



Software for Architecture,
Engineering and Construction



CYPE 3D

Calculations manual

Three dimensional bar structure project with steel, aluminium and timber sections, including foundations (pad footings, pile caps, strap and tie beams) and bracing systems against lateral loads, allowing ties that work only in tension. Joint and baseplate design for metallic structures.





Contents

1	Presentation	3
2	Calculations	3
2.1	Description of problems to resolve	3
2.2	Analysis carried out by the program	3
2.3	Units	5
2.4	Materials	5
2.5	Loads	6
2.6	Bar description	12
2.7	Checks carried out by the program	17
2.8	Tie design	18
2.9	I section joint design	21
2.10	Hollow structural section design	39
2.11	Composite beams	41
2.12	Fire resistance in CYPE 3D	42
2.13	Foundations	43
3	Code Implementation	62

1 Presentation

CYPE 3D is a powerful and efficient program brought about to carry out structural calculations in 3 dimensions of steel, aluminium and timber bars.

The program obtains the forces and displacements based on an automatic design. It possesses a laminated, welded and cold formed steel section database. It analyses any type of structure carrying out all the verifications and checks the selected code requires.

Thanks to the generation of structural views, the user can work with windows in 2D and 3D in a completely interactive manner. The structure can equally be redesigned and hence obtain its maximum optimisation. The element dimensions are created without the need of having to introduce coordinate systems or rigid meshes.

This *Calculations manual* explains the methodology followed by the program using the implemented codes.

2 Calculations

2.1 Description of problems to resolve

CYPE 3D designs three dimensional (3D) structures defined using bar type elements in space with nodes at their intersections.

Steel, aluminium, concrete, timber and generic section bars can be used and these are defined using their mechanical and geometrical properties.

The program automatically designs steel, timber and aluminium bars.

Reinforced concrete footings and pile caps and strap and tie beams are resolved for supports defined using vertical and inclined bars that bear on the on the foundation.

Data introduction and result consultation is done graphically.

2.2 Analysis carried out by the program

The program considers an elastic and linear behaviour of the materials. Bars defined are linear elements.

Loads applied on bars can be established in any direction. The program allows for the following types of loads to be introduced: uniform, triangular, trapezoidal, point, moments and different temperature increments on opposite sides.

Point loads can be applied at nodes in any direction. The type of node used is completely generic, whereby nodes with internal fixity can consist of fixed (moment) or pinned (simple) connections; and bar ends can be defined using fixity coefficients (between 0 and 1), rotational stiffness (moment/rotation) or defined as pinned connections.

Any type of support can be used, fixed or pinned, or as a function of its degrees of freedom. The support (or external fixity) can be elastic, by defining the constant corresponding to each restricted degree of freedom.

The number of loadcases is established depending on their origin and these can be assigned as dead, live, wind, seismic (static), snow and accidental loads. Dynamic seismic loads can be considered.

Using the basic loadcases, any type of combination can be defined with different combination coefficients in accordance with the selected code or defined by the user.

The limit states and combinations for each material and state are as follows:

- U.L.S. Concrete
- U.L.S. Foundation concrete
- Ground bearing pressures (characteristic loads)
- Displacements (characteristic loads)
- U.L.S. failure. Steel (rolled and welded)
- U.L.S. failure. Steel (cold formed)
- U.L.S. Timber
- U.L.S. Aluminium

All the combinations are generated for each case, indicating its name and coefficients, depending on the application code, material and use category.

Based on the geometry and loads that are introduced, the stiffness matrix of the structure is obtained, as well as the load matrices for each simple loadcase. The displacement matrix of the nodes of the structure is also obtained, inverting the stiffness matrix by frontal methods.

Having found the displacements for each simple loadcase, all the combinations are calculated for all the states and the forces at any sections based on the forces at the ends of the bars and any loads that have been applied.

2.3 Units

CYPE 3D allows for MKS system of units as well as the international SI system of units to define applied loads and obtain the corresponding forces.

2.4 Materials

The materials used in this program are classified into the following groups:

1. Rolled and welded steel
2. Cold-formed steel
3. Timber
4. Extruded aluminium
5. Concrete
6. Generic sections

1 and 2. If the selected material is **steel**, various steel files are present with their properties:

- Type of steel: rolled or welded
- Longitudinal modulus of elasticity: E
- Elastic limit σ_e , depending on the type
- Steel reduction coefficient g_s
- Poisson coefficient: ν . The transverse modulus of elasticity is calculated internally

$$G = \frac{E}{(1 + \nu) \cdot 2}$$

- Thermal expansion coefficient: α
- Specific weight: $\gamma = 78.5 \text{ kN/m}^3$
- Limiting slenderness

Finally, the material parameters for bolts are included in case baseplates are designed.

Steel sections used can be those belonging to the CYPE library or those that have been edited by the user.

If the default section library provided by CYPE is used, the existing section types can be used by selecting those to be used in each job.

If, on the contrary, the user wishes to create new section series and types, the x and y section geometry values will have to be indicated for each section, as well as the thicknesses of the plates and the remaining data for its correct definition.

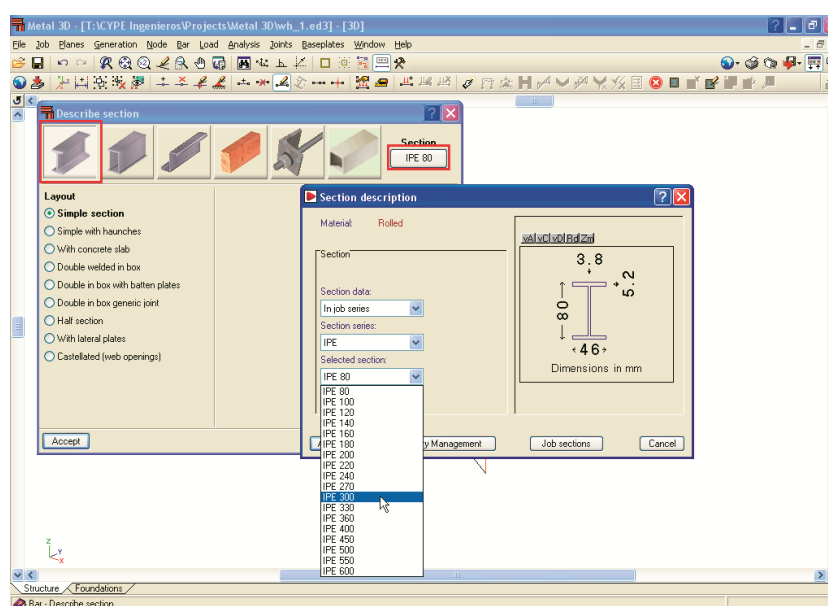


Figure 1

3. **Timber.** The type and resistance class of the timber is selected in accordance with the selected code, where its properties are defined.
4. **Extruded aluminium.** The aluminium alloy and temper is to be selected. These sections are designed in accordance to Eurocode 9.
5. **Concrete.** The type of concrete (resistance, control level, etc) is selected in accordance with the available codes.
6. **Generic section.** The user defines its geometry, mechanical characteristics and material properties.

2.5 Loads

Loads can be static or dynamic (in the case of earthquake loads) and are defined in accordance with their nature as simple loadcases.

2.5.1 *Additional loadcases*

CYPE 3D considers the characteristic loads for each of the simple loadcases defined as additional loadcases:

- Dead load
- Live load
- Wind load
- Seismic load
- Snow load
- Accidental load

Automatic loadcases are those generated by program itself, such as:

- The self weight of the bars
- Dynamic seismic loading, when activated.

The remaining loadcases are additional loadcases, with the option to be able to create different load dispositions for each loadcase. When different load dispositions are created, the user must define whether they are simultaneous, compatible or incompatible. When several additional loadcases of the same type are defined, it must be stated whether they can be combined or not. All this can be defined by the user.

2.5.2 *Limit states (combinations)*

Once a material, use category and code have been selected, all the combinations for all the limit states are generated automatically.

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

Different project situations, general code or user defined, can be defined for each limit state.

2.5.3 Loads

The following load types can be defined for each simple loadcase:

- **Loads on bars**

- Point
- Uniform
- Strip
- Triangular
- Trapezoidal
- Applied moment
- Increase in uniform temperature
- Increase in variable temperature

- **Loads on nodes**

Point, in any direction in accordance with the general axes, or by defining the direction vector with respect to those axes (X,Y, Z).

- **Loads on panels**

A panel is an area defined by coplanar nodes. Uniform or variable surface loads can be defined acting on the surface of a panel to simulate a one-way spanning slab. The span direction can be defined parallel to one of the straight lines joining the perimeter nodes of the panel. The resultant loads are applied on the bars after carrying out a simply supported simulation of the load distribution.

