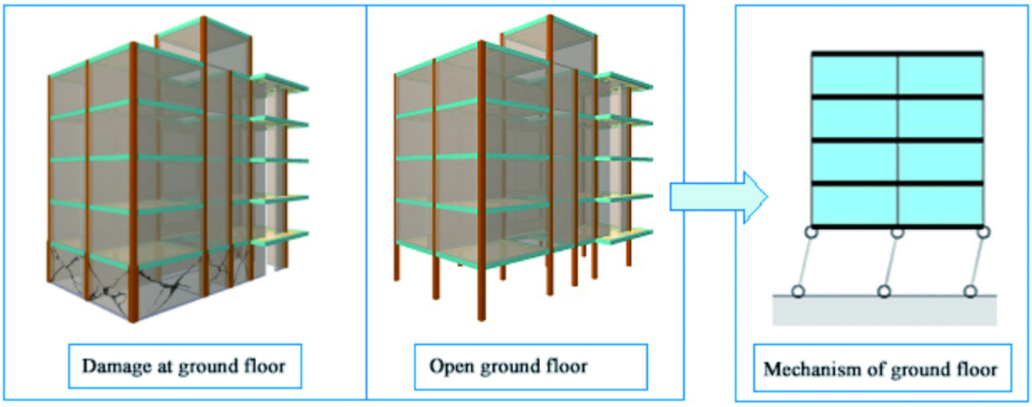



Figure 89. The non-uniform stiffness distribution between floors causes there to be greater forces acting on the columns of the less rigid floors, giving rise to fragile fractures if the elements have not been designed accordingly

Many design standards on today's market establish the need to have to consider how the non-structural elements affect the overall stiffness of the building.

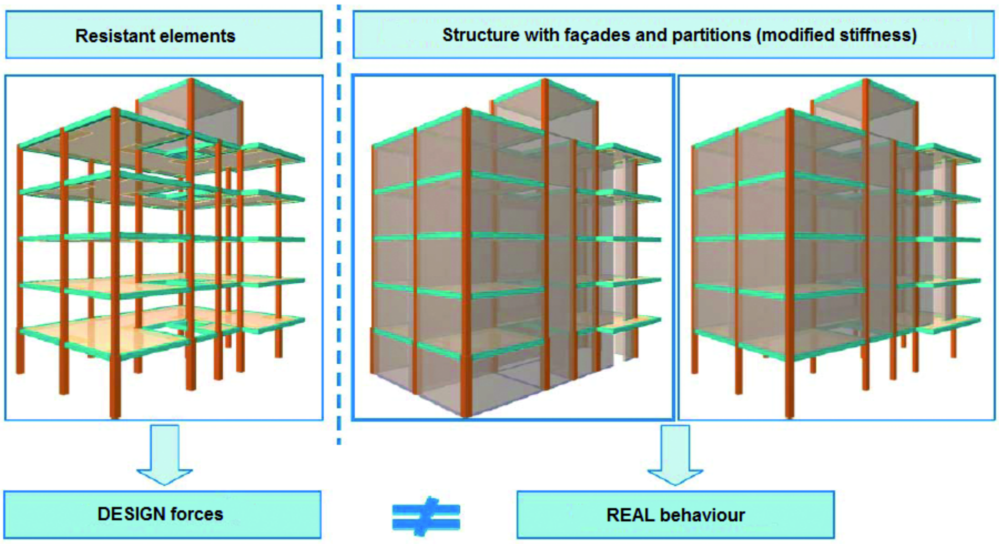



Figure 90. The actual design is based on models which only contemplate the structurally resistant elements. However, a more thorough analysis should take into account the stiffness and resistance provided by the façades and partitions

Hence, several linear analysis models should be defined and solved to cover the different situations that could occur in reality. Initially, it is not possible to know which is the worst-case situation as, in many cases, a change in the location of the partitions causes abrupt changes in the stiffness and, consequently, unfavourable effects on the structural elements.

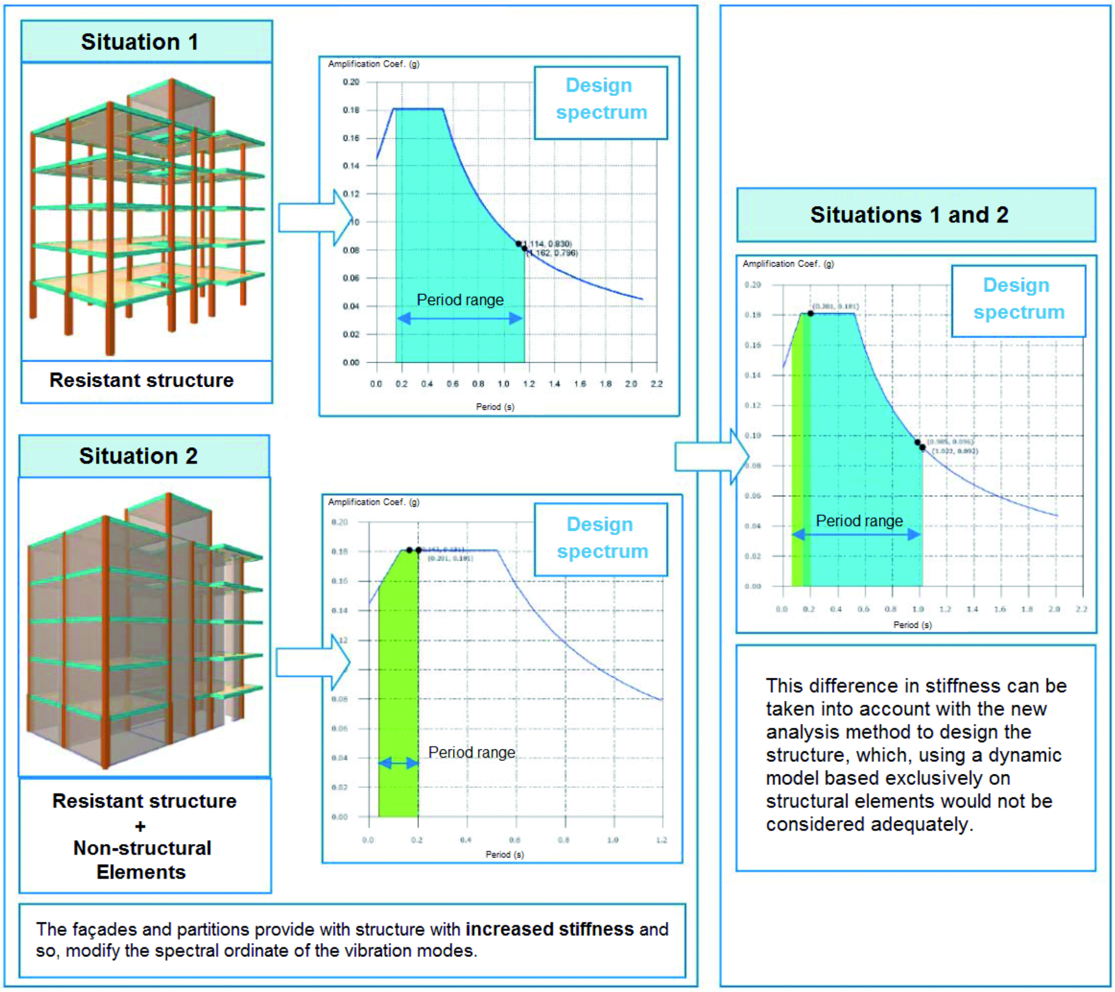



Figure 91. This new tool allows for the behaviour of the structure to be verified for different situations so to guarantee the correct response is obtained for all the possible behaviour loadcases during an earthquake

This way, the analysis method that has been developed is capable of estimating, in a sufficiently precise way, the behaviour of the building during an earthquake, considering the structural elements and taking into account how they may be influenced by the remaining construction elements. An example is analysed further on in this chapter, which displays how important it is to verify the possible states or situations of the structure due to its interaction with the non-structural elements.

## 14.1 Model used to analyse the effect of the non-structural elements

The analysis model, which includes the influence of the non-structural elements on the building in the presence of an earthquake, and how the façades and partitions behave bearing in mind they do not have a structural role, has been developed by the investigation team of *CIMNE (Centro Internacional de Métodos Numéricos en Ingeniería) of the UPC (Universidad Politécnica de Cataluña)*, together with the *company PRISMA, S.L.* and the technical team of *CYPE SOFT, S.L.*

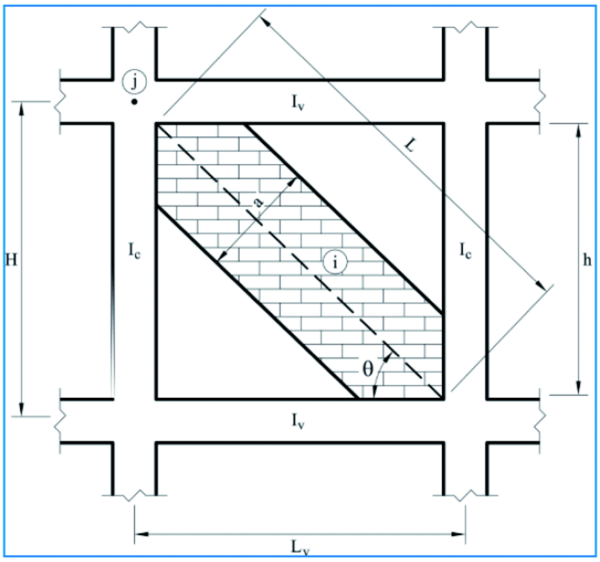



Figure 92. Representation of the model that has been considered to include the effect of the non-structural elements in the analysis: "equivalent masonry bar contained in a frame".

