User manual
Continuous beams

Continuous beams

User manual

cype

Software for
Architecture,
Engineering and
Construction

CYPE Ingenieros, S.A.
Avda. Eusebio Sempere, 5
03003 **Alicante**
Tel. (+34) 965 92 25 50
Fax (+34) 965 12 49 50
cype@cype.com

www.cype.com

This manual corresponds to the software version indicated by CYPE Ingenieros, S.A. as Continuous beams. The information contained in this document substantially describes the properties and methods of use of the program o programs accompanying it. The information contained in this document could have been modified after its mechanical edition without issuing a warning. The software accompanying this document can be submitted to modifications without issuing a previous warning.

CYPE Ingenieros, S.A. has other services available, one of these being the Updates, which allows the user to acquire the latest versions of the software and accompanying documentation. If there are any doubts with respect to this text or with the software User License or for any queries, please contact CYPE Ingenieros, S.A. by consulting the corresponding Authorised Local Distributor or the After-sales department at:

Avda. Eusebio Sempere, 5 – 03003 Alicante (Spain) • Tel : +34 965 92 25 50 • Fax: +34 965 12 49 50 • www.cype.com

CYPE Ingenieros, S.A.

Edited and printed in Alicante (Spain)

1. New job assistant

The program will, with the help of an assistant, request all the data required to create the project. First of all, it will ask for the name and description of the new project.

1.1. Design codes

The code to be applied by the program for each material must be selected:

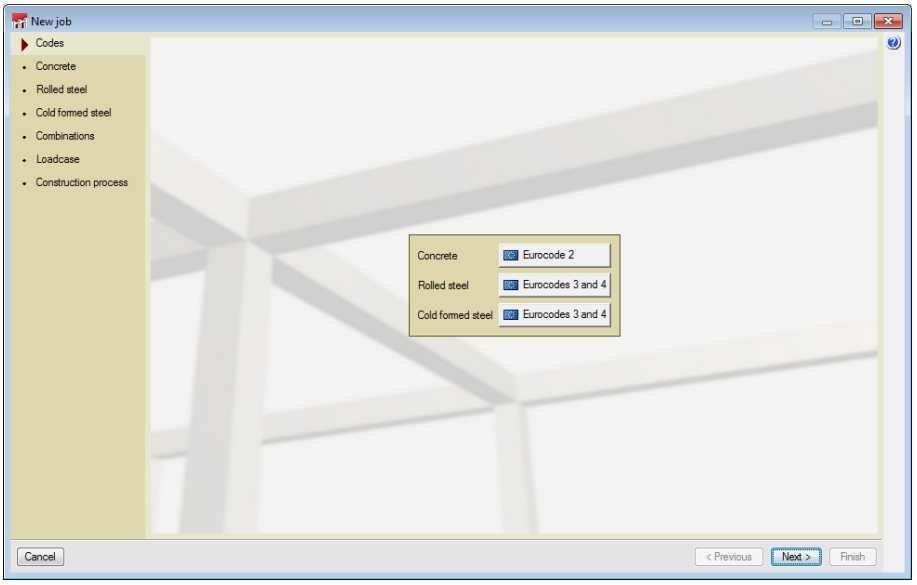

Fig. 1.1

1.2. Concrete

Click on **Next**, and in the **Concrete** section, indicate the type of concrete to be used for the frames, the maximum aggregate size and the steel to be used at each position (top assembly reinforcement, top additional reinforcement, bottom reinforcement etc.).

The exposure class must be indicated in the **Environment** tab and the program will display the maximum value for the crack width that will be considered in the checks.

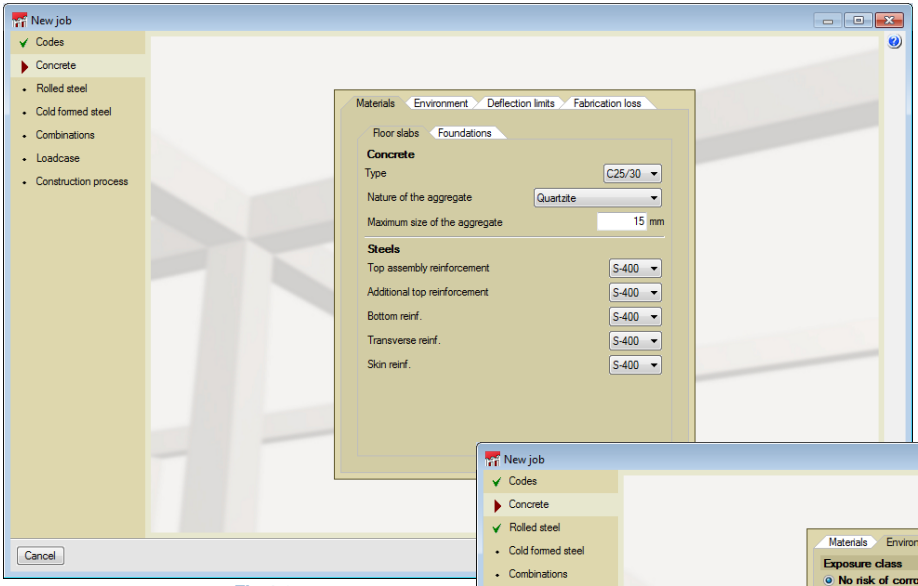

Fig 1.2

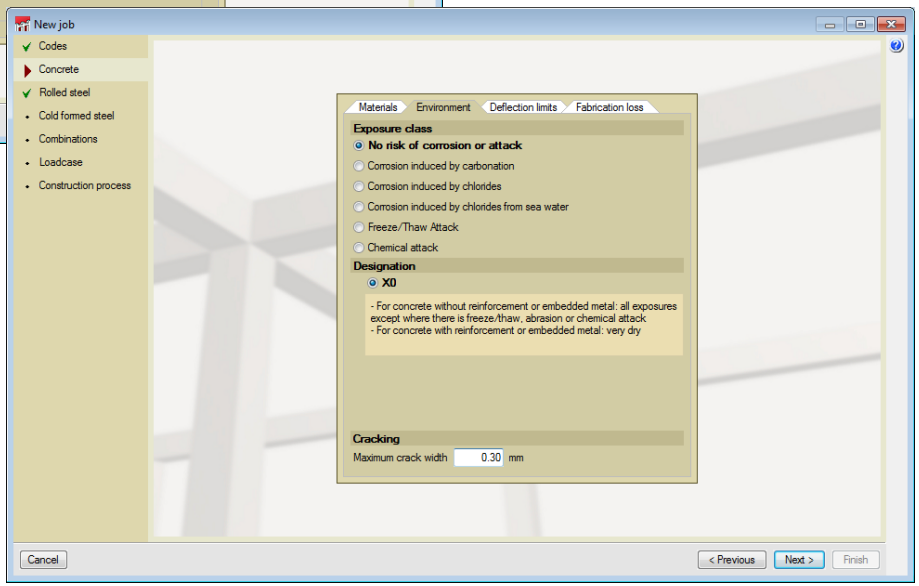

Fig 1.3

In the **Deflection limits** tab, the limits to be taken into account in the analysis are indicated. These are the instantaneous deflection, long-term deflection and long- term active deflection. For the long-term deflection, the construction period for the damageable elements must be specified.

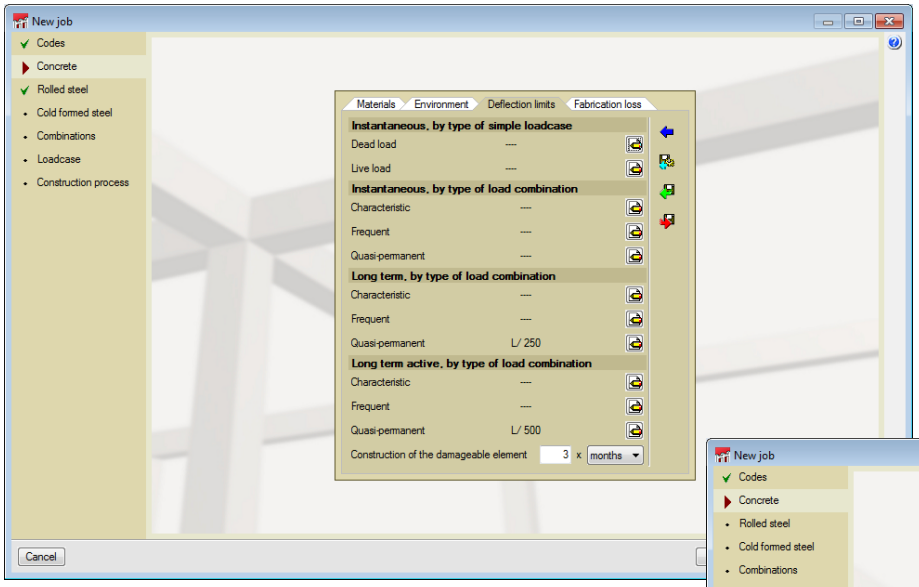

Fig 1.4

A button is available to import predefined values for the code, another to save the defined values and another to import them from another project.