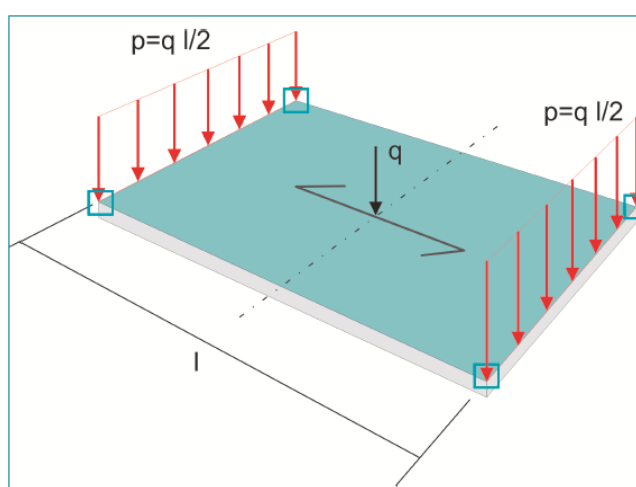


Figure 2

- **Surface loads**

This type of load acts in the same way as loads applied on panels, with the exception that the vertices of the surface can consist of nodes or any intermediate point of the bars of a panel.

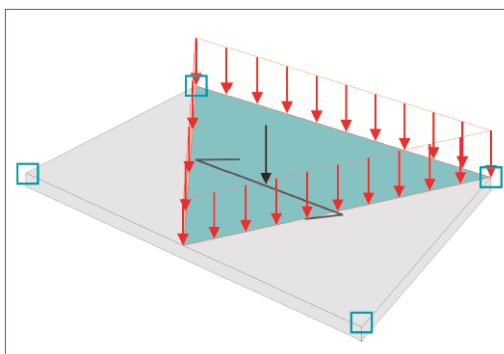


Figure 3

The load is distributed in the distribution direction indicated on the panel.

A surface load can only be defined on a single panel.

- **Prescribed displacements**

Displacements in the direction of the general axes can be defined on nodes that act as supports (displacement and rotation), for each loadcase. A common example would be the settling of a foundation, D_z , which would cause the corresponding forces to arise in the structure linked to that support.

2.5.4 *Consideration of 2nd order effects ($P\Delta$)*

Users can optionally choose to, when wind or seismic loadcases are defined, amplify forces produced by these horizontal loads. It is recommended this option be activated for the analysis.

This method is based on the P-delta effect due to the displacements produced by the horizontal loads, taking into account the second order effects, based on a first order analysis, a linear behaviour of the materials, with mechanical characteristics calculated using the gross section of the materials and their secant modulus of elasticity.

Due to the horizontal forces, at each node i , a horizontal force H_i is present, producing a displacement Δ_i . A load, P_i , due to gravitational loadcases, transmitted to the node by the structure acts at each node.

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

$$M_H = \sum H_i \cdot z_i$$

Similarly, a moment can be defined due to P-delta effects, $M_{P\Delta}$, due to loads transmitted by the bars to the P_i nodes, for each of the gravitational loadcases (k) defined, because of the displacements due to the horizontal load Δ_i .

$$M_{P\Delta k} = \sum_i P_i \Delta_i$$

where:

k: For each gravitational loadcase (dead load, live load...)

By calculating $C_k = \frac{M_{P\Delta k}}{M_{HK}}$, which is the stability index for each gravitational loadcase, an amplification coefficient can be obtained of the safety coefficient of the horizontal load loadcases for all the combinations for which these horizontal loads act.

This value, γ_z is calculated using the following formula:

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

where:

γ_{fgi} : Safety coefficient for dead loads belonging to loadcase i

γ_{fqj} : Safety coefficient for variable loads belonging to loadcase j

γ_z : Global stability coefficient

To calculate the displacements due to each horizontal load loadcase, the user must recall that a first order analysis has been carried out using the gross sections of the elements. If the forces have been calculated for ultimate limit state design, it seems logical that the displacements be calculated for cracked and homogenised sections. This is very laborious as it assumes the materials, geometry and load states are non linear. This cannot be taken on from a practical perspective using the normal means available for design. Therefore, a simplified method must be established, assuming a stiffness reduction of the sections,

which implies an increase of the displacements, as these are inversely proportional. The program asks the user to introduce the 'displacement multiplication factor' to be able to account for this stiffness reduction.

At this point, there is no single criteria, hence it is left to the user's judgement as to what value is to be introduced depending on the type of structure, estimated cracking grade, other stiffening elements that may be present, nuclei, stairs, etc, which in reality may reduce the calculated displacements.

In Brazil, a longitudinal elastic modulus reduction coefficient of 0.90 is usually considered as well as a cracked inertia reduction coefficient with respect to the gross section of 0.70. Therefore the stiffness is reduced by the product of these coefficients:

$$\text{Reduced stiffness} = 0.9 \cdot 0.70 \cdot \text{Gross stiffness} = 0.63 \cdot \text{Gross stiffness}$$

As the displacements are the inverse of the stiffness, the multiplying factor of the displacements will be equal to $1 / 0.63 = 1.59$, which is the value to be introduced in the program. It is generally considered that if $\gamma_z > 1.20$, the structure requires more stiffness, as it has high deformation and little stability in that direction. If $\gamma_z < 1.1$, its effect is small and practically negligible.

The NBR 6118:2003 code, using a simplified method, recommends amplifying the displacements by $1/0.7 = 1.43$ and limit the value of γ_z to 1.3.

The Model Code CEB-FIP 1990 applies a moment amplification method which recommends, due to the lack of a more precise calculation, to reduce the stiffnesses by 50% or, apply a displacement amplification coefficient equal to $14/0.50 = 2.00$. For this case, it may be considered that if $\gamma_z > 1.50$, the structure has to be stiffened in that direction, as the structure is very deformable and has little stability in that direction. If $\gamma_z < 1.35$, its effect is small and practically negligible.

ACI-318-95 describes a stability index Q per floor, not for the global stability of the building, even though a relationship could be established with the global stability coefficient if the floors are similar, using the following formula:

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

Regarding the limit it establishes to consider the floor as part of a non-sway frame, or what in this case would be the limit as to whether it is to be considered or not, it is taken as $Q = 0.05$, i.e. $1/0.95 = 1.05$.

For this case it has to be calculated and always taken into account when this value is exceeded, which definitively implies that it practically always has to be considered in the analysis and the forces are amplified using this method.

Regarding the displacement amplification coefficient, the code indicates that as the horizontal loads are temporary and short-term, a reduction of the inertia of 70% can be considered, and as the module of elasticity is smaller ($15100/1900 = 0.8$), i.e. a displacement amplification coefficient of $1 / (0.7 \cdot 0.8) = 1.78$ and, in accordance with the global stability coefficient, not exceeding a value of 1.35 would be reasonable.

It can be appreciated that the criteria of the model code is recommended and easy to recall, as well as advisable for all its application cases:

Displacement multiplication coefficient = 2

Global stability coefficient limit = 1.5

It is true that, on the other hand, stiffening elements are always present in buildings: façades, stairs, bearing walls, etc. which ensure a reduced displacement due to horizontal loads than those that have been calculated. Because of this, the displacement amplification coefficient is left at 1.00. The user is left to choose whether or not to modify it, as not all the elements can be discretised in the structure analysis.

Once the analysis has finished, a report can be obtained (**Reports > Global stability analysis**) where the maximum global stability coefficient in each direction can be seen.

It may occur that the structure is not stable, in which case, a message can be emitted before finishing the analysis, where the program warns that a global instability phenomenon is present. This will occur when the value of γ_z tends to ∞ or, which is the same using the formula, where it takes a null or negative value because:

$$\sum (\gamma_{fji} \cdot C_i + \gamma_{fqj} \cdot C_j) \geq 1$$

This may be studied for wind and/or seismic loads and its analysis is always recommended, as an alternative second order analysis calculation method, especially for buildings with sway or slightly sway frames, as is the case of most buildings.

2.6 Bar description

The user has to define the material, section shape (edited by the user or from the section library) and its disposition, which is how the bar section is adjusted with respect to the drawing axes.

- **Steel.** The user can choose amongst a range of simple or composed sections provided by the program for rolled, welded and cold-formed steel.

- **Timber.** Circular, square or rectangular (with constant or variable depth) bars can be defined
- **Aluminium.** I-sections, C-section, symmetrical angle, plate, T-sections, angle, rectangular and circular hollow sections, and, round and square bars are available.

2.6.1 Buckling

To study the buckling effects due to axial forces for each axis, the effective buckling length l_k or the coefficient β , where $l_k = \beta \cdot l$, where l is the distance between bar nodes. When the bar has intermediate nodes, the length or the coefficient corresponding to the real bar between its supports must be provided, bearing in mind it is being defined for a bar which is a fraction of the whole bar.

The user may also use the *"approximate calculation of buckling lengths"* by defining the structure as a sway frame or non sway frame structure, according to the simplified method and its validity for structures which are slightly orthogonal to each other, where the user can consult the accepted hypotheses in the help dialogue.

Additionally, the following hypotheses are assumed:

- The supports buckle simultaneously.
- The elastic shortening of the supports is ignored.
- The beams behave elastically and are joined in a rigid manner to the supports.

The stiffness of the beams is not modified due to normal forces.

