

Practical examples
CYPECAD

CYPECAD

Practical examples



**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

IMPORTANT: PLEASE READ THE FOLLOWING TEXT CAREFULLY.

The information contained in this document is property of CYPE Ingenieros, S.A. and cannot be reproduced nor transferred partially or completely in any way or by any means, be it electrically or mechanically, under any circumstances, without previous written authorisation of CYPE Ingenieros, S.A. Copyright infringement may be constituted as a crime (article 270 and onwards of the Penal code).

This document and the information it contains form an integral part of the documentation accompanying the User License of CYPE Ingenieros S.A. programs and cannot be separated. Therefore it is protected by the same laws and rights.

Do not forget to read, understand and accept the User License Contract of the software of which this documentation forms part of before using any component of the product. If the terms and conditions of the User License Contract are NOT accepted, immediately return the software and all the elements accompanying the product to where it was first acquired for a complete refund.

This manual corresponds to the software version indicated by CYPE Ingenieros, S.A. as CYPECAD. 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)

Windows is a registered trademark of Microsoft Corporation.

Presentation	5	15.2.5. Reinforcement modifications	38
15. Practical examples	7	15.2.5.1. Columns	38
15.1. Introduction	7	15.2.5.2. Beams	39
15.2. Example 1	7	15.2.5.3. Joist floor slabs	40
15.2.1. Data organising	8	15.2.5.4. Foundation	43
15.2.1.1. Column layout drawing	8	15.2.6. Application of finishing touches to texts before obtaining drawings	45
15.2.1.2. Building elevation section drawing	8	15.2.7. Drawings	46
15.2.1.3. Floor table	8	15.3. Example 2	49
15.2.1.4. Floor plans	9	15.3.1. Initial data	49
15.2.1.5. DXF or DWG template	9	15.3.1.1. Floor heights	50
15.2.2. Data introduction	9	15.3.1.2. Loads	50
15.2.2.1. Job creation	9	15.3.2. Job creation	51
15.2.2.2. General data	9	15.3.2.1. Automatic introduction DXF/DWG	52
15.2.2.3. Floor / Floor group definition	12	15.3.2.2. Automatic introduction IFC	55
15.2.2.4. Import DXF or DWG templates	13	15.3.3. Column introduction (Column Definition tab)	58
15.2.2.5. Column introduction	14	15.3.3.1. Job general data	58
15.2.2.6. Introduction of group 1: 1st floor	17	15.3.3.2. Creation of project floors and groups	60
15.2.2.7. Introduction of group 2: 2nd and 3rd floors	25	15.3.3.3. Column introduction	60
15.2.2.8. Introduction of group 3: Roof	28	15.3.4. Introduction of beams, walls and floor slabs (Beam Definition tab)	61
15.2.2.9. Introduction of group 4: Lift machinery roof	28	15.3.4.1. Walls	61
15.2.2.10. Introduction of group 0: Foundations	28	15.3.4.2. Beams	65
15.2.3. Analysis	31	15.3.4.3. Floor slab introduction	69
15.2.4. Results revision	31	15.3.4.4. Finishing touches	72
15.2.4.1. Columns	31	15.3.4.5. Load introduction	75
15.2.4.2. Beams	33	15.3.4.6. Stairs	78
15.2.4.3. Joist floor slabs	35	15.3.4.7. Analysis	82
15.2.4.4. Foundation	36		

15.3.5. Foundations	82
15.3.5.1. Pad footings	82
15.3.5.2. Strap and tie beams	83
15.3.6. Results	84
15.3.6.1. Deformed shape of the structure	84
15.3.6.2. Columns	85
15.3.6.3. Beams	88
15.3.6.4. Floor slabs	90
15.3.6.5. Foundations	94
15.3.7. Edition resources	96
15.3.8. Drawings	96
15.3.9. Reports	97

Presentation

CYPECAD is the software for reinforced concrete, steel, timber and aluminium structures which provides the spatial analysis, structural element design, reinforcement and section edition, and construction drawings of the structure.

It carries out the analysis of three dimensional structures composed of supports and floor slabs, including their foundations, and the automatic design of reinforced concrete, steel, timber and aluminium elements.

With CYPECAD, the engineer holds a precise and efficient tool to resolve all the aspects related to the analysis of the structure of any type of concrete as well as being adapted to the latest international Codes.

15. Practical examples

15.1. Introduction

Two practical examples are described below. Each example is approached in a different manner. There is no single correct way to introducing a job. With practice, the user will develop his or her own method of introducing and analysing a job. Nonetheless, the objectives of both the examples are to:

- Act as guides as to how the data of the structures should be organised.
- Show how they are to be introduced in the program.
- Analyse their results.
- Obtain the necessary drawings and reports.

15.2. Example 1

In this example, a reinforced concrete structure composed of columns, beams and joist floor slabs is to be introduced.

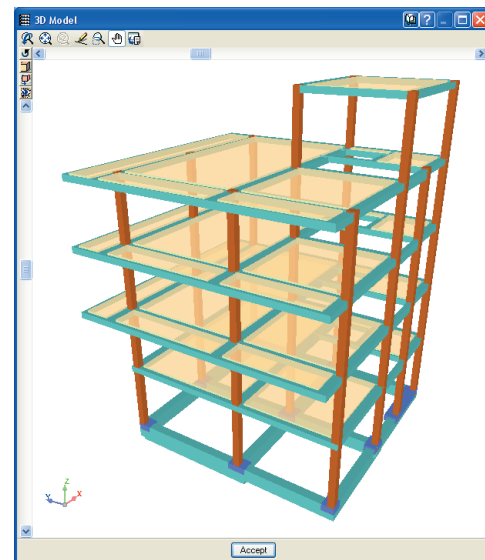


Fig. 15.1

N.B.: A more advanced course is available, published by CYPE : Practical course of CYPECAD, which can be studied once this example has been worked through.

The following practical example is included with the program. To access it and install the DXF file to be used as a template for the data introduction, follow the following steps:

- Open the program
- Click on **File > File manager**. The file manager window will open.
- Click on the **Example** button.

The File manager window will open displaying the job file at the following directory: **\CYPE Ingenieros\Examples\CYPECAD**, which will be opened later on during the example.

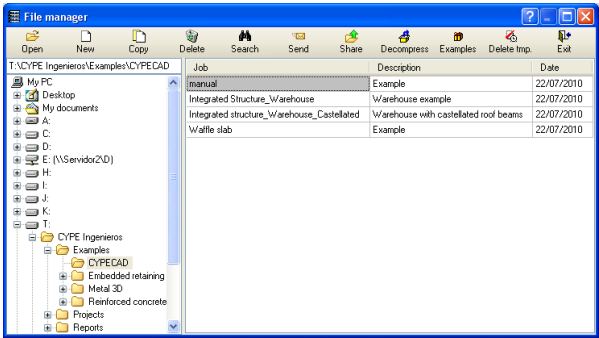


Fig. 15.2

15.2.1. Data organising

15.2.1.1. Column layout drawing

A drawing of this type should be created, which should display:

- The start and end elevations of each vertical column alignment, shear wall and wall.

- Their respective sections with their pre-designed heights.
- Angles
- Fixed points.

Please recall that the fixed point of a column is the corner, mid-point or axis from which the transverse section can be incremented. This point is defined so avoid the marked limiting boundaries of the structure to be surpassed and so to not invade openings. The user must know which column faces cannot be moved.

15.2.1.2. Building elevation section drawing

This drawing should contain:

- Floor numbering
- Floor group assignment
- Surface dead and live loading
- Difference in height between top elevations of finished consecutive floors

15.2.1.3. Floor table

Based on the previous drawing, the following table can be created:

Group	Group name	Floor	Floor name	Height (m)	Elevation (m)	Live load (kN/m²)	Dead load (kN/m²)
4	Lift machinery roof	5	Lift machinery roof	3.00	13.75	1.00	2.00
3	Roof	4	Roof	2.85	10.75	1.50	2.50
2	2nd and 3rd floors	3	3rd floor	2.85	7.90	2.00	2.00
		2	2nd floor	2.85	5.05		
1	1st floor	1	1st floor	3.10	2.20	2.00	2.00
0	Foundations				-0.90		

15.2.1.4. Floor plans

It is convenient for the type of floor, depth, materials, partition loads, stair loads, local surcharges, joist layout direction, etc. to be known beforehand by the user to increase the efficiency of the data introduction.

15.2.1.5. DXF or DWG template

Once the floor and column outlines, fixed point of the columns, etc. have been drawn, the building geometry can be introduced much more quickly with the aid of a DXF or DWG template in which these elements are already drawn. Although other methods can be used.

Use of the DXF or DWG as a template is an advantage compared to introducing the columns using coordinates or other methods (relative dimensions, layout lines or outlines). It also proves to be useful for edge beams and stair opening beams, patios, lifts, building services, etc.

If a DXF template is to be used, before exporting the file from the CAD program, please ensure the number of decimal places is set to 3.

Check the units of the drawing before importing it into the program and ensure it is in the same units and order of magnitude as the program.

15.2.2. Data introduction

15.2.2.1. Job creation

Follow this process to create the job:

- Click on **File > New**. In the window that opens introduce the name of the job and click on **Accept**.

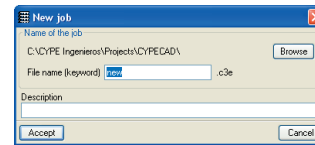


Fig. 15.3

- In the next window (**New job** window) the program will ask the user to choose the introduction mode amongst various options. Select **Empty job** (for this example) and click on **Accept**.

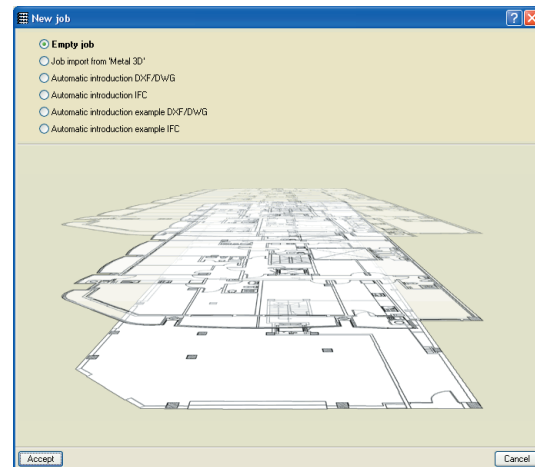


Fig. 15.4

15.2.2.2. General data

Having accepted the previous windows, the General data dialogue box will open. Here, a description of the job may be introduced, the codes to follow selected, materials to be used, horizontal loads to be applied, definition of load factors and additional loadcases, column effective length factors, edition of reinforcement tables and analysis options.

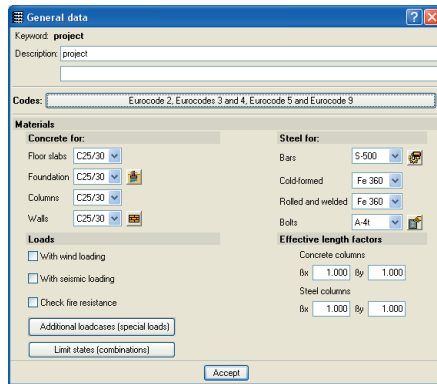



Fig. 15.5

15.2.2.2.1. Code and materials

The concrete, rolled-steel, cold-formed steel, wood, masonry and composite slab codes can be selected, as well as the type of concrete for each element of the job. The type of steel to be used for the reinforcement bars can also be defined.

For this example, the concrete to be used will be C25/30, steel for bars S-500 and for the codes, select the corresponding Eurocodes for each material by pressing on the **Codes** button at the top of the dialogue box.

Using the **By position** button , the reinforcement tables may be accessed and many analysis options can be edited and modified by the user.

In this example, columns will be designed in such a way that all four faces will contain symmetrical reinforcement and be continuous. Follow the steps to apply this arrangement:

- Click on the **By position** button
- Click on the **Vertical bar options** button

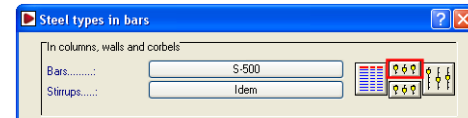


Fig. 15.6

A new window will open.

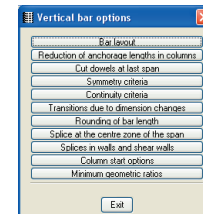


Fig. 15.7

- Click on **Symmetry criteria**. Introduce **300** in the window that appears and **Accept**.

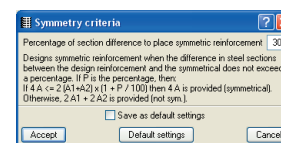


Fig. 15.8

- From the options window select **Continuity criteria**. Select **Apply from the next to last** from the scroll menu for the option **The number of bars in the faces must be greater than or equal to that of the higher floor**. **Accept** this option, **Exit** the **Vertical bar options** window and the program will return to the **General data** window.

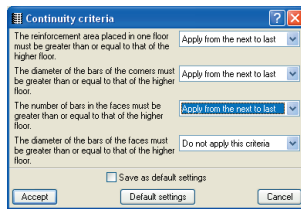



Fig. 15.9

- To define the allowable bearing pressure of the soil, click on the Foundation data button .
- In the emerging window, introduce an allowable bearing pressure of 0.20 MPa for persistent situations and 0.30 MPa for accidental situations.

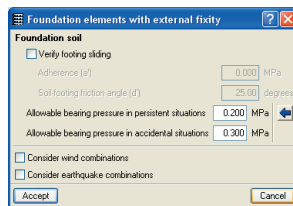


Fig. 15.10

15.2.2.2.2. Wind and earthquake loading

We recommend that a first analysis be launched with only vertical forces applied so that a general adjustment of the columns can be carried out. In the second analysis, the horizontal forces can be applied, if any, followed by the adjustment of all the resistant elements. In the case of an experienced user, predesign the columns so that only a single analysis is required.

In this practical example, no horizontal loads will be introduced.

15.2.2.2.3. Additional loadcases (special loads)

The use categories for the building are to be defined here. Click on the **Additional loadcases (special loads)** button and then on the use category icon. Select A.Domestic and household and household and H. Roofs from the emerging window and accept.

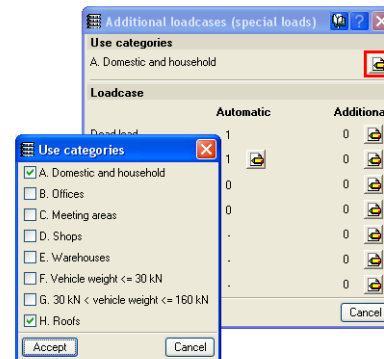


Fig. 15.11

The additional live loads of this example are introduced without the need of having to define an additional loadcase. Nonetheless, if other loadcases are to be created in another job, be they compatible, incompatible or simultaneous with other loadcases, click on the **Additional loadcases (special loads)** button.

15.2.2.2.4. Limit states (combinations)

The limit states (combinations) can now be defined:

- Click on the **Limit states (combinations)** button in the **General data** window.
- From the scrolling menus in the **Limit states** window which appears, choose the **Execution control**, **Use category** and **Snow level** for the job.

- If special combinations are to be defined for a specific material, click on the **Configure combinations for each limit state** button at the bottom of the **Limits states** window.
- In the new window, select the button corresponding to the option whose limit states are to be modified.

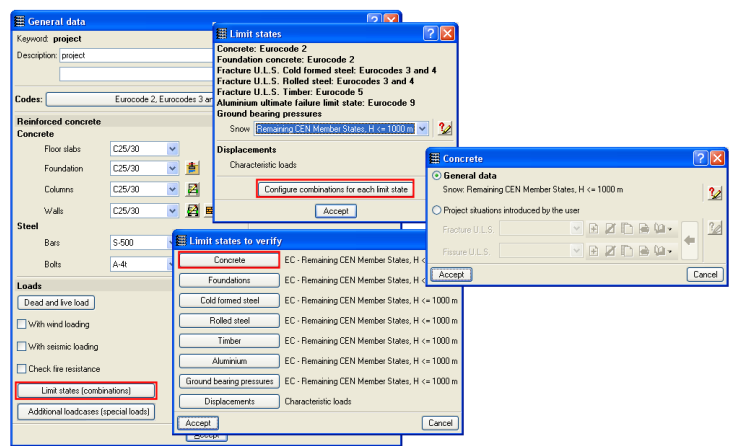


Fig. 15.12

15.2.2.2.5. Effective length factors

Leave the default coefficients.

If the structure is analysed with vertical forces, a value of 1 for these parameters provides results with an applied safety factor. If there are horizontal loads acting on the structure, this value is also reliable as long as the analysis is carried out taking into account second order effects.

15.2.2.3. Floor / Floor group definition

Floors and floor groups are defined as follows:

- Click on **Introduction > Floors /Groups**
- Click on **New floors** in the emerging window

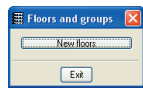


Fig. 15.13

- Click on **Single** and click on **Accept**.

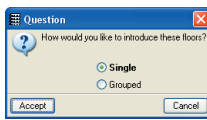


Fig. 15.14

- Introduce five floors and press the **Enter** key. Complete the table and click on **Accept**.

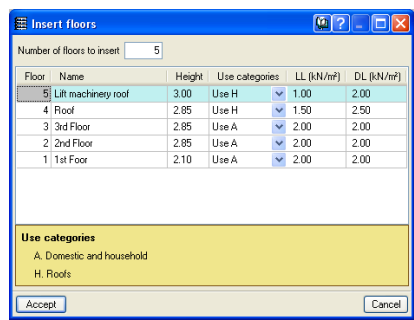


Fig. 15.15

- The **Floors and Groups** window will appear again but this time with more options.

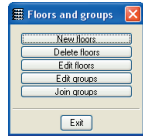


Fig. 15.16

- Group the second and third floors. Select **Join groups**. A new window will open to join the groups.
- Move the mouse cursor and click to select which floors are to be grouped.

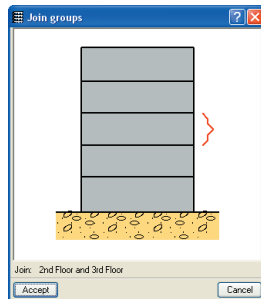


Fig. 15.17

- Click on **Accept**. From the dialogue box that appears, either option can be selected as no floor geometry has been defined yet (this option is intended for use if the floors are grouped once their geometry has been defined and differs between the two floors. Upon grouping them, since they both have to contain the same geometry, the program prompts as to which geometry of the floors is to remain).

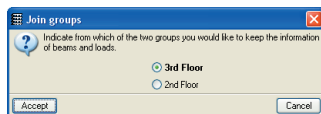


Fig. 15.18

- Click on **Accept** and the **Floors and groups** dialogue box will reappear.
- Click on **Edit floors** and introduce **-0.90** as the value for the **Elevation of the foundation plane**.

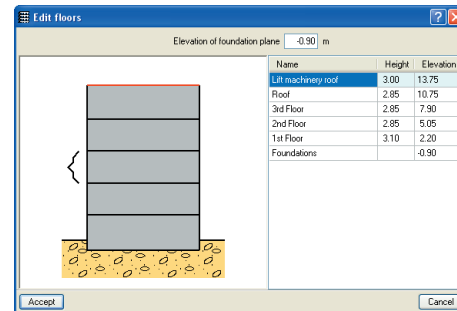


Fig. 15.19

- Click **Edit groups** and change the name of **Floors 2 and 3** to **2nd and 3rd floors**.

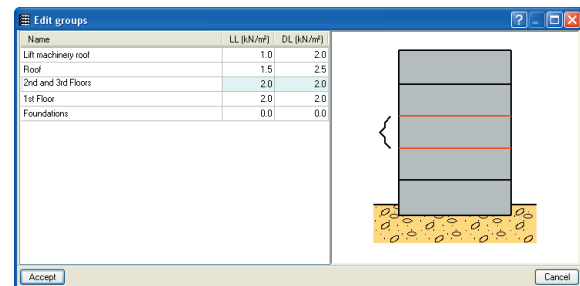


Fig. 15.20

15.2.2.4. Import DXF or DWG templates

As mentioned previously, the use of a DXF or DWG template is very useful when introducing the geometry of the structure. To import the DXF or DWG file:

- Select the **Edit templates** button  from the toolbar. The **Template views manager** window will open.

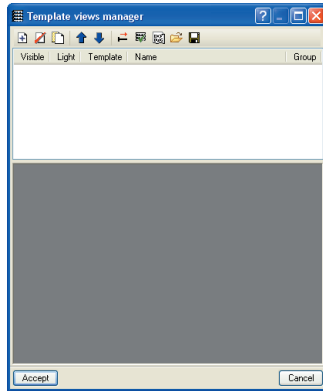


Fig. 15.21

- Click on the **Add** icon and the **Selection of templates to read** dialogue box will open asking for a DXF or DWG file to be selected. Find the **floor_geometry.dxf** file located in **(CYPE Ingenieros)\Examples (CYPE-CAD**. Select and click on **Open**.
- Click on **Accept** to return to the **Template views manager**.

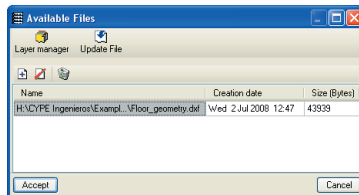


Fig. 15.22

- Click on **Accept** once again, to be able to view the template on screen. Upon doing so the program will ask: **Would you like to assign the view to all the groups?** Choose **Yes**.

For this example, a template has been created (with a CAD program) to be used for the floor geometry.

15.2.2.5. Column introduction

The columns will now be introduced.

- Click on **Introduction > Columns, shear walls and starts** from the menu in the **Column Definition** tab. The following dialogue box will open.

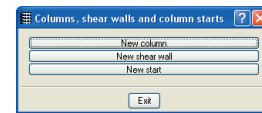


Fig. 15.23

- Click on **New column**. For the time being, all the columns will be introduced with the same dimensions from the **Foundations** to the **Lift machinery roof**. Therefore, accept the data that appears in the window.

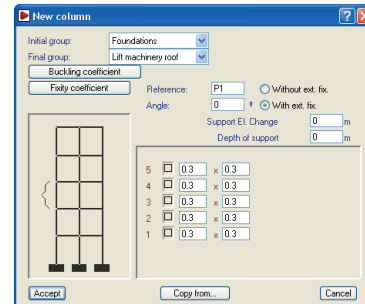


Fig. 15.24

- Upon accepting, the dialogue box will close and return to the main screen. Using the scroll wheel of the mouse or by opening a zoom window using the icon in the toolbar at the top of the screen, zoom in, so to view the first column in the DXF template.

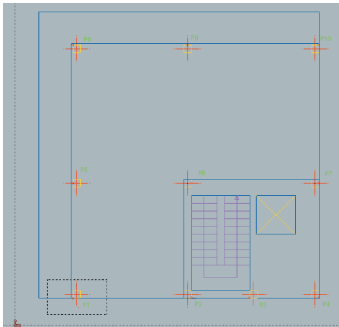



Fig. 15.25

- Click on the **Template object snaps**  icon in the toolbar at the top of the screen. The first column to be introduced is a corner column, therefore the fixed point of the column will be the bottom left hand corner of the column.
- In the **Object snap selection** dialogue box that opens, mark the **Intersection** box then click on **Accept**.

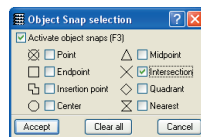


Fig. 15.26

- Place the mouse cursor over the bottom left hand corner of the column (its fixed point, marked on the template with a red circle). Notice how a cross appears when the mouse cursor is moved over two intersecting lines.

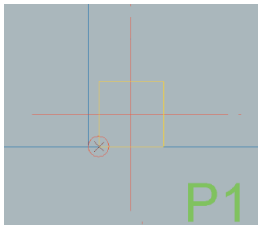


Fig. 15.27

- Click on the bottom left hand corner. The corner of the column has been snapped. However, the position of the column that has been introduced does not coincide with the column from the template.

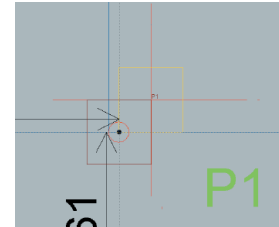


Fig. 15.28

- To adjust the column, click on the right mouse button. The **New column** window will open. Click on **Cancel** and the **Columns, shear walls and starts** dialogue box will appear.

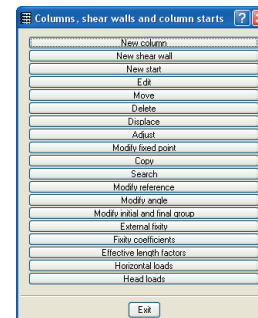


Fig. 15.29

- Click on **Edit** followed by a click on the corner column. In the **Edit** dialogue box, a diagram of the **Fixed point** of the column is shown in the top right hand corner. The possible fixed points of the column are displayed in **green**, and the selected fixed point is displayed in **red**.

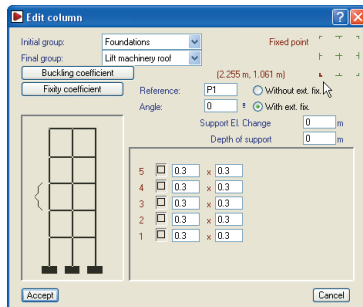


Fig. 15.30

- Click on the bottom left hand corner fixed point to select it.
- Click on **Accept**.
- Click on the **Full screen** icon from the toolbar to obtain a complete view of the floor.