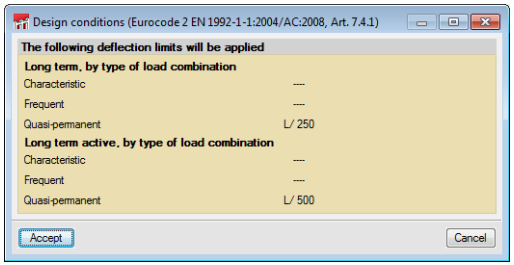

Fig 1.5

Finally, in the **Fabrication loss** tab, indicate the percentage steel to be added in the take-off reports, due to cuts and trims.

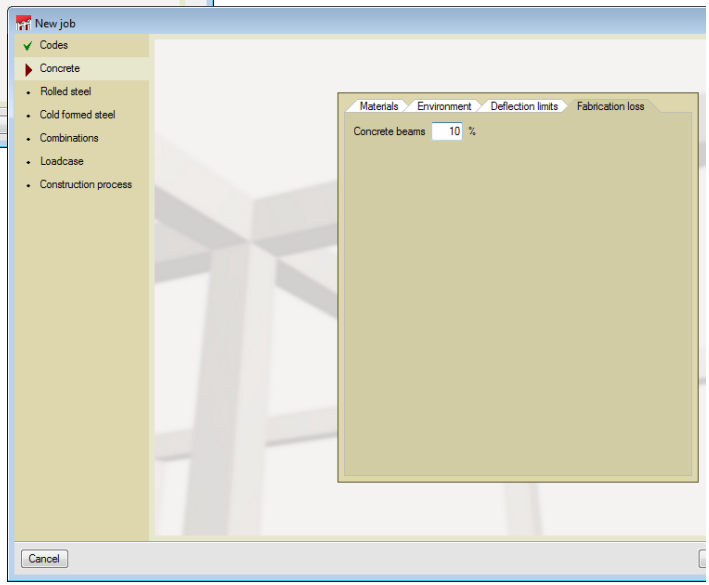

Fig 1.6

1.3. Rolled and cold-formed steel

As was done for the concrete section, in these two steps, the steel for the rolled and cold-formed steel sections is to be selected as well as the deflection limits for these materials. The fire resistance check can be activated in the **Fire situation** tab, and users can indicate the required resistance and type of protection coating.

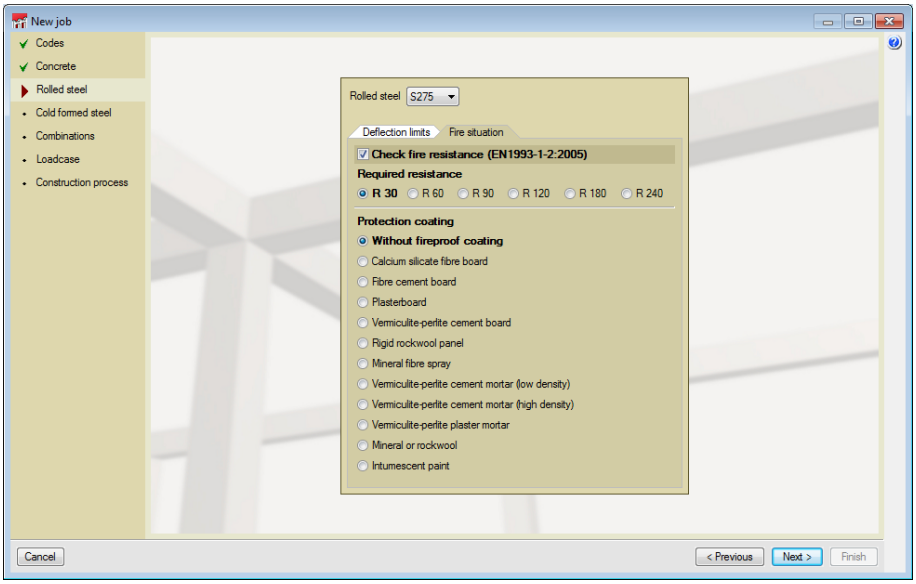

Fig 1.7

1.4. Combinations

The combinations carried out by the program can be consulted in this section.

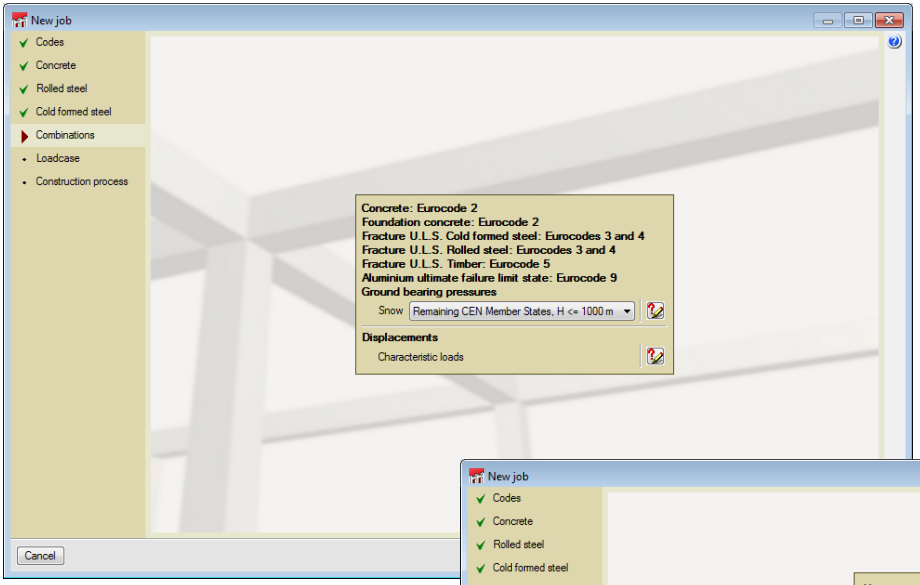

Fig 1.8

1.5. Loadcases

The different loadcases the frames will have are specified in this window.

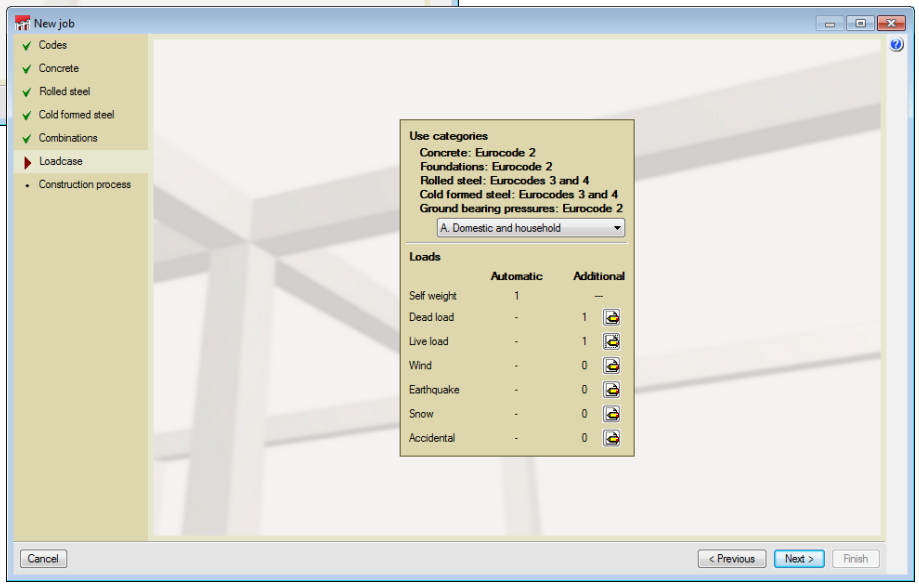

Fig. 1.9

1.6. Construction process

In this window, user can define the load percentage and when it is to be applied for the loadcases defined in the previous window.

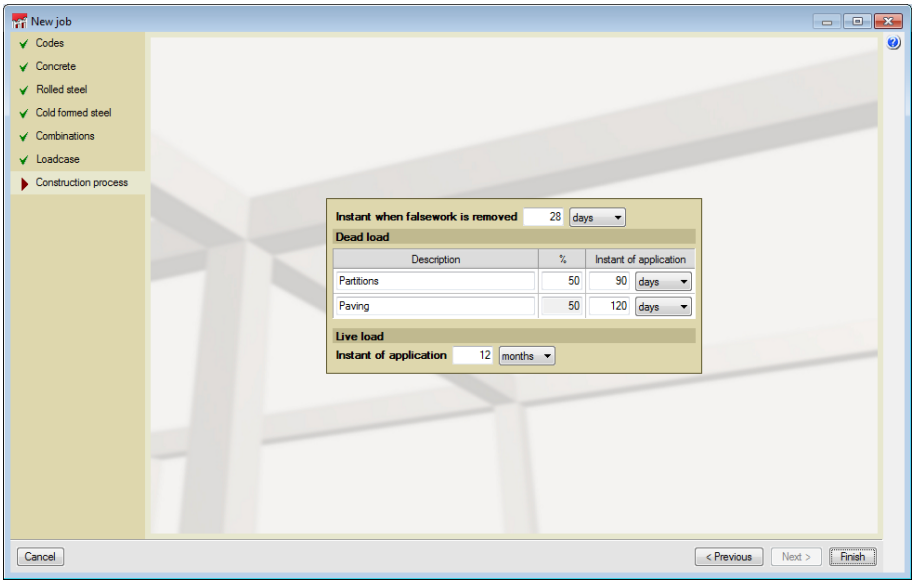

Fig 1.10

2. Program main screen

Once the assistant data has been completed, the main screen of the program is displayed. The next step will be to create the frames of the project. To do so, press the **Frames of the job button** , located in the top toolbar. A window will open displaying the empty list of frames included in the project. By clicking on **Add** , the assistant opens to create a frame with equal beams.