The formulas applied are:

Non-sway frames

$$\beta = \sqrt{\frac{1.6 + 2.4(K_M + K_N) + 1.1K_MK_N}{K_M + K_N + 5.5K_MK_N}}$$

where:

$$K_M = \frac{\sum \left(\frac{I_b}{L_b} \right)_M}{\sum \left(\frac{I_b}{L_b} \right)_M + \sum \left(\frac{I_c}{L_c} \right)_M}$$

$$K_N = \frac{\sum \left(\frac{I_b}{L_b} \right)_N}{\sum \left(\frac{I_b}{L_b} \right)_N + \sum \left(\frac{I_c}{L_c} \right)_N}$$

I_b : Inertia of the beams reaching the node

L_b : Length of the beams reaching the node

I_c : Inertia of the columns reaching the node

L_c : Length of the columns reaching the node

Sway frames

$$\beta = \sqrt{\frac{9.6 + (R'_M + R'_N) + 1.25R'_MR'_N}{R'_M + R'_N + 1.25R'_MR'_N}}$$

where:

$$R'_M = 6 \frac{K_M}{1 - K_M}$$

$$R'_N = 6 \frac{K_N}{1 - K_N}$$

Limitations of the approximate calculation

The following warnings should be taken into account by the users:

- The presence of intermediate nodes in continuous bars, to which no other bars reach, invalidates the method. Hence, in these cases, the appropriate manual corrections must be undertaken.
- To apply the approximate method, the structure must be classified as having sway or non sway frames, therefore care should be taken when defining the structure.
- All that has been described is only applicable to metallic bars.
- If the structure that has been introduced is a plane frame, the values obtained are valid in its plane, and may not be valid in the plane perpendicular to it, as there are no transverse elements defined, especially when there is symmetry in the geometry, such as a dual slope frame calculated in an isolated manner.

2.6.2 *Lateral buckling*

The user can define the distance or separation between bracing for both the top flange and bottom flange, or the coefficient that multiplies the length of the bars between nodes. The applicable moment coefficient, which varies in different codes and depends on the moment distribution between braced points, can also be defined.

2.6.3 *Elements and groups*

When a bar is initially introduced between two points. Its properties are maintained, even if later on they may intersect other bars. It is also possible to create an element later on consisting of aligned bars containing several spans by marking the ends.

By grouping, the user can assign common properties to a group of grouped bars, and they are designed so the most unfavourable bar of the group does not fail. These may be ungrouped later on.

2.6.4 *Deflections*

The 'deflection' is taken as the maximum distance between the straight line joining the end nodes of a bar, and the deformed shape of the bar, without taking into account the displacement of the end nodes if these have moved. This distance is measured perpendicularly to the bar.

The 'absolute deflection' is the value in millimetres of the deflection in the direction in question.

The 'relative deflection' is established as the span between intersecting points of the deformed shape with the bar, divided by a value to be defined by the user. There may be an intermediate point or points, as well as the end nodes of the bars with no displacements, depending on the deformed shape of the bar.

The 'active deflection' is the maximum absolute difference between the maximum deflection and the minimum deflection of all the combinations defined for the displacements state.

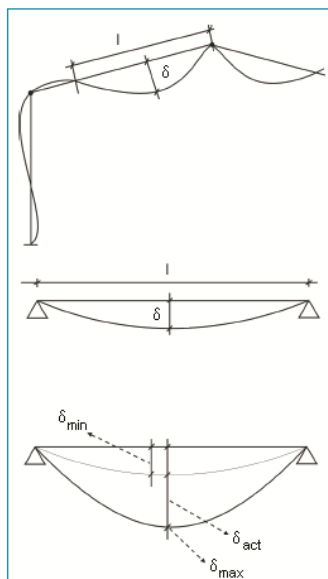


Figure 4

$$\delta_{act} = \delta_{max} - \delta_{min}$$

It is possible to establish a limit, be it a value for the maximum deflection, active deflection or relative deflection with respect to each of the local xy or xz planes of the bar.

2.6.4.1 Deflection groups

Bars can be grouped when these are aligned and so obtain the deflection between the ends of that particular group of bars whereby the deflection is calculated between the end nodes 'i' and 'f', instead of the local deflection between each pair of consecutive nodes.

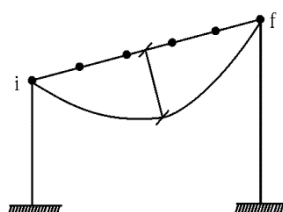


Figure 5. A deflection group

If the limit is exceeded, upon checking the bar after the analysis, it will be displayed in red, as well as all the other bars that fail.

2.7 Checks carried out by the program

In accordance with that explained previously, the program checks and designs the bars of the structure according to the criteria established in each code for each type of material:

- Steel
- Timber
- Aluminium

The program only obtains forces for concrete and generic sections.

If these limit states are exceeded, CYPE 3D will allow for a design to be carried out, whereby it will search through the section tables to find a suitable section which complies with all the conditions.

Any bars failing any checks are displayed in red.

The user is to bear in mind that when a bar is modified, the stiffness matrix will have changed and hence, should reanalyse the job and check the bars.

- **Check bars**

Upon selecting a bar, a table displaying the section series appears. Those sections which fail are marked with a red cross and those which are suitable with a green tick. Any errors that have been encountered are indicated.

- **U.L.S. Checks**

This is a detailed report indicating each and every code check that has been carried out, providing the article, formula and applied values for the worst case requirements of the consulted section.

As well as the resistance checks, the program carries out:

- Fire resistance checks
- Deflection checks

These two options are optional and have to be activated by the user if they are to be carried out.

2.8 Tie design

The program allows for bars to be defined as **Ties**. The fact the tie bars are composed of a straight axis and only admit tensile forces in the direction of the axis, implies that this model would only be strictly exact if a non-linear analysis were to be carried out, in which, all the bars resulting in compression would be deleted after each analysis.

Additionally, to undertake a dynamic analysis without taking into account the bars in compression, an analysis in the time domain with accelerograms would be required.

As an approximation to the exact method, we propose an alternative method whose results, with the requirement that all the conditions detailed below are met, are sufficiently acceptable for every day design of structures with bracing elements.

The method has the following limitations, which the program ensures are met:

1. The tie element forms part of a rigid bracing system framed at its four sides, or three of its sides if the bracing reaches external fixities. Additionally, each rigid frame must form a rectangle (four internal angles must be right angles).

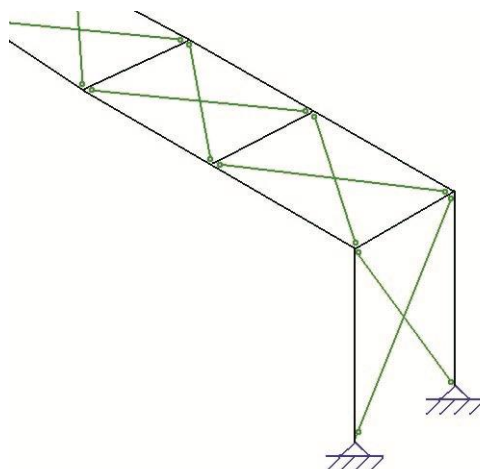


Figure 6

2. The axial stiffness of the ties (AE/L) is less than 20% of the axial stiffness of the elements framing the bracing elements.
3. The diagonal ties of a frame must both be of the same section.

2.8.1 Method application

The analysis method is linear and elastic using matrix formulae. Each tie is introduced in the stiffness matrix with only its axial stiffness term (AE/L), whereby this is equal to half the real axial stiffness of the tie. This way, displacements are achieved in the stiffness plane, similar to those obtained if the tie in compression were deleted from the matrix analysis and considering the real cross sectional area of the tie in tension.

The final forces in each tie are obtained for each loadcase and the following steps are taken for those ties in which the axial force is in compression:

- A. The axial force of the tie in compression is cancelled**
- B. This axial force is added to the axial force of the other diagonal tie of the same stiffening frame.**
- C. With this new axial force configuration, the equilibrium of the nodes is restored.**

Given that the method deals with force compatibility and not displacements, it is important to bear in mind the axial stiffness restrictions of the sections making up the stiffening frame indicating in part 2 of the previous section; the method is more exact the smaller the relative length differences are of the bars framing the ties. In all the cases analysed by CYPE, the discrepancies between the results obtained using this method and those obtained using linear analysis have been negligible. The following figures display a scheme of the previously described process. Forces resulting from each of the combinations:

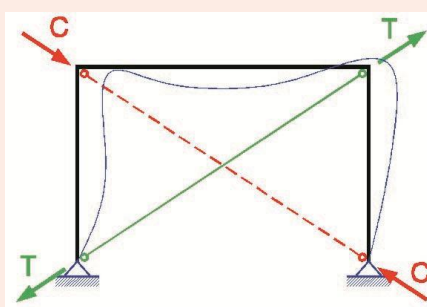


Figure 7

T: axial force corresponding to the tie in tension

C: axial force corresponding to the tie in compression

A. Cancelling out of the force of the tie in compression.

Assigning the compression value to the tie in tension

The axial force of the tie in compression ($C=0$) is eliminated and is added to the tie in tension ($T^* = T + |C|$)

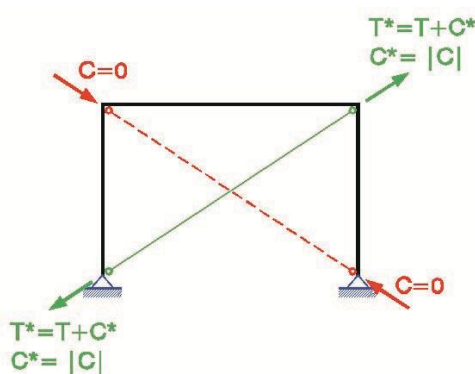


Figure 8

B. Distribution (by force resolution) of the increment of axial force in the tie in tension (C^*)

The increment in axial force (C^*) in the tie is resolved into the direction of the bars (or fixity reactions) reaching the nodes.