The program generates the equivalent masonry bars to consider the effect of each construction element. It is important to note that this stiffness is only developed if the construction element is laterally confined between columns, shear walls or walls.

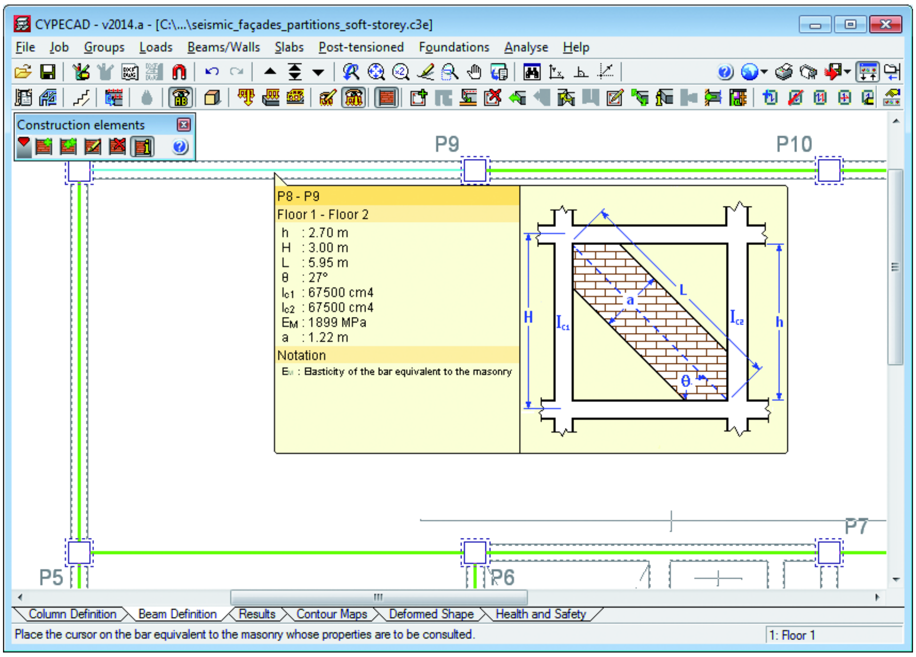



Figure 93

## 14.2 Cracked or fractured states

The analysis method that has been developed allows for the modal spectral analysis to be carried out of successive dynamic models that are generated automatically, which include the stiffness of the non-structural elements (façades and partitions) and their situation or state.

Users have the possibility to only consider two extreme analysis states: that corresponding to the structure without taking into account the effect of any construction elements, and that which considers all the construction elements that are laterally confined, assuming they are not cracked or fractured.

It is also possible to consider the automatic generation of states based on a fracture criterion which relates the damage suffered by an element with the relative displacement of its ends. These are intermediate states, in which each laterally confined element provides a percentage of its stiffness depending on the level of damage it has reached.

The intermediate states are generated automatically based on the model in which all the construction elements are considered to be effective. A modal spectral analysis of the model produces a relative displacement between the ends of each construction element which, upon applying the fracture criterion, is interpreted as being a specific level of damage. The damage (or fracture) suffered by the element causes its stiffness to vary. The new stiffness obtained for each construction element is included in the new dynamic model which is used for the next modal analysis. New relative displacements are obtained and each element reaches a new damage level, with their change in stiffness, and so the next model is generated. This process is repeated successively for each seismic loadcase that is considered.

This iterative process stops when the damage level stabilises between two states (which occurs when the difference between the level of damage of the last analysed state and the previous state is less than 5% for each element) or when the maximum number of iterations established by users is reached. This will be the last state of Final state.

If all the intermediate states that are generated are considered in the design of the structure, the time required to carry out the analysis can be substantially increased. Hence, the process can be configured so that only the final state is considered in the design, without taking into account the intermediate states. If the option to obtain intermediate states is considered, users can consult the Damage level option: **Result tab > Construction elements > Damage level**, for each construction element, which provides the relative displacements and associated damage level for each state that is generated for each seismic loadcase.

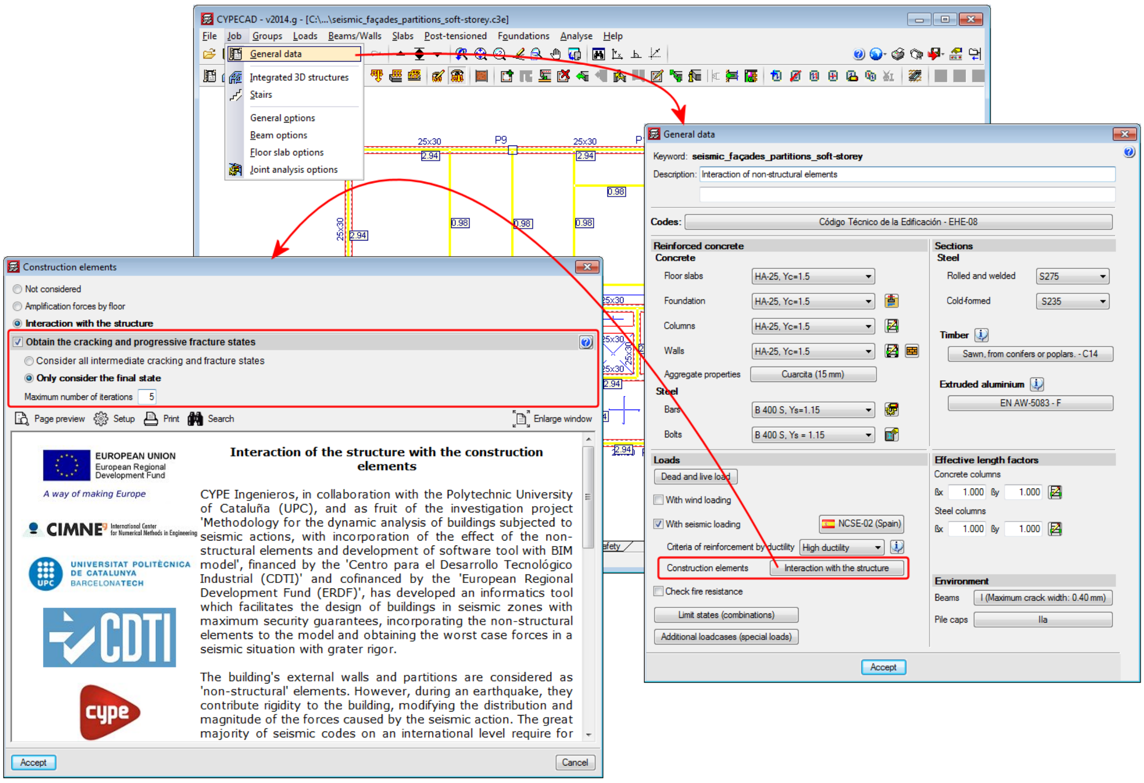



Figure 94

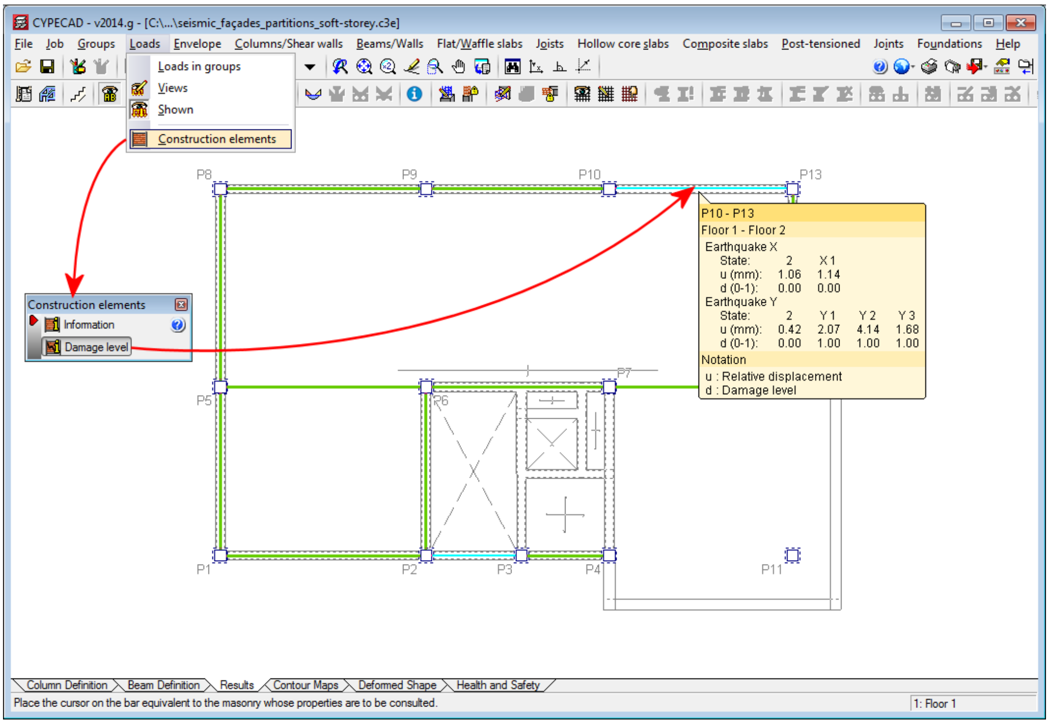



Figure 95

## 14.3 Analysis example

Using this analysis example, the states or situations of the structure due to its interaction with the non-structural elements will be verified. For this case, an open floor (Soft Storey) building will be used.