2.1. Assistant for creating a frame with equal beams

This assistant helps users introduce the basic data of the frame. A frame can be defined in three steps. The data defined can be modified later on to adapt it to the real frame of the job.

2.1.1. Description

The reference of the frame, number of beams it contains and the free span of the beams are defined here.

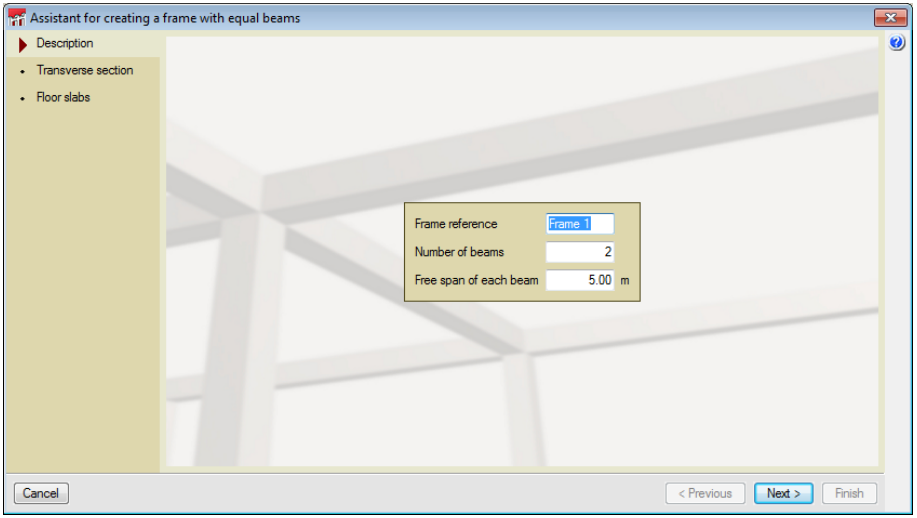

Fig 2.1

2.1.2. Transverse section

Users must select the beam family, then within this family, the type of beam to be introduced, specifying its dimensions.

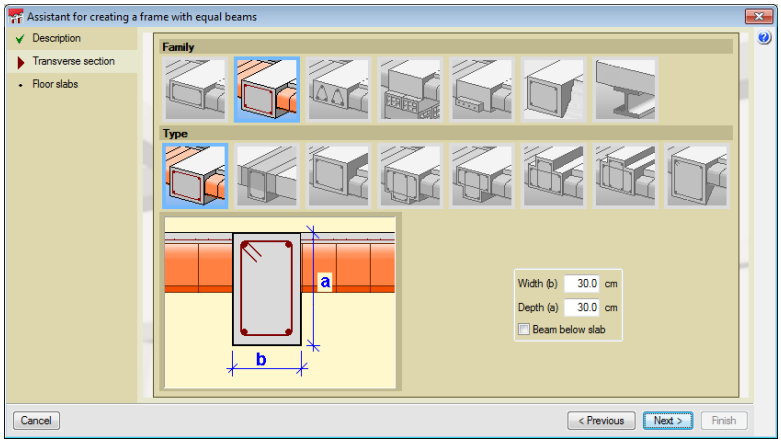

Fig 2.2

2.1.3. Floor slabs

Here, indicate whether or not there is a floor slab, its depth and how the floor slab reaches the beam (elevation change, sloped, etc.)

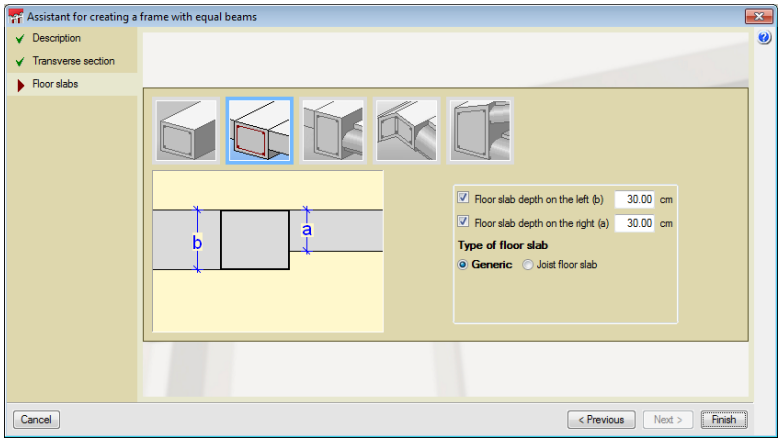

Fig 2.3

Once this assistant has finished, the Frames of the job window can be accepted and the basic geometry of the frame will be defined.

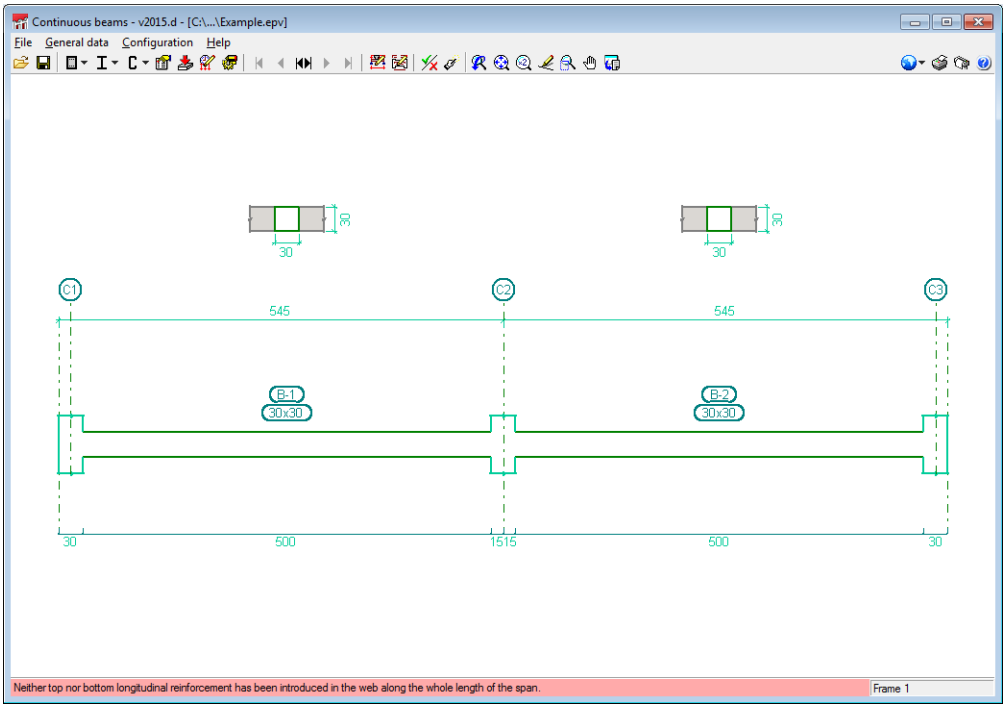

Fig 2.4

Once the equal beam frame has been created using the assistant, the generated frame will appear on the main screen of the program.

2.2. Description of the selected frame

By pressing the **Selected frame description** button , a window will open containing the options required to complete the geometry and loads conditions of the frame.

2.2.1. Add beams to the frame

Upon pressing this option, a window is displayed where the type of beam, its dimensions, the floor slab next to this new beam, free span, supports and elevation differences at either end have to be defined.

Once the type and its dimensions have been defined, indicate whether the new beam is added to the left or to the right of the selected beam. Each time a beam of the frame is selected, the introduction will repeat itself. If you wish to change the type of beam to be introduced, click on the secondary button of the mouse.

2.2.2. Transverse section

The geometry of the transverse section of the selected beam can be modified here.