$N_1, N_2, N_3, R_{1h}, R_{2h}, R_{3h}, R_{2v}$: forces and reactions of the elements forming the rigid frame without considering the increase in tension of the tie in tension.

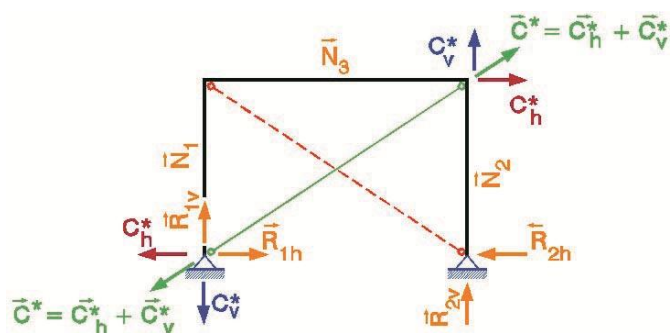


Figure 9

C. Equilibrium restoration in the end nodes of the ties. Force equilibrium

The vector sum of the components of the increment in tension (with the same absolute value as the compression of the tie in compression) is carried out.

The final force and reaction distribution is displayed in the following diagram:

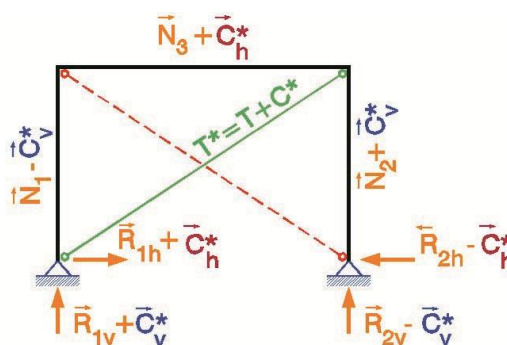


Figure 10

These values can be consulted in each bar or node for each loadcase or combination loadcase. Each loadcase is treated as a unit combination.

2.9 I section joint design

As of the 2008 version, CYPE 3D incorporates the analysis and design of I section joints, by means of welded and bolted connections.

Additionally, as of the 2009.1.g version, the program considers upon designing the connections that bar ends are rigid spans along the length taken up by the node.

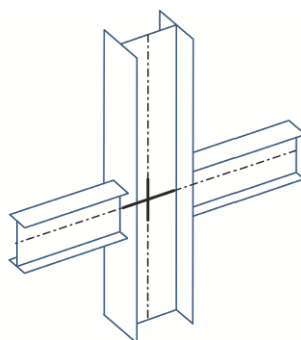


Figure 11

2.9.1 Types of I section joints

For I sections (Joints I, II, III and IV modules), the program provides two types of design:

Welded, e.g.:

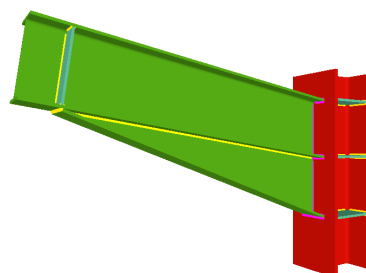


Figure 12



Figure 13

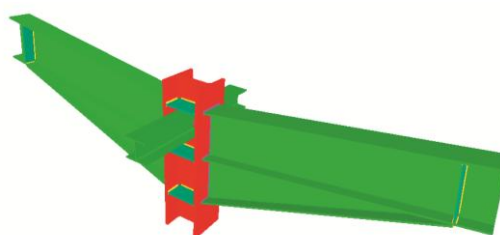


Figure 14

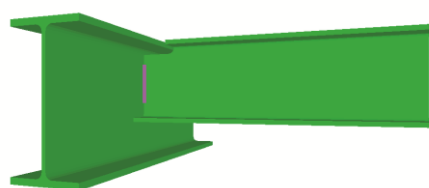


Figure 15

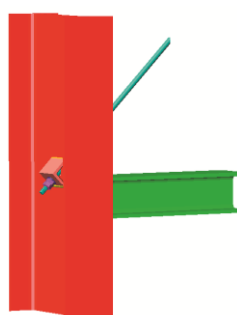


Figure 16

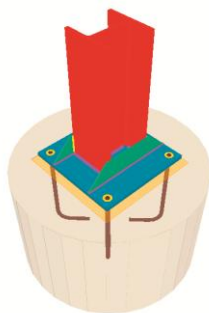


Figure 17

For a more complete list of available welded connections, please consult the corresponding website at

<https://info.cype.com/en/product/welded-joints-of-rolled-and-welded-steel-i-sections/>

<https://info.cype.com/en/product/welded-connections-of-rolled-and-welded-steel-i-sections-for-building-frames/>

Bolted (using ordinary or prestressed bolts), e.g.:

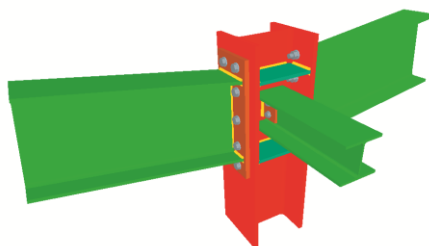


Figure 18

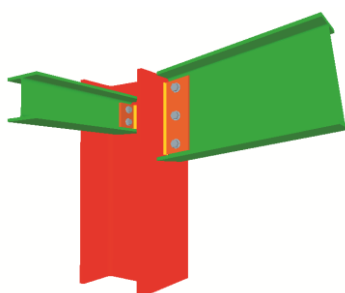


Figure 19

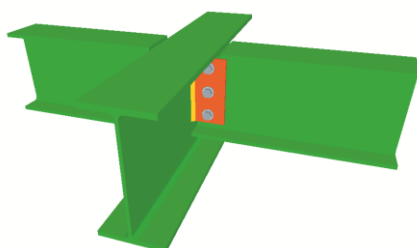


Figure 20



Figure 21

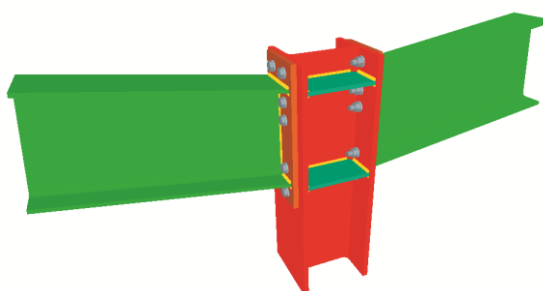


Figure 22

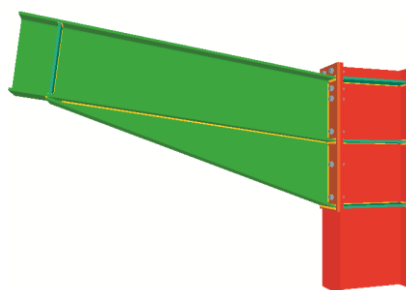


Figure 23

For a more complete list of available bolted connections please consult the corresponding website at

<https://info.cype.com/en/product/bolted-joints-of-rolled-and-welded-steel-i-sections/>

<https://info.cype.com/en/product/bolted-connections-of-rolled-and-welded-steel-i-sections-for-building-frames/>

2.9.2 *I section joint design*

When analysing the job, the program enquires whether the user wishes to activate the welded or bolted connection design. Nonetheless, if the option is not activated, the joints may be designed after the analysis using the option **Joints > Analyse**, bearing in mind what has already been indicated regarding node size consideration and the rigid consideration of the end span of a bar with the finite dimension of the node.

If nodes whose type of joint is resolved in the program are detected during the design process of the structure, the program will design the connections and will provide a detail drawing of the results.

Analysis of the joints of the job. EN 1993-1-8:2005, CTE DB-SE-A and ABNT NBR 8800:2006.

Joints are designed with welded connections or bolted connections using prestressed or ordinary bolts.

The program checks that the sections and components (stiffeners, bolts, plates etc.) of the joint do not interfere with one another: that the joints can be executed on site, that there is sufficient space to be able either weld or provide the required bolts and be able to tighten them correctly.

The complete node is designed considering the 6 forces at each bar end reaching the node, then optimising the components upon knowing the arrangement, behaviour and geometry of the node.

Detail drawings are provided as well as a report justifying all the checks that have been carried out, together with a take off summary of the joint and its components.

Options are available for each type of bolt, stiffeners and baseplates at foundation level.

Nodes possessing the same geometry, same sections and materials, same fixity coefficient or rotational stiffness with less than a 10% difference, and a similar structural behaviour, are automatically grouped. This simplifies the execution of the job by grouping similar joints.

In the case of bolted joints, the rotational stiffness of the designed connections is established and the program emits a warning if the difference between the stiffness obtained and that assigned is greater than 20%. It is important the user redesigns the joint if the rotational stiffness has been modified, as differences may be present that invalidate the analysis results.