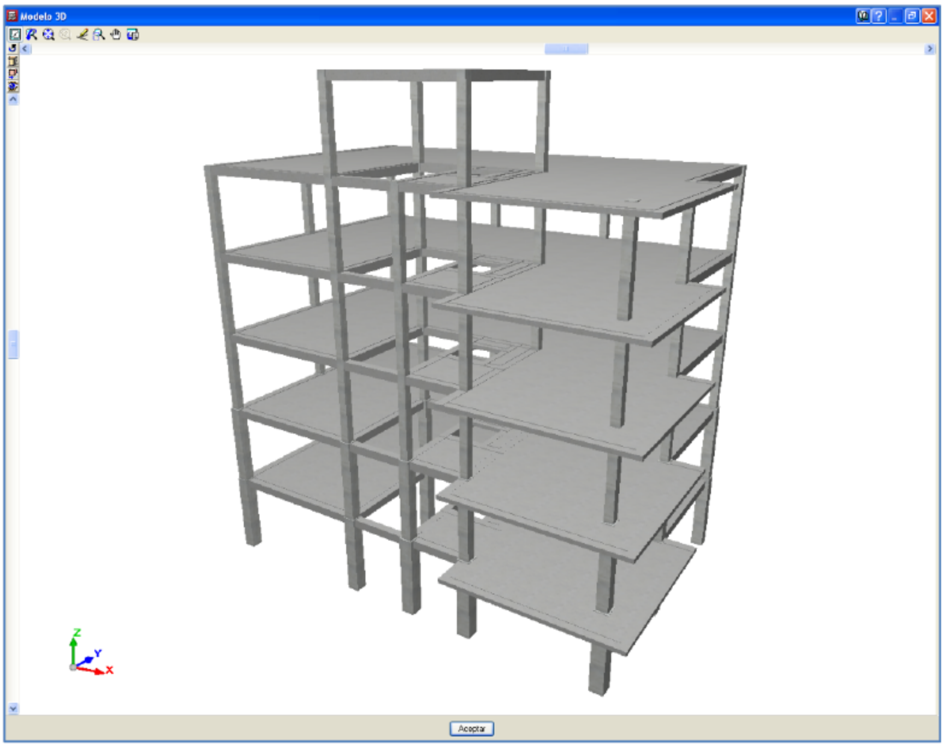



Figure 96

In this example, it is interesting to compare the behaviour of the structure with and without the influence of the construction elements being entered.

### 14.3.1 *Description of the structure*

The structure consists of a reinforced concrete building with 6 floors (5 + lift machinery room), made up of frames with spans ranging between 4.5 and 5.6 m and with 15 cm thick flat slabs. The frames are composed of columns which start at foundation level and measure 45×45 cm, and are reduced to 30×30 cm at the higher floors; the beams are 30×30 cm dropped beams. The height of each floor is 3 m.

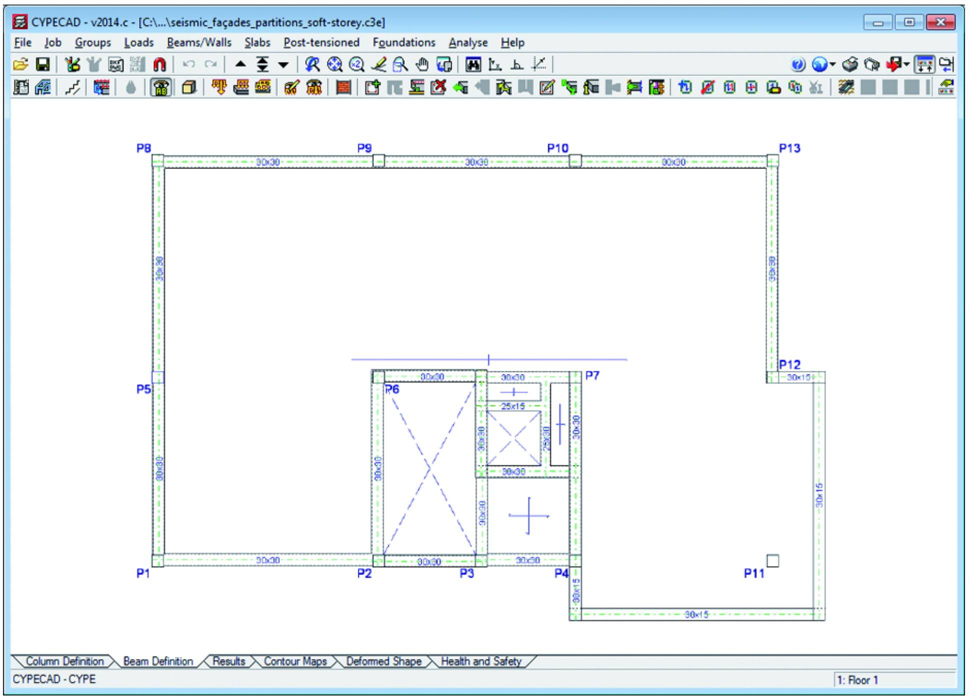



Figure 97

In addition to the structurally resistant system (columns-beams-slabs), partitions and façades are also entered (non-structural elements). These consist of 25 cm and 10 cm thick masonry walls. However, until now, only one case or the other could be analysed.