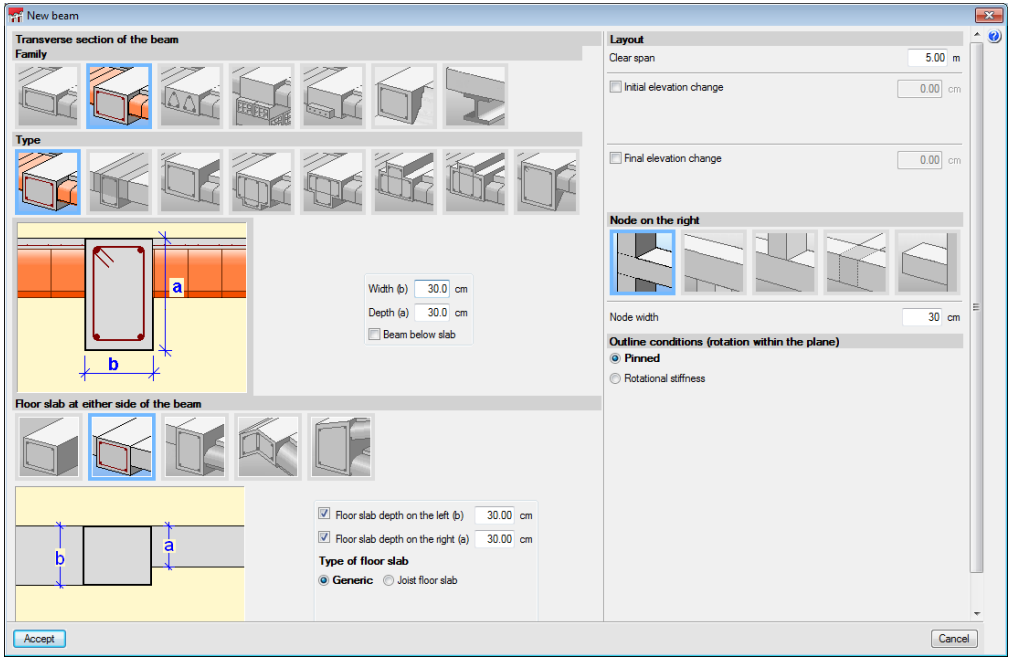

Fig 2.5

2.2.3. Beam layout

Here the free span and elevation difference at the ends of the selected beam can be defined and state whether this elevation change is with respect to the internal surface of the column or with respect to its axis.

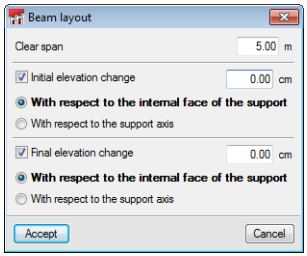

Fig 2.6

2.2.4. Floor slabs

Allows users to alter the type of floor slab reaching the selected beam. This option modifies the geometry of the beam if a flat beam has been selected, as it adapts itself to the geometry of the floor slab.

2.2.5. Node description

Allows users to modify the type of support and the view of the selected node. Here, users will have to indicate whether it is a continuous column, last floor column, column not starting at the foundation, secondary beam support or overhang. Similarly, its reference and dimensions will have to be indicated as well as the support conditions to be used in the analysis.

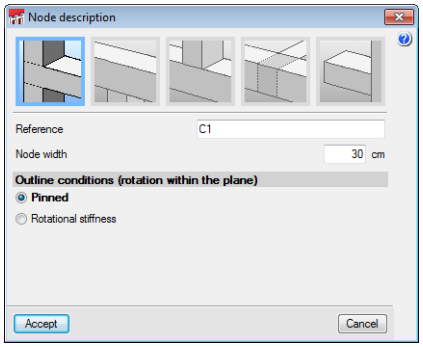

Fig 2.7

2.2.6. Copy beams

Using this option, users can select a beam and specify what is to be copied from it, then assign the data to selected beams.

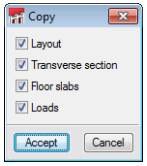

Fig 2.8

2.2.7. Divide beams

This option allows users to divide a beam at an indicated point. The program places the type of support the initial beam has to its right as the type of support at this point.

2.2.8. Delete beams

This option deletes the selected beam.

2.2.9. Fire situation

Using this option, users can specify, for one or several beams of the frame, different parameters to the general parameters, for the fire resistance check. This option is only active once the steel beam frame has been defined and the fire check option has been activated.

2.2.10. Viewed loadcase

Allows users to select the loadcase to be viewed on screen.

2.2.11. Introduce loads on beams

The beams of the frame to which the same loading is to be applied must be selected. Once the beams have been selected, click on the secondary mouse button and the “Load description” window will open. Here, specify the loadcase to which the load belongs, the type of load (point, line, etc.), and the value of the load. Also specify if the load is introduced in the direction of the global axes (X horizontal, Z vertical) or local axes (X direction of the beam, Z perpendicular to the beam).

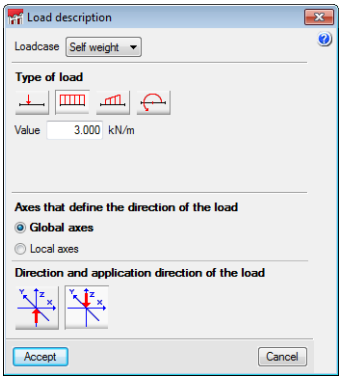

Fig 2.9

2.2.12. Introduce loads on nodes

This option will only be active if there are nodes due to secondary beams, columns starting on beams or overhangs. If one or several of the previously described nodes are selected followed by a click with the secondary button of the mouse, the “Load description” window will open, in which the value and direction of the point load are to be indicated.

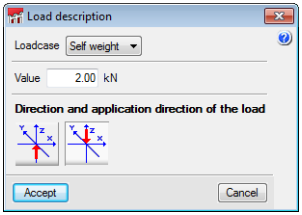

Fig 2.10

2.2.13. Edit loads

Allows users to select loads on nodes or on beams so they can be modified.

2.2.14. Delete loads

Users can select several loads, on beams and on nodes. Once selected, click on the secondary mouse to confirm they are to be deleted.

Now that all the geometry modifications have been carried out and all the loads defined, click on the **Accept** button located in the bottom right-hand corner of the "Frame description" window, to return to the main screen of the program.

2.3. Design

At this point, the frame can be designed by pressing on the **Design** button . If there is more than one frame in the project, a window will open in which you will have to indicate whether the "Selected element" (current frame) is to be designed or "All the elements" (frames).

2.4. Check

Once the design has concluded, the selected frame can be checked. The program will display a report detailing all the checks that have been carried out for each span in the worst case zones, and at zones where there is a change in the sign of the bending moments. The report also indicates whether the check meets or fails any code or user requirements.

2.5. Edit the selected frame

Using this option, users can modify the reinforcement of the selected frame and consult its corresponding force diagrams.

This window consists of a toolbar in the top area of the screen and a lateral menu to configure views on the left.

2.5.1. Toolbar

This toolbar contains the following buttons to modify and consult the reinforcement:

2.5.1.1. Save

Allows users to save the modified data without having to close the project.

2.5.1.2. Longitudinal reinforcement

When this option is activated, a window appears containing all the options required to modify the longitudinal reinforcement.

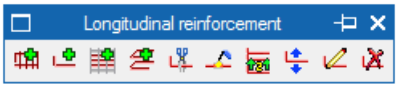

Fig 2.11

Introduce assembly reinforcement

The assembly reinforcement consists of a number of standard diameter bars placed along the top of the beam, a number of bars along the bottom of the same or other diameter and stirrups.

To introduce the assembly reinforcement, select the beam in question or the first and last nodes of the assembly reinforcement cage, as long as these nodes are aligned.

3 reference lines are displayed at the nodes for the introduction (axis of the node and the surfaces of the node minus the cover).

Once the beam or beams for which assembly reinforcement is to be introduced have been selected, the window in which it can be defined will open.

The assembly reinforcement can be defined in two ways:

1. The first way is to introduce the top and bottom reinforcement (e.g. 3ø10 at the top and 3ø12 at the bottom).

   Once the longitudinal reinforcement has been defined, the stirrup arrangement must be edited. To do so press the  button.

   The “Transverse reinforcement” window will open.

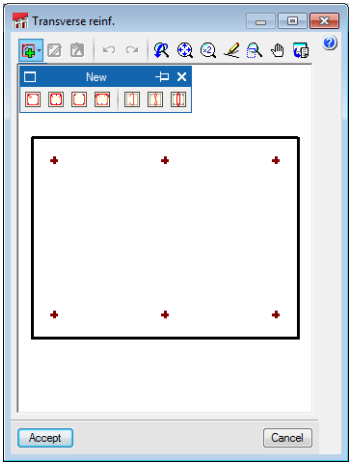

Fig 2.13

Click on the **New transverse reinforcement** button and select the type of transverse reinforcement to be defined. To introduce the reinforcement, place the cursor in the drawing area, and select the longitudinal bars the stirrup will surround. These will be displayed in a cyan colour. Confirm the selection by pressing on the main mouse button. Then, select the remaining longitudinal bars which are to have transverse reinforcement.

Once the transverse reinforcement has been defined, accept the window and finish defining the assembly cage by defining the anchorage bars at the top ends. To do so, click on the end of each bar with the mouse button.

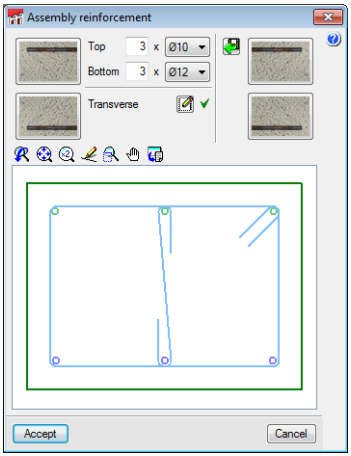

Fig 2.14

2. The other way in which the reinforcement can be defined is by selecting one of the reinforcement configurations from the reinforcement table incorporated by the program. Instead of defining the reinforcement, click on **Import reinforcement arrangement** from the table and the program will display the reinforcement table. Select a type and stirrup arrangement.