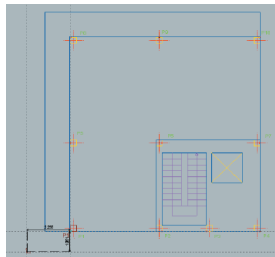


Fig. 15.31

- Zoom to view the column situated to the right of the column that has just been introduced. Click on the right mouse button.
- Click on **New column** in the **Columns, shear walls and starts** window. The **New column** window will open. Click on **Accept**.
- In this case the bottom right hand corner should be snapped to and then carry out the corresponding adjustment in the **Edit** window.

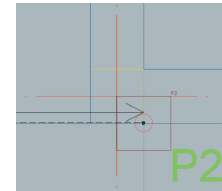


Fig. 15.32

- Introduce the remaining columns always using the object snap to snap to the fixed point, which is displayed with a red circle in the template. When introducing a column whose fixed point is at its centre, the adjustment will not have to be carried out.

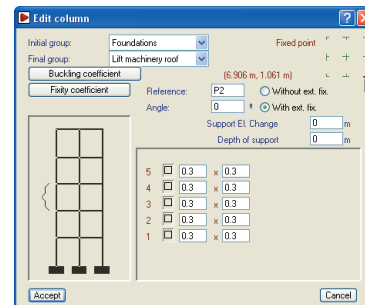


Fig. 15.33

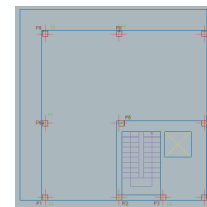


Fig. 15.34

This process may be slow and tedious. However, there is no need in carrying out a zoom on each column, a larger zoom window can be used to view several columns, intro-

duce them snapping to their fixed points and then adjust each one. In this example, the process has been described in detail so it may be understood well.

Another option is to introduce all the columns by snapping to their centre in the DXF template and then use the **Change fixed point** option, from the **Columns, shear walls and starts** window. This option modifies the fixed point without displacing the column.

When this tool is selected, the Fixed point dialogue box will appear. Here the fixed point of the column can be chosen and then assigned to the columns which are to have their fixed point at that position.



Fig. 15.35

This way of introducing the columns may result to be more comfortable, but remember that for the columns to be introduced correctly, the dimensions must be the same as they are in the DXF template. If this is not the case, the fixed point will not be in its correct position. Therefore if a DXF template is being used where the dimensions of the columns differ from one another, these must be changed each time a new column is introduced or alternatively, once they have all been introduced, edited one by one.

Now the start and end of the columns will be modified.

- Select **Modify initial and final group** from the **Columns, shear wall and starts** dialogue box.
- In the window that appears, deactivate the **Initial group** box and for the **Final group**, select **Roof**.

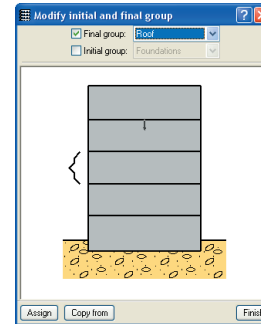


Fig. 15.36

- Bearing in mind that the columns are number from left to right and bottom to top, click on the **Assign** button and click on the columns **P1, P3, P5, P8, P9 and P10**, which will be marked in red. Click on the right mouse button and select **Finish**.
- Click on **Exit** to close the **Columns, shear walls and starts** window and to conclude the column introduction, click on the **Beam Definition** tab.

15.2.2.6. Introduction of group 1: 1st floor

Having clicked on the Beam Definition tab, the program will situate the user on the first floor.

Firstly, activate the visibility of the element references.

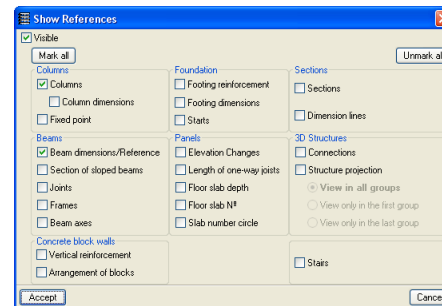



Fig. 15.37

15.2.2.6.1. Beams

- Click on **Groups > Show references** (or select the  icon from the toolbar at the top of the screen). A window will appear where the **Visible**, **Columns**, **Beam dimensions/ reference** boxes are to be activated. The rest left unmarked.

Now the beams will be introduced.

- Click on **Beams/Walls > Define beam**. The **Current beam** window will open.
- Select a flat beam by clicking on the top icon of the left hand column.
- Click on the dimension representing the width of the beam and change it to 0.30.

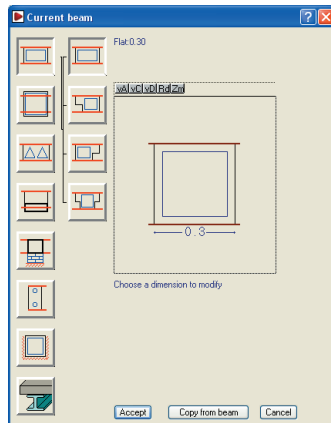



Fig. 15.38

- Click on **Accept** and the Define beam floating menu will appear on screen.







Fig. 15.39

This menu contains general and specific help dialogue boxes, which provide information on how to introduce straight and curved, continuous or discontinuous, program definitions, etc. To understand the introduction modes of the program, it is highly recommended the help dialogue boxes be consulted.


To change the dimensions of the beam to be introduced, click on the first button of the floating menu  and the **Current beam** window will reappear.

For this example, ensure the options of the floating menu are set as follows:

- : Should be in **Simple** mode
- : Any of the three options can be activated depending on the user's beam introduction preference.
- : The **Without displacement** option should be activated.
- : Should have **Straight** option activated.

Whilst the floating menu is visible, beams are ready to be introduced. If the menu disappears, it can always be activated by reselecting the **Define beam** option from the **Beams/Walls** menu.

- Beams can be introduced and adjusted to their respective positions in the DXF template in two different ways. One way consists in snapping the axes, faces or edges of the previously introduced columns and then adjusting the beam to the DXF template. The other way is by snapping to the elements of the imported DXF template, and so the beam is adjusted when it is being introduced.
- Introduction of beam P1-P5 by snapping to columns.**
 - To snap to the axes, faces or edges of the columns, the DXF or DWG object snaps must be deactivated.

To do so, click on the  icon in the toolbar and deactivate any objects snaps which may be activated. Alternatively, pressing F3, will deactivate/activate all the object snaps.

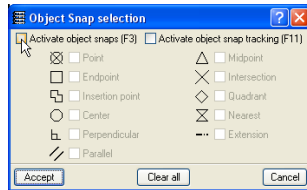



Fig. 15.40

- o The third button of the Define beam floating menu  must be set to Centre. Click on column P1. When the centre of the column has been selected, the mouse cursor will change to a red point.
- o Click on column P5.

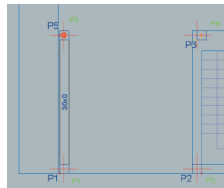


Fig. 15.41

- o Having introduced this beam, its side has to be adjusted to the façade drawn on the template. To do so, activate the **Nearest** object snap from the **Object snap selection** window.

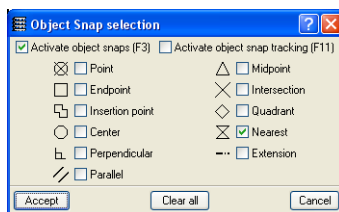


Fig. 15.42

- o Select **Beams/Walls > Adjust**.
- o Click on the left side of the beam

Note: The mouse cursor must be situated outside the width of the beam but close to its left side, otherwise if it is placed within the width of the beam, the axis of the beam will be adjusted to the façade line.

The beam is displaced automatically so that its side coincides with the line drawn in the template.

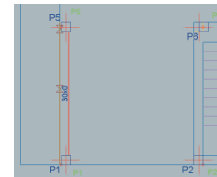




Fig. 15.43

This adjustment, as well as displacing the beam so its side face coincides with the template also establishes its fixed side, which due to it being an edge beam at the façade of the building is displaced 5 cm with respect to the face of the column.

It is normal for the width of the beam to vary as the analysis of the structure proceeds, and if a side of the beam is to be maintained at a fixed position after it has been introduced, it must be adjusted (to a template line, column face, etc.)

• Introduction of beam P1-P5 by snap to DXF template

- o Delete the beam P1-P5 (**Beams/Walls > Delete**) to practise introducing the beam using a different way.
- o Select the option **Beams/Walls > Define beam**.

- o Click on the  button in the toolbar to activate the **Nearest** object snap and deactivate any other object snap which may be selected.
- o The third button from the **Define beam** floating menu  should be set to **Left** adjustment. Position the cursor at a point on the façade line of the template between columns P1 and P5.

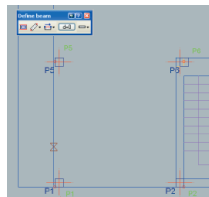


Fig. 15.44

- o The symbol \propto which appears when the cursor is brought close to a line of the DXF template indicates that if the user clicks on the left mouse button at that point, the start of the beam will be at that point. Introduce the initial and final points of the beam by snapping to two points along the façade line of the template, the first close to **P1** and the second close to **P5**.

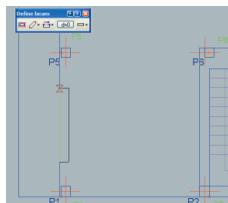


Fig. 15.45

- o For the beam to span between column **P1** and column P5, it must be extended using the **Extend beam** command found in **Beams/Walls > Extend beam**. The beam may also be introduced without the need of having to extend it later on. If the initial and final points of the beam are snapped to points

close enough to the columns, in such a way that the width of the beam overlaps them, the beam connecting columns **P1** and **P5** will be obtained.

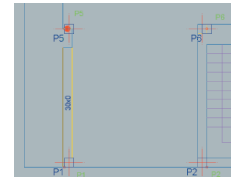


Fig. 15.46

- To introduce a beam using any of the methods described above, beam spans do not necessarily have to be introduced from one column to the next column. Any beam length can be introduced and the beam will remain joined to any column it overlaps.

Now introduce the remaining beams:

- Beam **P5 - P8** (from column **P5** to column **P8**). The option **Beams/Walls > Extend beam** can be used to extend the end of beam **P1-P5** or introduce a new beam using **Beams/Walls > Define beam** and then adjust it to the template line using the option **Beams/Walls > Adjust**, as was done with beam **P1-P5**. Each of the spans introduced must be adjusted; in this case only the second as the first was adjusted previously.
- Beam **P8-P10**, flat 0.40m width. External side adjusted to the template line. In this case, the two ends of the frame are introduced as column **P9** is aligned with columns **P8** and **P10**.
- Beam **P4-P10**, flat 0.30m width. External sides (right) of both spans adjusted to the template line.
- Beam **P1-P2**, flat 0.30m width. External sides (bottom) of the three spans adjusted to the template line.
- Beam **P2-P3**, dropped 0.20 x 0.40 m. External sides (bottom) of the three spans adjusted to the template line.

- Beam **P3-P4**, flat 0.40 m width. External sides (bottom) of the three spans adjusted to the template line.
- Beam **P5-P7**, flat 0.50 m width. Not to be adjusted, therefore the program maintains the longitudinal axis fixed in case its width varies.

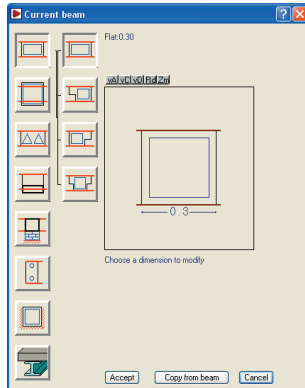


Fig. 15.47

The beam that has just been introduced contains a span, **P6-P7**, which partly defines the stair opening, but its side does not coincide exactly with the opening.

What must be done in the job (the user should correct and reflect it in the drawing) is to infill the area between the side of the beam up to the edge of the opening.

It does not make sense to move, in this case, the beam so that its side coincides with the stair opening (as it would have to be moved a large distance), and therefore, the axis of the beam would not lie within the width of the columns.

- Beam **P2-P6**, flat 0.30 m width. External side (right) adjusted to the template line which defines the stair opening.
- Beam **P3-(P6-P7)**, flat 0.25m width. The beam spans from P3 and ends perpendicular to beam **(P6-P7)**. The **Orthogonal on/off** option can be used from the toolbar or roughly introduce the second end of the beam and then adjust to the template line.

- 0.25 m width flat beam closing off the bottom of the lift opening. Similar to the previous case, it is introduced at its approximate position, then adjusted.
- 0.25 m width flat beam which closes off the right hand side of the lift opening. Similar to the previous case, it is introduced at its approximate position, then adjusted.
- Click on the **Edit templates** icon from the toolbar. In the window that opens, deactivate the **Visible** box from the top table.

The floor layout should be as follows:

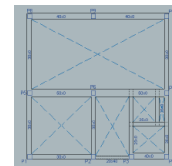


Fig. 15.48

It can be seen that beams introduced as flat beams do not have depth. They will automatically adopt the depth of the panel they hold.

If two panels reach the same beam, the beam will adopt the greatest depth of the two.

As there are important section variations in the same beam alignment, it is convenient that the reinforcement be divided at these section changes.

- Click on **Beams/Walls > Beam alignments > Divide an alignment**.
- Place the cursor over **P2**. A red dot and a scissor symbol will appear. Click on the column with the red mouse button.
- Repeat over **P3**.

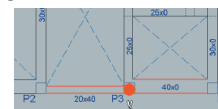


Fig. 15.49


15.2.2.6.2. Floor slabs

Follow these instructions to introduce the floor slabs:

- Click on **Slabs > Panel manager**. A floating menu will appear. Initially and whilst no slabs are introduced, nearly all the icons in this window will appear deactivated. This window contains most of the tools required to manage slab introduction.



Fig. 15.50

- Click on the first icon from the floating menu . The **Panel manager** dialogue box will appear.

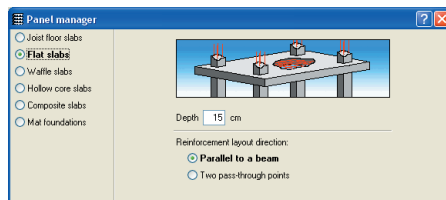


Fig. 15.51

- Select the **Joist floor slabs** option. The dialogue box will change to display the options associated with joist floor slabs (these will vary depending on the selected construction codes).

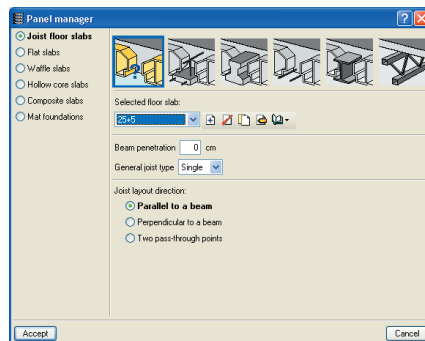




Fig. 15.52

The types of joist floor slabs available can be selected by clicking on their respective image. By clicking on the  icon and then on the image, the name of the type of joist slab is displayed. To be able to select some of these types, the user must have acquired the corresponding modules. The selected default option is that of a concrete joist floor slab to be used to introduced generic precast joist slabs. This corresponds to the first image. Click on it if it is not already selected.

- The properties of the slab that is going to be introduced have to be defined. To do so, click on the  button which lies below the joist images. A new dialogue box opens where the corresponding data has to be introduced.

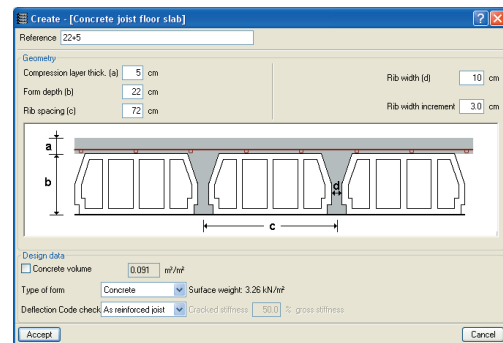



Fig. 15.53

- Having done so, click on **Accept** and the program will return to the **Panel manager** window. In the bottom part of the screen, the layout of the joists is defined. For this example, select the first option: **Parallel to a beam** and accept the dialogue box.
- The slab can now be introduced. Click on the opening defined by columns **P1-P2-P5-P6**.
- The direction of the joists has to be defined. Click on beam **P1-P5**. The slab as been introduced.

The remaining slabs can be copied from the one that has just been introduced. This is recommended so that joists from adjacent slabs have a continuous layout.

- To do so click on the **Copy panel** icon  from the floating menu. Click on the slab that has just been introduced. A new window will open from which the data to copy can be selected. Accept the window with the proposed selection.

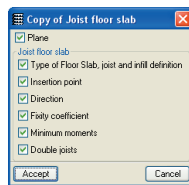


Fig. 15.54

Now click on the openings to introduce the slabs displayed in the following figure.

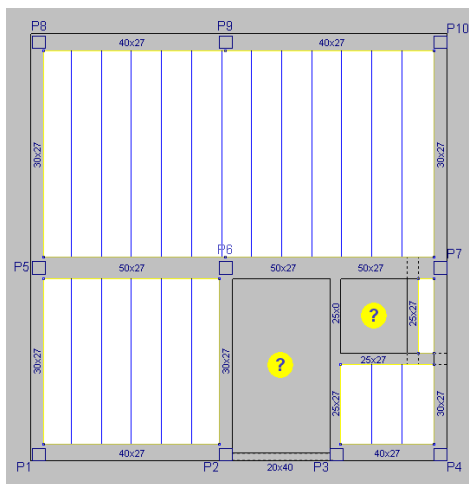



Fig. 15.55

- To introduce an opening where a slab has been introduced, select **Delete panel**  from the floating menu, then click on the slab to be eliminated.

15.2.2.6.3. Loads

There are two general methods to introduce loads. The first is to introduce the loads manually using the options in the **Loads** window (**Loads > Loads**). The second is available for line and surface load introduction which consists of an automatic introduction of loads on beams in the case of line loads (**Loads > Line loads on beams**) and on slabs in the case of surface loads (**Loads > Surface loads on slabs**). The latter option is the quicker option and is useful only if the loads are to be assigned to the entire element. In the case of loads partially assigned to the element, the user will have to judge whether to introduce the loads manually or automatically and then edit them to obtain the real distribution.

Follow this process to introduce the line loads:

- Click on **Loads > Line loads on beams**
- First introduce the dead loads. In the window that opens, type in a load of 7.00 kN/m. Select **Dead load** for the Loadcase.

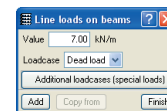


Fig. 15.56

- Click on **Add** and the window will close so the load can be introduced.
- Assign the load to all the perimeter beams.

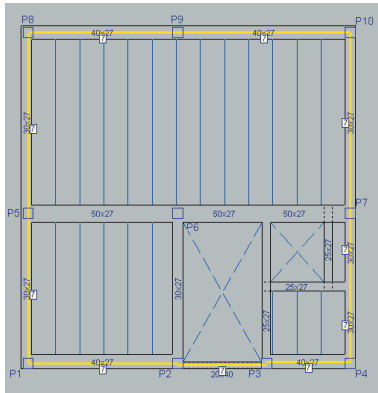


Fig. 15.57

- Click on the right mouse button and select **Finish** from the window that appears.
- Click on **Loads > Loads**.
- In the new window, select **Line** load, type in 12.00 kN/m, leave the **Loadcase** as **Dead load** and select **New**. This will correspond to the self weight of the staircase on both supports:

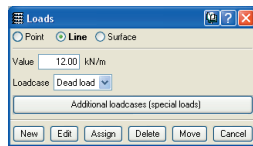


Fig. 15.58

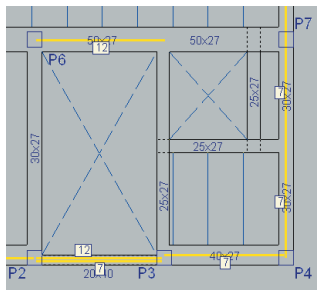


Fig. 15.59

- Complete the remaining loads.
- To introduce the live loads transmitted by the staircase, select the Live load in the Loadcase compartment from the Loads window.
- Introduce the two 6.00 kN/m line loads on the stair supports.

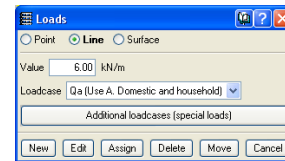


Fig. 15.60

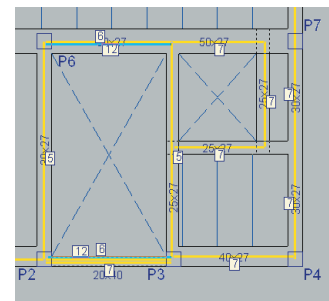


Fig. 15.61

The floor introduction has been completed. The second floor can now be defined:

- Click on the **Up a group arrow** in the toolbar.
- It may occur that, if the described beam and slab introduction order has not been followed exactly, upon going up to a different group, the program may ask for the depth of the beam separating the stair and lift openings. In this case, if the beam remains divided in two spans, it has been defined as flat but does not have a slab on either side of it.

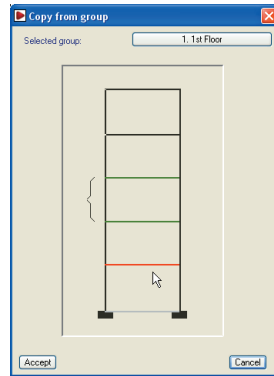
This is of no great importance. Introduce, in this case, the same depth as has been defined for the slab. This way, it remains a flat beam.

15.2.2.7. Introduction of group 2: 2nd and 3rd floors

As this group is very similar to the previous group, it may be copied and then edited to carry out the necessary changes.

- Click on **Groups > Copy from another group**. In the window that opens, place the mouse cursor over the first group. The text corresponding to the **Selected group** at the top of the screen will display **1. 1st floor**. Click on that floor or press **Accept**.

Fig. 15.62



- Activate the template by carrying out the opposite process to that explained earlier to deactivate it. Deactivate the visibility of the loads by pressing **Loads > Shown**.

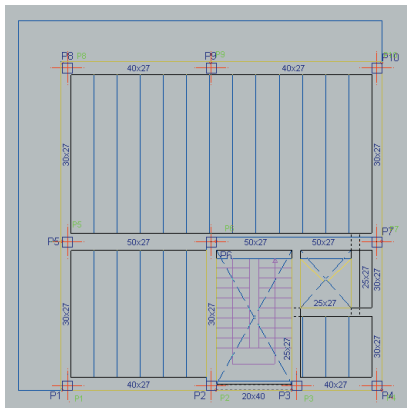


Fig. 15.63

15.2.2.7.1. Beams

Beams **P1-P5-P8** and **P8-P9-P10** are not to be adjusted to the template line as they are in the group below, as they do not coincide with the façades. They must be adjusted to the column axes. To do so:

- Deactivate all the template object snaps (in the toolbar).
- Select **Beams/Walls > Adjust** followed by a click on the longitudinal axis of the beam at the centre of its span. This will adjust its axis with the axis of the columns on which it is supported.





If the cursor is not placed at the middle of the span, i.e. the cursor is close to one of the beam ends, only that end will be adjusted. However, the same can be done at the other end.
- Repeat this process at each of the aforementioned spans.

It must be noted that adjustment to columns (be it to their face or axis) can only be done if the template object snaps have been previously deactivated.

The following beams will now be introduced. When the connection of several beams does not occur at a column, introduce beam segments shorter than their real length. These beam spans can be adjusted to the DXF template lines whilst they are being introduced or first introduced and then adjusted later. Their ends are then extended and finally the unwanted ends can be cut off.

An example of what has been described can be seen in the introduction of the beams parallel to the west and north façades. These beams define the two overhanging slabs of this floor group. To introduce them, follow the instructions described below:

- Activate the **Nearest** object snap from the toolbar.

- To introduce the beam parallel to the west façade, select a flat beam width 0.25 m. To do so, click on the first button of the floating menu **Current beam selection**. If this window is not visible, activate it using the **Define beam** option from the **Beams/Walls** menu.
- Select the **Left adjustment** by clicking on the  button of the **Define beam** window. This button may differ depending on the current adjustment at the time:  or .
- Introduce the first point of this beam by bringing the cursor near to the DXF template line to be snapped to until the snap symbol  appears on the DXF template line as is displayed in the image.

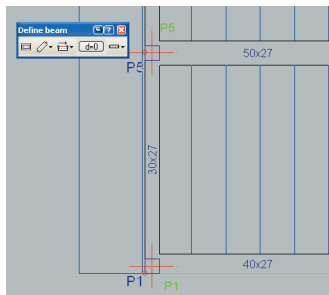


Fig. 15.64

- Once the object snap has appeared, click on the left mouse button. If the mouse is then moved, the beginning of a beam can be seen anchored on its left side to the point that has just been snapped to.

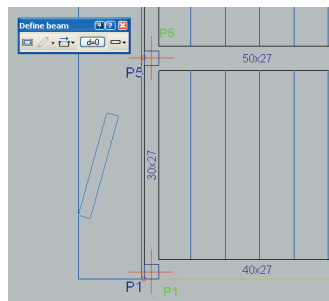


Fig. 15.65

- Now the final point of the beam fragment that is being defined has to be defined. This can be done by approximating the cursor to the DXF template line where the object snap symbol will appear, followed by a click with the left mouse button at that moment. The beam will have the following appearance:

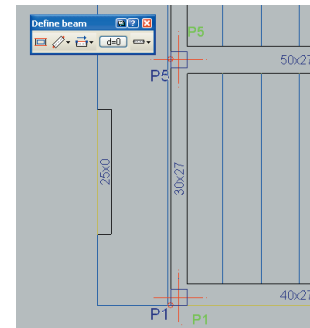


Fig. 15.66

- Introduce a beam segment parallel to the north façade, following the same procedure as in the previous example. In this case use a 0.10 m wide non-structural beam with a width of 0.10 m.