Welded I section joint checks

The following checks are carried out on the components:

- Shear and slenderness of the panel made up by the web of the column. It is reinforced using a plate welded to the web.
- Combined stresses at the simply connected column web.
- Combined stresses at stiffeners and plates which are coplanar with beams fixed to the column web.
- Shear-bending interaction at separation plates with sections with different depths (splices, column transitions, beams fixed to the web).
- Sufficient net area for round threaded ties.
- Bending and punching shear due to perpendicular forces of the web or flange of the supporting elements of simply connected sections.
- Stiffener buckling resistance.

Below are the checks carried out on the welds:

- Minimum thickness of elements to be welded
- Real and absolute minimum effective lengths and relative to the throat weld.
- Minimum and maximum (0.7t_{min}) throat weld thickness.
- The contributing material must have at least, the same resistance as the materials to be joined.
- Minimum and maximum angle between the surfaces to be welded.
- Weld resistance: The 3 stress components in the plane of the throat and the equations are checked for all the load combinations of the code. In the case of double angle welds, the check is carried out for the acute angle and obtuse angle.

$$\sqrt{\sigma_{\perp}^2 + 3 \cdot (\tau_{\parallel}^2 + \tau_{\perp}^2)} \leq \frac{f_u}{\beta_w \cdot \gamma_{M2}}$$

$$\sigma_{\perp} \leq K \cdot \frac{f_u}{\gamma_{M2}}$$

K = 0.90 (EC3)

K = 1.00 (CTE)

Bolted I section joint checks

	Check	M.C.E.	S.C.E.	S.C.L.
1	Distances (maximum and minimum) to edge, between bolts and to elements	×	×	×
2	Bolt tension (including prying forces)	×	×	×
3	Shear in ordinary bolts and U.L.S. slipping for prestressed bolts	×	×	×
4	Shear-tension interaction of ordinary bolts	×	×	×
5	Crushing of the bolts and plate	×	×	×
6	Crushing of the bolts and simply connected section web			×
7	Crushing of the bolts and supporting section flange	×	×	
8	Punching shear of the plate	×	×	×
9	Punching shear of the simply connected section web			×
10	Punching shear of the supporting section flange	×	×	
11	Bending of the end-plate	×	×	
12	Bending of the supporting section flange	×	×	×
13	Bending of the supporting section web			×
14	Bending of the simply connected section web		×	×
15	Buckling of the simply connected section web		×	
16	Buckling of the plate			×
17	Shear and slenderness of the panel made up by the column web	×		
18	Tearing of the plate		×	×
19	Tearing of the simply connected section web			×
20	Tension of the section flange welded to the plate	×		
21	Tension of the section web welded to the plate	×	×	
22	Tension of the column web or continuous beam	×	×	
23	Tension or compression of the stiffeners	×	×	
24	Compression with shear interaction and torsion of the section flange welded to the plate	×		
25	Compression with shear interaction and torsion of the section web welded to the plate	×		
26	Combined stresses at the simply connected section web		×	
27	Combined stresses of the plate			×
28	Punching shear of the supporting section web due to the plate			×
29	Punching shear of the supporting section flange due to the plate			×
30	Effective weld throat stresses. The bolt stress influence is taken into account	×	×	×
31	Rotation capacity. Resistant moment Mj.Rd. Critical mechanism 1 or 2 (double or single hinge)	×		

M.C.E.: With end-plate

S.C.E.: Simple connection with end-plate

S.C.L.: Simple connection with lateral plate

Baseplates

Baseplates are designed using the welded joints module: dimensions, stiffeners, welds and bolts according to the selected code.

$$\sqrt{\sigma_{\perp}^2 + 3 \cdot (\tau_{\parallel}^2 + \tau_{\perp}^2)} \leq \frac{f_u}{\beta_w \cdot \gamma_{M2}}$$

$$\sigma_{\perp} \leq K \cdot \frac{f_u}{\gamma_{M2}}$$

2.9.3 Check of I section joints

The joints that have been resolved can be seen after the analysis using the **Consult** option within the Joints menu. Upon activating this option and clicking on a node whose joint has been designed (displayed in green), the details of the joint will be displayed.

If the user clicks on a node whose joint has not been designed (displayed in red), the program will display a 3D view with the bars reaching the joint, so the user may see whether or not the bars interfere with one another so as to prevent the joint from being designed.

By clicking on a node that has been partially resolved (displayed in orange), only the resolved joints will be displayed.

The following tabs may be consulted:

- **Detail.** These are the detail drawings of the designed joints.
- **Report.** This is a report summarising the analysis that has been carried out and its take off.
- **3DView.** This is a three dimensional view of the node and its connections.

Even if the user does not dispose of the joints module in his/her license, the program allows the user to activate the joint design. Once concluded, the 3D views of the joints that could be designed with the non-acquired modules can be visualised. The details or check and take off reports are not displayed. When the cursor is placed over one of these nodes, a warning is displayed indicating the non-acquired modules that could design the joint.

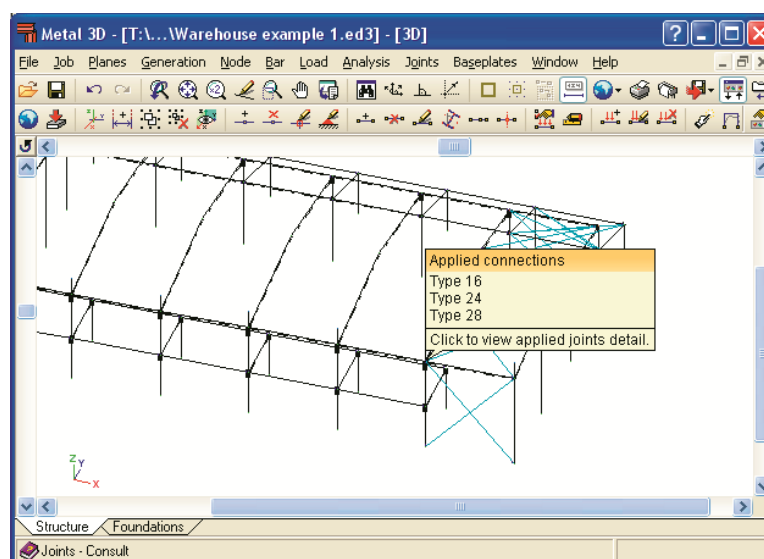


Figure 24

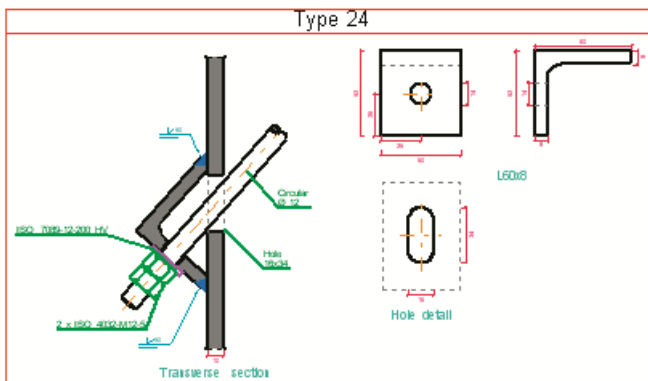


Figure 25

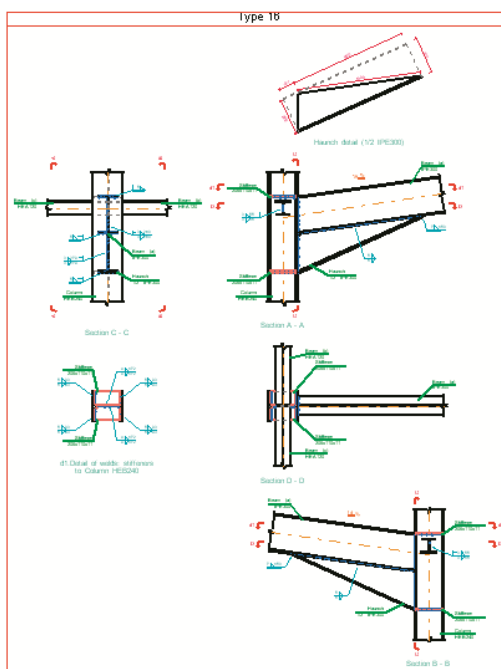


Figure 26

2.9.4 Rotational stiffnesses

When performing the global analysis of the structure, it is important to take into account the behaviour of the joints to obtain the correct distribution of the internal forces, stresses and deformations.

To establish how the effect of the behaviour of the connections should be taken into account in the analysis of the structure, three different simplified models are used:

- Simple supports: those in which no moments are transmitted.
- Continuous: those in which the behaviour of the connections does not intervene significantly in the analysis of the structure.

- Semi-continuous: those in which the behaviour of the connection must be taken into account in the global analysis of the structure.

As the program carries out an elastic analysis of the structure, the following relationships arise from the previous three models:

	Connection classification		
Type of model	Simple support	Continuous	Semi-continuous
Elastic analysis	Pinned connection	Rigid connection	Semi-rigid connection

The assumptions corresponding to semi-rigid connections are analysed below.