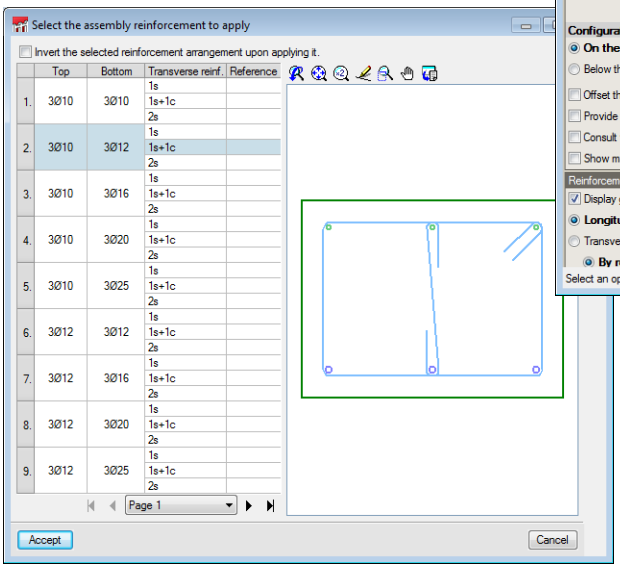

Fig 2.15

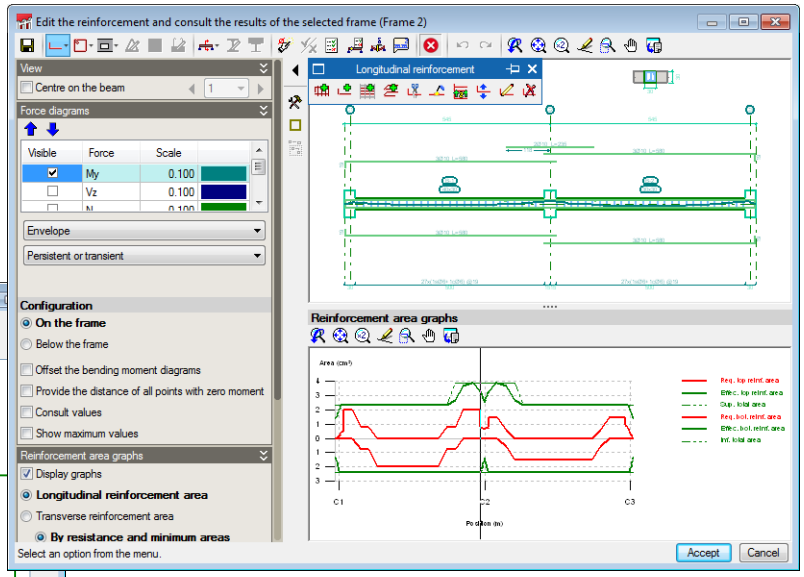

Fig 2.16

By accepting the dialogue box, the reinforcement is introduced in the selected beam. If other beams of the frame are selected, the reinforcement will also be introduced if these are the same type of beam. If the geometry were to be different, the dialogue will appear for the reinforcement to be defined.

Introduce reinforcement

Upon activating this option, a window appears in which the reinforcement to be introduced can be configured, as well as the number of bars and their diameter. Users can also indicate whether they are anchored or not and specify the layer at which they are to be introduced.

This configuration is defined for the top reinforcement and bottom reinforcement.

To introduce the reinforcement, click, using the main mouse button, at the start of the reinforcement bar, validate its distance with respect to the nearest node, move the mouse in the direction of the bar and press the mouse button to establish the end point.

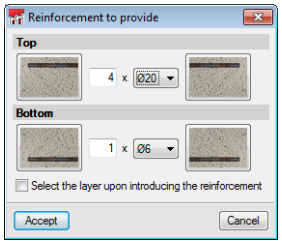

Fig 2.17

Introduce skin reinforcement

Once this option has been selected, click on, using the main mouse button, the main beam where the skin reinforcement is to be introduced. The program will display a dialogue box where the number of bars at each surface and their diameter can be selected.

Introduce stitching reinforcement

Having selected this option, click on, using the main mouse button, the end of the beam where the stitching reinforcement is to be added, then, move the cursor towards the inside of the span and click on the main mouse button again. The program will display a dialogue box where the number of bars, their spacing and anchorage lengths can be indicated.

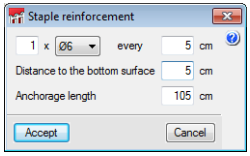

Fig 2.18

Introduce overlaps

Using this option, users can indicate where bar overlaps are to be located. To do so, a bar must be introduced preciously and then, using this option, divide it at the point in question. This way, the program will be able to verify if these splices comply with the requirements stated in the selected code.

To create an overlap, bring the cursor near a bar; it will be displayed in cyan. Then, click on the point where the splice is to be located. A window will open where the arrangement of the splice, its length and distance from the origin of the bar can be defined.

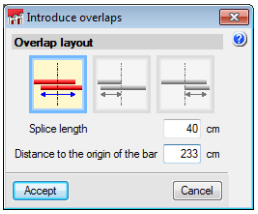

Fig 2.19

Join

Using this option, reinforcement bars of the same type and diameter can be joined. To do so, activate the option, select the first pack of bars to be joined followed by the second pack. If the packs contain a different number of bars, the program will ask how many bars are to be joined.

Divide

This option allows user to divide a pack of bars and create two separate packs. Click on the pack of bars to divide and specify the number of bars that are to be separated to form a new pack.

Place in another layer

Users can choose the layer at which a reinforcement pack is to be positioned. To do so, select the bar and a window will open indicating the current number of available layers in the beam. Now specify the new layer at which the bar is to be placed.

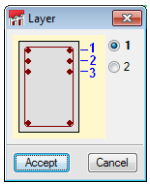

Fig 2.20

Edit

This option allows users to edit all the parameters that have been introduced for a bar (length, diameter, number of bars, etc.). By clicking at the end of a bar, the minimum anchorage length will be provided and, if the end already has a hook, its configuration window will open.

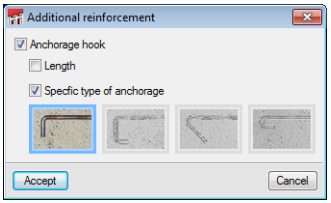

Fig 2.21

The specific length of the hook and type of anchorage hook can be modified in this window.

If the cursor is placed next to the end of a reinforcement bar, a double arrow will appear. If it is selected, the length of the bar can be increased or decreased by moving the cursor, and the program will simultaneously indicate the total length of the bar.

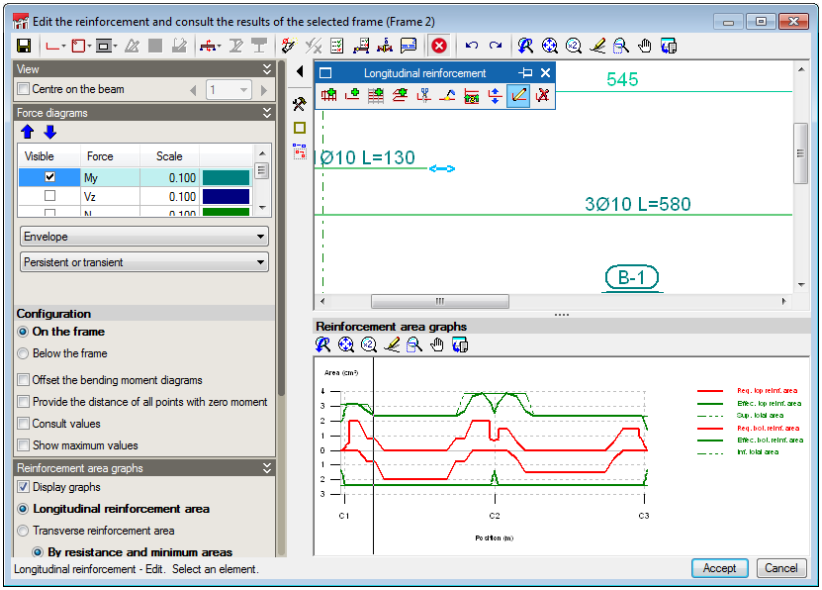

Fig 2.22

Delete

Using this option, users can delete the reinforcement selected with the mouse cursor.

2.5.1.3. Transverse reinforcement

When this option is activated, a window opens containing all the transverse reinforcement modification options.

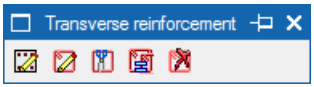

Fig 2.23

Edit template

If no base reinforcement has been defined for the beam, this option allows users to indicate the number of bars that will be tied by the stirrups of the span. Upon activating this option, the beam should be selected to define the stirrup arrangement and indicate the number of bars it contains.