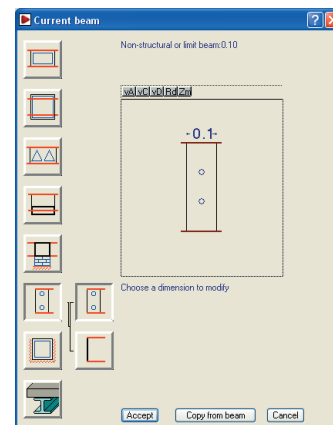


Fig. 15.67

- The non-structural beam must be introduced as shown in the following figure.

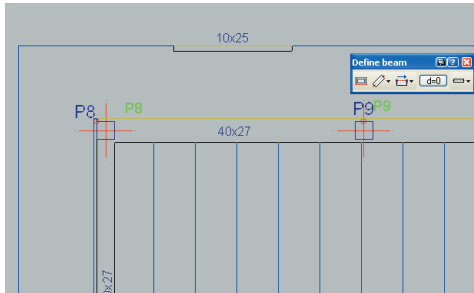


Fig. 15.68

- Extend the two beams (further than is required) using the option **Beams/Walls > Extend beam**.

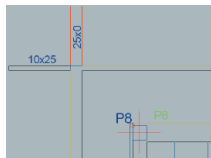


Fig. 15.69

- Delete the unnecessary beam segments using the option **Beams/Walls > Delete**.

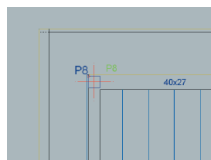


Fig. 15.70

- Extend and delete beam segments until the following layout is obtained.

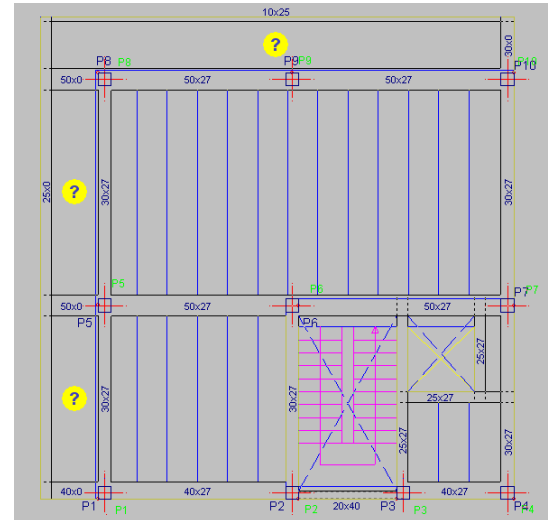


Fig. 15.71

15.2.2.7.2. Floor slabs

Introduce the floor slabs by copying those that have already been introduced. This way the joists are continuous as was explained earlier.

15.2.2.7.3. Loads

The perimeter line loads of the north and west façades change position.

- Use the option **Loads > Loads > Move** to obtain the following layout.

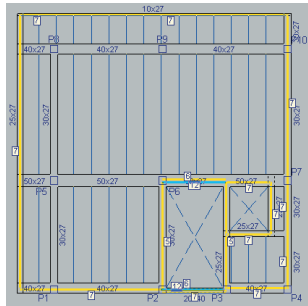


Fig. 15.72

- Click on the **Up a group** icon in the toolbar to proceed to the next group.

15.2.2.8. Introduction of group 3: Roof

The roof layout is very similar to the previous group.

- Copy the group below. The line loads must be modified.
- The external line load values are changed from 7.00 kN/m to 4.00 kN/m corresponding to the roof wall. This can be done using the option **Loads > Loads > Edit**.
- The roof wall has a brief interruption when it reaches the wall of the lift machinery room. The loads must be moved. Do so using the option: **Loads > Loads > Move**.
- The load to introduce due to the wall of the Lift machinery room is 6.00 kN/m.

The loads transmitted by the staircase can be reduced as the stair only reaches the floor. The start loads of the stair can be deleted.

The loads around the perimeter of the lift opening are increased to 8.00 kN/m. These correspond to the machinery slab where it is supported and the walls that support the slab. This slab must be designed in a separate file. The load distribution should look as follows:

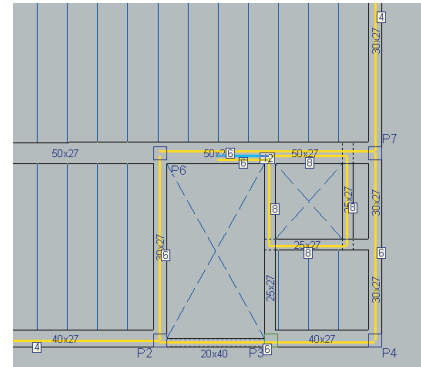


Fig. 15.73

- Click on the **Up a group** icon in the toolbar.

15.2.2.9. Introduction of group 4: Lift machinery roof

Introduce the geometry of the Lift machinery roof, which is shown in the diagram below. There are no line loads.

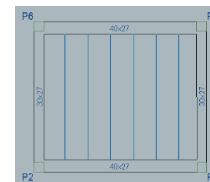


Fig. 15.74

15.2.2.10. Introduction of group 0: Foundations


15.2.2.10.1. Pad footings

The foundations of the building will consist of pad footings.

- Click on the **Go to group** icon in the toolbar and in the window that opens select **Foundations**. The first elements to introduce will be the pad footings followed by the strap and tie beams.
- Select **Foundation > Foundation elements**. The following floating window will appear:



Fig. 15.75

- Whilst no pad footings have been introduced, the floating menu only displays the first option as being available. This window contains helps which inform the user on the tools provided to introduce footings and pile caps.
- Click on the first icon  and the following dialogue box will appear:

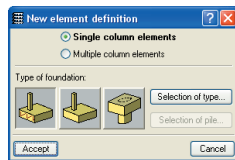


Fig. 15.76

- Select the **Single column elements** option and the first of the three buttons which represents a reinforced concrete footing. The other two buttons are used to introduce mass concrete footings and pile caps.

- Click on **Accept**. Introduce a corner footing at **P1**. To do so, bring the mouse cursor close to the column and move it around the column. It can be observed that the cursor changes indicating what type of footing would be introduced if the user were to click at that moment.

This way, if the cursor is placed in the top right hand corner, the symbol which is displayed represents a footing with the column situated in the bottom left hand corner.

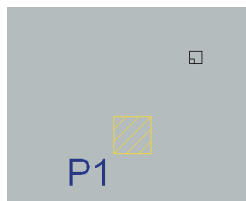


Fig. 15.77

- Click and the footing will be introduced.

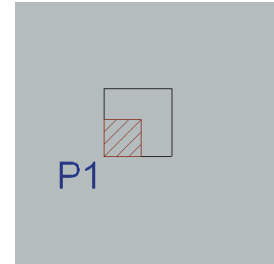


Fig. 15.78

- To introduce an edge footing at column **P2**, place the cursor in the following way. This way the edge footing is obtained.

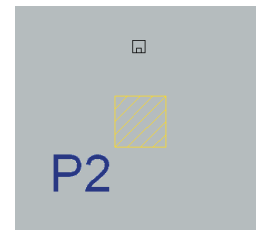


Fig. 15.79

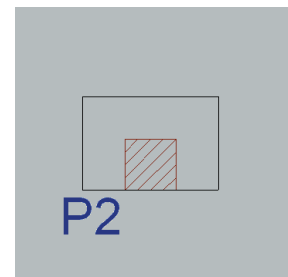


Fig. 15.80

- To introduce a centred footing at column **P6**, the cursor must be placed directly over the column. The cursor will display two concentric squares. Click to introduce the footing.

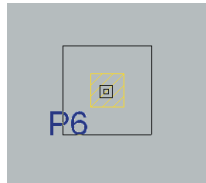


Fig. 15.81

- Now a combined pad footing will be introduced at columns **P3-P4**. To do so, click on the right mouse button and in the window that opens select **Multiple column elements**.

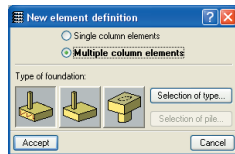


Fig. 15.82

- Click on **Accept**. Select, by means of a capture window, columns **P3** and **P4** or by clicking on one and then the other. The selected columns should be displayed in yellow.



Fig. 15.83

- Now click on the right mouse button. A cross and circle will appear, indicating the insertion point of the footing. Move the cursor of the mouse until the cursor symbol changes to that of a corner footing.
- Click and the footing will be introduced.

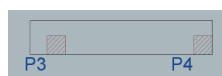


Fig. 15.84

- Introduce the remaining footings until the layout has the following appearance:

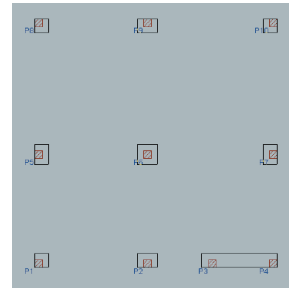


Fig. 15.85

15.2.2.10.2. Strap and tie beams




Fig. 15.86

Having introduced the footings, the strap and tie beams are to be introduced. This order is followed so that once the pad footings have been introduced (corner, edge, centred, etc. footings) the program will automatically balance the beam only at the end at which there is an eccentric column on the footing (eccentric in the direction of the beam).

- Click on **Foundation > Strap and tie beams**.

The strap and tie beam floating menu will appear.

Only the first icon will be activated, the rest will be activated once a beam has been introduced. This window provides help and information on all the tools used in the introduction of strap and tie beams.

- Click on the first icon in the floating menu . A dialogue box will open.
- When the new window opens, the selected default option is that of the automatic balancing beam. Leave this option as the selected beam.

Nothing has to be selected. The scroll menus display the selection of beams the program contains in its tables. CYPECAD selects the optimum sections of the beams parting from the smallest that exist in the table.

- Therefore, click on **Accept**.
- Introduce the first beam between **P1** and **P5**. It can be seen that the balancing symbol only appears at the bottom end of the beam, which is the only position where the balancing action is required. Now introduce the remaining beams as shown.

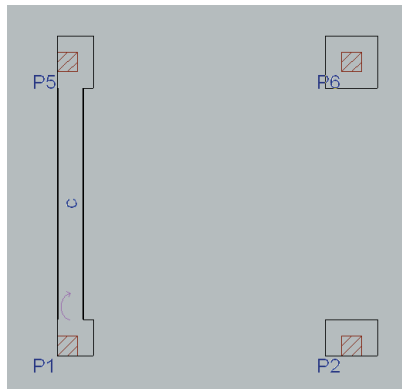


Fig. 15.87

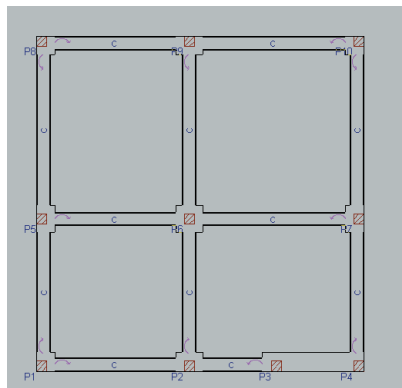


Fig. 15.88

15.2.3. Analysis

Now that all the data has been introduced, the structure can be analysed. If the data has not been completely introduced as has been instructed up to this point, this example is available at **CYPE Ingenieros\Examples\CYPECAD**.

- Click on **Analyse > Analyse the job (except foundation)**.

The reason why the foundation should not be analysed, for the time being, is due to the reason that, usually, columns, beams, etc require section modifications after the first analysis, and therefore a new analysis of the structure must be launched to account for any load changes.

The foundation of the structure can be designed, as will be seen later on, having completed the rest of the structure.

Once the analysis has concluded, a report appears on screen informing the user of any errors that have arisen.

- To check the results, click on the **Results** tab.

15.2.4. Results revision

15.2.4.1. Columns

It is recommended that the columns be the first elements to check.

15.2.4.1.1. Displacements (only with horizontal loads)

Even though horizontal loads have not been used in this job, it is necessary for the analysis to be carried out considering second order effects (the coefficient that multiplies the displacements will be left the assigned value of 1).

When the analysis has ended, it should be checked that the global stability coefficient is greater than 1.20. If so, the

stiffness of the structure must be increased in that direction, as it is easily deformed and has little stability in that direction. If the value is less than 1.1, its effect is negligible.

15.2.4.1.2. Force check

The user may consult axial forces, moments, shear and torsion forces by loadcase at any column elevation, analytically and graphically.

The worst case forces can also be consulted (loadcase combinations which result in having to provide maximum reinforcement) for any span.

Click on Envelopes > Forces in columns and shear walls. Click on a column, for example **P6**.

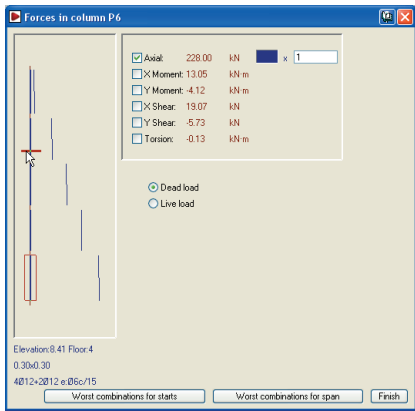


Fig. 15.89

It is recommended that, if not all, at least some of the columns of the structure are checked.

The axial force at the start of the column provided by the program must be approximately equal to that calculated by hand. The variance is due to the difference in the estimated tributary area due to the hiperstaticity of the structure in general and especially of the slabs.

It must be taken into account that the forces provided by the program belong to simple loadcases and, therefore, they do not have an applied safety factor and are independent for dead loads, live loads, etc.

15.2.4.1.3. Section check

- Select the option **Columns > Edit**, followed by a click on the first column. The **Column edition** window will appear.

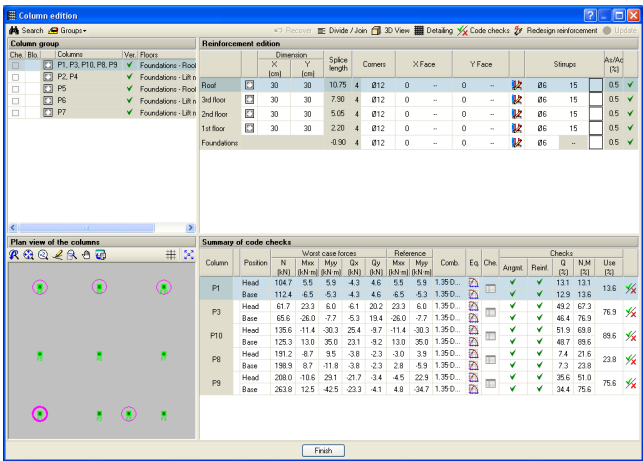


Fig. 15.90

In the bottom left hand corner of the window, the **Plan view of the columns** can be seen. Here, the columns sharing the same reinforcement (i.e. which belong to the same column group) are highlighted with a circle; one of them in a bold circle. In this case, P1 is in the bold circle. This means P1 is the master column of the group, hence any modifications carried out on P1 will also be carried out on the other columns of the.

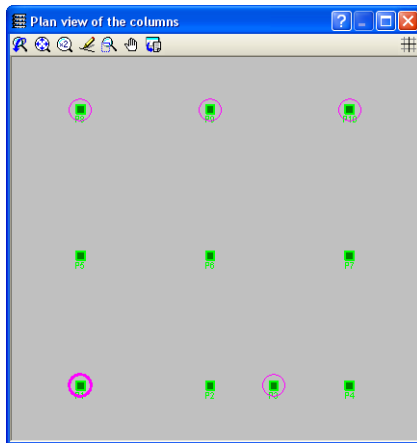


Fig. 15.91

At the end of the column information line of a floor, a tick or cross appears indicating the column span is adequate or it fails, respectively.

A more detailed report of the checks carried out on the column can be found in the **Summary of code checks** section on the bottom right hand corner of the Column edition window. If there are any errors, the column section has to be redesigned.

Upon modifying the column section in this window, the reinforcement redesign of the columns will be carried out using the simple loadcase combinations belonging to the last analysis.

If the section modifications are important, or if they are of little importance but horizontal wind and/or seismic loading is present, the structure must be reanalysed with the new sections as their inertia will have varied.

- Run through all the columns by selecting them in the Column group section in the top left hand corner of the Column edition window, up to column P10, and check that they do not contain any errors.

If horizontal loads are present, it is recommended the section of the columns be increased when reinforcement bars with a diameter greater than 16mm appear. This way, it will avoid, after the second analysis, 25mm reinforcement bars being used.

15.2.4.2. Beams

15.2.4.2.1. Force check

It is convenient to check at least the bending moment envelopes of all the beams of the structure in a graphical way.

- Go to group 1. Deactivate the visibility of the joist top reinforcement by clicking on **Joists > Views** and deactivating the **Top reinforcement** box.

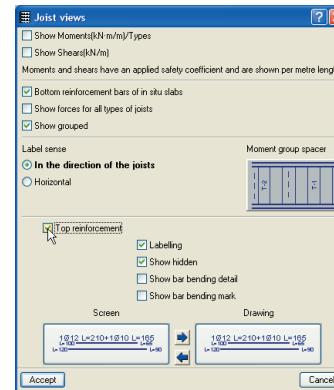


Fig. 15.92

- Click on **Envelopes > Forces in beams**. In the window that opens, activate the negative and positive moment boxes.

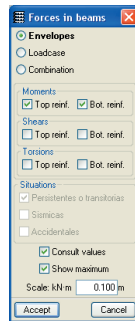


Fig. 15.93

- Click on a beam.

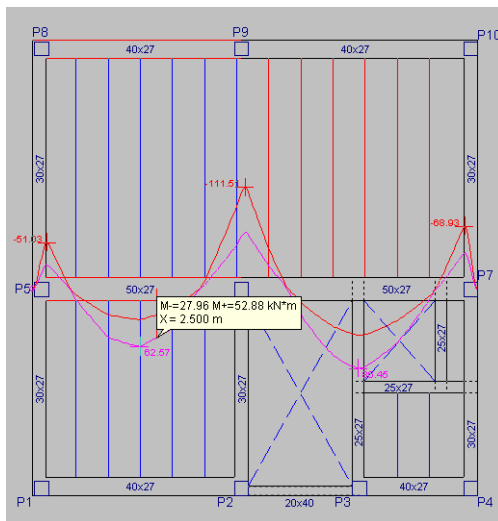


Fig. 15.94

- The remaining beams should be consulted on all floors. If the diagram of an envelope is not coherent, it is convenient that this span be checked and to ensure that the sum of half of the moments at its ends plus the whole of the centre moment approximately correspond to the simply supported moment.

Bear in mind that these envelopes have an applied safety factor and, as was said previously in the case of forces in columns, the tributary area considered by the user when carrying out manual calculations is only an approximation to what occurs in reality, due to the hyperstaticity of the slab. The user must also be aware that the simply supported moment $W L^2 / 8$ is for line loads but what the joists really transmit to the beam is a series of point loads.

15.2.4.2.2. Section check

Beams containing an error with their reinforcement, deflection, etc. or that stand out for a reason will be displayed in **red**.

- To know the reason of the error or warning click on **Beams/Walls > Beam errors**, followed by a click on the beam displayed in red. In our example, one of these beams is **P9-P10** in Group 2.

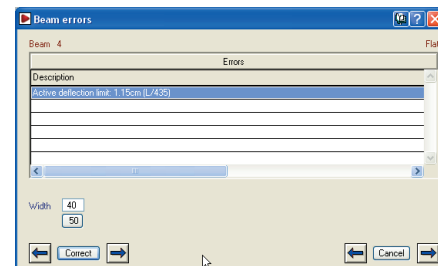


Fig. 15.95

This beam has a deflection problem. In this case, as only this problem is present, the reinforcement of the beam can be edited and increased, or increase the width of the beam. Here, the second solution will be applied.

- Select the Beam Definition tab. Click on **Beams/Walls** > **Assign beams**.

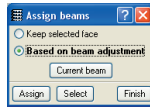


Fig. 15.96

- Select **Based on beam adjustment** and click on **Current beam**. In the new window that opens select a flat beam with a width of 0.5 m.
- All three beam spans making up the frame must be clicked on.

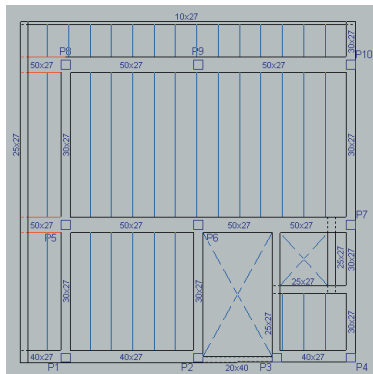


Fig. 15.97

- As section changes have occurred, the beams must be redesigned. Click on **Analyse** > **Redesign frames with changes**. Upon doing so, a warning message will appear which should be read carefully.
- Click on **Yes**. Select the Results tab and check the new deflection of the beam.
- Repeat this process with the same span in Group 3, only with a width of 0.55 m.

15.2.4.3. Joist floor slabs

15.2.4.3.1. Force check

It is convenient for some joist bending moment envelopes to be checked.

- Return to Group 1. Click on **Envelopes** > **Forces in joists**. In the window that opens only activate the positive and negative moment boxes.
- Click on a joist.

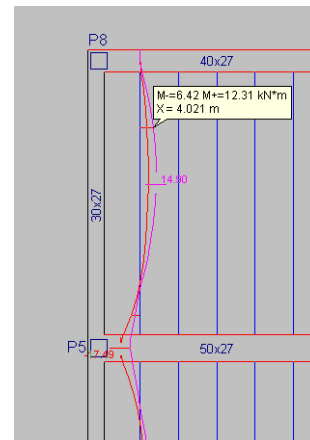


Fig. 15.98

- The user should continue checking the remaining joists on all floors and, if the diagram of an envelope is not coherent, it is convenient the same check be carried out as with the beams.

15.2.4.3.2. Section check

Joists containing errors such as shear, deflection, etc. or that stand out for a reason will be displayed in red.

- To know the reason for the error or warning click on **Joists** > **Errors**, followed by a click on a joist in red.

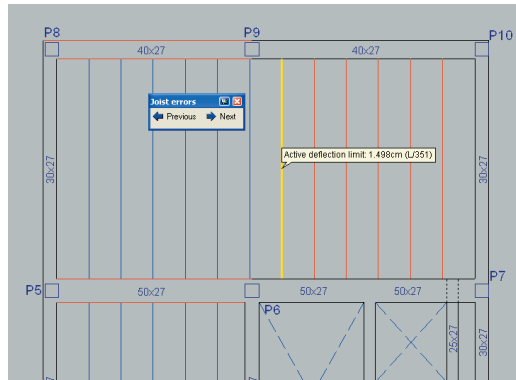


Fig. 15.99

The deflection of this joist and others adjacent to it overcome the deflection limits. There are various solutions to deflection problems. In this case a double joist will be placed where the joists only have one span. To do so, follow the following steps:

- Select the **Beam Definition** tab. Open **Slabs > Panel manager**. In the window that opens select **Enter double joist**.
- The previous window will close and the joists are to be entered from right to left sequentially on the joists of the top right slab.

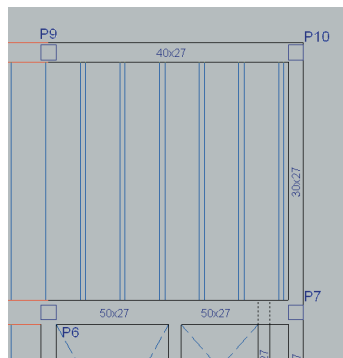


Fig. 15.100

- Reanalyse the structure. Click on the **Results** tab. Select **Joists > Information**.
- Click on the joists which had deflection problems. The problem has now been solved.

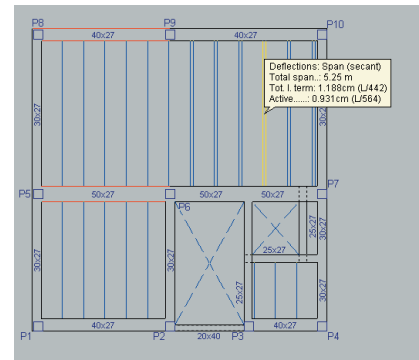


Fig. 15.101

15.2.4.4. Foundation

15.2.4.4.1. Design

First of all the foundation has to be analysed, if it has not been already. But, as a previous step, the pad footings will be assigned a minimum depth of 1m. To do so:

- Go to group 0 (foundation).
- Click on **Job > General data**.
- Click on the **By position** button.
- Click on the **Foundation** tab.
- Click on the **Strip and pad footing options**.
- Click on the **Pad footing options**.
- In the window that opens, introduce a value of **100** in the **Minimum depth** box.

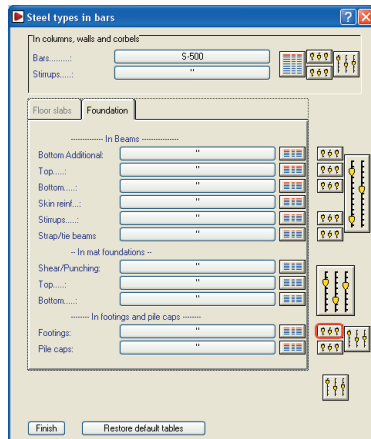


Fig. 15.102

- Close all the windows until the following option can be activated.
- Click on **Foundation > Design**.
- The following window will open. Activate the same options that have been activated here (please observe that the **Iterative design** option has been selected).

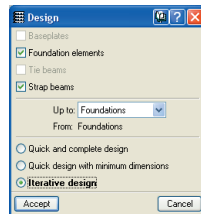


Fig. 15.103

15.2.4.4.2. Errors check

- Click on **Foundation > Code check errors**. Elements that do not contain any design errors will be displayed in black. Those containing errors in red. In any case, by clicking on any elements drawn in red, a message will be displayed containing the checks that have been carried out and any design or check errors, if any.