2.9.4.1 Moment rotation diagram

The behaviour of the connections is studied by analysing the moment- rotation properties diagram, which allows to define the three main structural properties of the connection:

- Bending moment resistance $M_{j,Rd}$: Maximum ordinate of the diagram.
- Rotational stiffness S_j : the secant stiffness for an acting bending moment value $M_{j,Ed}$, defined up to rotation ϕ_{xd} which corresponds to the point at which $M_{j,Ed}$ is equal to $M_{j,Rd}$.
- Rotation capacity ϕ_{Cd} : represents the maximum rotation of the moment-rotation diagram.

The following figure displays a typical connection moment-rotation:

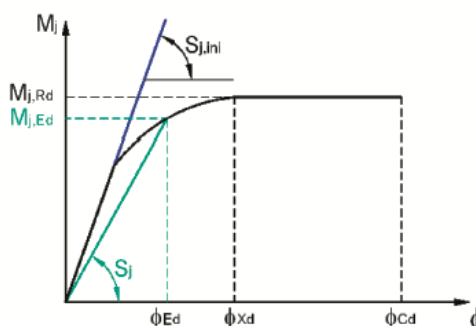


Figure 27

According to its rotational stiffness S_j connections are classified as: pinned connections, rigid connections or semi-rigid connections. The boundaries between one type and another are shown in the figure below:

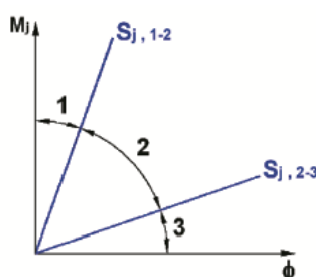


Figure 28

Where:

Zone 1. Rigid connections

Zone 2. Semi-rigid connections

Zone 3. Pinned connections

2.9.4.2 Connection rotational stiffness analysis

The program constructs the characteristic Myy moment - rotation in the xz plane diagram for each joint at the end of an element where its rotational stiffness has been calculated due to the presence of deformable components in the node.

2.9.4.2.1 Initial stiffness calculation $S_{j,ini}$

For axial forces not exceeding 5% of the transverse section's capacity, the stiffness S_j of the connection for an acting moment $M_{j,Ed}$ can be obtained using the following expression:

$$S_j = \frac{E \cdot z^2}{\mu \sum_{i=1}^{i=n} \frac{1}{k_i}}$$

Where:

E: Steel elastic modulus

z: Connection lever arm

k_i : Stiffness coefficient for the basic i th component

μ : Stiffness ratio:

$$\mu = \frac{S_{j,ini}}{S_j}$$

μ can be found as follows:

When $M_{j,Ed} \leq \frac{2}{3}M_{j,Rd} \rightarrow \mu = 1.00$ (elastic behaviour)

When $\frac{2}{3}M_{j,Rd} < M_{j,Ed} \leq M_{j,Rd}$;

$$\mu = \left(1.5 \cdot \frac{M_{j,Ed}}{M_{j,Rd}} \right)^{2.7} \quad (1.00 \leq \mu < 3.00)$$

To calculate the initial stiffness of the connection $S_{j,ini}$ the component method is used, according to which:

For column-beam joints:

$$S_{j,ini} = \frac{E \cdot z_{eq}^2}{\frac{1}{k_{eq}} + \frac{1}{k_1}}$$

For splices and ridge intersections:

$$S_{j,ini} = \frac{E \cdot z_{eq}^2}{\frac{1}{k_{eq}}}$$

Where:

z_{eq} : Equivalent connection lever arm:

$$z_{eq} = \frac{\sum_{r=1}^{r=n} k_{eff,r} \cdot h_r^2}{\sum_{r=1}^{r=n} k_{eff,r} \cdot h_r}$$

h_r : Distance between row r and the compression centre (which is considered to coincide with the compressed flange)

n : Number of rows in tension

k_1 : Fitted joints of the beam and the column flange: Stiffness coefficient of the shear-resisting column web.

Fitted joints of the beam and the column web: Stiffness coefficient of the shear-resisting column flanges.

$$k_1 = \frac{0.38 \cdot A_{vc}}{\beta \cdot z_{eq}}$$

A_{vc} : Shear area of the column panel

β : Force distribution coefficient, taken as $\beta = 1.0$

k_{eq} : Equivalent stiffness coefficient of the rows in tension of the connection:

$$k_{eq} = \frac{\sum_{r=1}^{r=n} k_{eff,r} \cdot h_r}{z_{eq}}$$

$k_{eff,r}$: Effective stiffness coefficient of row r

For column-beam connections:

$$k_{eff,r} = \frac{1}{\frac{1}{k_3} + \frac{1}{k_4} + \frac{1}{k_5} + \frac{1}{k_{10}}}$$

For splices and ridge intersections:

$$k_{eff,r} = \frac{1}{\frac{1}{k_5} + \frac{1}{2 \cdot k_{10}}}$$

k_3 : Fitted joints of the beam and the column flange: Stiffness coefficient of the tension-resisting column web.

Fitted joints of the beam and the column web: Stiffness coefficient of the tension-resisting vertical plate.

$$k_3 = \frac{0.7 \cdot l_{eff,1} \cdot t_{wc}}{d_c}$$

k_4 : Fitted joints of the beam and the column flange: Stiffness coefficient of the flexure-resisting column flange.

Fitted joints of the beam and the column web: Stiffness coefficient of the flexure-resisting support plate.

$$k_4 = \frac{0.9 \cdot I_{\text{eff},1} \cdot t_{\text{fc}}^3}{m^3}$$

k_5 : Stiffness coefficient of the moment-resisting end-plate:

$$k_5 = \frac{0.9 \cdot I_{\text{eff},1} \cdot t_p^3}{m^3}$$

k_{10} : Stiffness coefficient of the bolts in tension

$$k_{10} = \frac{1.6 \cdot A_s}{L_b}$$

$I_{\text{eff},1}$: Smallest value of the effective length of the bolt row, considered individually or in groups of rows.

m : Distance from the bolt to the plastic hinge formed next to the section or next to the stiffener.

d_c : Fitted joints of the beam and the column flange: Depth of the column web.

Fitted joints of the beam and the column web: Width of the vertical plate.

t_{wc} : Fitted joints of the beam and the column flange: Thickness of the column web. Fitted joints of the beam and the column web: Thickness of the vertical plate.

t_p : Thickness of the end-plate

t_{fc} : Thickness of the support plate

A_s : Tensile resistance area of the bolts

L_b : Bolt elongation length

2.9.4.2.2 Calculation of the plastic moment resistance $M_{j,Rd}$

The plastic moment resistance of the joint $M_{j,Rd}$ is obtained as follows:

- The minimum resistance of the zone in tension, for which the tensile resistance is obtained for each row of bolts, starting at the row furthest from the centre of compression, which supposedly coincides with the centre of the compressed flange to

be joined. The smallest of the following values is taken for the resistance of each row of bolts:

- The tensile resistance of the column web (Fitted joints of the beam and the column flange).
 - The tensile resistance of the vertical plate (Fitted joints of the beam and the column flange).
 - The tensile resistance of the beam web.
 - The bending resistance of the column flange (Fitted joints of the beam and the column flange)
 - The bending resistance of the support plate (Fitted joints of the beam and the column web).
 - The bending resistance of the end-plate.
 - The tensile resistance of the bolts.
- b) The minimum resistance of the zone in compression is obtained, for which the smallest of the following values is taken:
- The compression resistance of the stiffeners (column-beam connections with stiffeners).
 - The compression resistance of the column web (column-beam connections without stiffeners)
 - The compression resistance of the section flange.
- c) The shear resistance of the column panel (column-beam connections) is obtained.
- d) The maximum admissible force of the group is taken as the smallest value amongst:
- The minimum resistance of the zone in tension.
 - The minimum resistance of the zone in compression.
 - The shear resistance of the panel of the column.
- e) The maximum force is shared amongst the rows in tension, starting at the row furthest away from the centre of compression in such a way that the force assigned to each row $F_{TEd,i}$ does not exceed the previously calculated capacity.
- f) The moment resistance of the connection $M_{j,Rd}$ is established using the following expression:

$$M_{j,Rd} = \sum_{i=1}^n h_i \cdot F_{T,Ed,i}$$

Where:

h_i : Distance between the bolt row with index i and the centre of compression.

$F_{T,Ed,i}$: Tensile force in the i th row.

n : Number of bolt rows situated in the tension zone of the connection.

2.9.4.2.3 Moment-rotation diagram representation

The moment rotation diagram of the connection is drawn using the initial stiffness $S_{j,ini}$ and the moment resistance $M_{j,Rd}$, for positive and negative values and the forces acting at the ends of the element. The result (curve A) is shown in the figure below:

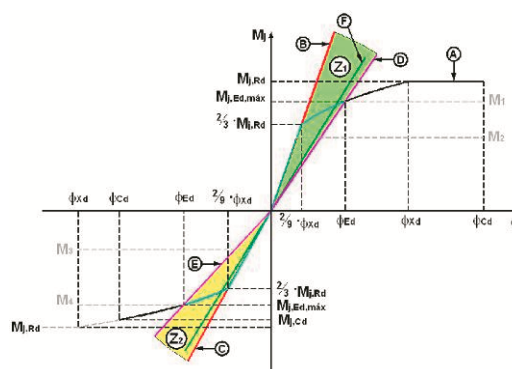


Figure 29

The following data can be seen in the graph:

a) Characteristic points of the diagram A:

$M_{j,Rd}$

Plastic moment resistance of the connection.