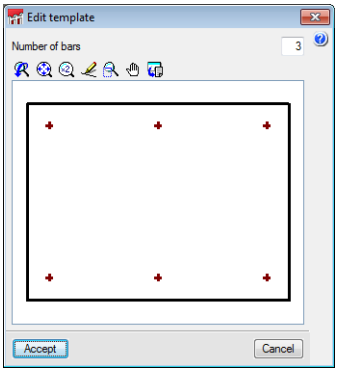

Fig 2.24

Once the number of bars of the template has been defined, the geometry of the stirrups is established.

Edit reinforcement

The geometry of the stirrups is defined in the same way as was explained for “Introduced assembly reinforcement”.

Divide reinforcement spans

Using this option, concentrated stirrup zones can be created as well as change their diameter. To do so select the point at which the stirrups are to be divided in the stirrup representation zone located below the frame. Validate the length of the span.

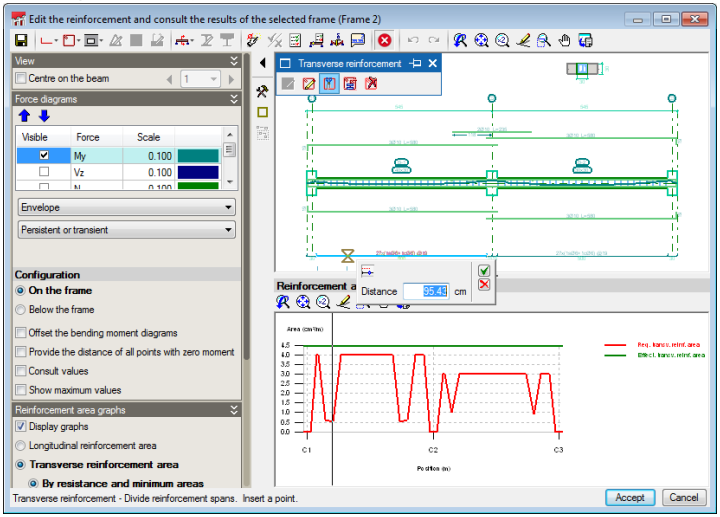

Fig 2.25

Match reinforcement spans

Applies the diameter or spacing of the selected reinforcement to the reinforcement selected afterwards.

Delete reinforcement spans

This option allows users to delete the reinforcement bars defined in a beam.

2.5.1.4. Transverse sections

Allows users to introduce sections at any point of the span. The sections can be defined by a single point or by two

points if top reinforcement is to be provided in a zone and bottom reinforcement in another. To recover the automatic sections generated by the program, use the **Generate transverse sections** option.

2.5.1.5. U.L.S. and S.L.S. checks at the worst case point

Upon activating this option, select the span for which the ULS and SLS checks at the worst case point are to be displayed.

Whilst this option is active in the View menu (2.5.2), the deflections produced in the continuous beam can be consulted.

2.5.1.6. U.L.S. and S.L.S. checks at a point

Having activated this option, select the point on the span where the ULS and SLS checks are to be displayed.

Whilst this option is active, the deflections produced in the continuous beam can be consulted in the View menu (2.5.2).

2.5.1.7. Display error messages

Using this option, users can view the errors not allowing for the frame to be designed or checked.

2.5.2. View configuration menu

This menu is situated to the left of the drawing area and contains the option **Centre on the beam** to aid users in consulting the spans of the beam. If the option is activated, users can move along the spans of the frame by pressing the next or previous buttons, or a specific frame can be selected.

Below the Centre on the beam option, the view options are located, with which users can choose which force diagram is to be viewed, as well as their scales. These diagrams can be displayed for each selected loadcase, combination or envelope.

The force diagram configuration option allows for force diagrams to be drawn on the frame or below it. The bending moment diagram can be seen on the frame or below it, and users can mark the distance to the null bending moment points. By activating the **Consult values** option, the values are displayed as a tag next to the cursor when using the options to add reinforcement bars or divide stirrup zones.
Whilst the ULS/SLS check options are selected, the options to view the deflections appear and can be consulted in the same way as the force diagrams.

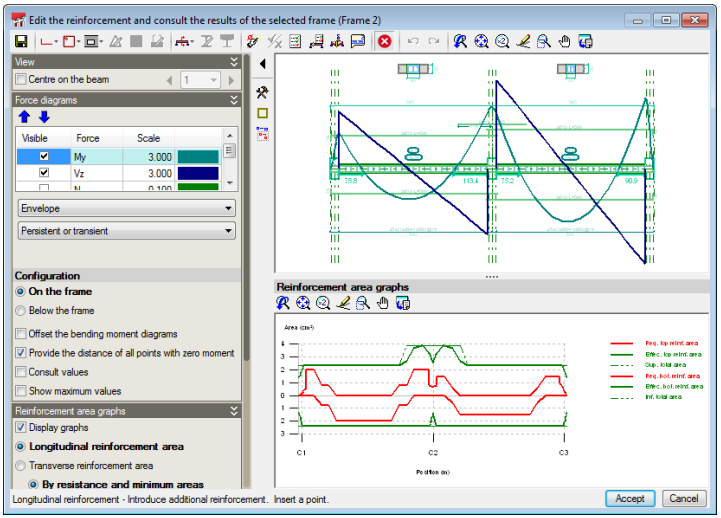

Fig 2.26

In the bottom part of the view menu, users can configure the view of the reinforcement area graphs.

The reinforcement area graphs are displayed in two colours: the red graph corresponds to the required steel area in cm2, and the green graph corresponds to the reinforcement that has been provided in the beam. The graph is updated when the reinforcement is edited, stretched or increased.

The worst case points or points where the steel is being used to a maximum are those where the provided reinforcement graph (green) is closest to the required reinforcement graph (red).

2.6. General data menu

This menu is located on the top toolbar of the main screen of the program and contains a series of options where users can modify materials, the environment, deflection limits and fire resistance for each type of material. The combinations and loadcases can be modified for the deflection loadcases.

2.6.1. Options

When this option is selected, a window is displayed in which the reinforcement tables, the reinforcement arrangement, force options and design options can be modified.

2.6.2. Reinforcement tables

2.6.2.1. Assembly reinforcement tables

The program interprets the assembly reinforcement as being the reinforcement that forms the base reinforcement cage that will be taken to the job. Users have to indicate the number of bars and their diameters for the top and bottom reinforcement, and the stirrup arrangement. A reference can be given to each assembly reinforcement configuration.

For each stirrup arrangement, users can indicate the range of lengths of the beam that are to contain the arrangement.

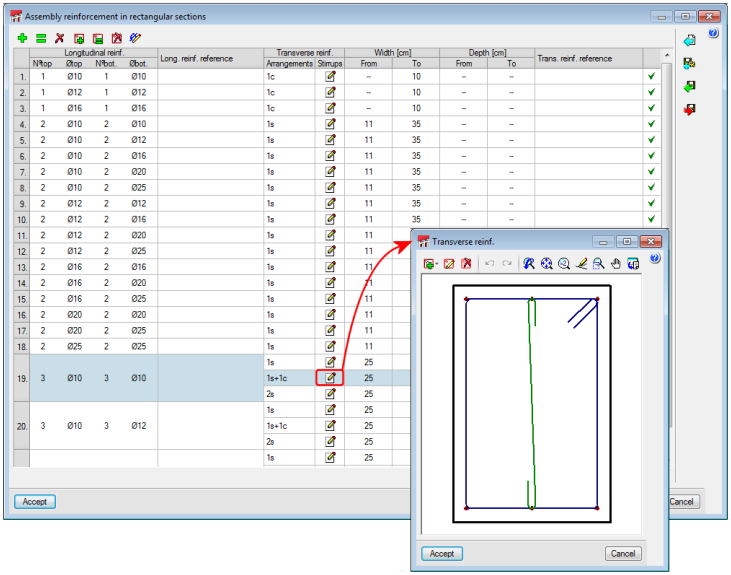

Fig 2.27

2.6.2.2. Transverse reinforcement tables

The diameters available for selection can be configured as well as the spacing range to be used during the design.

2.6.2.3. Stirrup-carrying and skin reinforcement tables

The available diameters can be configured and indicate the assembly reinforcement in the flanges.

2.6.2.4. Top and bottom reinforcement tables

The available diameters for this type of reinforcement can be specified in this type of tables, as well as the compatibility between the assembly reinforcement and the first reinforcement bar, the second reinforcement bar and the first reinforcement bar. Users can also indicate the minimum spacing between bars, so that the program will find reinforcement combinations with larger diameters, so the limit is verified.

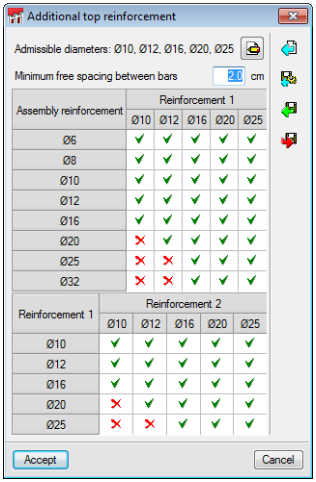

Fig 2.28

2.6.3. Reinforcement arrangement options

A series of options are included in this section where users can specify the reinforcement arrangement. These are:

2.6.3.1. Cover for beams and foundation beams

The top, bottom and side covers are specified here.