- Click on, for example, the footing below **P6**.

Criteria of CYPE	Maximum	Calculated	Minimum	Calculated	Verdict
Pressures on the soil:					
-Mean stress in persistent situations:	Maximum: 0.2 MPa	Calculated: 0.192668 MPa			Verified
-Maximum pressure gravity loads:	Maximum: 0.249959 MPa	Calculated: 0.197573 MPa			Verified
Bending in the pad footing:					
-In X direction:	Moment: 317.94 kN-m				Verified
-In Y direction:	Moment: 316.91 kN-m				Verified
Overturning of footing:					
<i>If the safety reserve % is greater than zero, then the overturning safety factors are greater than those strictly required for all equilibrium combinations.</i>					
-In X direction:	Safety reserve: 18076.5 %				Verified
-In Y direction:	Safety reserve: 19762.4 %				Verified
Oblique compression in the footing:					
Criteria of CYPE	Maximum: 5000.06 kN/m²	Calculated: 1290.31 kN/m²			Verified
Shear in the footing:					
-In X direction:	Shear: 61.80 kN				Verified
-In Y direction:	Shear: 61.61 kN				Verified
Minimum depth:					
Criteria of CYPE	Minimum: 15 cm	Calculated: 100 cm			Verified
Space to anchor starter bars in foundation:					
-P6:	Minimum: 28 cm	Calculated: 92 cm			Verified
Minimum geometric ratio:					
Article 5.4.2.1.1 (EUROCODIGO-2 Code)	Minimum: 0.0015	Calculated: 0.0016			Verified
-Bottom reinf. X direction:	Calculated: 0.0016				Verified
-Bottom reinf. Y direction:	Calculated: 0.0016				Verified
Minimum diameter of the bars:					
-Bottom reinforcement mesh:	Minimum: 10 mm	Calculated: 16 mm			Verified
Criteria of CYPE					
-Bottom reinf. X direction:	Maximum: 30 cm	Calculated: 13 cm			Verified
-Bottom reinf. Y direction:	Calculated: 13 cm				Verified
Minimum bar spacing:					
Recommendation from the book "Cálculo de estructuras de cimentación", J. Calavera, ed. INTEMAC, 1991	Minimum: 10 cm				Verified

Fig. 15.104

15.2.4.4.3. Footing overlaps

Overlapping does not occur in this example however, it is useful to know that if it does arise, an option is available to create a combined footing by selecting the route **Foundation > Eliminate overlaps**. They can also be joined manually (**Join** option in the Foundation elements dialogue box) if the distance between them is less than 50 cm.

15.2.4.4.4. Results check

Follow the steps to obtain quick information on the data entered and results of the footings:

- Click on **Foundation > Foundation elements > Information**. A window will appear with options of what information to show.

- Click on the footing below **P6**.

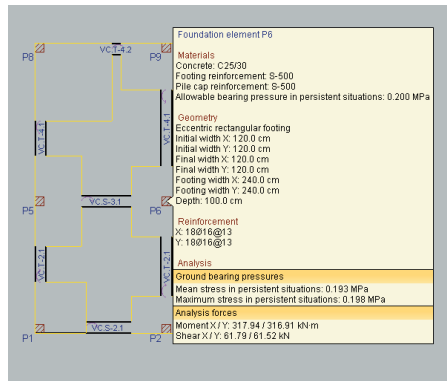



Fig. 15.105

15.2.5. Reinforcement modifications

Having checked the forces, sections, etc., the reinforcement can be edited.

15.2.5.1. Columns

The column reinforcement can now be matched.

- Select the **Results** tab.
- Click on **Columns > Edit**. Click on a column. Click on the  icon in the bottom left hand corner of the **Column edition window**.
- In the emerging window, **P1** is marked with a **bold** circle, which indicates it is the selected column of the group. Columns **P3-P8-P9-P10** are also marked with circles to indicate they belong to the same group as the selected column. If column **P5** is selected, for example, it will be marked with a **bold** circle, to indicate it is the selected column.

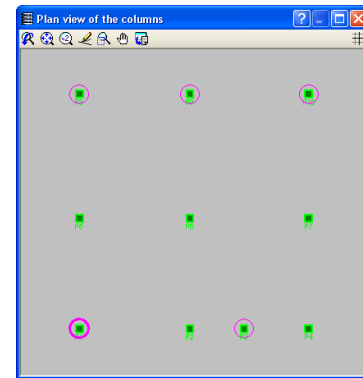


Fig. 15.106

- Click on **P4**. It will be displayed in a **bold** circle (because it is the selected column), **P2** can be seen displayed in a circle. This indicates that **P2** belongs to the same group as **P4** because they have the same geometry and have obtained the same reinforcement after the analysis. They have been grouped by CYPECAD so they may appear as a single entry in the column schedule in the drawings.

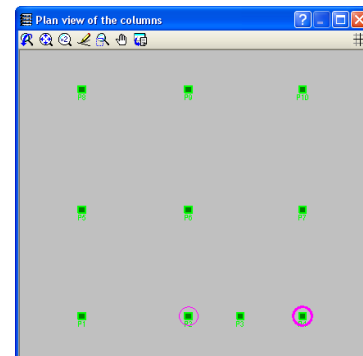


Fig. 15.107

Column sections and reinforcement can be copied from one group to another. **Click on Groups > Copy geo-**

metry and reinforcement to other groups (in the top left hand corner of the **Column edition** window). The Group selection window appears. In this case, the cursor was situated on the P2 and P4 row in the main window, hence the grouping possibilities of this group are displayed i.e. columns P6 and P7.

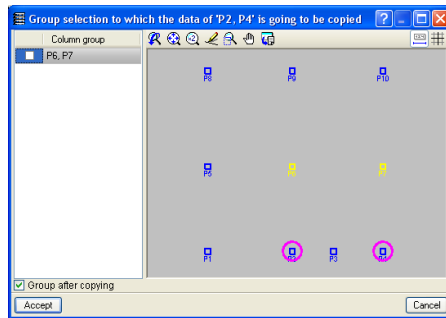


Fig. 15.108

Columns P6 and P7 are displayed in yellow on the floor layout plan and listed in the column on the left. Mark the boxes in the column and close the window. P2 and P4 are now grouped with P6 and P7.

The program now displays these two newly added columns in red and red crosses appear in the **Reinforcement edition** section. This is because they fail with their newly assigned reinforcement. Click on the **Redesign reinforcement** in the top right hand corner of the **Column edition** window and the program will provide the required reinforcement for the new group.

Manual modifications can also be carried out. If any modifications cause the column span to fail, a cross will appear at the end of the span in the **Reinforcement edition** section and in the **Column group** section. The columns will also be displayed in red in the **Plan view of the columns** section and their corresponding **Summary of code checks** is highlighted in red.

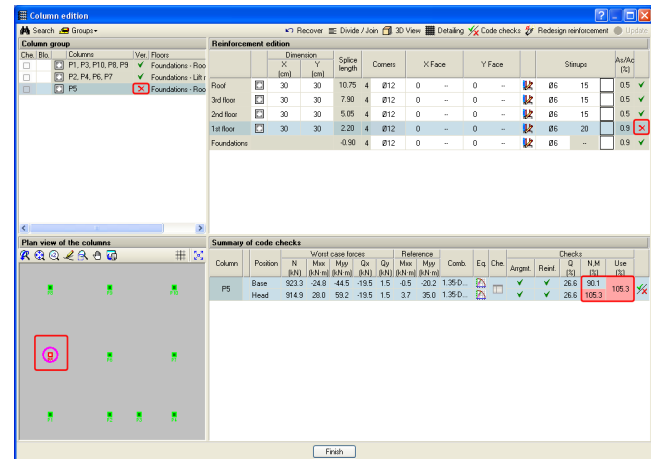


Fig. 15.109

- Press **Finish** to conclude.

15.2.5.2. Beams

- Go to group 3. It can be seen that beam **P9-P10** has a deflection error that can be solved by slightly increasing its reinforcement.
- Click on **Beams/Walls > Edit beams**, followed by a click on the aforementioned beam. The Beam reinforcement editor window will open.

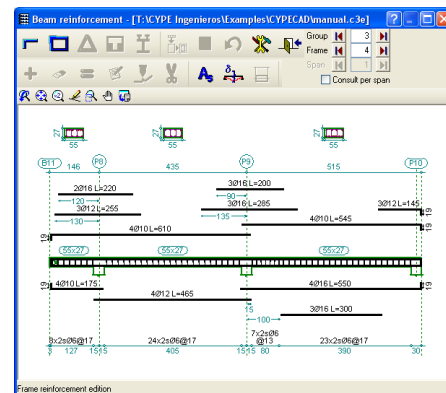





Fig. 15.110

- Click on the deflection icon  and place the mouse cursor over beam **P9-P10**. Its current deflection is displayed; in this case the active and long term deflection fail (displayed in red).
 - Click on the **Longitudinal** reinforcement icon  followed by the **Edit** icon  **d=0**.
- This operation will be used to modify the longitudinal reinforcement.
- Click on the diameter symbol of the top reinforcement of beam **P9-P10** and change the diameter of the top and bottom bars to 20 cm.

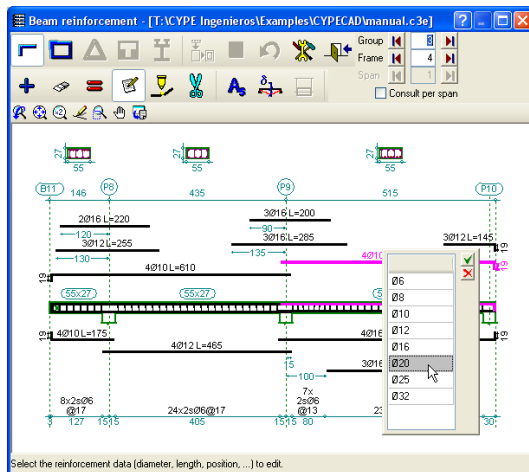



Fig. 15.111

- Click on the **Resituate** icon  for the program to check the new reinforcement.
- Check the deflection of the beam to ensure it lies within the limit.

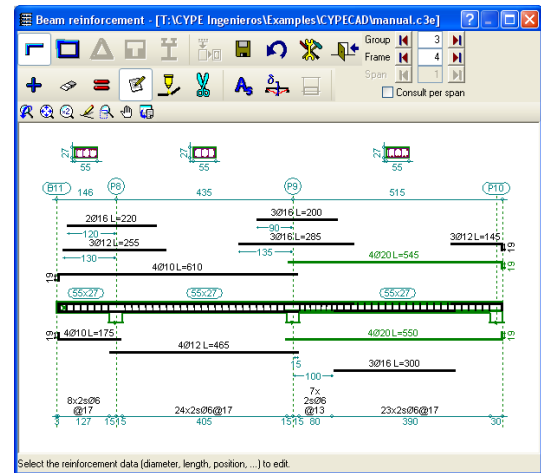


Fig. 15.112

As can be seen, by consulting the beam reinforcement of the beam, it can be modified without having to change the section of the beam to solve errors that arise in the design, for example with deflection problems if it is not excessive.

If the beam only fails due to deflection, the top or bottom reinforcement can be modified until the error is solved.

The deflection is automatically recalculated upon indicating the new reinforcement.

15.2.5.3. Joist floor slabs

It is convenient that moments (or reinforcement) be matched, as the analysis provides individual bottom reinforcement for each joist.

- Go to Group 1. Click on **Joists > Bottom reinf. > Match**. A new window will open with the percentage difference for the match and the criteria to be taken for the match; whether it should take the maximum or average value.

In the case, the **Average** value will be selected and the percentage set at **50%**. This is because joists passing by columns, upon the stiffness of the support being greater, the amount of top reinforcement increases greatly and the bottom reinforcement is equally reduced.

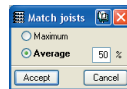


Fig. 15.113

That is what really occurs, however construction criteria is of greater importance than the theoretical analysis values.

- Click on **Accept** and the match will be carried out.

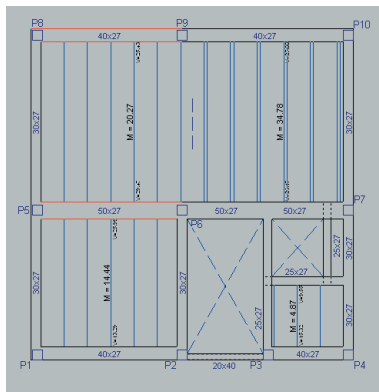


Fig. 15.114

- Now match the reinforcement on the remaining floors. All that has been said for the bottom reinforcement is applicable to the top reinforcement.
- Activate the visibility of the top reinforcement: **Joists > Views**.
- Click on **Joists > Top reinf. > Match**. A window will open displaying the percentage difference for the

match. In this case, only the lengths will be matched (25% difference), therefore different steel areas will not be matched. The lengths will be matched using the average value criteria.

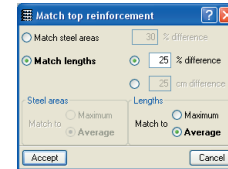


Fig. 15.115

- Click on **Accept** and the match will be carried out. Bars shown in magenta are those that have been matched with others and will not be displayed in the drawings.

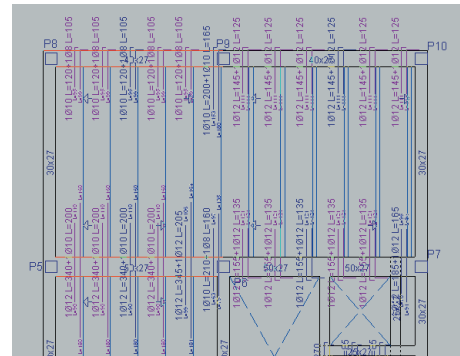


Fig. 15.116

- To deactivate the visibility of the hidden bars click on **Joists > Views** and deactivate the **Show hidden** box.

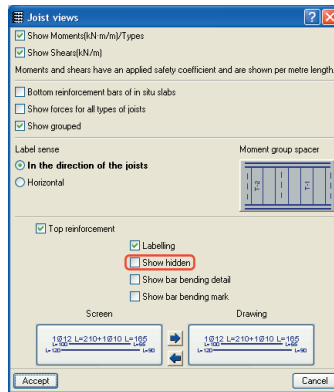


Fig. 15.117

If the match that has been carried out is not sufficient, another option can be used which allows to copy the reinforcement of other joists by following the steps indicated below:

- Click on **Joists > Top reinf. > Modify**.
- Select the reinforcement to modify. A window will appear with a description of the reinforcement.

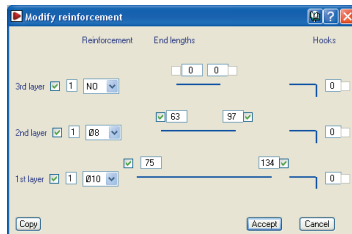


Fig. 15.118

- Click on the **Copy** button.
- Click on the reinforcement to copy. The Modify reinforcement window will open and display the reinforcement of the selected bar(s).

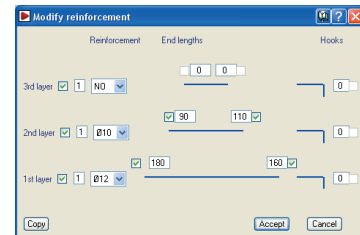


Fig. 15.119

- Click on **Accept**.
- Once all the necessary modifications have been carried out, click on **Joists > Top reinf. > Match**.
- Continue matching the reinforcement on the remaining floors.

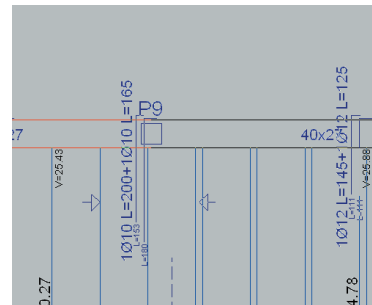


Fig. 15.120

The top reinforcement of the joists can also be matched by matching the steel areas and lengths of the bars. To do so, in the Match top reinforcement window that appears, select the **Match steel areas** option.

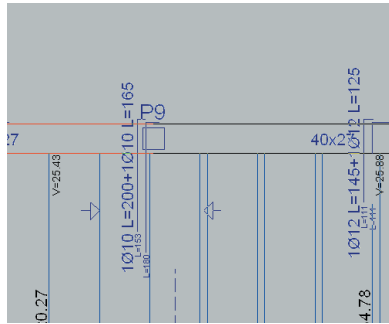


Fig. 15.121

A more uniform result is obtained by using this matching method. In some occasions, the negative moment of a joist that coincides with a column is substantially different to the other joists of the same panel. By matching the steel areas, the negative moment difference is shared amongst the joists adjacent to the joist that coincides with the column (it is logical for the joist to behave like this in reality), and so the difference with the remaining beams of the slab is not so large. This, occasionally, implies that all the top reinforcement of a slab must be matched with the same reinforcement or that, at least, the difference between the top reinforcement of the joists belonging to the same slab will be smaller.

15.2.5.4. Foundation

- Place the view of the screen at group 0
- When there are small differences in the size of the pad footings (rounded to multiples of 25cm), it is convenient to group them. This is done using the option **Foundation > Foundation elements > Match**.

No matching will be carried out in this exercise as there are no footings of the same type. Nonetheless, the size of the footings can be increased so that their total dimensions are multiples of 25 cm. This can be done in two different ways:

First option:

- Click on **Foundation > Foundation elements > Edit**.
- Click on the footing below **P6**. The following window will open.

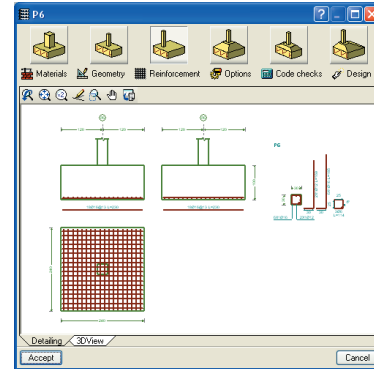


Fig. 15.122

- Click on **Geometry**. The dimensions of the footing will appear.
- Change the dimensions to what is shown in the following figure.

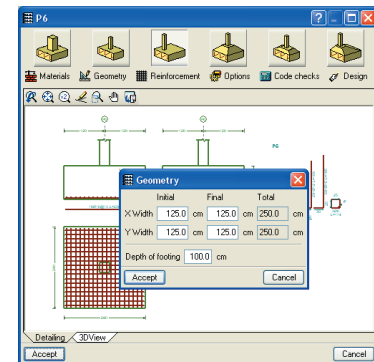


Fig. 15.123

- Now, to redesign the reinforcement, click on **Design > Redesign**.

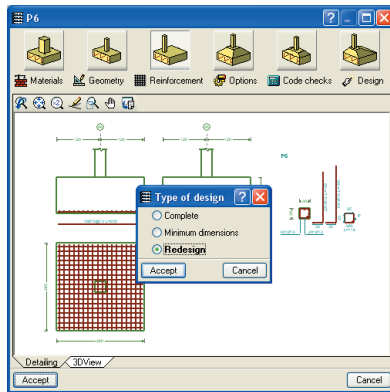


Fig. 15.124

Having done so, continue modifying the remaining pad footings.

Second option:

This option has the advantage that it is carried out on all the pad footing simultaneously.

- Click on **Job > General data**.
- Select the **By position** button.
- Click on the **Foundation** tab.
- Click on the **Strip and pad footing options** button.
- Select the **Pad footing options** button.
- In the window that opens introduce a value of 25 in the **Width increment** box.
- Redesign the foundation (activating the **iterative design** option).

It is also recommended that the strap (and tie beams, if there are any present) be matched. For the time being, there is not an option that carries out an automatic match, and so, it must be done manually. For example, to match

all the beams with the same section, but different reinforcement:

- Make a note of the strap footing with the greatest reinforcement for the same depth. In this example for type VC.T-4, the VC.T-4.2 and VC.T-4 are available. The first beam will be used as it has a greater stirrup reinforcement area.
- Select **Foundation > Strap and tie beams > Edit beam design**.
- Click on a VC.T-4 beam, for example the one joining **P6** and **P9**. The Strap beam edition window will appear.

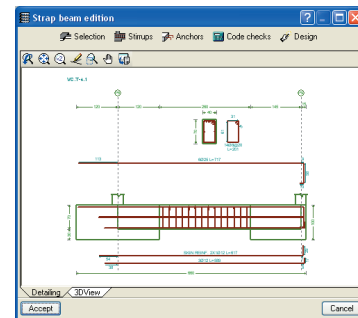


Fig 15.125

- Click on the **Selection** button.

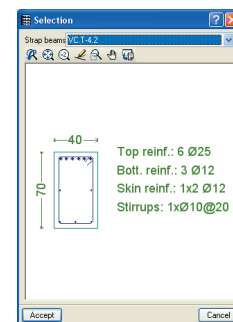


Fig. 15.126

- In the new window, a scroll down menu appears where a different strap beam may be selected amongst those available from the table. In this case, select the **VC.T-4.2** beam.

Repeat his process on all the necessary beams.

15.2.6. Application of finishing touches to texts before obtaining drawings

To avoid texts overlapping in the drawings, their positions may be modified. Texts overlapping reinforcement must also be avoided, etc. Therefore, all the reinforcement should be activated as visible beforehand. The following steps should then be taken:

- Click on **Groups > References** and right click.
- The **Column, joint and frame references** window will appear.

The **Modify position** option allows for any texts that are visible to be changed. But first, the visibility of these references must be activated.

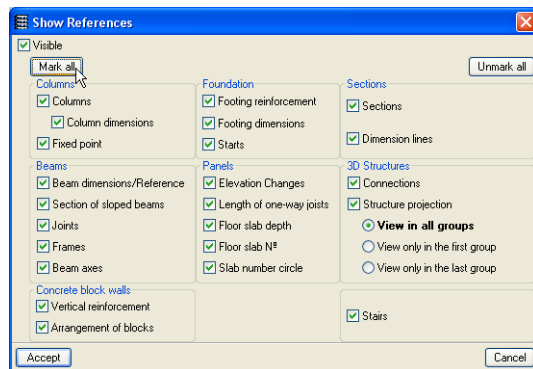


Fig. 15.127

- Click on the **Show references** button and activate all the boxes (Click on the **Mark all** button).
- Accept all the windows.
- Click on and move each of the overlapped texts or any that are to be moved.

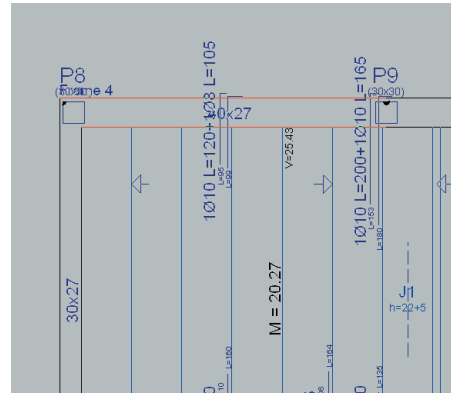


Fig. 15.128

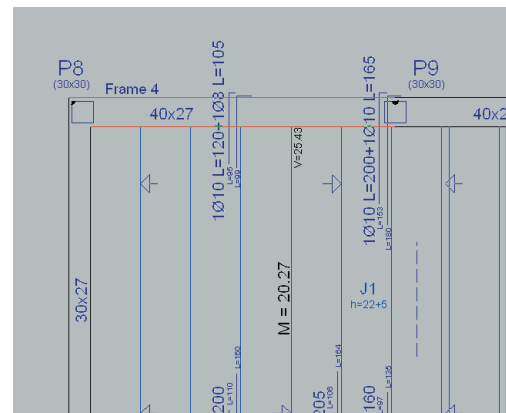


Fig. 15.129

As a final comment, the position of the texts can also be modified when the drawings are obtained.


15.2.7. Drawings

The next step consists in creating the job drawings.

The formats of the drawings do not have to be configured previously; the program carries this out automatically based on the format configuration defined in the Control Panel of the system, to which the program also adds the DXF and DWG configuration.

However, as there are various drawing sizes for DXF and DWG files (in this example the drawings to be obtained will be DWG files), the first thing to do is select a specific drawing size.

To do so:

- Access the following option: Click on the **Configuration icon**  > **Drawings**
- In the window that opens select the **A0-A4** format. This is the format created by the program to obtain DXF and DWG files.
- In the Sizes column, click on **A0**. Having done so, the size should appear marked in red in the window on the right, where all the sizes defined for the format are drawn.

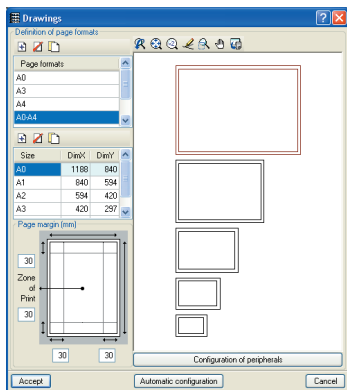


Fig. 15.130

- Close this window by clicking on **Accept**.

Now the drawings will be obtained. Follow the steps indicated below:

- Click on **File > Print > Job drawings**.
- In the Drawing selection window that opens, add a new element to the list. The **Drawing editor** window will open.
- Activate **Floor plans** from the **Drawing type** menu.
- Select for the drawings to be drawn from floors 1 to 4.
- Click on **Accept**.

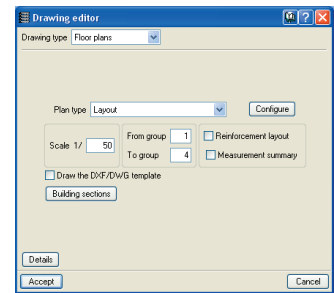


Fig. 15.131

The program will return to the Drawing selection window. The selected drawing will be displayed in the table.