$2/3 \cdot M_{j,Rd}$

Elastic moment resistance of the connection.

$M_{j,Cd}$

Moment resistance corresponding to the rotation capacity of the connection ϕ_{Cd} .

ϕ_{Xd}

Rotation corresponding to the plastic moment resistance.

$2/9 \cdot \phi_{Xd}$

Rotation corresponding to the elastic moment resistance.

ϕ_{Cd}

Rotation capacity of the connection.

When rotation ϕ_{Cd} is less than ϕ_{Xd} , diagram 'A' is interrupted at the corresponding point $(M_{j,Cd}; \phi_{Cd})$, as the connection cannot reach rotation ϕ_{Xd} for the moment resistance $M_{j,Rd}$

b) Lines:

B Line whose gradient is the rotational stiffness corresponding to the smallest acting positive moment.

C Line whose gradient is the rotational stiffness corresponding to the smallest acting negative moment.

D Line whose gradient is the rotational stiffness corresponding to the greatest acting positive moment.

E Line whose gradient is the rotational stiffness corresponding to the greatest acting negative moment.

F Line whose gradient is the rotational stiffness used in the analysis of the structure.

Also displayed are the lines whose gradients are the initial rotational stiffnesses for positive moments as well as for negative moments.

c) Zones:

Z1 Zone which includes the rotational stiffness values for the range of positive moments.

Z2 Zone which includes the rotational stiffness values for the range of negative moments.

2.9.4.2.4 Application of the moment-rotation diagram

Rotational stiffness in the xz plane of the connection

The program proposes to adopt a rotational stiffness value equal to the smallest of those corresponding to lines D and E of the previous figure.

When, at an element end, there is a significant interaction of Myy moment with the axial forces and/or Mzz bending moments (xy plane of the connection), this type of stiffness analysis loses precision and the program warns of this situation if the following condition is not met:

$$\frac{M_{z,Ed}}{M_{z,pl,Rd}} + \frac{N_{Ed}}{N_{pl,Rd}} \leq 0.05$$

Where:

$M_{z,Ed}$ Mzz moment.

$M_{z,pl,Ed}$ Plastic moment resistance of the section.

N_{Ed} Axial force.

$N_{pl,Rd}$ Plastic axial force resistance of the section.

Rotational stiffness in the xy plane of the connection

Bearing in mind that the Mzz moments acting on a connection are usually negligible compared to the Myy moments, the program proposes a rotational stiffness value corresponding to the initial rotational stiffness $S_{j,ini}$ of the xy plane, calculated in a similar way as for the xz plane.

2.9.5 *Reasons why a joint has not been designed*

If the program does not design a joint, which initially appears to correspond to one implemented in the program, it may be due to one of the circumstances explained below:

a) Section fixed to the web of another section

If a section is to be fixed to the web of another section, the connection cannot be solved. Bar ends connecting to the web of a section must always be a pinned connection.

b) Interference between sections and stiffeners

If the section joining to the web of another section intersects with the stiffeners placed by the program to guarantee the bars reaching in the orthogonal plane are fixed.

c) Thickness of the elements

If the necessary throat thickness of the weld is greater than 0.7 times the thickness of the joining element.

d) Orthogonal elements

If the webs of the bars are not contained in the same plane or are not perpendicular to one another, the program will not resolve the joint.

e) Angle

If the previous point is complied with, the angle between the surfaces of the bars to be welded must be greater or equal to 60°, otherwise the joint is not designed.

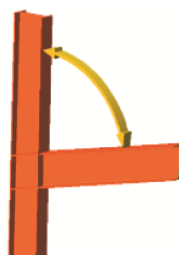


Figure 30

f) Trusses

If the node has a geometric configuration composed of bars for which the resistance formulas of the code cannot be applied.

2.10 Hollow structural section design

2.10.1 *Types of hollow structural sections*

Below are some of the types of hollow structural sections that have been implemented in CYPE 3D:

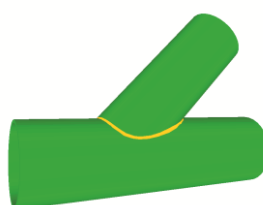


Figure 31

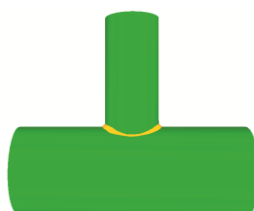


Figure 32



Figure 33

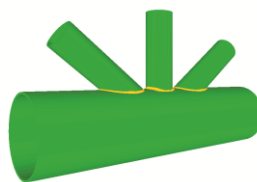


Figure 34



Figure 35



Figure 36

For a more complete list of available hollow structural section connections please consult the corresponding website at

<https://info.cype.com/en/product/joints-v-flat-trusses-with-hollow-structural-sections/>

2.10.2 Checks

Two sets of checks are carried out, one corresponding to the geometric checks stated in the application codes and the other carries out resistance checks.

The geometric checks that are undertaken at a joint consisting of circular hollow structural sections are:

For the chords:

1. Maximum elastic limit.
2. Ratio between the diameter of the chord and its thickness.
3. Section class (for chords in compression).
4. Minimum and maximum chord thickness.

For the infill bars:

1. Maximum elastic limit.
2. Ratio between the diameter of the bar and its thickness.
3. Section class (for bars in compression).
4. Maximum and minimum bar thickness.
5. Minimum angle between the infill bars and the chord.
6. Ratio between the diameter of the infill bar and the diameter of the chord.
7. Minimum spacing between infill bars.
8. Minimum overlap between infill bars.

The resistance checks carried out on circular hollow structural sections are:

1. Plastification of the chord surface.
2. Punching due to shear of the chord surface.
3. Interaction between the axial force and bending moment.
4. Interaction between the axial force and shear at the chord.
5. Shear at the chord surface in the case of overlapped bars.
6. The overlapping bar is checked as if it were the chord of the overlapped bar (for an overlap greater than the overlap limit indicated in the code).
7. Local plastification of the overlapping bar.

Welds:

The welds are designed so their resistance is, at least, the smallest resistance of the bars to be joined.

2.11 Composite beams

The design of composite beams is carried out according to UNE ENV1994-1-1: June 1995 – Eurocode 4: Design of composite steel and concrete structures – Part 1-1: General rules and rules for buildings.

Steel I sections with connectors can be introduced below floor slabs ('with concrete slab' option within the describe section box), hence creating composite behaviour.

Regarding the analysis, the width of the concrete has to be defined, as this determines the weight. The effective width has to be indicated to be able to design for bending and to determine its stiffness.

The program assumes the existence of the connectors. If it is not possible to provide the adequate number of connectors, a partial connection can be established by indicating a coefficient between 0 and 1 (1 being complete collaboration).

The creep coefficient is also to be indicated to establish the steel /concrete equivalence coefficient of the composite section, to take into account the duration of the loads.

For the design of the selected steel section and concrete slab, the corresponding codes are used, for both the steel and concrete, in accordance with that indicated in Eurocode 4.

2.12 Fire resistance in CYPE 3D

2.12.1 *General properties*

Using the Fire resistance check module (also used in CYPECAD), CYPE 3D carries out the fire resistance check and designs the protection coating of the structural steel elements making up the job in accordance with the selected code.

The fire resistance check for structural timber elements was already possible in previous versions and is undertaken by a different module: the analysis of timber structures module (available for CYPE 3D and integrated 3D structures of CYPECAD). This module designs the timber sections against fire so to comply with the selected code.

2.12.2 *Code selection*

When designing using rolled and cold-formed steel in accordance with Eurocode 3, and its corresponding National Application Documents for France and Portugal: Eurocode 3 and 4 (France) and Eurocode 3 and 4 (Portugal), the prescriptions of the Eurocode are applied.

If the Spanish CTE DB-SE A code is selected for the design of steel sections, then the code applied for the fire resistance check is the CTE DB-SI 6 code. For the remaining steel and cold-formed codes, the fire resistance check, for the time being with this current version, is not undertaken.

New codes are constantly being implemented into the program.

Please consult our website at

<https://info.cype.com/en/product/design-codes-implemented-in-cype-programs/> for the list of available design codes

To activate the fire resistance check in CYPE 3D, select the corresponding box within the **Job > Steel sections and Job > Timber sections dialogue boxes**.

The Fire resistance module carries out the following checks:

- For bars with defined protection coatings, the program designs the minimum required thickness so the requirements of the selected code are met.
- For bars for which the protection coating has not been defined, the program checks the element with the assigned fire resistance data.
- If a protection coating is assigned to a structural element and the program checks it is not required to meet the requirements stated in the selected code, the program emits a warning and considers a minimum coating thickness due to constructive reasons.

2.13 Foundations

The present chapter indicates the general considerations that have been taken into account for the check and design of the foundation elements that can be defined in CYPE 3D below the vertical supports of the structure which have been defined as external fixities.

These may be designed simultaneously with the rest of the structure or independently. As these are support elements with no displacement, they do not affect the analysis of the structure.

As they can be designed independently, the user is reminded that modifications can be carried out on the structure without affecting the foundations.

It may also be used as an editor, whereby foundation elements may be introduced without having to design them and obtain drawings and take-offs.