2.6.3.2. Separation between longitudinal reinforcement layers

The free distance between the reinforcement layers can be indicated. The program will always respect the minimum bar spacing indicated in the selected code.

2.6.3.3. Stirrup selection

Users can indicate how to arrange the transverse reinforcement inside nodes (columns or secondary beams).

2.6.3.4. Anchorage length for stirrup closings

The closing length of stirrups can be indicated depending on their diameter.

2.6.3.5. Lattice beams

Defines the properties of the lattice beams.

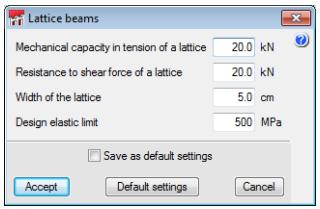

Fig 2.29

2.6.3.6. Path of the longitudinal reinforcement anchorage

By selecting this option, the anchors adapt to the geometry of the frame.

2.6.3.7. Type of anchorage

Allows users to select the default type of anchorage hook the program will place.

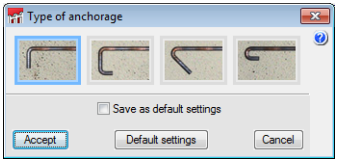

Fig 2.30

2.6.3.8. Prestressed beam library

Allows users to create a library of prestressed beam families. By pressing the New button, a window opens in which the data defining the prestressed beam family can be introduced.

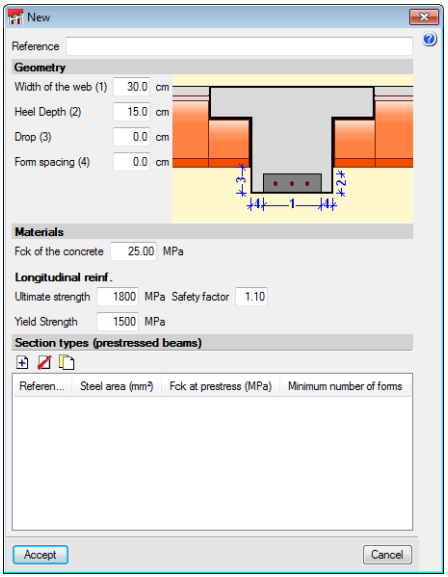

Fig 2.31

2.6.3.9. Rounding the bar lengths

Even though the program calculates the length required for each bar, it may be convenient to round-off the excess lengths to multiples of a specific length to simplify the execution process.

2.6.3.10. Maximum length of a bar

The program, when designing the continuous beam, will respect the value introduced in this dialogue box and create reinforcement overlaps so this length is not exceeded. If a bar with a length exceeding this maximum value is introduced, the program will highlight it with a yellow triangle.

2.6.4. Force options

This section contains all the force result options.

2.6.4.1. Minimum moments to cover with reinforcement in beams

This option allows for a minimum moment to be considered to reinforce the section. By activating it, the fraction of the WL2 to be considered as the minimum moment is indicated, and can be done so for positive moments and negative moments.

2.6.4.2. Coefficients to multiply forces

Using this option, users can introduce a value which increases the force diagrams. This coefficient can be applied only to the maximum values or to the entire diagram.

2.6.4.3. Force redistribution coefficients

This option allows users to redistribute the negative moment at the supports, in such a way that the negative moment will be reduced by the indicated coefficient and the value of the positive moments will be increased.

2.6.5. Design/Code check options

All the options that can be used to design the frames are in this section.

2.6.5.1. Beam reinforcement

This option is used to indicate the minimum lengths of each reinforcement pack depending on whether the reinforcement is at the support or in the span. The maximum length of the top reinforcement can be indicated, and if it is exceeded the top assembly reinforcement would be increased.

The option: Anchor at support a reinforcement equal to shear, is available which increases the bottom reinforcement at the support.

The maximum aggregate size and vibrator diameter are also specified.

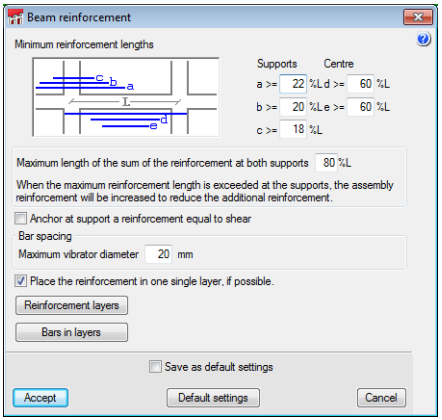

Fig 2.32

2.6.5.2. Assembly reinforcement

In this option, users indicate how the assembly reinforcement is to be considered during the design of the beam. If users select the base reinforcement as collaborating reinforcement, the required steel area to be covered by the assembly reinforcement can be specified.

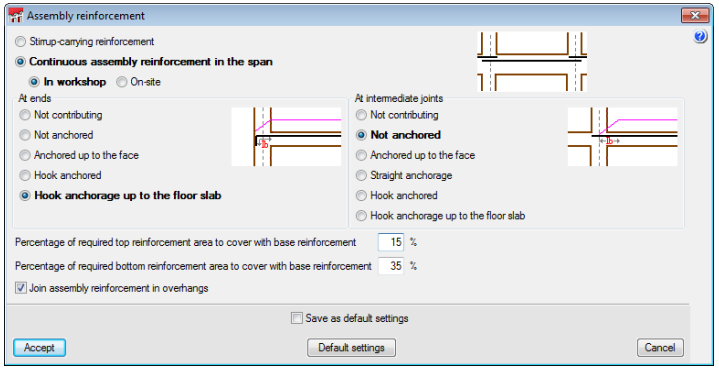

Fig 2.33

2.6.5.3. Top reinforcement

This option contains the options that manage the top reinforcement.

Symmetrical top reinforcement in single span beams
This option simplifies the execution of the beam on site and reduces the risk of errors occurring during its execution.

Percentage difference for symmetrical top reinforcement
The program, using the force diagrams obtained, calculates the bar lengths at either side of a node. This may complicate its execution on site and not allow for the control tasks to be undertaken as easily. Using this option, a maximum percentage difference can be indicated so that, as of this difference, the top reinforcement will not be symmetrical.

Anchorage length
The length of the top reinforcement anchorage hooks to be generated can be indicated here.

2.6.5.4. Bottom reinforcement

This option contains all the options that manage the bottom reinforcement.

Hooks at end of alignment
By activating this option, the program will always place hooks at the ends of the frames, whether they are necessary or not.

Anchorage length
Users can indicate the length of the anchorage hook to be generated for the bottom reinforcement during the design.

2.6.5.5. Transverse reinforcement

Minimum length of stirrups
Users can indicate the minimum reinforcement length to be placed if transverse reinforcement is required.

Symmetry in stirrup reinforcement
If this option is activated, the transverse reinforcement will always be symmetrical, simplifying the execution of the stirrup cage on site and avoiding errors.

Stirrups of different diameter in a beam
If this option is activated, the program can combine different shear reinforcement diameters in a beam during the design.

Tie all longitudinal bars
This option generates a reinforcement arrangement in which all the longitudinal bars are tied using crossties or stirrups.

Multiple stirrup layout
Users can choose to place perimeter stirrups and smaller internal stirrups, or place the same size overlapping stirrups.

Section for the shear check
Allows for shear checks to be carried out a closer to the support, compared with that indicated in the design code.

2.6.5.6. Stirrup-carrying reinforcement

The various transverse reinforcement arrangements, which depend on the dimensions of the beams, can be defined using these options. The program will use these reinforce-

ment arrangements as templates when designing the reinforcement of the beams using stirrup carrying reinforcement.

The reinforcement arrangements have to be created for each type of beam (rectangular, rectangular section with one flange or rectangular section with two flanges), and the number of longitudinal bars of the web and flange must be indicated. The geometry of the stirrup can be defined by pressing the **Layout** button, then in **Application**, indicate the dimension range for which this reinforcement is to be used.

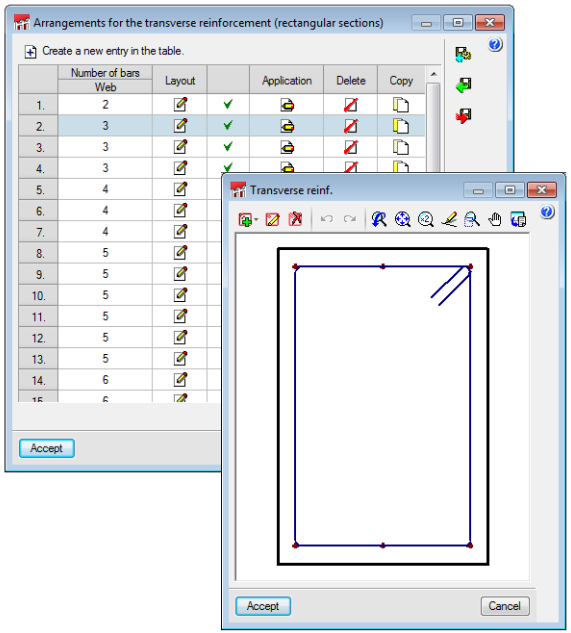

Fig 2.34

2.6.5.7. Skin reinforcement

The depth as of which skin reinforcement is to be placed is indicated in this option. The maximum spacing of the reinforcement is also to be indicated. The program will then place the reinforcement the reinforcement with the spacing stated in the selected code or the value provided by users, whichever is the smallest.