- Select **DWG** in the **Peripherals** column.

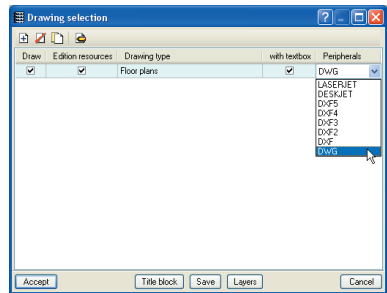


Fig. 15.132

In this case, the beam drawings should be drawn together with the floor plans. To do so:

- Add another element to the list in the **Drawing selection** window.
- Activate **Beam drawings** from the **Drawing type** menu.
- Select for the drawings to be drawn from floors 1 to 4.

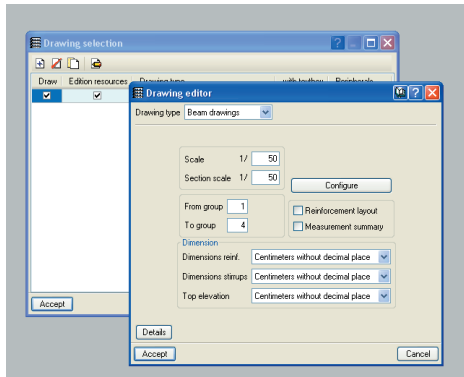


Fig. 15.133

A construction detail is to be added to this drawing.

- Activate the **Details** button. The **Details** window will open.

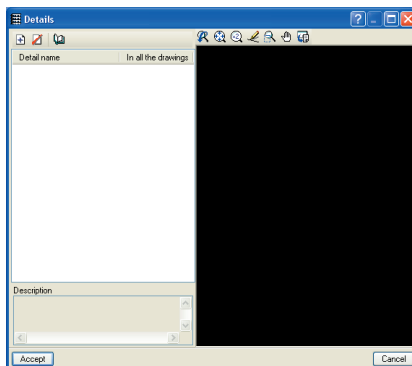


Fig. 15.134

- Add a new element to the list; the **Detail selection** window will open, which contains the construction details included in the program.
- Activate the **EHU** tab.
- Select detail **EHU399**.

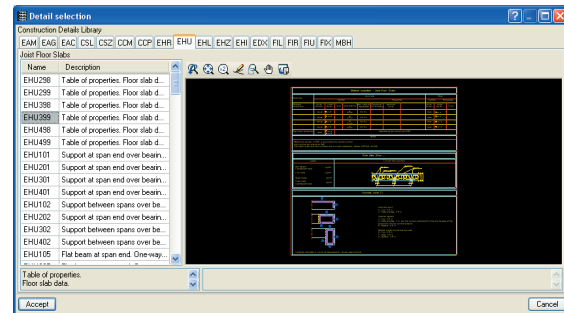


Fig. 15.135

- Click on **Accept** consecutively until the **Drawing selection** window is displayed.
- Choose **DWG** from the **Peripherals** menu.

A title block for the drawings is to be selected. In this case, the CYPE title block will be chosen.

- Click on the **Title block** button.
- From the **Title block selection** window that opens, activate **CYPE**.
- Click on **Accept**.

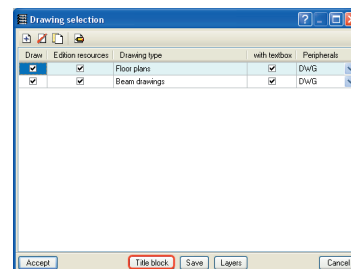


Fig. 15.136

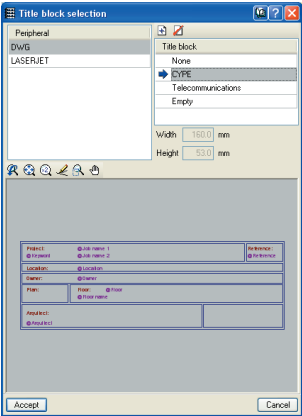


Fig. 15.137

The program returns to the **Drawing selection** window. A preliminary view of the drawings can now be seen.

- Click on **Accept**. The **Fields editor** window will appear where the data to be displayed in the title block can be entered.



Fig. 15.138

- Click on **Accept**.

The **Drawing composition** window will open, which offers a preliminary view of the drawings to be generated and allows for certain actions to be carried out, such as: change the scale, move the position of the image within the drawing, change text positions, etc.

Each floor, frame, etc. drawing is initially displayed as an object lacking in details, that is, a drawing where only its border is shown. This allows for the object to be redrawn more quickly if modified than if it were entirely displayed.

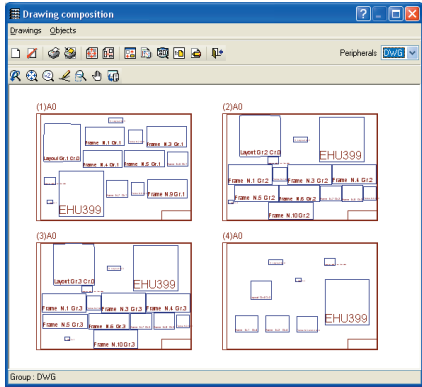


Fig. 15.139

- To view the real image of the drawing, click on **Objects** > **Drawing detail** and select the objects to be seen.

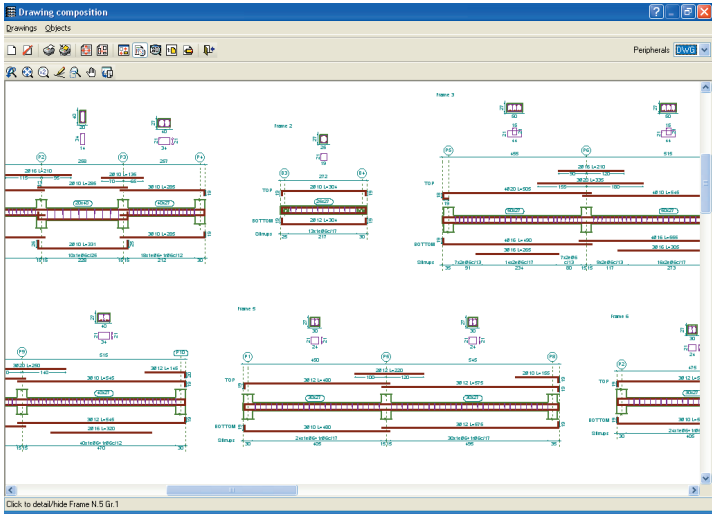


Fig. 15.140

- To change the position of the texts, select **Objects > Edit object** and click on an object. The Drawing editor window will open, in which a text can be moved by clicking on it and then at its new position.

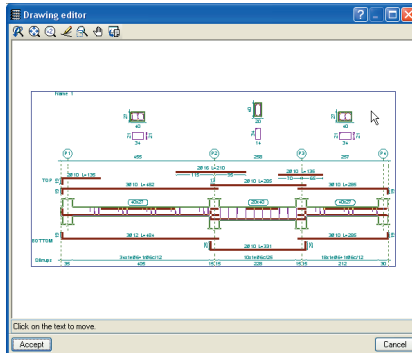


Fig. 15.141

To finally print the drawings:

- Click on **Drawings > Print > Print all**.
- The **Filenames** window will appear, where the file-names and directory can be specified.
- Click on **Accept**.

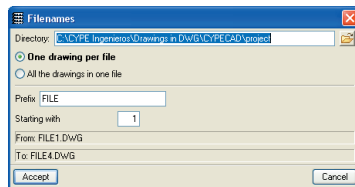


Fig. 15.142

15.3. Example 2

The following example is to be used as a guide for the correct introduction of a building, consisting of nine apartments, with its ground floor to be used for business pur-

poses (a shop or similar) and basement. Each project has its individual properties and generally cannot be extrapolated to other projects. However, in this example, solutions will be proposed to specific properties which can later be applied to future projects.



Fig. 15.143

15.3.1. Initial data

The project should be studied before it is introduced so to avoid errors and hence not waste any time having to repeat entering and redefining data.

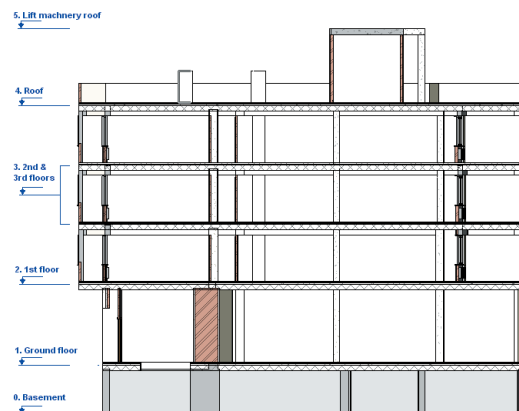


Fig. 15.144

First and foremost is to identify how many groups (levels) there are in the building. This example will contain 6: ground floor, first, second and third floor, roof and lift machinery roof. As floors 2 and 3 are the same, geometrically speaking, and their loading is also, these two floors can be grouped, resulting in a building of 5 groups and 6 floors.

NB: It must be taken into account that if there is going to be a start or end of a structural element (column, wall, stair nucleus or sloped beam), it cannot occur at a grouped floor; it must be defined as an individual floor.

15.3.1.1. Floor heights

When there is an elevation change in a floor, as occurs in this example where there is an area on the ground floor 42 cm higher than the rest of the floor, depending on the elevation difference and surface area of the floor slab, there are different ways as to how it can be defined in the program.

Group	Floor name	Elevation (m)
5	Lift machinery roof	15.7
4	Roof	12.2
3	3rd floor	9.4
	2nd floor	6.6
2	1st floor	3.8
1	Ground floor	0
0	Basement	-2.5

If the elevation difference is small (<0.65 m) or the surface of the higher area is not too large, the user may opt to infill the area, without altering the height of the floor slab. In this case, the only thing that must be done is apply a surface

load to the slab equivalent to the increase in dead loading corresponding to the infill. This is the case in this example, and so a load will be introduced to represent the infill.

If the elevation difference or surface area is large, an elevation change should be assigned to the floor slab using the option **Groups > Sloped floor slabs/EI. Changes**. This way, the program creates a floor slab with two elevations and if these are separated by a flat beam, the beam will change to acquire the depth of the slab plus the elevation change between the two slabs. If the elevation difference is greater and the user does not wish for there to be a connection between the two parts, an intermediate group will have to be created between the current and the following floor. In this case, the height of the intermediate floor will be the elevation change between the two floor areas.

15.3.1.2. Loads

15.3.1.2.1. Permanent loads

First the permanent loads acting in the example will be analysed so to then introduce them when defining the project in the program. Note that the slab loads, once the slabs to be introduced have been defined, will be applied internally by the program. Therefore, the user does not have to define them when introducing the dead loads to be applied.

Element	Load
Waffle slab with lost form 25+5cm	4.35kN/m²
Waffle slab with lost form 30+5cm	4.97 kN/m²
Partitions	1.00 kN/m²
Screed	1.10 kN/m²
Ground floor external walls	1.00 kN/m²
Remaining floor external walls	8.00 kN/m²
Guardrails	3.00 kN/m²
Infills	16.00 kN/m²

15.3.1.2.2. Live loads

The live loads acting on the floor slabs will depending on what they will eventually be used for. For this example, the following loads have been taken:

Group	Floor name	Use	Loading (kN/m ²)
5	Lift machinery roof	Roof accesible only for maintenance purposes	1.00
4	Roof	Roof accesible only for maintenance purposes	1.00
3	3rd floor	Residential	2.00
	2nd floor	Residential	2.00
2	1st floor	Residential	2.00
1	Ground floor	Residential	2.00
0	Basement	Shop (or similar)	5.00

15.3.1.2.3. Wind loading

For the wind loading, the generic code will be used. Here, the shape factors of the building will have to be established as well as the tributary widths.

15.3.1.2.4. Earthquake loading

The modal spectral analysis will be used to apply the seismic loads to the building. An acceleration of 0.10 g will be assumed.

15.3.1.2.5. Snow load

A snow load of 0.2 kN/m² will be taken for the analysis.

15.3.1.2.6. Fire resistance

The basement floor will have to verify a fire resistance of 120 min and the remaining floors a resistance of 60 min.

15.3.2. Job creation

Click on **File > New**. A dialogue box will appear where the name and description of the new project are to be defined.

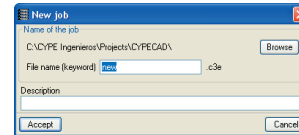


Fig. 15.145

Upon accepting, a new dialogue box will open where the user can choose to enter the structure in four different ways (the remaining two options are examples).

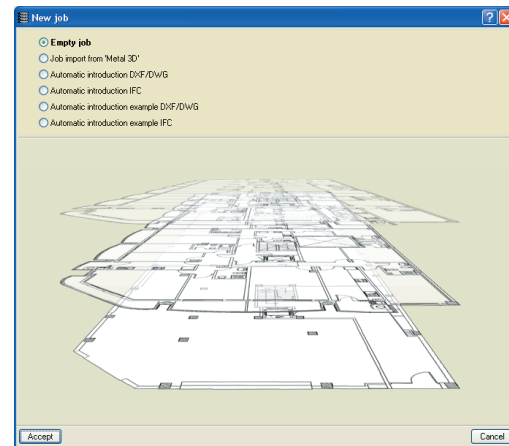


Fig. 15.146

- **Empty job.** If an IFC or DWG/DXF file of the project or is not available, the Empty job option should be selected.
- **Job import from Metal 3D.** If the project that is going to be created is based on one whose structure has previously been created in Metal 3D, then the option **Job import from Metal 3D** should be selected. This way, the program will ask for the Metal 3D job to be selected

which will then create a job in CYPECAD with the codes and loadcases that were defined when the steel structure was defined.

- **Automatic introduction option.** If DXF/DWG files of the project are available, the project can be introduced using the **Automatic introduction DXF/DWG option**.
- Finally, if the IFC file of the job is available, the work carried out previously in the CAD program can be made the most of to introduce the project using the **Automatic introduction IFC option**.

The job will be introduced using two methods (**15.3.2.1. Automatic introduction DXF/DWG** or **15.3.2.2. Automatic introduction IFC**) so to view the differences. Having concluded, the introduction will continue at point **15.3.3. Column introduction (Column Definition tab)**.

15.3.2.1. Automatic introduction DXF/DWG

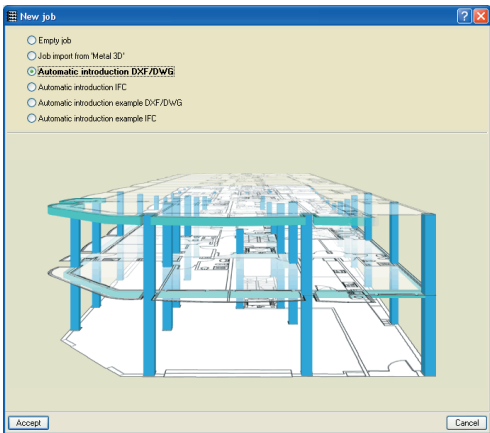


Fig. 15.147

Select the option and accept. A new dialogue box appears where the data of the building is to be defined.

15.3.2.1.1. Assistant: Building data

The Basements box will remain activated with a 1 introduced in the adjacent box indicating there is one basement. It is to have a height of 2.50 m and no live load or dead load as the structure is going to have pad footings.

The Ground floor is to have a height of 3.80 m with a live load of 5 kN/m² and a dead load of 2.1 kN/m² (partitions + screed). For the Floors, the box is to remain activated, introducing a 3 in the adjacent box to indicate there are three floors. They are to be assigned a height of 2.8m, a live load of 2 kN/m² and a dead load of 2.1 kN/m².

The Roof is also to have a height of 2.8 m with an applied live load of 1.2 kN/m² (live load + snow load) and a dead load of 1.5 kN/m² (roof screed).

Finally, the Lift machinery slab box is to remain unactivated. The Lift machinery roof box is to be activated and is allocated a height of 3.5 m. Its loading will be the same as that of the Roof.

Before accepting and moving on to the next window, the **Columns have been introduced with external fixity** box will remain activated as the foundations of the building will consist of pad footings which require for the columns to be introduced with external fixity.

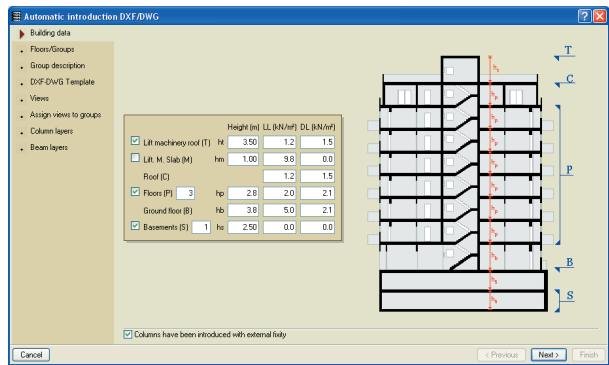


Fig. 15.148

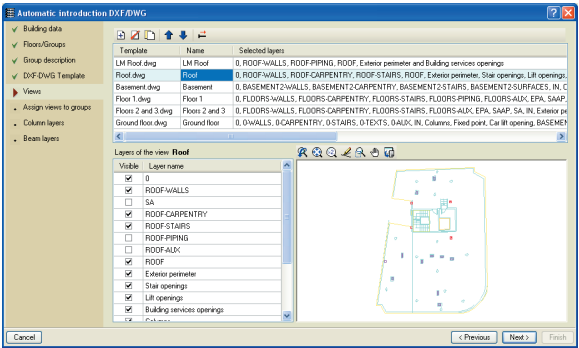


Fig. 15.152

15.3.2.3.1.6. Assistant: Assign views to groups

In this window, each group will be assigned a view or CAD file. If the name of the file coincides with that of the group, it will be assigned automatically, otherwise a group must be selected from the list on the left and activate the box of the corresponding view or file to be assigned from the list on the right. The process is to be repeated for each of the floor groups of the project.

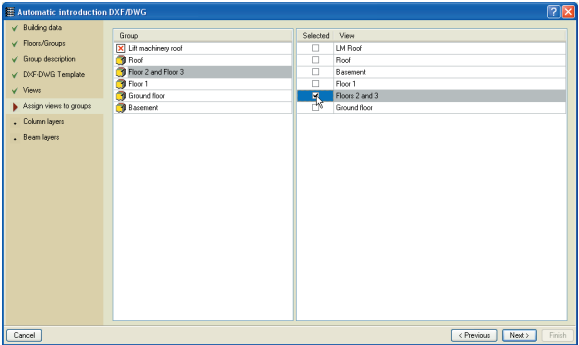


Fig. 15.153

15.3.2.3.1.7. Assistant: Column layers

The columns have been drawn in the same layer for all the groups and so the first of the two options at the top is to be marked. The layer in which the columns have been drawn (**Columns**) is to be selected from the table and and, as the fixed point of the columns has been drawn in the CAD file, the **Layers defining the fixed point** box should be activated and then the corresponding **Fixed point** layer selected.

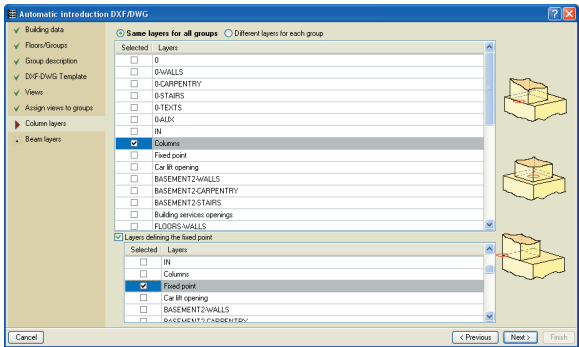


Fig. 15.154

15.3.2.3.1.8. Assistant: Beam layers

The same is to be done for the beams. First the layer corresponding to the exterior perimeter of the floors is selected. Once this has been done, the beams are assigned a width of 30 cm and due to the drawing template being that of the finished floor, a 5 cm displacement towards the inside is to be applied.

For the openings, select the following layers: Lift opening, stair opening and car lift opening. As in the case of the beams, these are assigned a width of 30 cm but without an applied displacement. Finally, select the Building services openings layer and assign it a non structural 15cm wide beam.

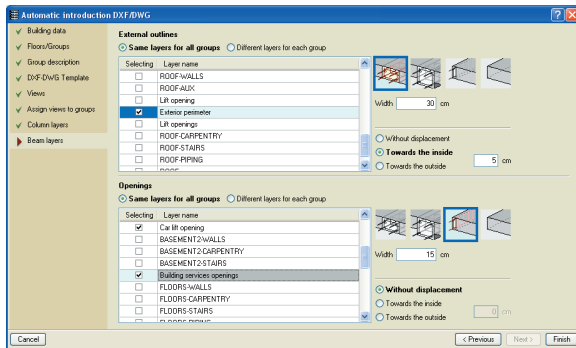


Fig. 15.155

Click on **Finish** to conclude the automatic introduction. To continue with the example, proceed to point **15.3.3. Column introduction (Column Definition tab)**.

15.3.2.2. Automatic introduction IFC

15.3.2.2.1. IFC Assistant: IFC file selection

Using the Automatic introduction using an IFC file, a dialogue box opens where the IFC file has to be selected. In this case select the file: Project.ifc.

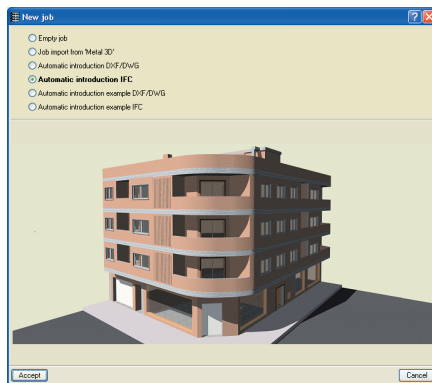


Fig. 15.156

Once the file has been imported, a view of the file is displayed. Below it, a description of any errors that have been incurred during its import.

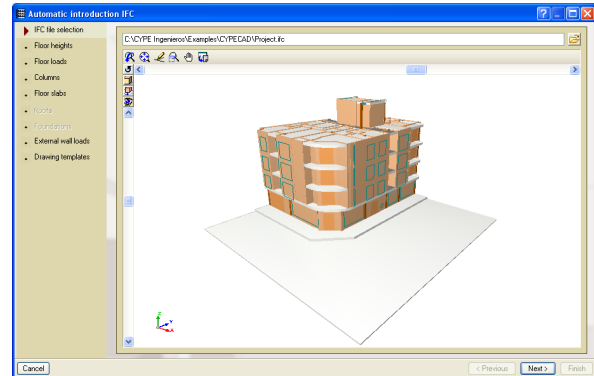


Fig. 15.157

15.3.2.2.2. IFC Assistant: Floor heights

By clicking on the **Next** button, the Floor heights window opens where the floor structure with which the IFC file has been created can be seen. Below the diagram, the elevation of the foundation plane can be modified. In this case introduce an elevation of -2.5 m. Due to the type of foundation to be used (pad footings), the **Fixed (without external fixity)** option is selected.

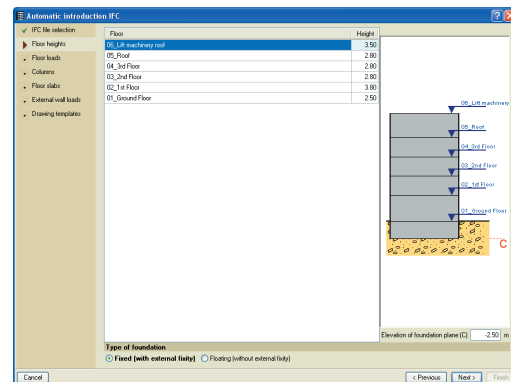


Fig. 15.158

15.3.2.2.3. IFC Assistant: Floor loads

Here, the live and dead loads will be introduced. The Basement does not have any loading as the structure's foundation elements will consist of pad footings.

The ground floor is to have a live load of 5 kN/m² and a dead load of 2.1 kN/m² (partitions + screed). The residential floors are to have a live load of 2 kN/m² and a dead load of 2.1 kN/m². Finally, the roof and lift machinery roof are to have an applied live load of 1.2 kN/m² (live load + snow load) and a dead load of 1.5 kN/m² (roof screed).

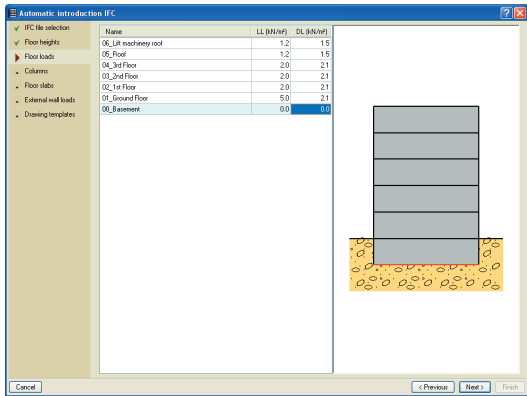


Fig. 15.159

15.3.2.2.4. IFC Assistant: Columns

Having defined the floor loads, the columns are now to be defined. The assistant displays the different materials with which the columns have been defined; in this case only one. The columns made of the same material as that displayed in the list are drawn in red.