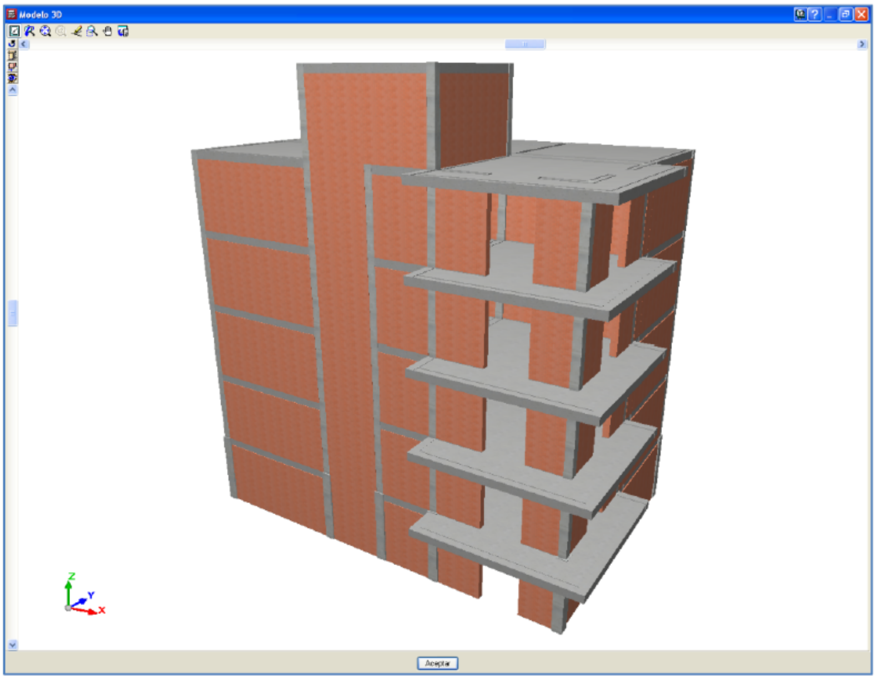


Figure 98

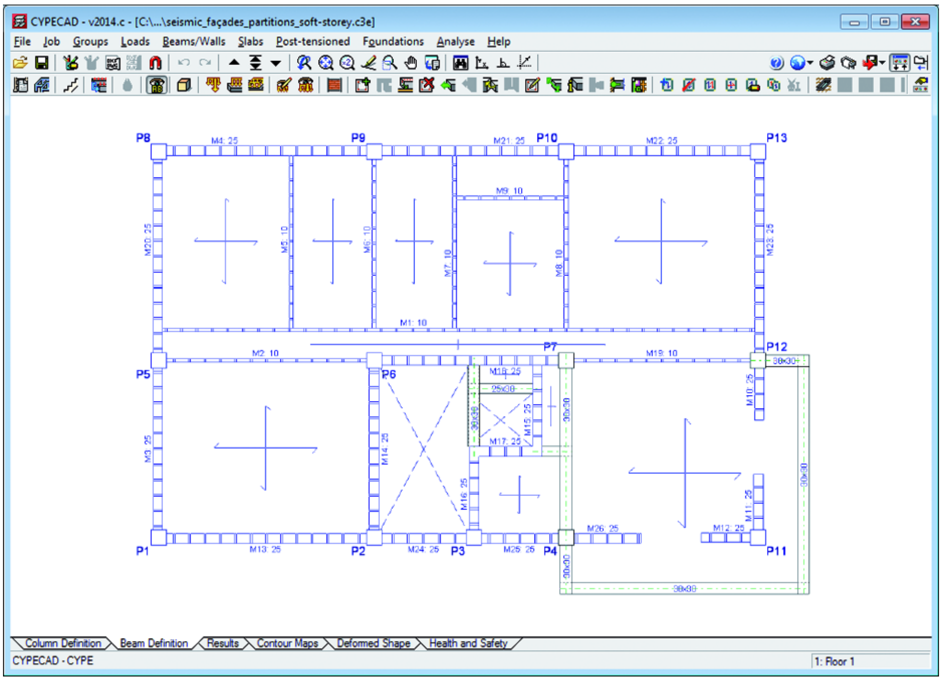



Figure 99

## 14.3.2 *Construction elements*

Using this analysis method, the mechanical and elastic properties are assigned to each construction element, and so not only are the loads associated to these elements generated and considered in the analysis method, but also their stiffness, using a data entry interface similar to that of line loads.

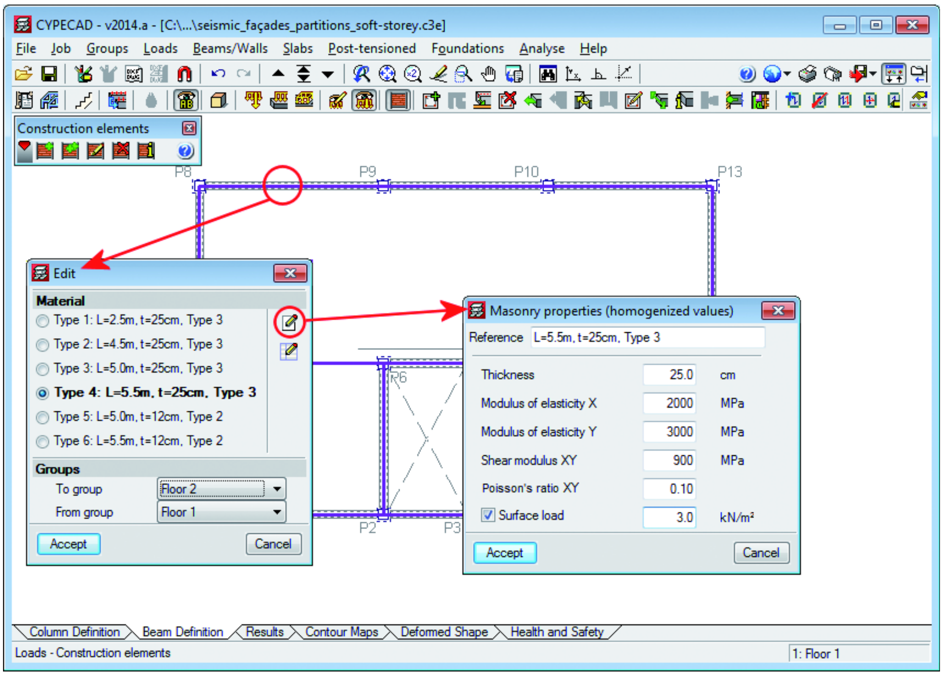



Figure 100. Properties of the types of masonry considered. Assistant to help users with the most frequently used values

The stiffness of the elements in the analysis is taken applying the method proposed by CIMNE. The construction element will only develop its stiffness if it is confined otherwise the program will generate the corresponding line load but will not be assigned a stiffness.

The program generates an “equivalent masonry bar” for each confined element. Depending on its properties, it will be assigned an area, a length and a homogenised modulus of elasticity which define the stiffness of the element to be included in the dynamic analysis model of the structure.

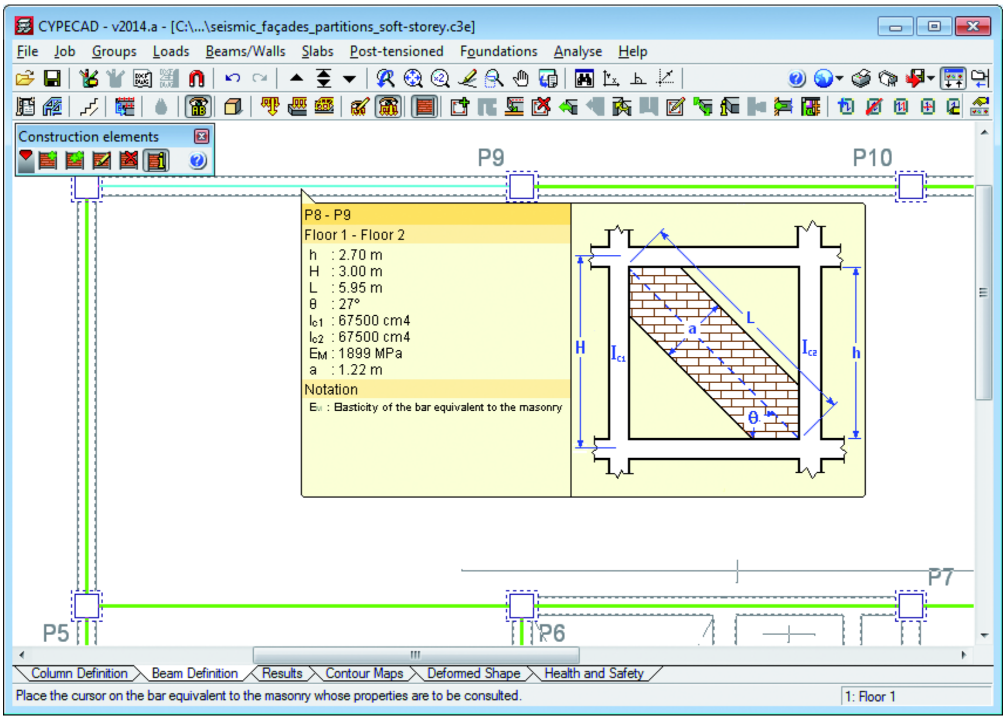


Figure 101. Properties of the equivalent masonry bars that are generated

## 14.3.3 Seismic action

A modal spectral analysis shall be considered for the seismic action, applying the *NCSE-02 Spanish code*. The location, type of soil, properties of the structure and other parameters shall be selected; the program then generates the corresponding spectrum with which the analysis is carried out.