2.13.1 Pad footings

CYPE 3D allows for the design of reinforced concrete and mass concrete footings. The footings can be of the following types:

- Footings with constant depth
- Tapered footings or footings with variable depth

Geometrically on plan, they are classified as:

- Square
- Centred rectangular
- Eccentric rectangular (specific cases: edge or corner footings)

The loads transmitted by the supports are transported to the centre of the footing, obtaining its resultant. The forces transmitted may be:

N: Axial

M_x : X moment

M_y : Y moment

Q_x : X shear

Q_y : Y shear

T: Torsion

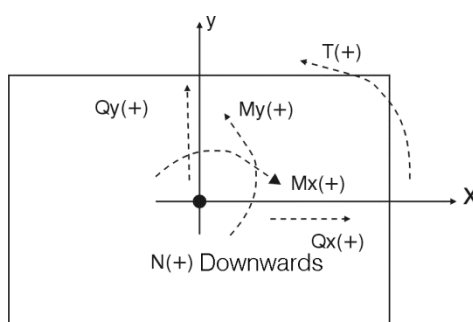


Figure 37

The loadcases considered include: Self weight, live load, wind, snow and seismic loading. The states that are checked are:

- Ground bearing pressures
- Equilibrium
- Concrete (bending and shear)

A design can be carried out using the default dimensions that have been defined within the program options or other dimensions provided by the user.

Alternatively, the reinforcement can be obtained with a specific geometric configuration. The check consists in verifying that the geometry and reinforcement of the footing meets the requirements of the selected code.

2.13.1.1 *Ground bearing pressures*

A flat deformation diagram is assumed for the footing and hence, depending on the forces, a trapezoidal force distribution will be obtained acting on the ground. Tensile forces are not admitted, therefore, when the resultant lies outside of the central nucleus, areas without pressure will be present.

The resultant force must lie within the footing, otherwise there will not be equilibrium. The self weight of the footing is taken into account by the program.

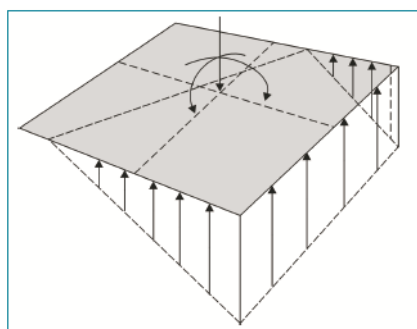


Figure 38

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: 33%
 - With seismic loads: 50%

These values are optional and may be modified.

2.13.1.2 *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.

2.13.1.3 *Concrete states*

Bending in the footing and the tangential forces are verified.

2.13.1.4 *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.

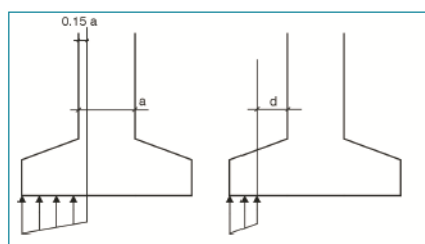


Figure 39

2.13.1.5 *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.

2.13.1.6 Reinforcement anchorage

The anchorage of the reinforcement ends is checked, providing the corresponding anchorage lengths for each case and position.

2.13.1.7 Minimum depths

The minimum depth specified by the selected code is checked.

2.13.1.8 Reinforcement spacing

The minimum spacing between reinforcement bars stated in the selected code is checked. However when designing a minimum of 10cm is taken.

2.13.1.9 Minimum and maximum steel areas

The minimum mechanical and geometric steel areas specified in the selected code are checked.

2.13.1.10 Minimum diameters

The program checks the reinforcement diameters are at least those indicated in the selected code.

2.13.1.11 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.

2.13.1.12 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. The user can optionally choose to not place the top reinforcement mesh if the design is successful with mass concrete.

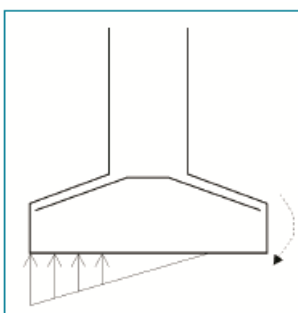


Figure 40

2.13.2 *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)

2.13.2.1 *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 introduced 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-98 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.

2.13.2.2 Sign criteria

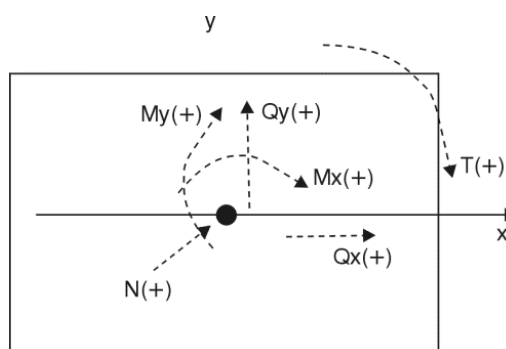


Figure 41

2.13.2.3 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 $\geq 10\%$ maximum compression
- Bending moment warning: strap beams must be provided (pile cap types A and B)
- 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 introduced, 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:**
 - 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.

2.13.3 *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 45° 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.

2.13.4 *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.

2.13.4.1 *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.

2.13.4.2 *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.

2.13.4.2.1 *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.

2.13.4.2.2 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:

$$f_{ct,d} = \frac{0.21}{1.5} \cdot \sqrt[3]{f_{ck}^2}$$

2.13.4.2.3 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:

$$\tau_{sd} \leq \tau_{rd}$$

$$\tau_{sd} = \frac{F_{sd,ef}}{u_0 \cdot d}$$

$$F_{sd,ef} = \beta \cdot F_{sd}$$

$$\tau_{rd} = f_{tcd} = 0.30 \cdot f_{cd}$$

Where:

- f_{cd} is the design compression resistance of the concrete
- F_{sd} is the axial force the support transmits to the footing
- β 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.

	β
Internal supports	1.15
Edge supports	1.4
Corner supports	1.5

- 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.

2.13.4.3 *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.

2.13.4.4 *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.

2.13.4.5 *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.

2.13.4.6 *Maximum slope angle check*

This check is similar to that carried out in the case of reinforced concrete footings.

2.13.4.7 *Check for overturning*

The overturning check is similar to that carried out in the case of reinforced concrete footings.

2.13.4.8 *Soil bearing pressures check*

The soil bearing pressures check is similar to that carried out in the case of reinforced concrete footings.

2.13.4.9 *Check for bending*

The data displayed in the check report is indicated below.

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.

2.13.4.10 Check for shear

The shear check is carried out in accordance to what has been explained in section 1.8.4.2. 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.

2.13.4.11 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*.

2.13.4.12 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.

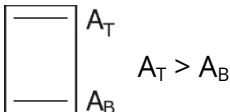
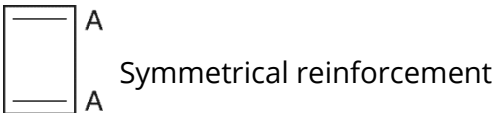
In this check, the program verifies the separation between the reinforcement axes is equal to or greater than 10cm. This value is imposed regardless of the selected code as criteria of CYPE.

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.

2.13.5 *Strap and tie beams*

2.13.5.1 *Strap beams*

The program designs reinforced concrete strap beams between foundation elements. Strap beams are using to balance the moments in footings and pile caps. Two types are available:

- Negative moments: 
- Positive moments: 

The forces acting on the strap beams are:

- Moments and shears required for their balancing action.
- Loads cannot be introduced 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.
- They support axial forces and compaction surcharges in the same way as tie beams.

If their length is less than 25cm, 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 spacing between longitudinal reinforcement
- Minimum spacing between stirrups
- Maximum spacing between longitudinal reinforcement
- Maximum spacing between stirrups
- Minimum beam width ($\geq 1/20$ span)
- Minimum beam depth ($\geq 1/12$ span)
- Check for cracking (0.3 mm)
- Minimum steel area in tension
- Minimum mechanical steel area (reductions are accepted)
- Maximum longitudinal reinforcement steel area
- 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 (15°).

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.

2.13.5.2 Tie beams

The program designs tie beams between reinforced concrete foundation elements.

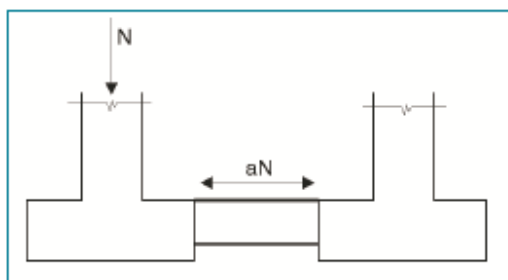


Figure 42

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 ($a \cdot N$).

The user can optionally choose to design these for bending due to a uniformly distributed surcharge of $w = 10 \text{ kN/m}$, 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 ($\geq 1/20$ span)
- Minimum beam depth ($\geq 1/12$ 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.

3 Code Implementation

CYPE 3D allows its users to analyse and design their structures in accordance to a wide variety of codes which have been implemented in the program:

<https://info.cype.com/en/product/design-codes-implemented-in-cype-programs/>

New codes are constantly being added to the existing list and so users are recommended to consult the program updates website (www.downloads.en.cype.com) for these new implementations.