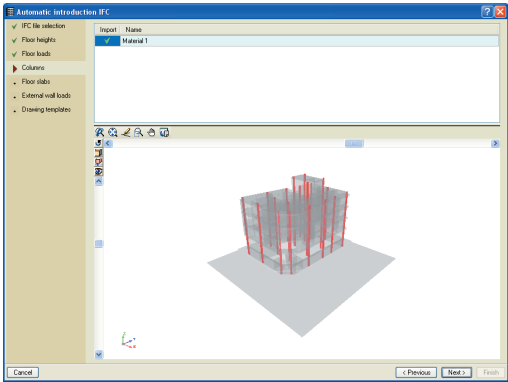


Fig. 15.160

15.3.2.3.2.5. IFC Assistant: Floor slabs

Now that the column materials to be imported have been selected, the floor slab materials are to be selected as well as their perimeter beams and the beams defining the openings.

The assistant displays a similar list to that of the columns, with the floors/floor slabs defined in the IFC file. Upon clicking on one of these, it is displayed in red in the 3Dview. Those that are not to be imported are deactivated in the Import column (Floor: road, Floor: pavement, Floor: Generic 100, Floor: Generic 200 mm).

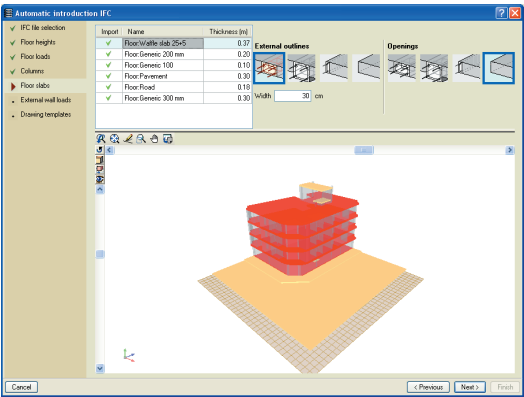


Fig. 15.161

Once these have been deactivated, click on Floor: Waffle slab. To the right, in External Outlines, select the flat beam (first on the left), assign a thickness of 30 cm to it and for Openings select the limit beam (first on the right) with 0 cm thickness.

Repeat the process for Floor: Generic 300.

15.3.2.2.6. IFC Assistant: Partition loads

Now that the floor slabs have been defined, the partitions to be imported are selected. These are assigned a material which has an associated surface load. The program will then calculate the corresponding line load to apply on the floor slab depending on the height of the partition.

In a similar way as with slabs, upon clicking on a partition from the list, it will appear marked in red. To assign the material, click on the Not defined button to select the type. In the example a generic material has been used with a load of 2.86 kN/m².

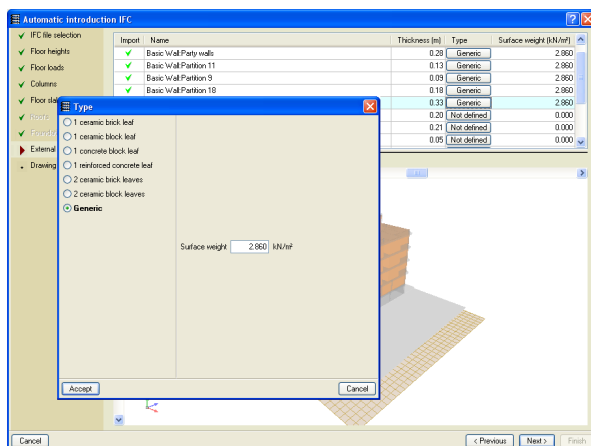


Fig. 15.162

All the materials corresponding to partitions will be deactivated in the Import column.

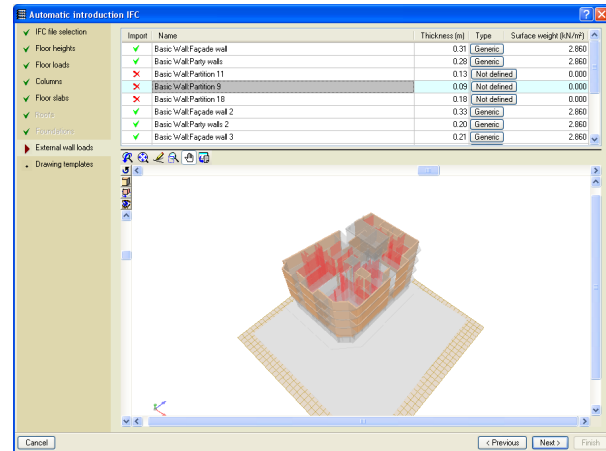


Fig. 15.163

15.3.2.2.7. IFC Assistant: Drawing templates

Lastly, the elements which are to appear in the DXF/DWG template generated by the program based on the imported IFC file will be selected.

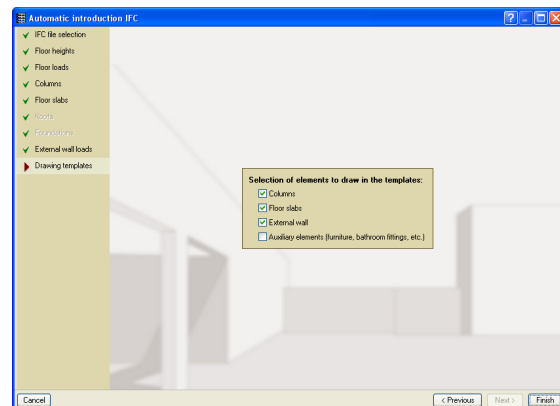


Fig. 15.164

15.3.3. Column introduction (Column Definition tab)

15.3.3.1. Job general data

15.3.3.1.1.Code and materials

Once a name has been given to the project, the **General data** window appears. The codes to use, materials and loads to be applied in the job are to be specified in this dialogue box.

For this example the Eurocodes will be used. By clicking on the Codes button, a new dialogue box will appear where the codes to be used for each material can be selected:

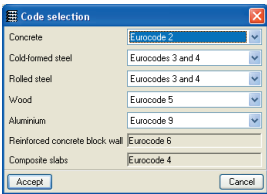


Fig. 15.165

The type of concrete to be used is C25/30, steel for bars S-500 and for cold-formed, rolled and welded S 275.

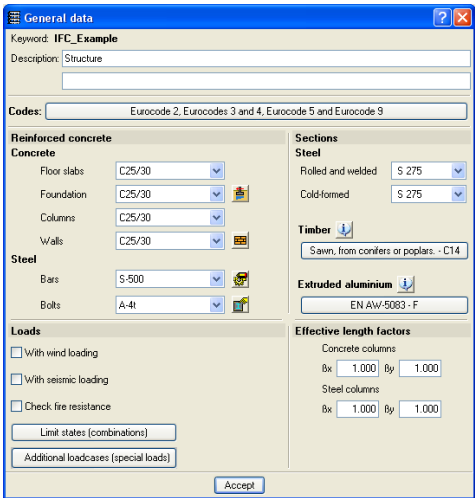


Fig. 15.166

15.3.3.1.2. Wind

In the Loads section of the dialogue box, activate the **With wind loading** box. The **Generic code** is to be used. In the new window that opens, introduce the tributary widths of the building. Click on the **By floor** button and introduce a Y value of 23 m and an X value of 17m for groups 1 to 5 and Y = 4 m and X = 7 m for group 6. Change shape factor x to 1 and shape factor y to 1.35. Leave the gust factor and wind action values as they appear in the program.

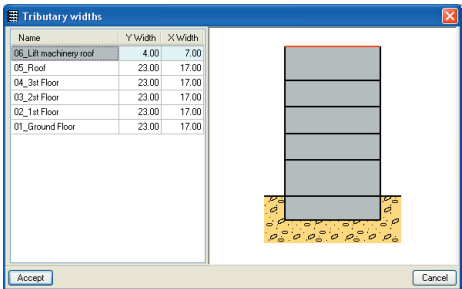


Fig. 15.167

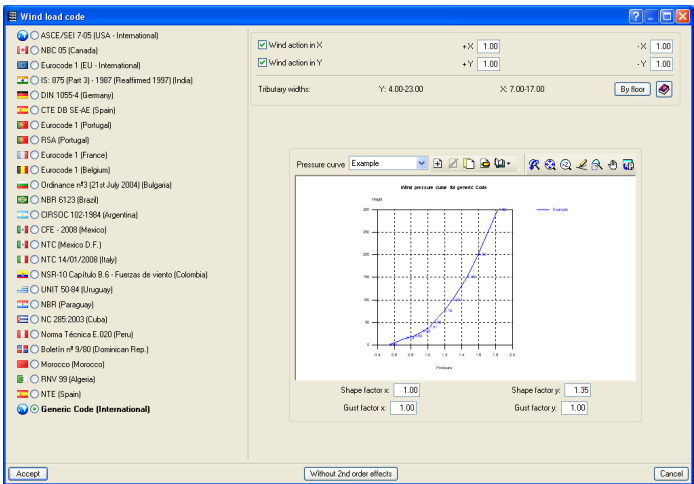




Fig. 15.168

15.3.3.1.3. Earthquake loads

The seismic loading can be activated by selecting the **With seismic loading** box in the **General data** dialogue box. From the window that opens, select the **Modal spectral analysis** option on the left. The corresponding screen will appear on the right where the data can be defined.

For this example, an acceleration due to gravity of 0.10 g will be used. The ductility shall be left at 2.50 and the part of snow and live loads to consider will be left at 0.5.

As for the number of modes, the number to be selected must be such that the whole mass of the structure is displaced. 3 modes of vibration can be considered for each independent zone, i.e. for this example 18 vibration modes should be considered (6 floors with 3 vibration modes each). Bear in mind that the higher the number of modes is considered, the longer the analysis will take as this number affects the number of combinations the program undertakes when designing the reinforcement. Therefore it is preferable, for the initial analysis whereby what is sought is to obtain a rough idea as to the eventual dimensions of the elements, that the 6 default modes be left. Once the first analysis has concluded, the report containing the participation coefficients may be consulted and if the percentage of the displaced mass is greater than 95%, then the number of modes considered is adequate, otherwise it must be increased.

To select the spectrum, click on the Library manager button  and select Import predefined data . From the list that appears, mark the box corresponding to EUROCODE 8 Type:II Subsoil:A.

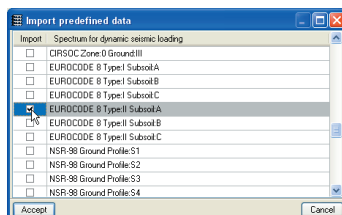


Fig. 15.169

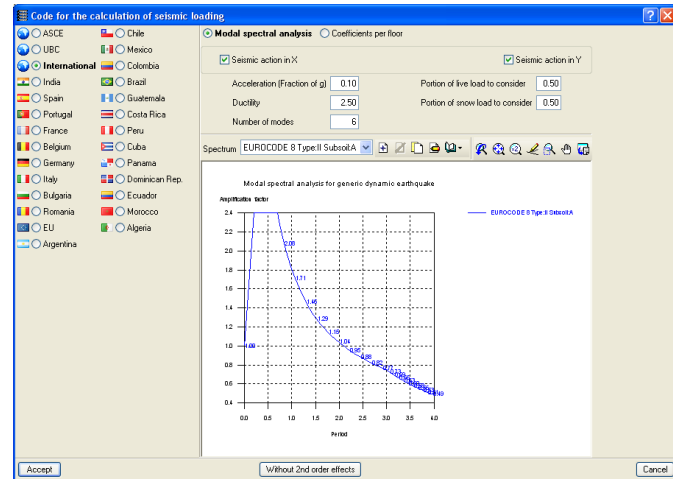


Fig. 15.170

15.3.3.1.4. Fire resistance

To carry out the fire resistance checks and the structural design with the correct fire resistance, select the Check fire resistance box. A window will open where R120 must be activated for the Ground floor and R60 must be activated for the remaining floors. Floors with compartmentation function must also be activated.

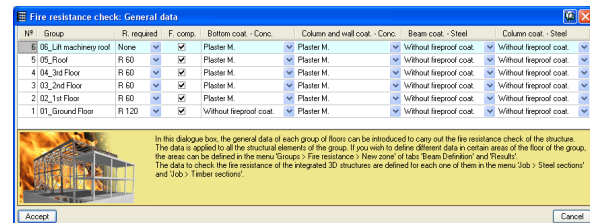


Fig. 15.171

Accept all the dialogue boxes that appear (one of which may include a report displaying introduction errors and warnings) until the floor layout can be seen on screen.

15.3.3.2. Creation of project floors and groups

15.3.3.2.1. Group floors

If the job has been imported using an IFC file, any floors that are to be grouped should be specified. As was mentioned in section 15.3.1., floors 2 and 3 are to be defined as a single group. To do so, go to option **Introduction > Groups > Join groups** in the **Column definition** tab. In the window that opens, a diagram of the floors is displayed with a red bracket indicating which floors are to be joined. Select floors 2 and 3.

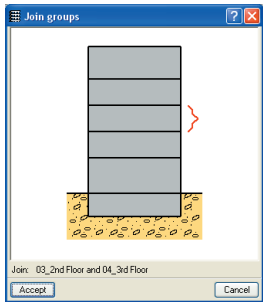


Fig. 15.172

Having selected the floor and accepted the window, a dialogue box will appear asking from which floor the information for beams and loads is to be kept. As both are the same, any can be selected.

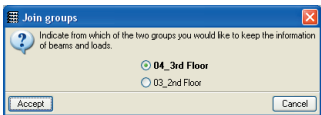


Fig. 15.173

15.3.3.3. Column introduction

Columns may be introduced manually or via automatic introduction. In this second example, the structure has been introduced automatically. Please refer to the first example for a more detailed explanation as to how to introduce the columns manually.

After importing the job using one of the two automatic assistants, the columns must be reorganised so that their references follow an order which allows for an easy consultation. This is especially important for large jobs where having to locate a column without these following a specific order can be very complicated.

To edit the references, click on **Introduction > Columns, shear walls and column starts > Modify reference**. Click on the bottom left hand column; this will be the first reference. In the dialogue box that appears, provide a reference for this column and select the **Number consecutively** box. Then, one by one, click on the columns in order, from left to right and bottom to top.

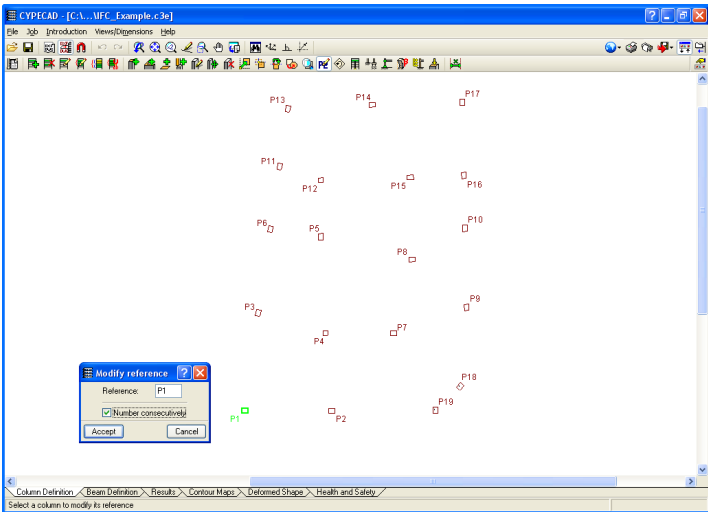


Fig 15.174

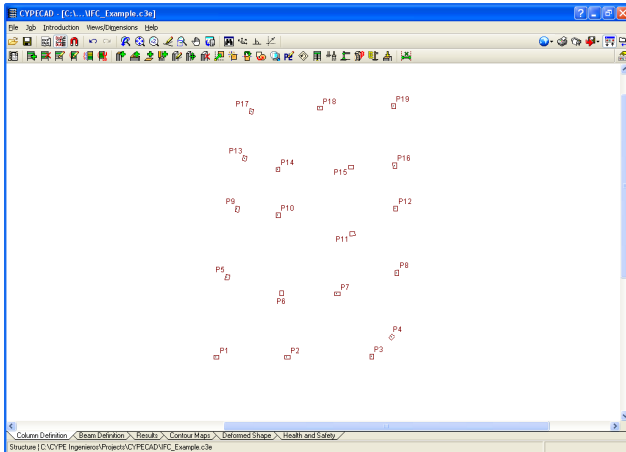


Fig. 15.175

15.3.4. Introduction of beams, walls and floor slabs (Beam Definition tab)

15.3.4.1. Walls

Situate the main view of the screen on the Ground floor to introduce the basement wall. If the previous data introduction has been carried out using the Automatic introduction option or by IFC file import, the beams marking the perimeter of the building must first be deleted: **Beams/Walls > Delete** and click on the beams to eliminate.

So that no overlapping problems arise when introducing the wall, the beams perpendicular to it will also be deleted. If any beams have been introduced in the Basement, these too must be deleted.

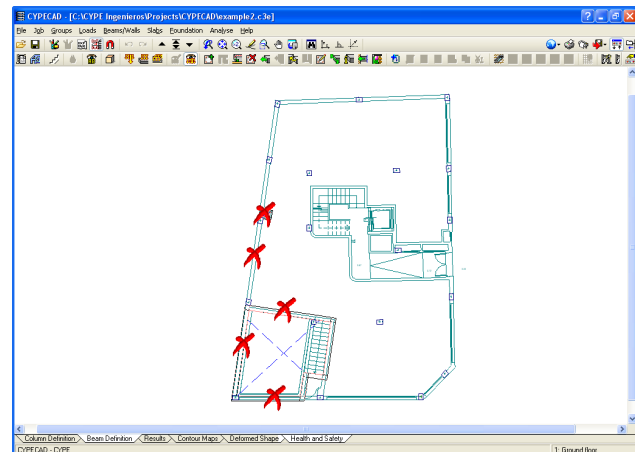


Fig. 15.176

Now the wall can be introduced. (Trick: the entire beam does not necessarily have to be deleted; using the option **Beams/Walls > Define beam**, introduce a beam segment perpendicular to the internal beam that is to be deleted at roughly its mid-point. Delete the half that would be in contact with the wall and the beam segment. Later on, once the wall has been introduced, all that has to be done is extend the beam (**Beams/Walls > Extend beam**). This way the original position of the beam is not lost.)

To introduce the wall go to **Beams/Walls > Define wall**. A dialogue box displaying four options will appear:

- Reinforced concrete wall
- Normabloc wall
- Reinforced concrete block wall
- Masonry walls



Fig. 15.177

The first wall type is to be selected (reinforced concrete wall). In the emerging window, indicate the wall begins at group 0 and reaches group 1. It will be assigned a thickness of 0.15 m to the left and right.

Now to define the pressures acting on the wall due to the soil: Click on the **Lateral pressures** button. The **Wall Pressure edition** dialogue box will appear. Click on the Edit soil pressure button.

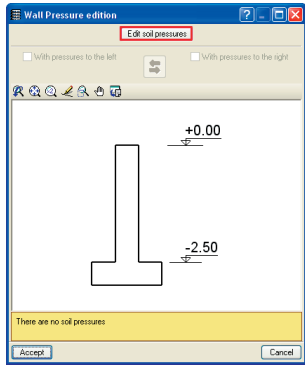


Fig. 15.179

The **Infill situation** dialogue box will open. The pressures will be defined as dead loads with no additional loadcases. Click on the **Soil** button so to define it.

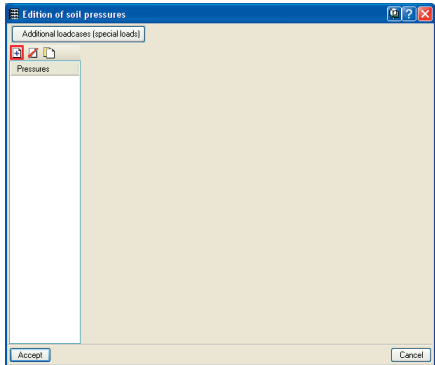




Fig. 15.180

In the **Soil edition** dialogue box, mark the **With fill** box. Indicate it is to reach an elevation of 0m, and leave the remaining parameters as they are. In the **Loads** section, a uni-

form load of 10 kN/m² is to be defined: click on the  icon. A default load will appear in the table. If required, this can be edited using the  icon and clicking on it.

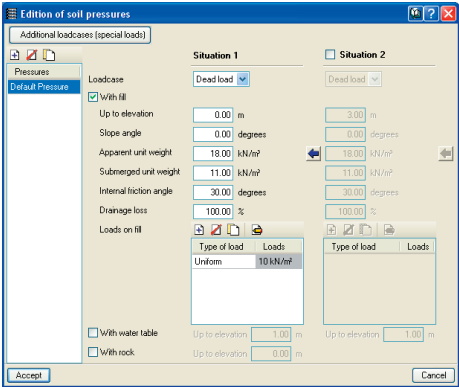


Fig. 15.181

Having accepted the **Earth pressures on walls** dialogue box, the **Wall pressure edition** window will emerge in which the direction of action of the pressure has to be defined. The wall is going to be introduced in an anticlockwise sense and so, the **With pressures acting to the right** box should be marked. **Accept** the window.

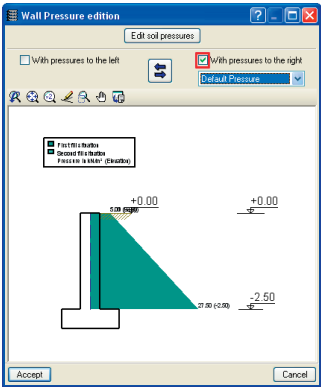



Fig. 15.182

Finally, before introducing the wall, the foundations must be defined. In this case the **With external fixity** box will be marked and click on the **Edit foundation** button  to establish the type of footing. The **Foundations. With external fixity** window will open. Here, mark the **Strip footing** and **Overhang only on the left** options. The depth and overhang dimensions are to be left as they are. **Accept** the dialogue box.

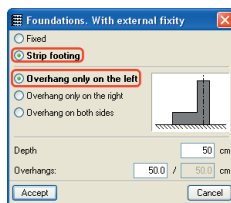


Fig. 15.183

The defined wall can now be introduced. Click on P1 and move the cursor to P3 (There is no need to click on P2 first due to columns P1, P2 and P3 being aligned). Finish off the introduction of the wall with a click on P3.

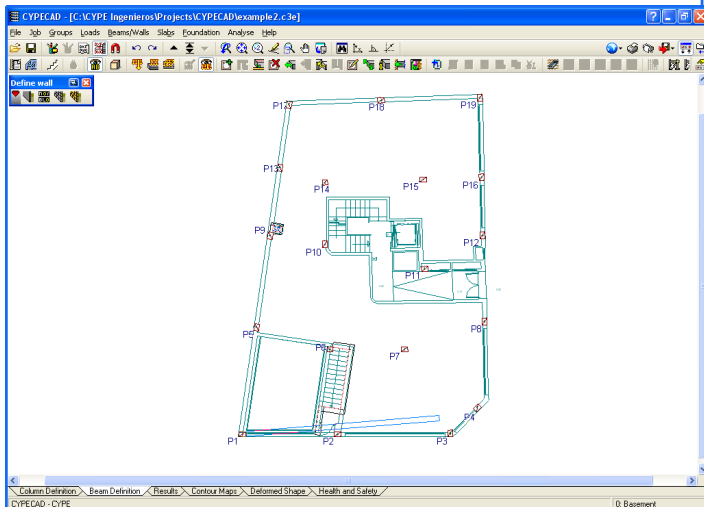



Fig. 15.184

Click on P3 to start the new wall span and then on P4 to finish it off. As there is an irregular intersection between columns P4 and P8, the wall will not be introduced and will be dealt with later on. Continue introducing the rest of the wall: P8-P19, P19-P17 and P17-P1.

The wall must now be adjusted. Click on **Beams/Walls > Adjust**. Once the option has been selected, press the F3 button on the keyboard (or the magnet icon on the toolbar ) which will activate the **Object snap selection** window. **Mark the Activate object snaps (F3)** and the **Nearest** boxes and **Accept**.

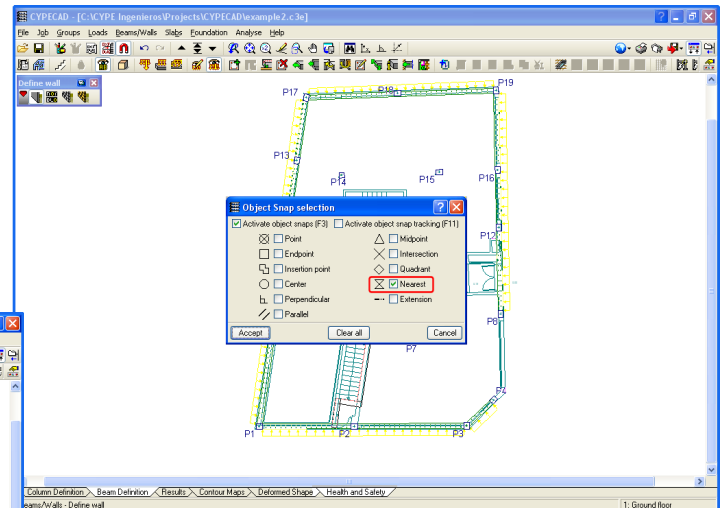


Fig. 15.185

Place the mouse cursor on the side of the wall which is to be adjusted and close to the CAD template. The nearest symbol will appear. At that moment click with the left mouse button and the wall will be adjusted to the template. Repeat this process with all the walls.