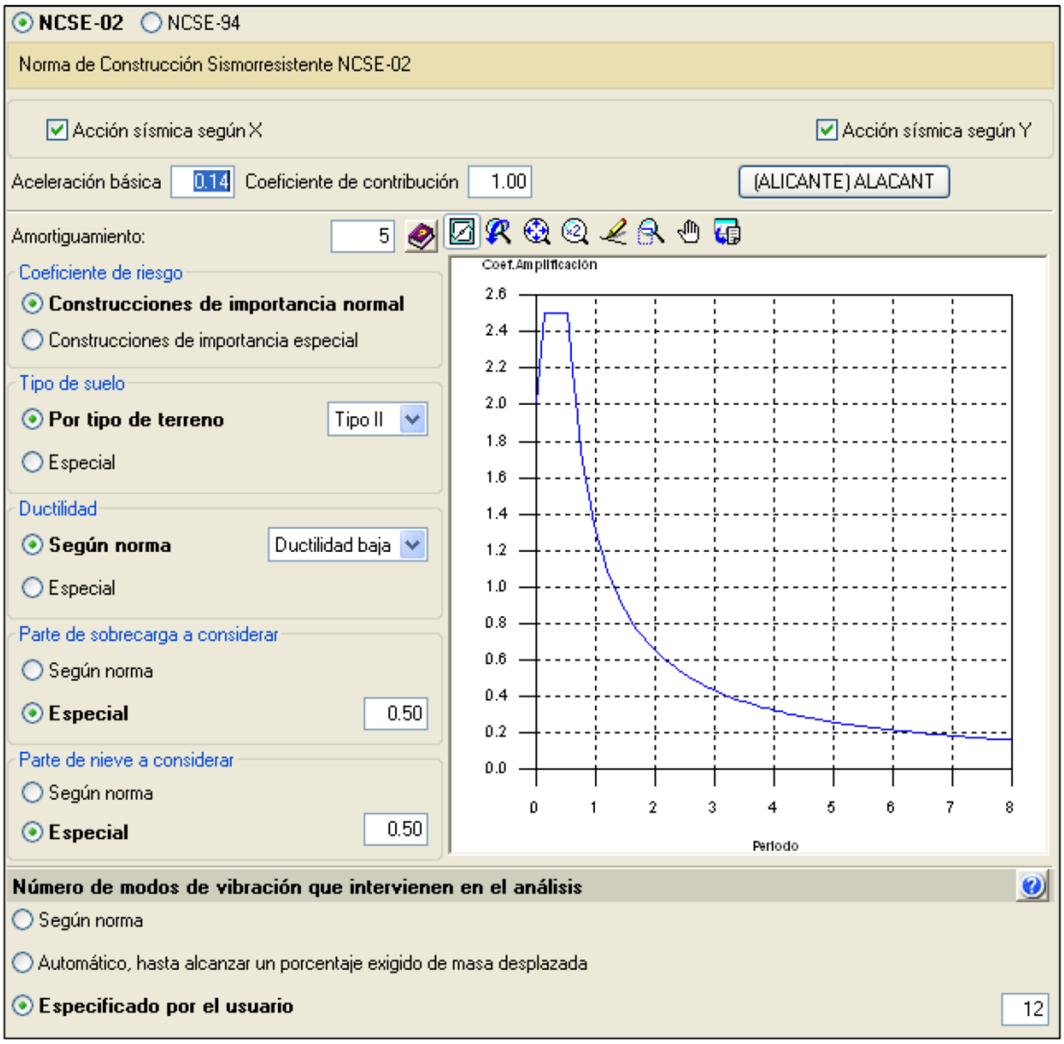



Figure 102. Seismic action definition dialogue box for the NCSE-02 Spanish code

The aim of this example is to demonstrate how important it is to consider how the stiffness differs along the height of the building, as it causes greater forces to arise in the resistant elements of the floors with less stiffness.

As has been indicated previously, it is those buildings whose ground floor is for retail use that, generally, have a stiffness irregularity, making them weaker at that floor. The difference in stiffness is due to their height usually being greater than that of the floor above and, that due to their use, is a much more open floor. Even if the stiffness of the ground floor were to be similar to that of the floors above it, during the first instants of an earthquake, the partitions of the lower zones of the building are the first to fail, which causes abrupt changes in the stiffness and, therefore, an irregularity similar to that described previously. Therefore, the stiffness provided by the various non-structural elements can change during an earthquake, due to the cracks and fissures that appear successively.

A uniform distribution of partitions and façades has been entered along the building's height, except for the ground floor. The program automatically analyses two models or states: the model in which only the structural elements are considered and the model that includes the structural elements and their interaction with the non-structural elements, contemplating the stiffness of the latter elements in the stiffness analysis.

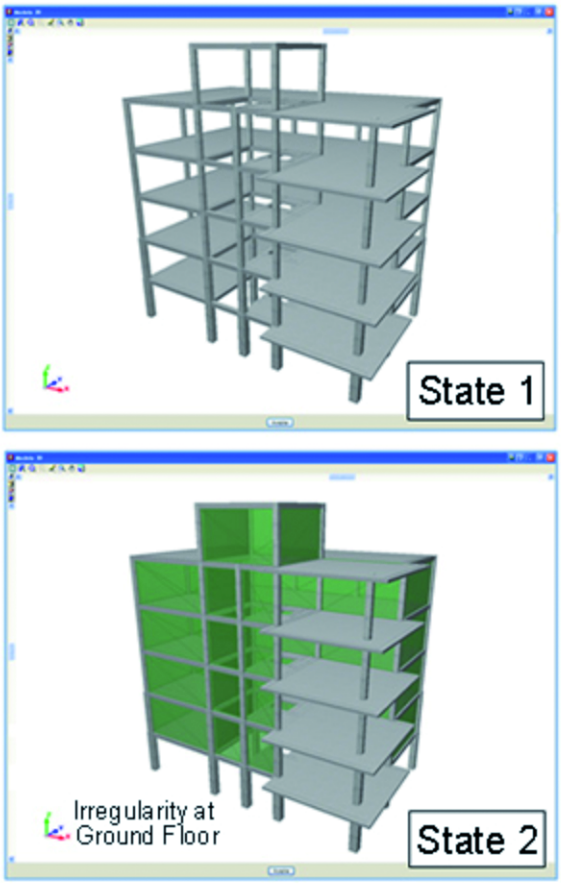



Figure 103. Analysis models or states that are generated automatically

The dynamic modal spectral analysis provides two mode groups corresponding to the states that have been considered. For each state, the modal responses (forces, displacements, distortions, etc.) are combined using the CQC method to obtain the response for each seismic loadcase (Earthquake X and Earthquake Y) and for each state, in such a way that the following dynamic loadcases are considered:

- Earthquake X (State 1)
- Earthquake X (State 2)
- Earthquake Y (State 1)
- Earthquake Y (State 2)

Both states are considered in the combinations of the seismic action with the other static actions, designing each structural element for the worst case situation it is exposed to.

Displayed below are some sections of the "Justification of seismic action" report provided by the program. The two calculated mode sets are displayed for the two states of the structure that have been considered with their participation coefficients, percentage of displaced mass in each direction and associated spectral acceleration. The information contained in the tables on the design spectrums used in the analysis is then displayed on a graph, and the period intervals that have been studied for each state are represented.

If the results are observed, it can be seen that the periods of State 2, in which the effect of the non-structural elements is considered, are smaller than those obtained for State 1, i.e. State 2 takes into account a model with greater stiffness than that in State 1.

Depending on the stiffness of the models, the intervals and, therefore, their associated accelerations will vary. Additionally, the vibration modes are different for each state, affecting the resistant elements in a different manner. This explains why a state can be the worst case state for a specific resistant element, but not for another. Hence, the new implemented tool will analyse the elements considering both states.

**1.3.1.- State 1**

| Mode | T | $L_x$ | $L_y$ | $L_{gz}$ | $M_x$ | $M_y$ | Loadcase X(1) | Loadcase Y(1) |
|---|---|---|---|---|---|---|---|---|
| Mode 1 | 1.022 | 0.0363 | 0.9659 | 0.2565 | 0.11 % | 77.64 % | R = 2<br>A = 0.905 m/s²<br>D = 23.9274 mm | R = 2<br>A = 0.905 m/s²<br>D = 23.9274 mm |
| Mode 2 | 0.985 | 0.9406 | 0.0386 | 0.3373 | 77.61 % | 0.13 % | R = 2<br>A = 0.938 m/s²<br>D = 23.0747 mm | R = 2<br>A = 0.938 m/s²<br>D = 23.0747 mm |
| Mode 3 | 0.836 | 0.0106 | 0.0158 | 0.9999 | 0.3 % | 0.17 % | R = 2<br>A = 1.106 m/s²<br>D = 19.591 mm | R = 2<br>A = 1.106 m/s²<br>D = 19.591 mm |
| Mode 4 | 0.362 | 0.0791 | 0.9506 | 0.3003 | 0.06 % | 9.33 % | R = 2<br>A = 1.776 m/s²<br>D = 5.90505 mm | R = 2<br>A = 1.776 m/s²<br>D = 5.90505 mm |
| Mode 5 | 0.352 | 0.972 | 0.0827 | 0.2201 | 9.17 % | 0.07 % | R = 2<br>A = 1.776 m/s²<br>D = 5.57428 mm | R = 2<br>A = 1.776 m/s²<br>D = 5.57428 mm |
| Mode 6 | 0.301 | 0.0013 | 0.0056 | 1 | 0 % | 0.01 % | R = 2<br>A = 1.776 m/s²<br>D = 4.08155 mm | R = 2<br>A = 1.776 m/s²<br>D = 4.08155 mm |
| Mode 7 | 0.235 | 0.0026 | 0.998 | 0.0638 | 0 % | 3.14 % | R = 2<br>A = 1.776 m/s²<br>D = 2.48555 mm | R = 2<br>A = 1.776 m/s²<br>D = 2.48555 mm |
| Mode 8 | 0.230 | 0.5256 | 0.0097 | 0.8507 | 3.04 % | 0 % | R = 2<br>A = 1.776 m/s²<br>D = 2.38692 mm | R = 2<br>A = 1.776 m/s²<br>D = 2.38692 mm |
| Mode 9 | 0.195 | 0.146 | 0.0887 | 0.9853 | 1.36 % | 0.5 % | R = 2<br>A = 1.776 m/s²<br>D = 1.71211 mm | R = 2<br>A = 1.776 m/s²<br>D = 1.71211 mm |
| Mode 10 | 0.191 | 0.3277 | 0.5988 | 0.7308 | 0.6 % | 2 % | R = 2<br>A = 1.776 m/s²<br>D = 1.64613 mm | R = 2<br>A = 1.776 m/s²<br>D = 1.64613 mm |
| Mode 11 | 0.176 | 0.0804 | 0.0283 | 0.9964 | 0.68 % | 0.08 % | R = 2<br>A = 1.776 m/s²<br>D = 1.39608 mm | R = 2<br>A = 1.776 m/s²<br>D = 1.39608 mm |
| Mode 12 | 0.145 | 0.1079 | 0.0846 | 0.9905 | 0.08 % | 0.05 % | R = 2<br>A = 1.776 m/s²<br>D = 0.94615 mm | R = 2<br>A = 1.776 m/s²<br>D = 0.94615 mm |
| Total | | | | | 93.01 % | 93.12 % | | |