2.6.6. Concrete, rolled steel and cold-formed beams

In this section, users can modify the values that were indicated in the introduction assistant (1.2. Concrete and 1.3 Rolled and cold-formed steel).

2.6.7. Combinations

Users can modify the values that were indicated in the introduction assistant (1.4. Combinations).

2.6.8. Loadcases

Users can modify the values that were indicated in the introduction assistant (1.5. Loadcases).

2.6.9. Construction process

Users can modify the values that were indicated in the introduction assistant (1.6. Construction process).

3. Job reports

By pressing on the reports option, reports of the frames of the job can be obtained as well as a list of the fabrication tags used.

3.1. Job reports

By pressing this option, a window opens displaying a tree diagram containing all the chapters that can be included in the report. If there are any that are not required, users can uncheck the box situated to the left of the chapter for it not to appear.

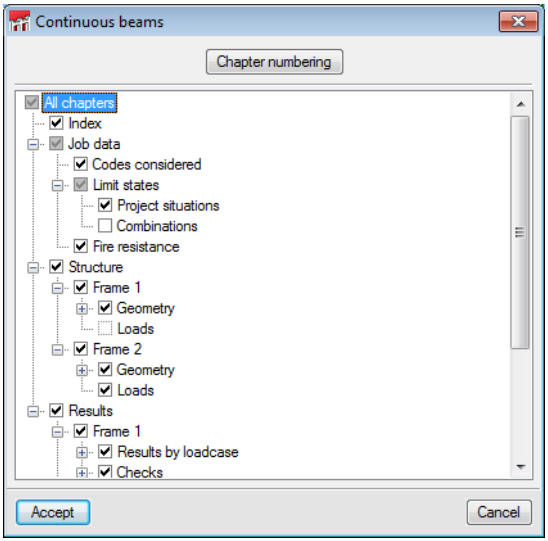

Fig 3.1

3.2. Fabrication tag list

Using this option, a report is generated in which all the reinforcement diagrams of each continuous beam span are organised. The scale and size of the tag can be configured, as well as specify the bar display configuration.

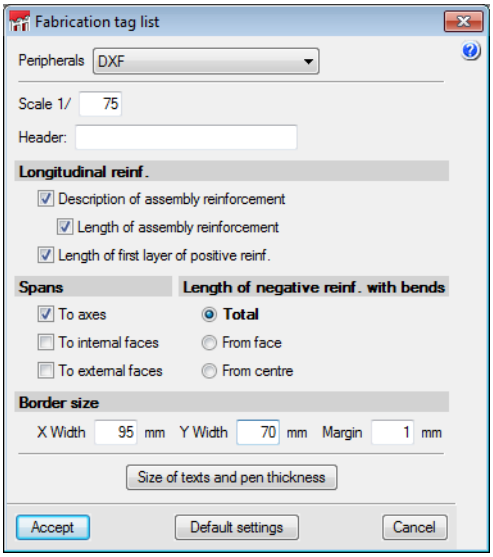

Fig 3.2

4. Drawings

By pressing on the drawings option  , a window is displayed in which, having selected the  option, users are to choose which frames are to be drawn, their scale and the scale of the section.

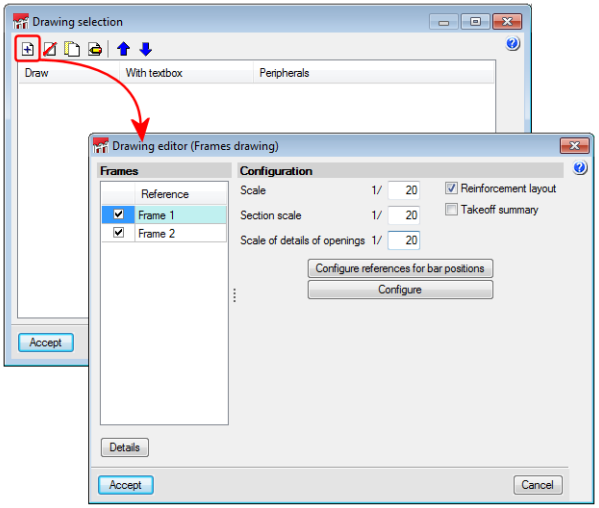

Fig 4.1

Using the **Configure** button, the drawings can be adapted to user preferences and construction details (user details or from the CYPE library) can be added to the drawing by pressing the **Details** button. To add details from the CYPE details library, press the  button within the details window. The user library will be displayed, where details can be selected or new ones can be added to the library.

Once the drawing has been configured, the peripheral to be used for the drawing must be selected. Click on **Accept** and the program will display the drawing in the “Drawing composition window”, where it can be viewed before it is printed.

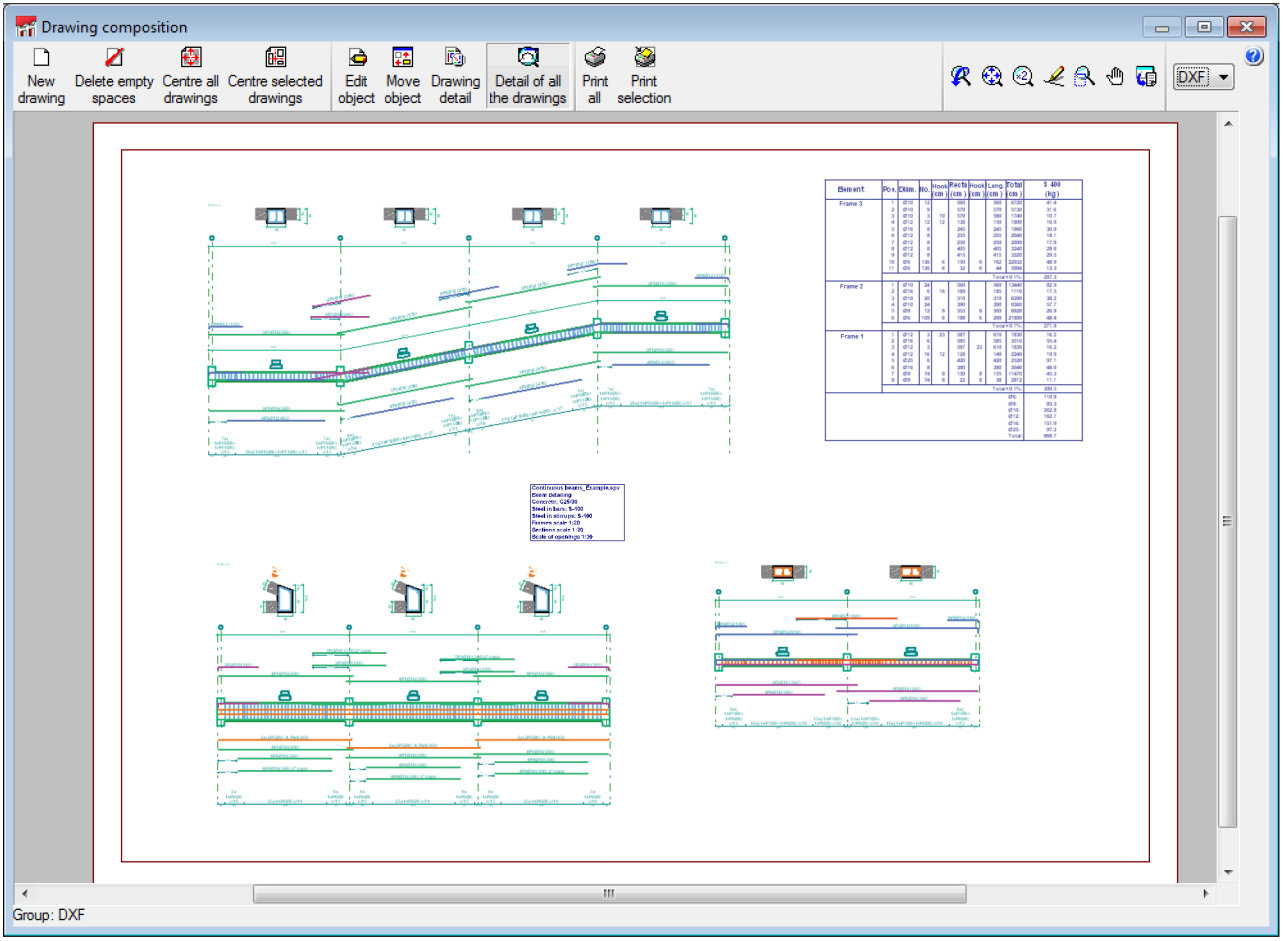

Fig 4.2