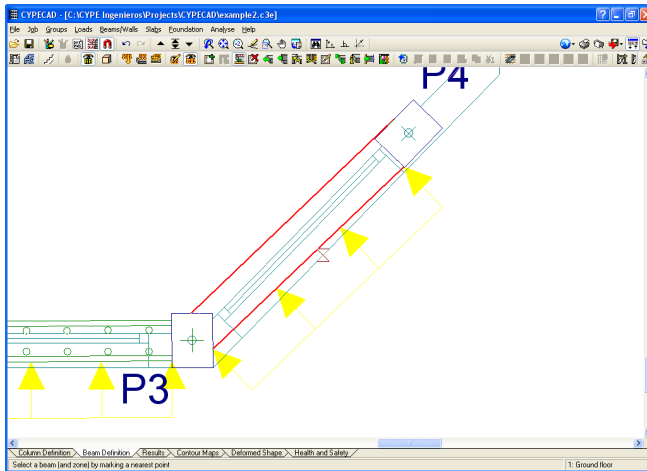


Fig. 15.186

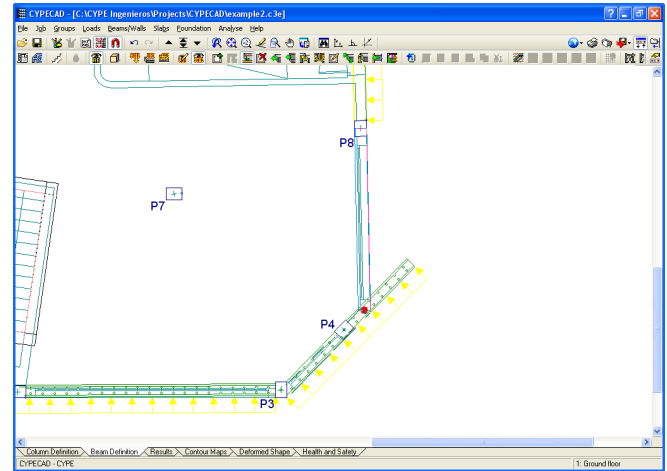


Fig. 15.188

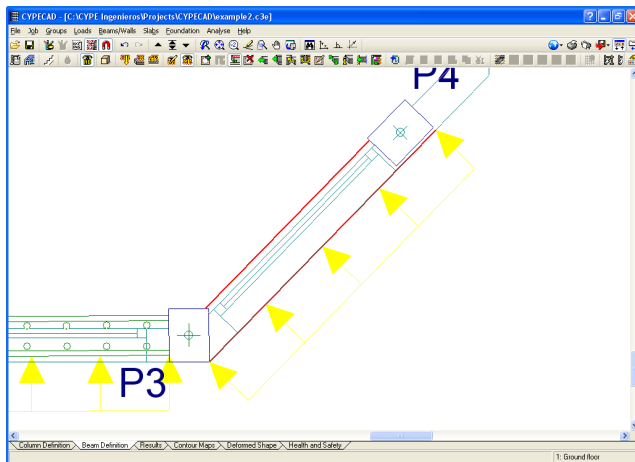


Fig. 15.187

Click on **Beams/Walls > Extend wall**. Click on the end of the wall reaching P4 and extend it by dragging it past the intersection. Click to end the extension. Now click on the end of the wall reaching P8 and drag until a red dot is displayed at the point where it intersects with the wall that has just been extended. Click with the left mouse button to confirm that is where the wall is to end.

Now, using the option **Beams/Walls > Delete > Delete the selected wall segment in all the groups** to delete the unwanted wall segment past the intersection.

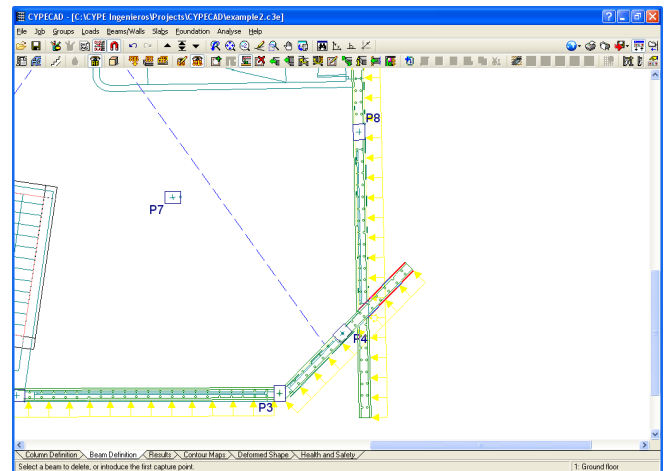


Fig. 15.189

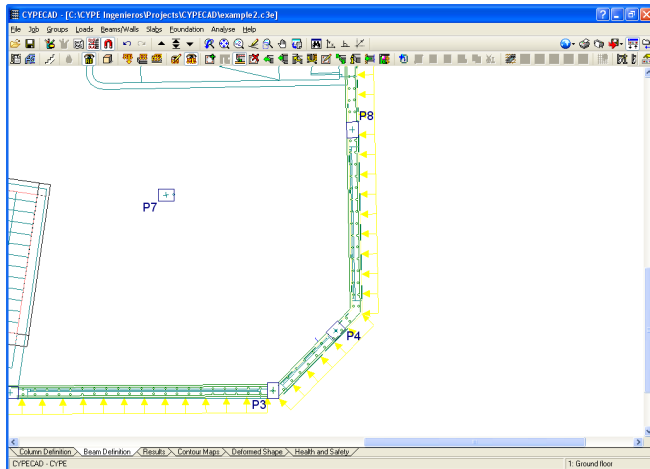


Fig. 15.190

Now that the wall has been completed, a question mark appears indicating there is a closed precinct in which a floor slab is awaiting definition.

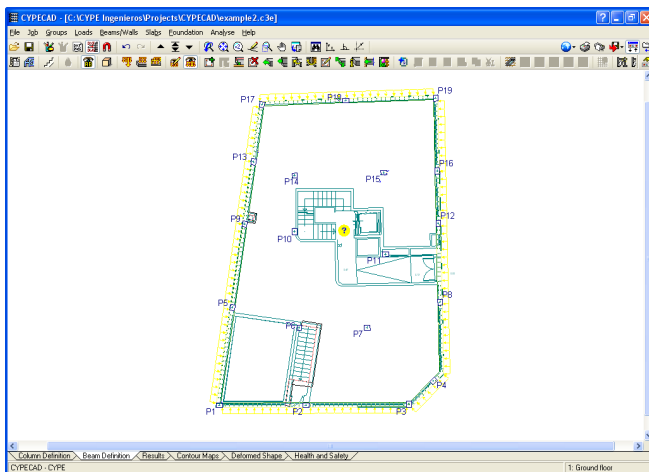


Fig. 15.191

Remember to deactivate the object snap settings by pressing on F3.

15.3.4.2. Beams

If the job has been introduced using Automatic introduction DXF/DWG go to the following section 15.3.4.2.1. Alternatively, if the Automatic introduction IFC has been used, go to section 15.3.4.2.2.

15.3.4.2.1. Data introduction using Automatic introduction DXF/DWG

The beams that were previously deleted to introduce the wall will now be re-introduced. Zoom into the car lift area. Using the option **Beams/Walls > Extend beam** (select the **Beam that is extended** option and leave the **Fixed length** box unmarked), extend the beam reaching P6 so it connects with the wall. When it reaches the intersection with the wall, a red dot will appear. When this occurs, click with the left mouse button to confirm the point.

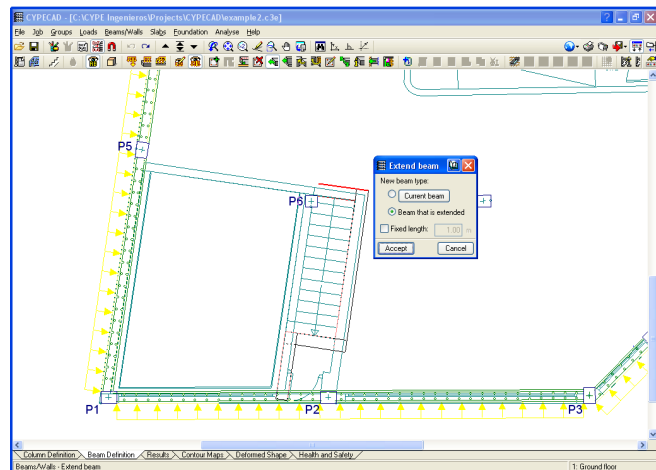


Fig. 15.192

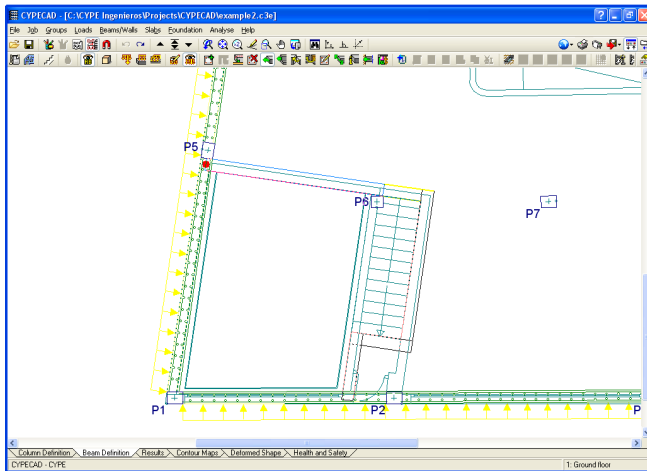


Fig. 15.193

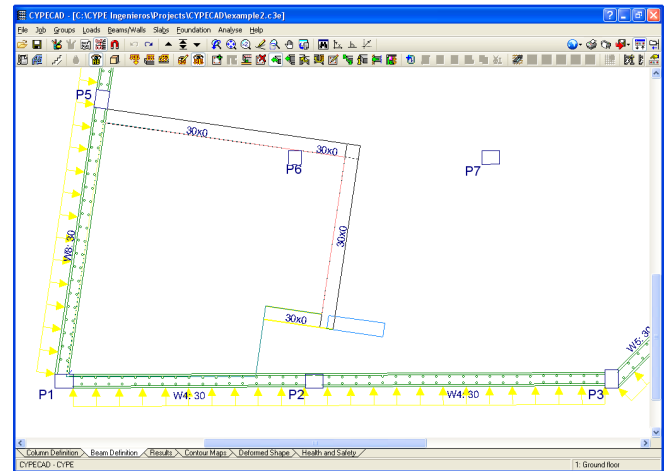


Fig. 15.194

Now to introduce the beam spanning from P6 to the wall at the bottom of the screen. First of all, the beam closing off the stairs opening must be deleted otherwise it will overlap with the beam to be introduced as it lies in its path. (Trick: Instead of completely deleting it, it is best to extend it to the right, delete the segment on the left and then introduce the beam. This way, a segment of the beam will remain adjusted to the outline and can then be extended). Now the beam spanning from P6 can be introduced.

Click on **Beams/Walls > Define beam**. Select the **Current beam** icon (first on the left) from the floating menu. The Current beam dialogue box will appear. Select a dropped beam (second option) with a width of 20 cm and depth of 35 cm. **Accept** the dialogue box and click on P6 and then at its other end at the wall. Adjust it to the CAD template by activating the object snaps and clicking on **Beams/Walls > Adjust**.

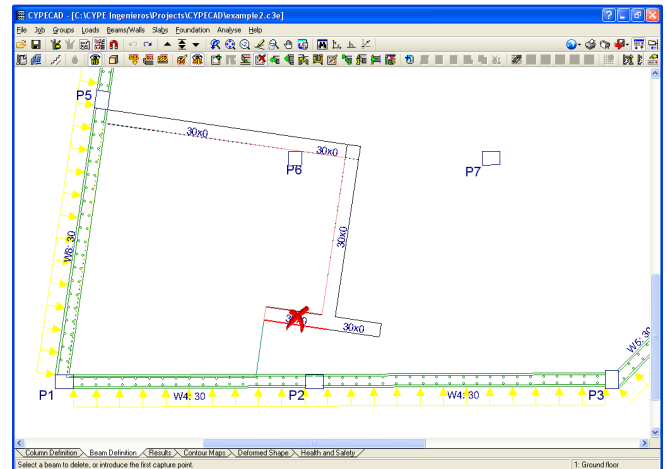


Fig. 15.195

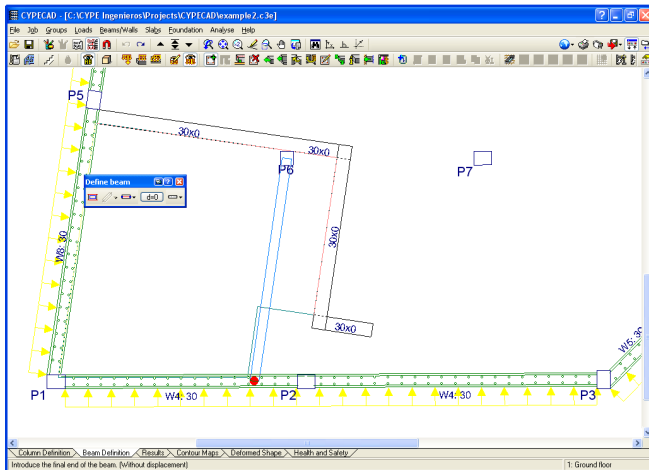


Fig. 15.196

Extend the small beam closing off the stairs opening and delete the unwanted segment on the right. Finish off by introducing the limit beams that were deleted defining the basement ventilation opening.

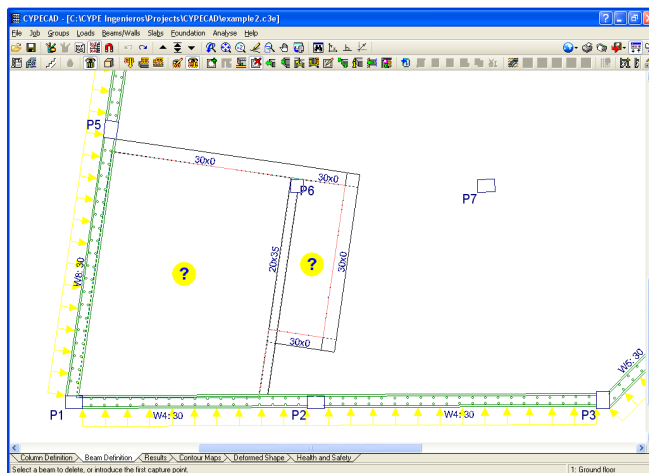



Fig. 15.197

Now the beams of the Lift machinery roof will be defined. Click on the **Go to group** icon  in the toolbar. In the **Go to group** window that appears, click on **Group 5: Lift machinery roof**.

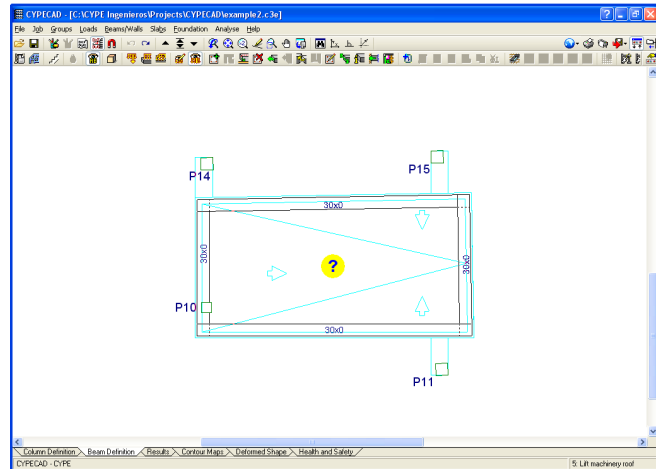


Fig. 15.198

Using the option **Beams/Walls > Assign beams** the width of the vertical beam reaching P10 will be modified. In the Assign beams dialogue box, select the **Based on beam adjustment option** and click on the **Current beam** button. Edit the beam in the **Current Beam** window so that it is a flat beam with a width of 50 cm. Click on the **Accept** button and then on the **Assign** button in the following window. Bring the cursor close to the beam in question and it will be displayed in yellow. Click with the left mouse button and it will modify the beam. Modify the small beam segment on the other side of P10 in the same way. Right click with the mouse button to end the command.

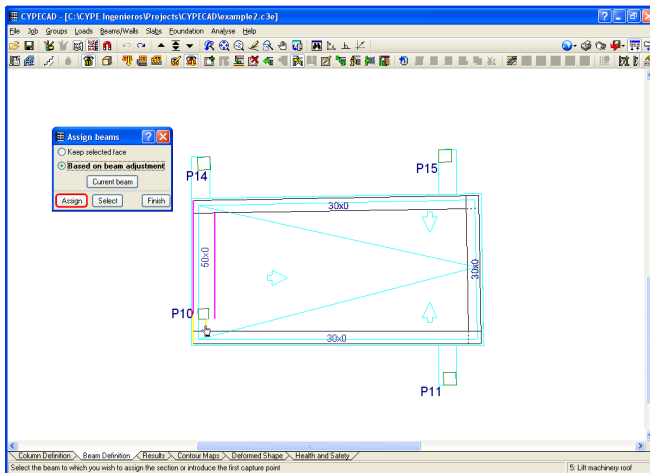


Fig. 15.199

Having modified the beam, its end shall be extended to P14 using the option **Beams/Walls > Extend beam** (leaving the default option **Beam that is extended** activated).

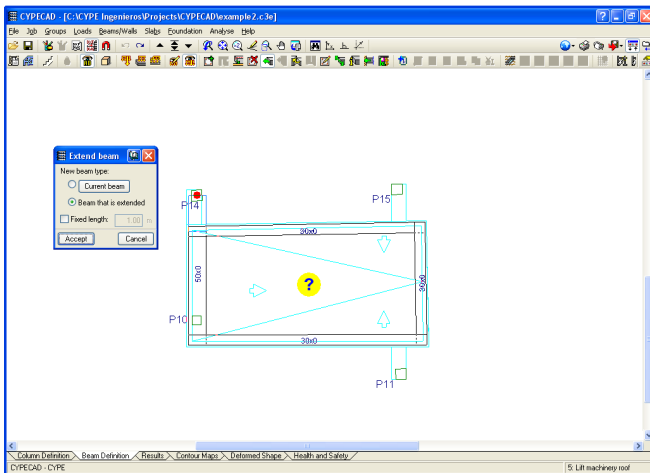


Fig. 15.200

Finally, using the option **Beams/Walls > Define beam**, introduce a flat 50 cm wide beam from columns P11 to P15.

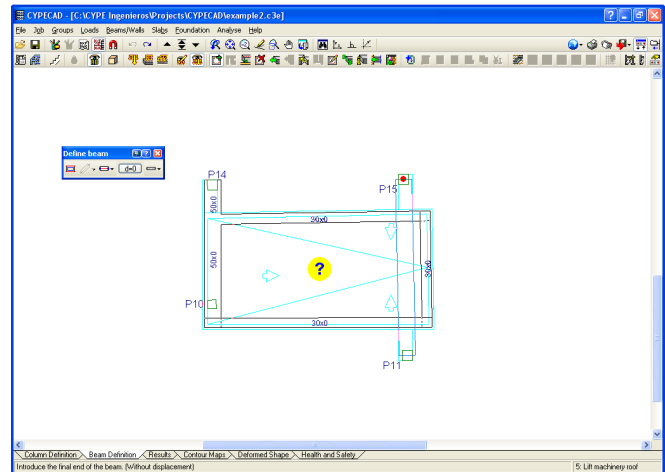


Fig. 15.201



Once all these beams have been introduced, the user can proceed to introduce the floor slabs (explained in point 15.3.4.3)

15.3.4.2.2. Data introduction using Automatic introduction IFC


The beam introduction process for this option is the same as that indicated above in point 15.3.4.2.1. in the case of Automatic introduction DXF/DWG, only here a previous step must be taken before carrying out the process.

When the IFC file was imported, the beams to be generated were to be limit beams. Therefore using the option **Beams/Walls > Assign beams**, the beams must first be modified to what they should be in each case.

15.3.4.3. Floor slab introduction

Using the **Go to group** icon in the toolbar  situate the main view of the screen on the Ground floor. Go to **Slabs > Panel manager** and from the floating menu select **Define panel** .

In the Panel manager dialogue box that opens, select **Waffle slab**. A window will open offering the user to choose amongst predefined waffle slabs with lost forms. Click on **Cancel**. A second window will appear offering waffle slabs with removable forms. Click on **Cancel**. Two empty rectangles should be visible on the right of the Panel manager dialogue box.

For the **Type of form**, select **Lost**. Click on the **Add new element to the list** icon . In the new window that opens type the name of the new slab: Waf. 30+5 rib 12 spacing 82 and Accept. In the new dialogue box, the properties of the slab have to be defined.

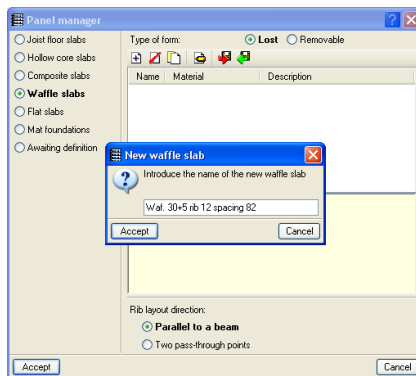


Fig. 15.202

First it has to be assigned a reference. Label it 30+5. The type of form has to be defined: its material and layout. In this case the form will be made out of concrete and will contain three elements. The geometric data will be the

same in both directions, the total depth 35 cm, compression layer thickness 5 cm, 12 cm ribs, and a rib spacing of 82 cm.

The program will automatically calculate the concrete volume and weight of the slab depending on the parameters introduced. However, these values can be modified by marking the **Concrete volume** and **Dead load** boxes and introducing the new values.

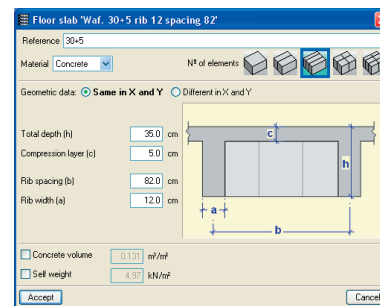


Fig. 15.203

For the **Rib layout direction**, leave the **Parallel to a beam** option selected. Accept the window.

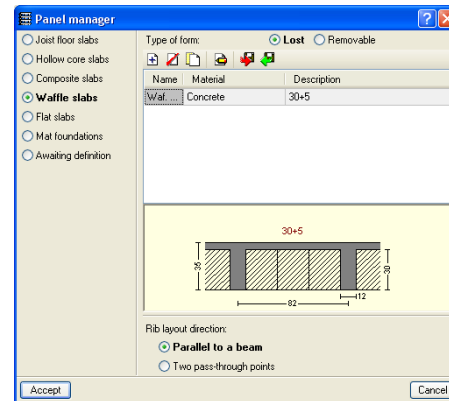


Fig. 15.204

If the cursor is brought over an opening, it will be highlighted in yellow. By clicking in the opening and then on a beam to which the ribs are to be parallel, the slab will be introduced.

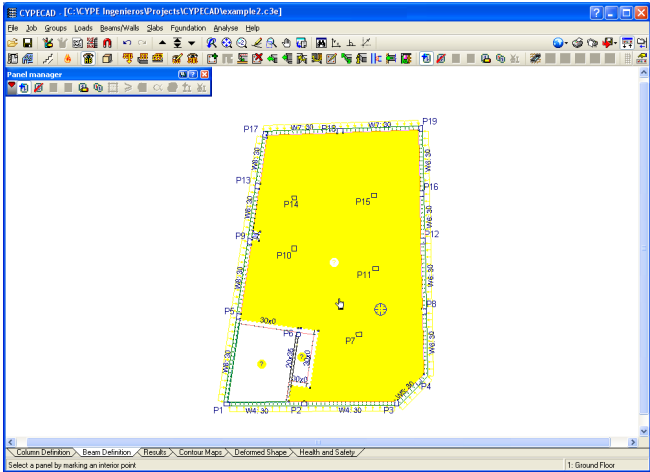


Fig. 15.205

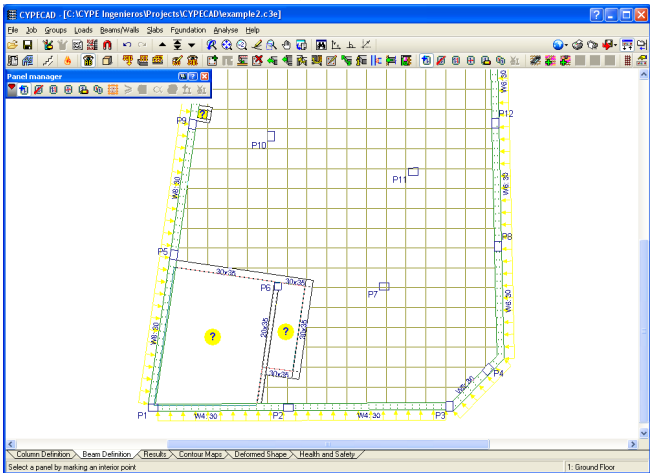


Fig. 15.206

Now that the floor slabs have been introduced, the drop panels can be generated using the option **Slabs > Drop**

panels > Generate drop panels. When this order is executed, drop panels are generated at each column with dimensions equal to 16% of the distance between columns that lie within the angle of vision (40°) with a maximum of 5 times and a minimum of 2.5 times their depth. These maximum and minimum values can be configured in the menu: **Slabs > Drop panels > Configuration of drop panel generation.**

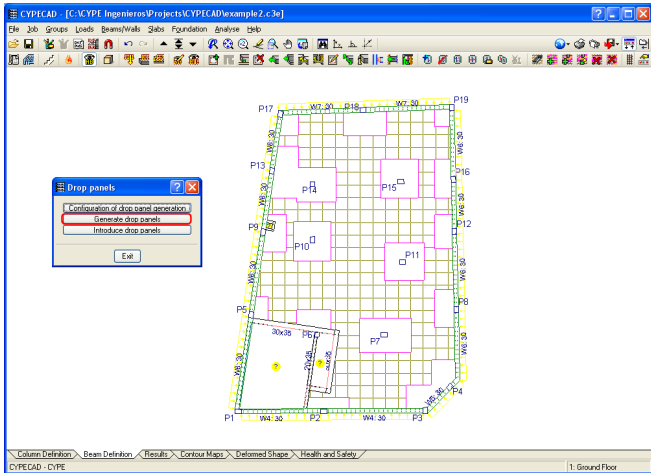


Fig. 15.207

Now to assign a top base reinforcement of 12 mm bars in both directions. Go to **Slabs > Assign base reinforcement** and click on the waffle slab. From the right hand column in the Waffle slab assembly window that opens, select 1Ø12 and press the arrow buttons to assign it as top longitudinal and transverse reinforcement. Click on **Assign all** then on **Finish**.