Figure 104. Participation coefficients for State 1

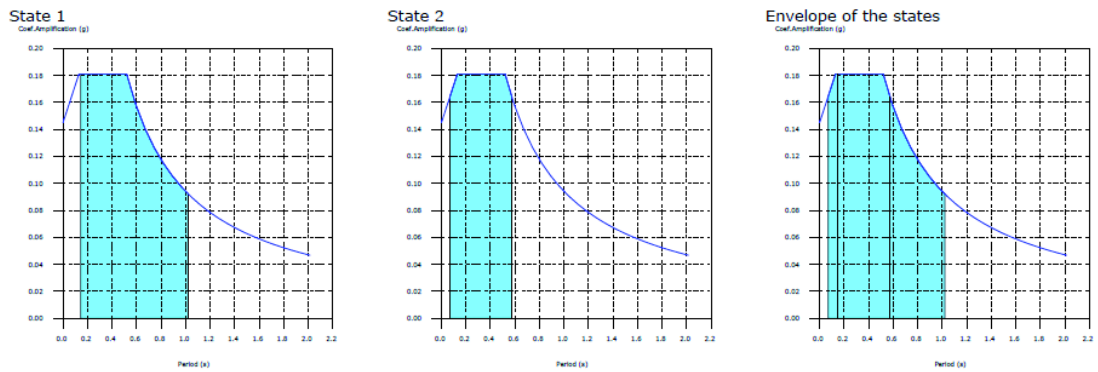
#### 1.3.2.- State 2

| Mode | T | $L_x$ | $L_y$ | $L_{gz}$ | $M_x$ | $M_y$ | Loadcase X(1) | Loadcase Y(1) |
|---|---|---|---|---|---|---|---|---|
| Mode 1 | 0.575 | 0.0307 | 0.5917 | 0.8056 | 0.23 % | 85.19 % | R = 2<br>A = 1.613 m/s²<br>D = 13.4947 mm | R = 2<br>A = 1.613 m/s²<br>D = 13.4947 mm |
| Mode 2 | 0.480 | 0.9183 | 0.0654 | 0.3905 | 93.7 % | 0.48 % | R = 2<br>A = 1.776 m/s²<br>D = 10.3573 mm | R = 2<br>A = 1.776 m/s²<br>D = 10.3573 mm |
| Mode 3 | 0.410 | 0.0165 | 0.0406 | 0.999 | 0.84 % | 5.12 % | R = 2<br>A = 1.776 m/s²<br>D = 7.57823 mm | R = 2<br>A = 1.776 m/s²<br>D = 7.57823 mm |
| Mode 4 | 0.193 | 0.0294 | 0.4256 | 0.9044 | 0.03 % | 6.56 % | R = 2<br>A = 1.776 m/s²<br>D = 1.67286 mm | R = 2<br>A = 1.776 m/s²<br>D = 1.67286 mm |
| Mode 5 | 0.152 | 0.5783 | 0.0712 | 0.8127 | 4.19 % | 0.06 % | R = 2<br>A = 1.776 m/s²<br>D = 1.03962 mm | R = 2<br>A = 1.776 m/s²<br>D = 1.03962 mm |
| Mode 6 | 0.127 | 0.0274 | 0.0402 | 0.9988 | 0.09 % | 0.19 % | R = 2<br>A = 1.767 m/s²<br>D = 0.71837 mm | R = 2<br>A = 1.767 m/s²<br>D = 0.71837 mm |
| Mode 7 | 0.119 | 0.0363 | 0.3913 | 0.9195 | 0.01 % | 1.59 % | R = 2<br>A = 1.746 m/s²<br>D = 0.62384 mm | R = 2<br>A = 1.746 m/s²<br>D = 0.62384 mm |
| Mode 8 | 0.092 | 0.4628 | 0.1024 | 0.8805 | 0.61 % | 0.03 % | R = 2<br>A = 1.672 m/s²<br>D = 0.35626 mm | R = 2<br>A = 1.672 m/s²<br>D = 0.35626 mm |
| Mode 9 | 0.089 | 0.0482 | 0.3199 | 0.9462 | 0.01 % | 0.54 % | R = 2<br>A = 1.664 m/s²<br>D = 0.33273 mm | R = 2<br>A = 1.664 m/s²<br>D = 0.33273 mm |
| Mode 10 | 0.076 | 0.0684 | 0.1278 | 0.9894 | 0.04 % | 0.13 % | R = 2<br>A = 1.628 m/s²<br>D = 0.23689 mm | R = 2<br>A = 1.628 m/s²<br>D = 0.23689 mm |
| Mode 11 | 0.075 | 0.0366 | 0.105 | 0.9938 | 0.01 % | 0.08 % | R = 2<br>A = 1.625 m/s²<br>D = 0.22877 mm | R = 2<br>A = 1.625 m/s²<br>D = 0.22877 mm |
| Mode 12 | 0.071 | 0.1961 | 0.0381 | 0.9799 | 0.12 % | 0 % | R = 2<br>A = 1.615 m/s²<br>D = 0.20654 mm | R = 2<br>A = 1.615 m/s²<br>D = 0.20654 mm |
| Total | | | | | 99.88 % | 99.97 % | | |

Figure 105. Participation coefficients for State 2

#### 1.3.3.- Summary of the modal periods per state

Representation of the ranges of the modal periods for each analysis phase.




$T_{min}$ = 0.145 s $T_{max}$ = 1.022 s

$T_{min}$ = 0.071 s $T_{max}$ = 0.575 s

$T_{min}$ = 0.071 s $T_{max}$ = 1.022 s

Figure 106. Graphical representation of the period intervals for each state. Superposition of the states

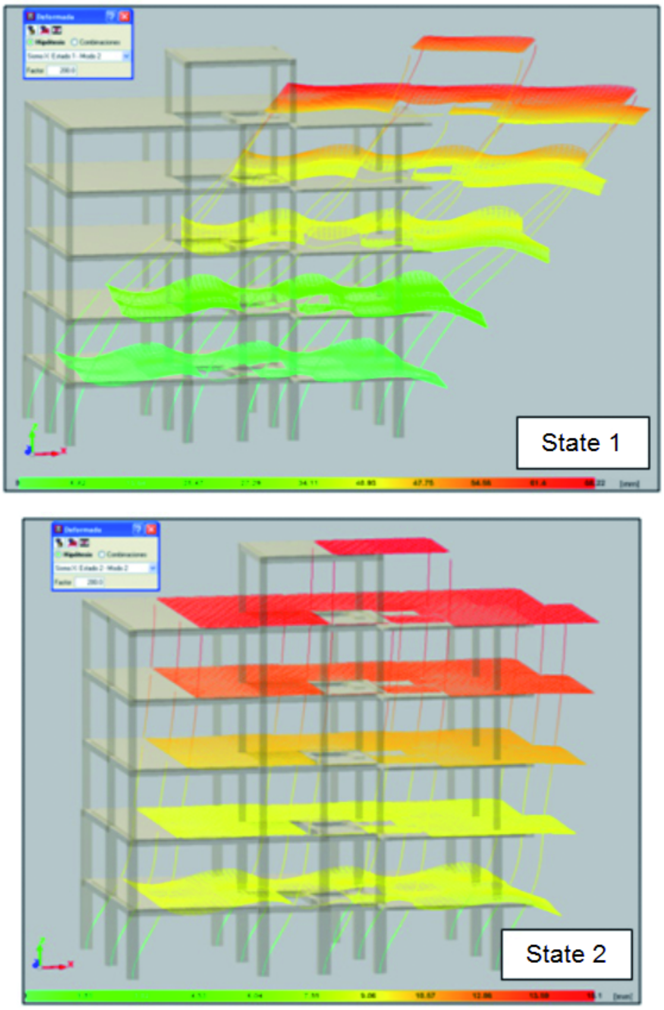



Figure 107. Deformed shape associated to the mode that displaces the most mass in the X direction for each state

The forces for each mode and state, for each seismic loadcase are displayed in the “Forces and reinforcement of columns, shear walls and walls” report.

## Column, shear wall and wall forces and reinforcement

Interaction of non-structural elements

### 1.- SUM OF THE FORCES OF THE COLUMNS, SHEAR WALLS AND WALLS BY LOADCASE AND FLOOR

- Only the forces of the columns, walls and shear walls are taken into account, hence if the job contains beams with external fixity, sloped beams, ties or integrated 3D structures. the forces of these elements are not displayed in the following report.
- This report is useful to know which forces act above the base elevation of the supports on a floor, and so in the case of columns starting on a beam, which are in tension, the forces of these columns will be influenced not only by loads acting from above, but also from those they receive from floors below.

#### 1.1.- Summarised

| Values referred to the origin (X=0.00, Y=0.00) | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| Floor | Elevation (m) | Loadcase | N (kN) | Mx (kN·m) | My (kN·m) | Qx (kN) | Qy (kN) | T (kN·m) |
| Roof | 15.00 | Self weight | 211.6 | 2063.1 | 698.3 | 0.0 | -0.0 | -0.0 |
| | | Dead load | 25.2 | 245.3 | 83.0 | 0.0 | -0.0 | -0.0 |
| | | Live load | 25.2 | 245.3 | 83.0 | -0.0 | -0.0 | -0.0 |
| | | Earthquake X State 1 - Mode 1 | 0.0 | 0.1 | -3.4 | 0.0 | -1.1 | -11.3 |
| | | Earthquake X State 1 - Mode 2 | 0.0 | 92.4 | 4.0 | 30.8 | 1.3 | -90.4 |
| | | Earthquake X State 1 - Mode 3 | 0.0 | 3.3 | -0.6 | 1.1 | -0.2 | -3.7 |
| | | Earthquake X State 1 - Mode 4 | 0.0 | 0.6 | -8.5 | 0.2 | -2.8 | -28.2 |
| | | Earthquake X State 1 - Mode 5 | -0.0 | -99.9 | -8.8 | -33.3 | -2.9 | 83.1 |
| | | Earthquake X State 1 - Mode 6 | -0.0 | -0.5 | 0.1 | -0.2 | 0.0 | 0.7 |
| | | Earthquake X State 1 - Mode 7 | 0.0 | 0.0 | 0.3 | 0.0 | 0.1 | 0.9 |
| | | Earthquake X State 1 - Mode 8 | 0.0 | 101.4 | 0.4 | 33.8 | 0.1 | -113.0 |
| | | Earthquake X State 1 - Mode 9 | 0.0 | 30.8 | -23.0 | 10.3 | -7.7 | -103.8 |
| | | Earthquake X State 1 - Mode 10 | 0.0 | 13.9 | 27.3 | 4.6 | 9.1 | 73.9 |
| | | Earthquake X State 1 - Mode 11 | -0.0 | -17.3 | 4.9 | -5.8 | 1.6 | 46.0 |
| | | Earthquake X State 1 - Mode 12 | -0.0 | -0.1 | -0.5 | -0.0 | -0.2 | -16.9 |
| | | Earthquake X State 2 - Mode 1 | -0.0 | 0.0 | 0.1 | 0.0 | 0.0 | 0.4 |
| | | Earthquake X State 2 - Mode 2 | -0.0 | 10.2 | -0.4 | 3.4 | -0.1 | -13.3 |
| | | Earthquake X State 2 - Mode 3 | 0.0 | 0.3 | -0.3 | 0.1 | -0.1 | -1.1 |
| | | Earthquake X State 2 - Mode 4 | 0.0 | -0.0 | -0.1 | -0.0 | -0.0 | -0.4 |
| | | Earthquake X State 2 - Mode 5 | -0.0 | -3.2 | 0.3 | -1.1 | 0.1 | 4.6 |
| | | Earthquake X State 2 - Mode 6 | -0.0 | -0.1 | 0.2 | -0.0 | 0.1 | 0.6 |
| | | Earthquake X State 2 - Mode 7 | -0.0 | 0.0 | 0.1 | 0.0 | 0.0 | 0.4 |
| | | Earthquake X State 2 - Mode 8 | -0.0 | -2.1 | 0.2 | -0.7 | 0.1 | 3.0 |
| | | Earthquake X State 2 - Mode 9 | -0.0 | -0.0 | -0.2 | -0.0 | -0.1 | -0.5 |
| | | Earthquake X State 2 - Mode 10 | 0.0 | -0.2 | 0.5 | -0.1 | 0.2 | 1.7 |
| | | Earthquake X State 2 - Mode 11 | 0.0 | 0.1 | 0.0 | 0.0 | 0.0 | 0.0 |
| | | Earthquake X State 2 - Mode 12 | -0.0 | -1.6 | -0.0 | -0.5 | -0.0 | 1.9 |
| | | Earthquake Y State 1 - Mode 1 | 0.0 | 1.8 | -91.7 | 0.6 | -30.6 | -301.0 |
| | | Earthquake Y State 1 - Mode 2 | 0.0 | 3.8 | 0.2 | 1.3 | 0.1 | -3.7 |
| | | Earthquake Y State 1 - Mode 3 | 0.0 | 4.9 | -0.9 | 1.6 | -0.3 | -5.6 |
| | | Earthquake Y State 1 - Mode 4 | 0.0 | 7.1 | -101.9 | 2.4 | -34.0 | -339.0 |
| | | Earthquake Y State 1 - Mode 5 | -0.0 | -8.5 | -0.7 | -2.8 | -0.2 | 7.1 |
| | | Earthquake Y State 1 - Mode 6 | -0.0 | -2.0 | 0.5 | -0.7 | 0.2 | 2.9 |
| | | Earthquake Y State 1 - Mode 7 | 0.0 | 2.4 | 105.8 | 0.8 | 35.3 | 340.3 |
| | | Earthquake Y State 1 - Mode 8 | 0.0 | 1.9 | 0.0 | 0.6 | 0.0 | -2.1 |

Loadcase Earthquake X

Loadcase Earthquake Y

State 1

State 2

Figure 108. Forces and reinforcement of columns, shear walls and walls

The element check reports display that the static load and dynamic load combinations of both the states have been taken into account in the analysis. This way, different seismic behaviour loadcases are considered; the worst case being taken to design the element. For example, below is the check of "Failure due to shear (seismic combinations)" for a column span between floor 4 and roof, and a span between foundation and floor 1. It can be seen that the worst case situation for the first example is for State 1, and for the second example is for State 2.

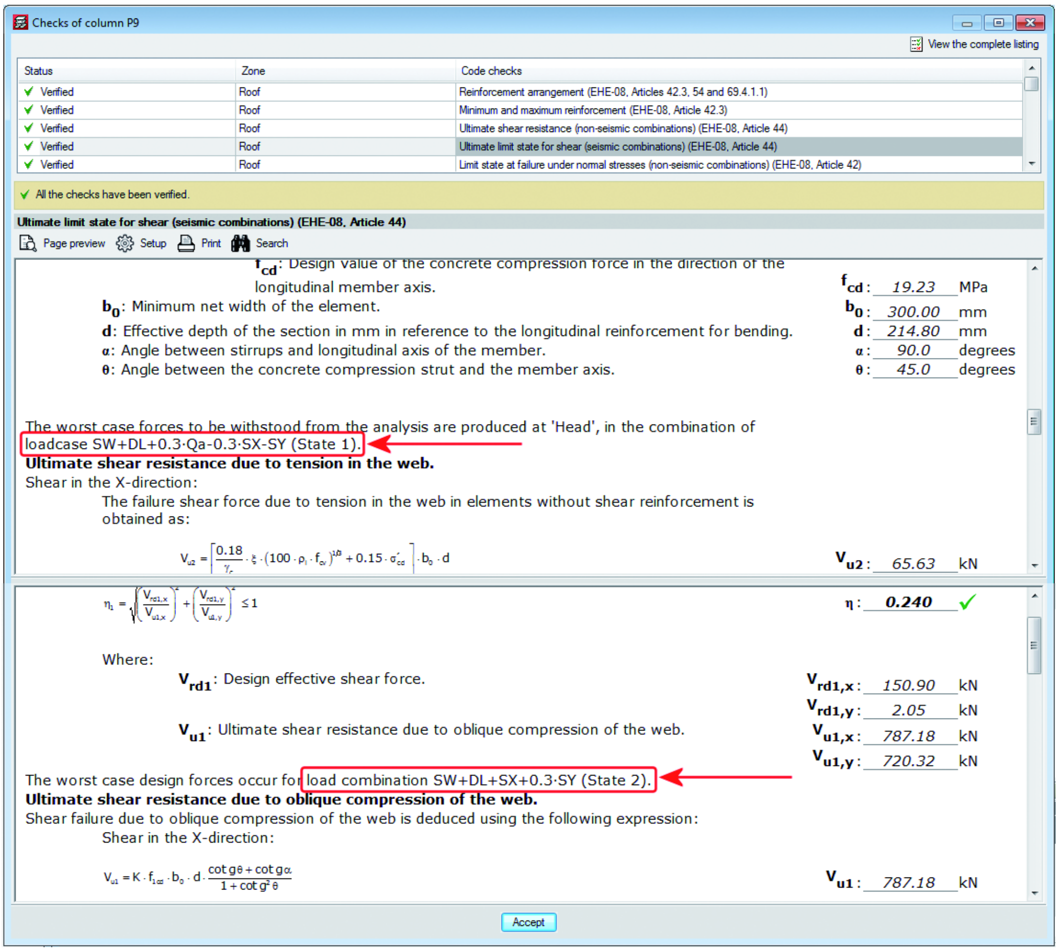


Figure 109. Check on "Failure due to shear (seismic combinations) for column P9

To conclude, a comparison is carried out of the total shear per floor, in each analysis direction, produced by the seismic loadcase for each stiffness state that is considered. By considering the effect or interaction of the non-structural elements, the irregularity of the structure can be automatically taken into account in the analysis. When there is a non-uniform distribution between floors of the stiffness associated with the partitions, the horizontal forces have a greater impact on the columns belonging to the floors with less stiffness, producing shear forces of a high magnitude in the columns. If these have not been designed accordingly, the forces can cause a fragile fracture, endangering the stability of the building, even leading to its collapse.

The new tool allows users to place the partitions and façades on the various floors, taking into account their stiffness If they have not been distributed in the same way on each floor, a stiffness irregularity will automatically be generated, which is considered directly in the analysis. In this example, it can be seen with the ground floor, which is an open floor and therefore has less stiffness. The following graph displays the total shear per floor produced by the seismic loadcase for each stiffness state that has been considered. It can be seen that, for the case in which the stiffness irregularity is considered, the forces at the ground floor are much greater (approximately twice as large) than those obtained if this irregularity is not considered. Hence the analysis results generated before applying this new method (only State 1 was provided) provided smaller forces that those which could arise.

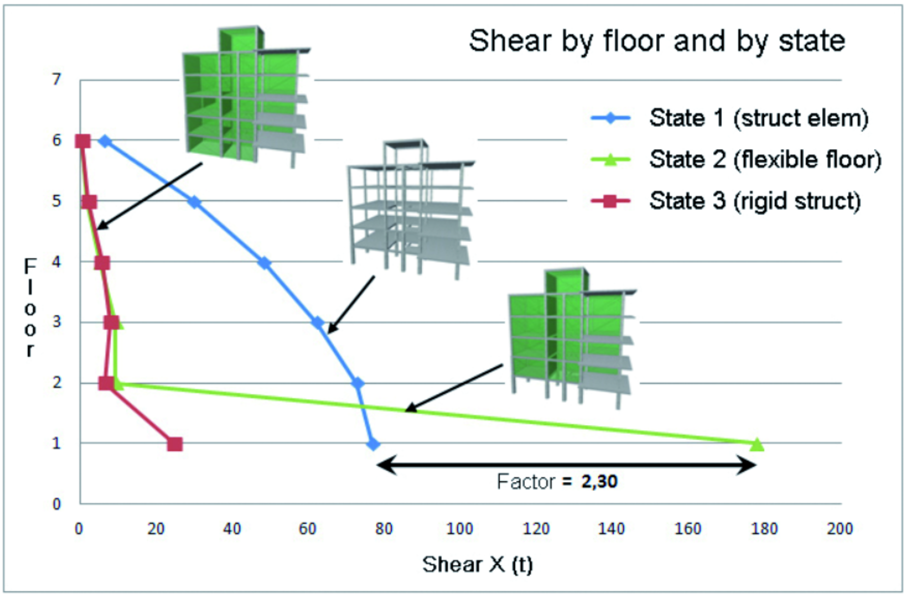



Figure 110. Total X shear by floor and state. (Modal combination = CQC; Combination of directions = SRSS)

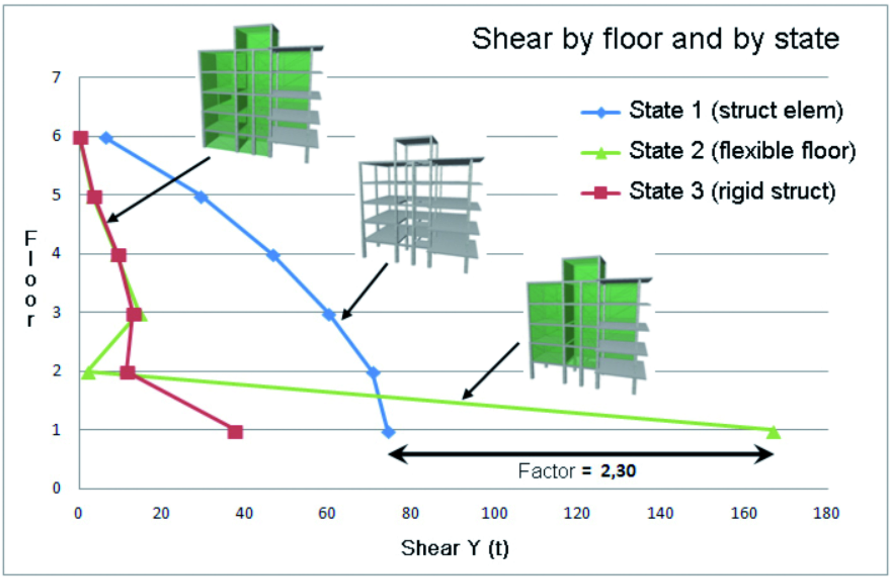



Figure 111. Total Y shear by floor and state. (Modal combination = CQC; Combination of directions = SRSS)

This unfavourable effect was not taken into account in the analysis until now and many design codes offer the possibility to simulate it by amplifying the forces of a floor that is considered as a soft-storey, by a specific factor. The shears and moments resulting from the analysis in which only the structural elements have been considered, are amplified by a factor that varies depending on the seismic design code that has been considered. The improvement provided by this new tool is that the analysis itself tells us directly which floor is soft, without the designer having to establish it before the analysis. Additionally, the force amplification factor is no longer left to the designer's judgement, nor is it applied in a deterministic manner based on a design code; the analysis itself provides us with the value of the amplification factor. As can be seen in the example, the factor obtained from the analysis is 2.3 for Earthquake X and 2.25 for Earthquake Y. These values are within the range of values provided by design codes for which shear forces are to be amplified due to "soft-storey" effects, which range between 1.5 and 2.5. Therefore, the analysis itself, upon including the stiffness irregularity, provides greater forces in the least rigid floor, without the need of having to consider the effect indirectly by amplifying specific forces later on.

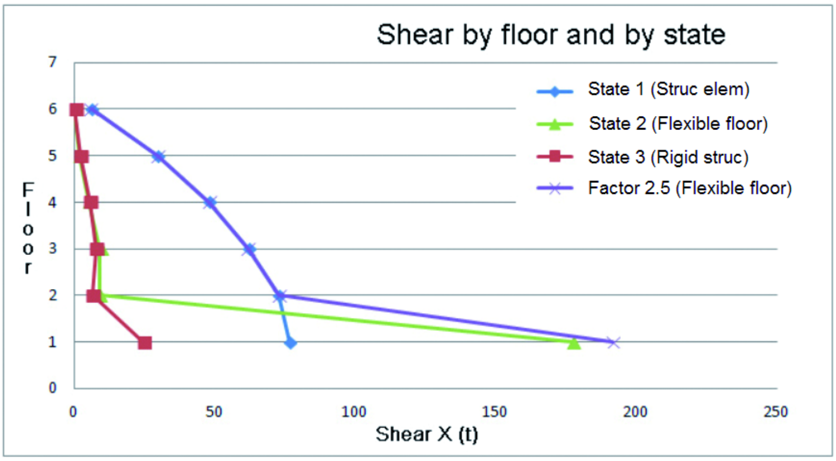



Figure 112. Total X shear by floor and state, considering an amplification factor of 2.5 for the forces of the ground floor. (Modal combination = CQC; Combination of directions = SRSS)

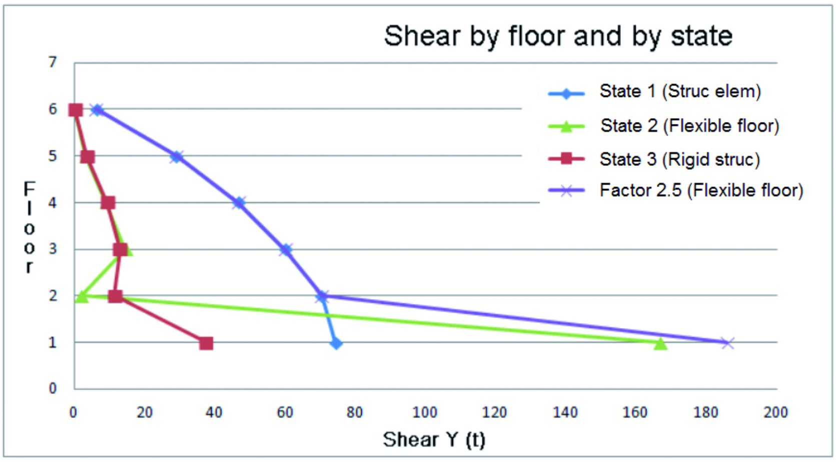



Figure 113. Total Y shear by floor and state, considering an amplification factor of 2.5 for the forces of the ground floor. (Modal combination = CQC; Combination of directions = SRSS)