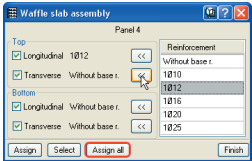



Fig. 15.208

Finally, to define the openings, click on **Slabs > Panel manager** and from the toolbar click on the **Delete panel** icon  and click on the three openings which display a question mark.

The Ground floor elements have been defined. Go up to the next group where a new slab will be defined and introduced: Waf. 25+5 rib 12 spacing 82. It has the same properties as the previous slab only, in this case, the total height is 30 cm.

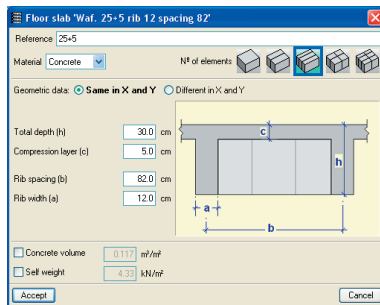


Fig. 15.209

It is important when defining the floor slabs that the positions of the ribs coincide on plan. This way, when downpipes are positioned on a floor, we can be sure it will not come across any other ribs on other floors. Therefore, it is best to copy the slab distribution from one floor to another.

In this case use the following option to copy the relevant properties from the Ground floor: **Slabs > Panel manager** and from the floating menu select **Copy panel**. Right click with the mouse button and the **Go to group** window will appear. Click on the Ground floor and when it appears on screen click on the waffle slab. A dialogue box will open asking which properties are to be copied. Select the **Insertion point**, **Direction** and **Base reinforcement**. The program will return to the original floor group where the properties can be copied onto the slab by clicking on it.

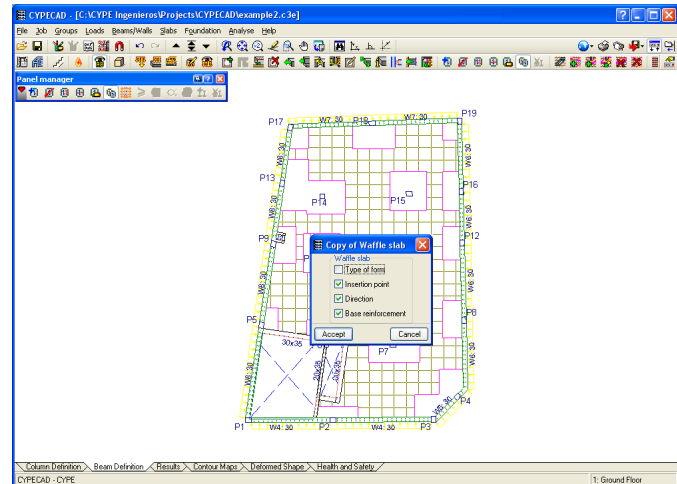


Fig. 15.210

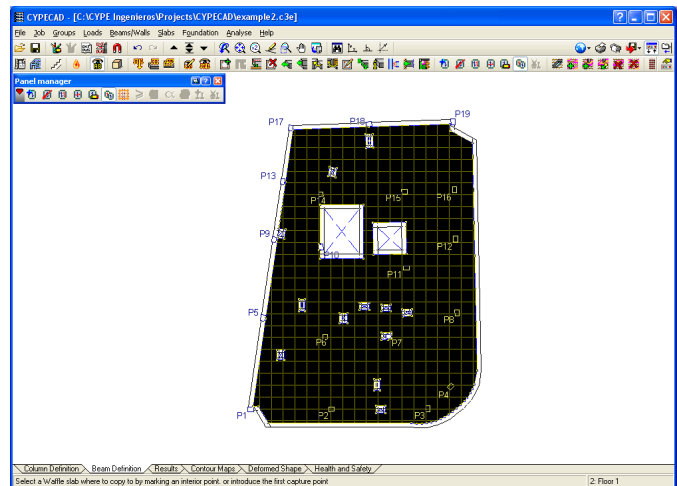


Fig. 15.211

Finally, generate the drop panels for the group: **Slabs > Drop panels > Generate drop panels**.

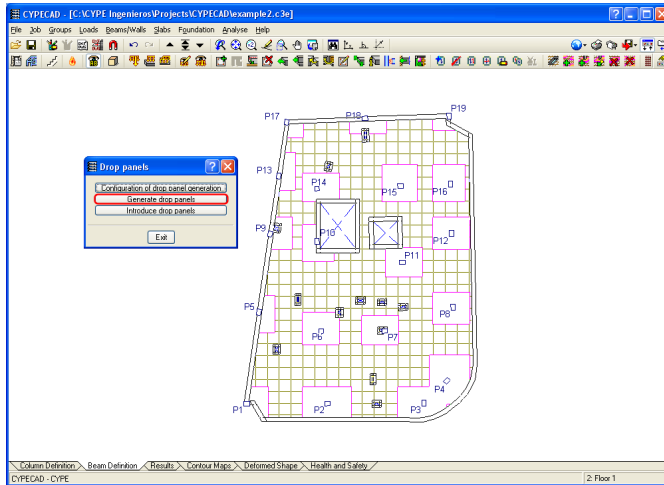


Fig. 15.212

Go up to the next group and repeat the same process only this time select all the properties to copy from the group below. Repeat the process for the remaining groups.

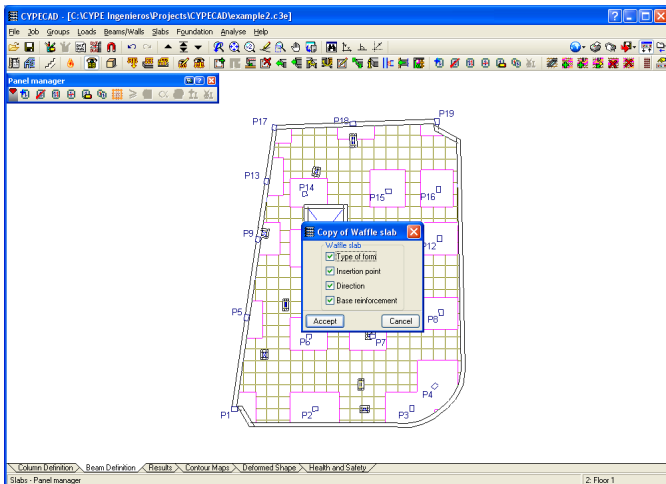


Fig. 15.213

15.3.4.4. Finishing touches

Now that the beams, slabs and drop panels have been introduced, the necessary finishing touches can be applied so to have a more adequate structural behaviour. For this example, the following guidelines should be considered:

- All overhanging beams will be extended at least their overhanging length towards the inside.
- Any main beams defining openings will have their ends extended to the closest support band.
- Secondary opening beams will have their ends extended a little after their intersection.

Starting off at group 1: Ground floor. The car lift and stairs area is separated by a beam which reaches P6. This beam is to be extended to the edge of the drop panel. The beam parallel to it on the other side is to be considered a main beam. Therefore it is to be extended to the wall at one end and the other to the edge of the drop panel. All this is to be done using the option: **Beams/Walls > Extend beam**, leaving the option **Beam that is extended** activated.

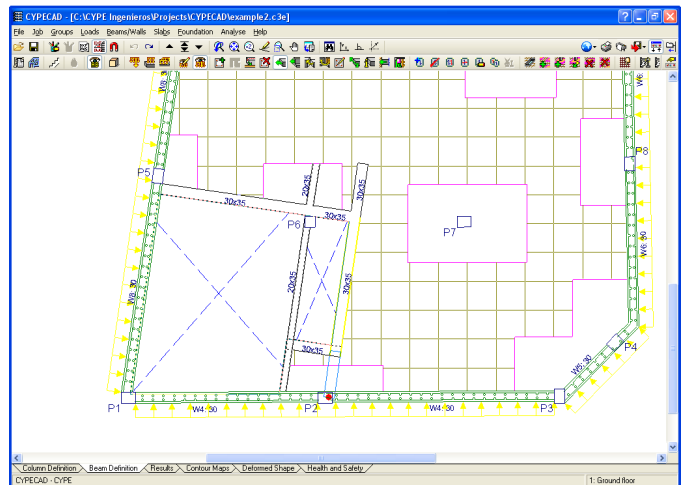


Fig. 15.214

NB: The position of the drop panels after being generated depends on the position of the ribs of the waffle slabs. Problems may arise if the edge of a drop panel is overlapped when extending a beam; the program may warn of an incorrect introduction. To solve this problem, temporarily move the edge of the drop panel using the option **Slabs > Drop panels > Move corners**. Having moved the edge, extend the beam past the original position of the drop panel edge, then move the edge back to its position. It may occur that due to there being a small area between the original edge and the extended beam that it is best to completely extend the drop panel into the beam. The beam segment lying outside the drop panel can now be deleted.

To complete the finishing touches of the group, the stair beams are to be extended 65 cm from the axis of the stair beams. To do so, click on **Beams/Walls > Extend beam** and in the dialogue box select **Beam that is to be extended**, mark the **Fixed length** box and type in 0.65. Click on Accept and click on the two ends to be extended. Right click to end.

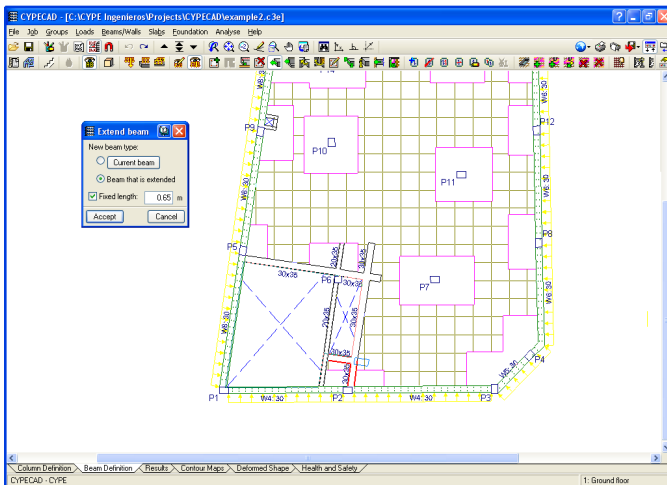


Fig. 15.215

Now to group 2: 1st Floor. The first beam to modify is the beam supported by P10. The beam is to have its top end extended to the top of the drop panel, past P14 and similarly with the bottom end.

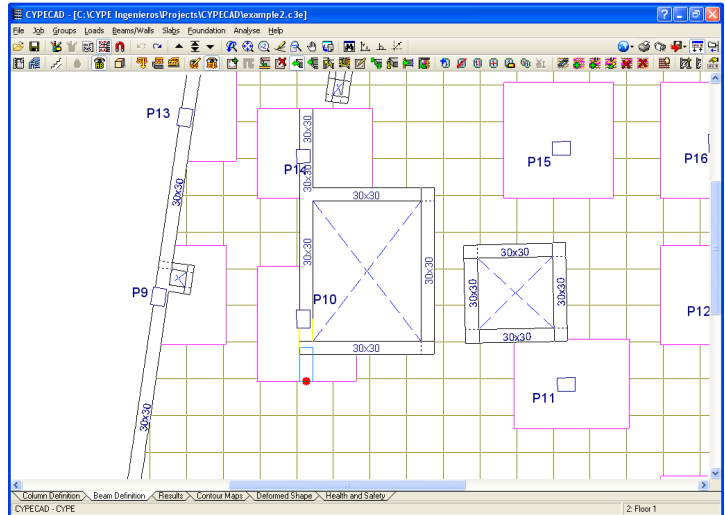


Fig. 15.216

The beam parallel to the previous beam will also be extended: its top end to the next rib after the imaginary line connecting P14 and P15, and the bottom end to the next rib.

The beam to defining the right side of the lift opening is also to be extended. The ends will be taken to the edges of drop panels of columns P11 and P15.

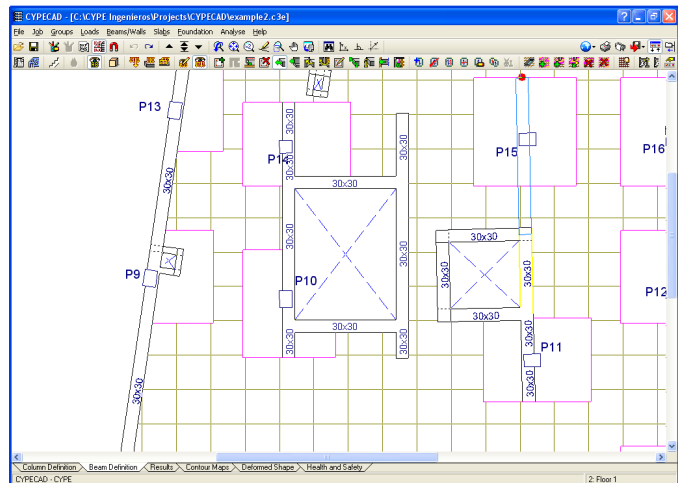


Fig. 15.217

Using the option **Slabs > Drop panels > Move corners**, move the top right hand corner of the drop panel of column P11 so that it lies above the beam intersection.

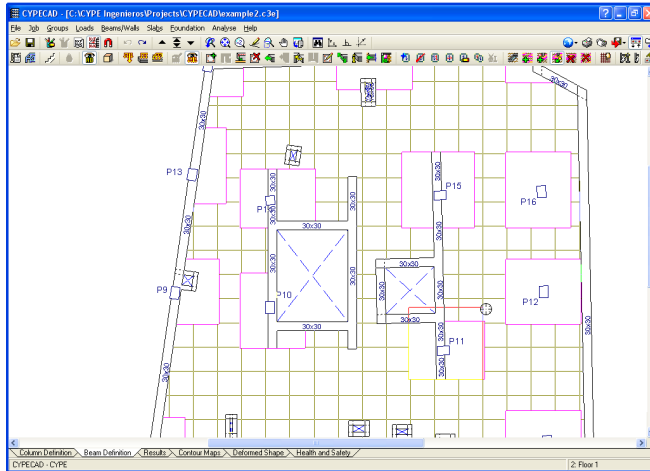


Fig. 15.218

The next step is to edit the bottom beam of the lift opening and change it to a 15 cm wide non-structural beam using the option: **Beams/Walls > Assign beams > Current beam** and selecting the non-structural beam and leaving the **Based on beam adjustment** box marked.

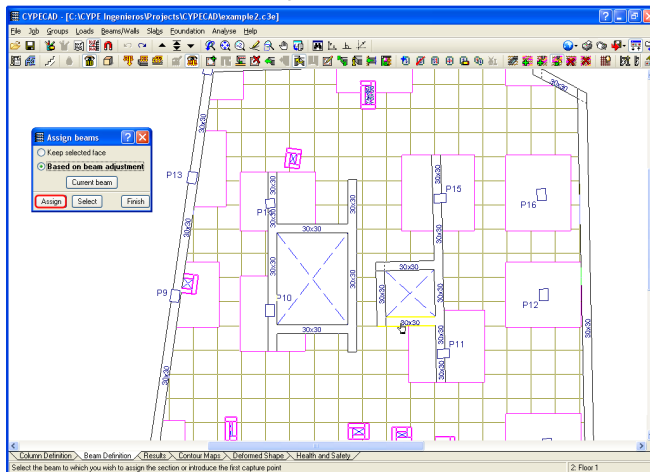


Fig. 15.219

The beam defining the bottom of the stairs opening will now be extended to the right until it reaches the vertical beam to the right of the lift opening.

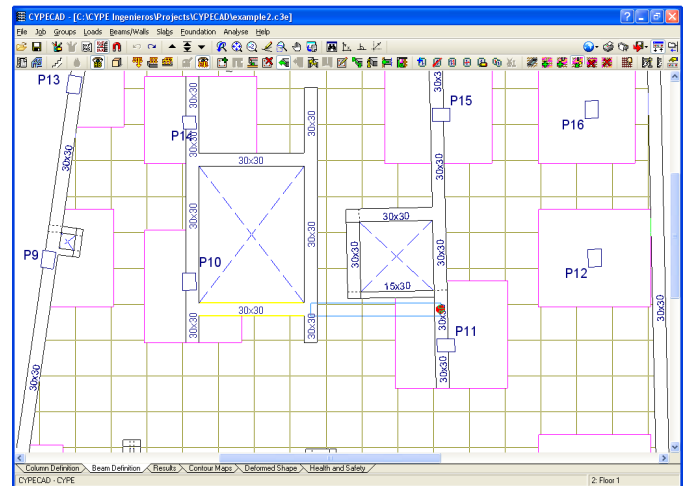


Fig. 15.220

The top beam of the lift opening will also be extended to the left until it reaches the vertical beam defining the right side of the lift pit.

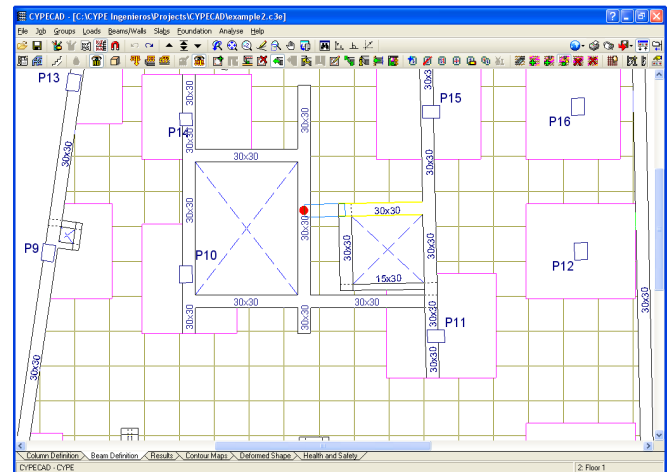


Fig. 15.221

Finally, the secondary beams reaching the main beams will be extended 85 cm using the Fixed length command within the Extend beam option.

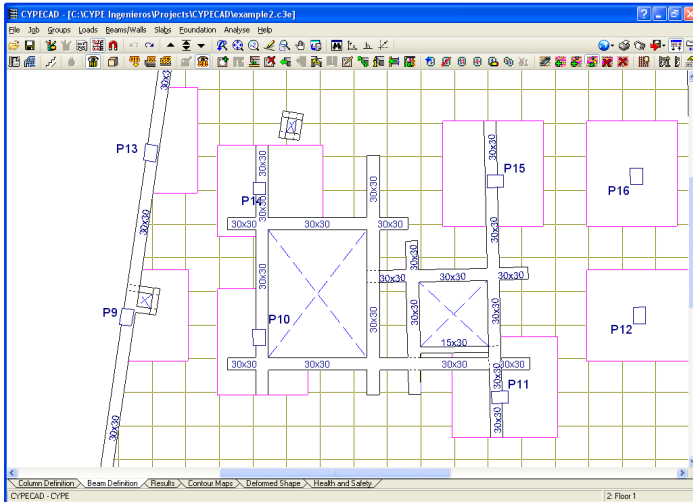


Fig. 15.222

Repeat the entire process with the remaining groups.

15.3.4.5. Load introduction

15.3.4.5.1. Surface loads

The surface loads due to the infill on the ground floor and the access stair to the machinery room on the Roof have to be introduced. Using the **Go to group** option, situate the screen on the Ground floor. Apply the option **Loads > Loads**, select surface load and assign a value of 6.72kN/m² (0.42 m of infill at 16 kN/m³), leave it as a dead load. Click on the **New** button and introduce the load by clicking on the vertices of the perimeter of the infill area, finishing off the introduction by clicking on the right mouse button.

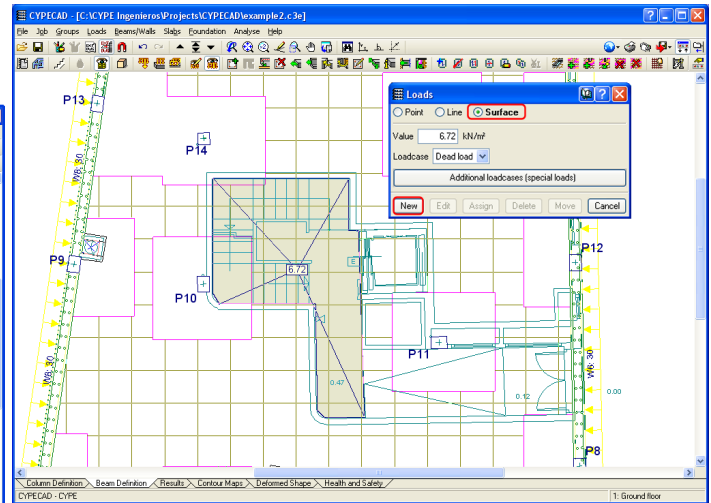


Fig. 15.223

By following the same steps, introduce the surface load in the area of the ramp applying a load corresponding to its average height: 3.36 kN/m² ($0.5 \times 0.42 \text{ m} \cdot 16 \text{ kN/m}^3$). This concludes the surface load introduction on the Ground floor.

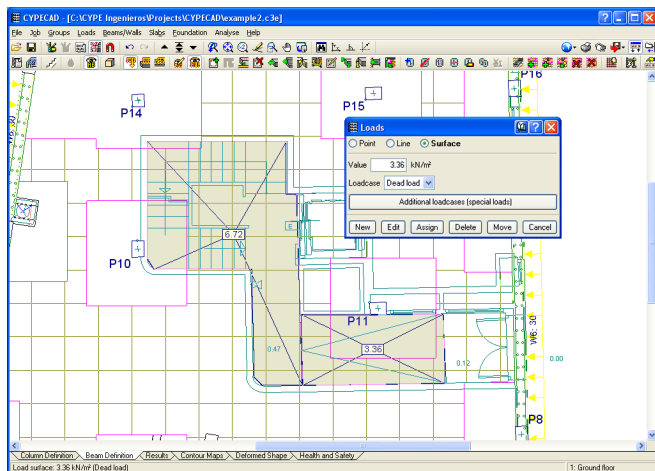


Fig. 15.224

Moving on to the Roof, the surface load due to the machinery room will now be applied. In a similar way as to what was done in the Ground floor, a load corresponding to 49 cm of infill (average height of the stair) i.e. 7.84 kN/m^2 ($0.49 \text{ m} \cdot 16 \text{ kN/m}^3$) will be introduced.

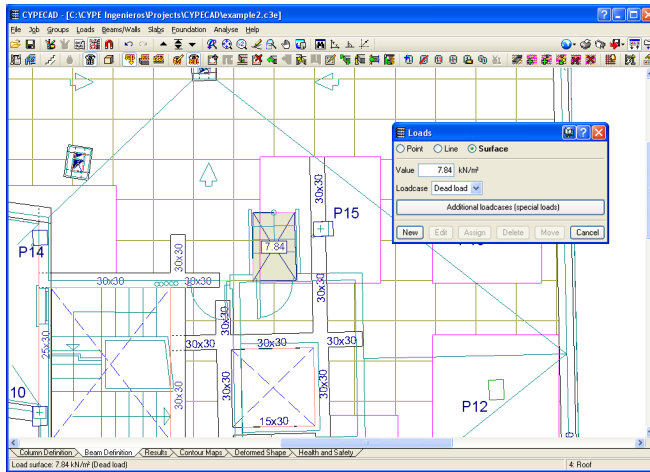


Fig. 15.225

15.3.4.5.2. Line loads

Line loads are to be introduced corresponding to partitions or walls that due to their importance cannot be addressed as general partitions whose load is included in the applied loads of the building.

The introduction process will commence at the Ground floor. Click on **Loads> Line loads on beams** and insert a value of 11 kN/m , leaving the loadcase with the **Dead load** option. Select **New** and click on the beams defining the perimeter of the floor, i.e. beams on which an external wall will be supported. Click on **Loads>Loads** and select a line load of 11 kN/m to introduce the loads surrounding the stairwell and ramp.

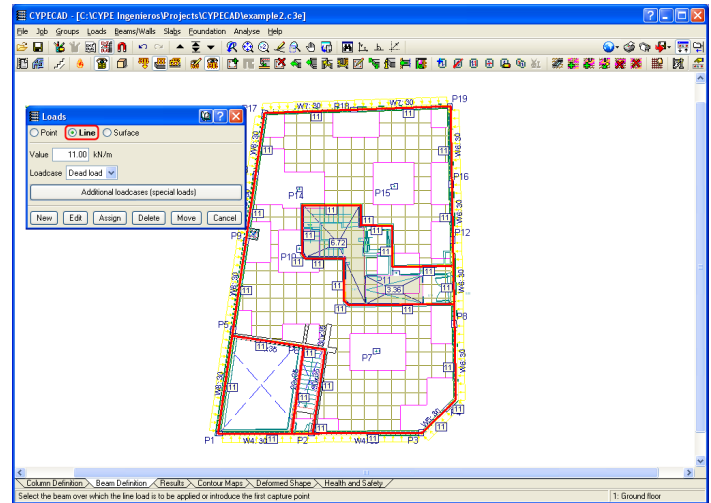


Fig. 15.226

Having introduced the line loads on this floor, move up to the next floor and introduce, in a similar manner, the line loads corresponding to the external walls (8 kN/m) and guardrails of the balconies (3 kN/m).

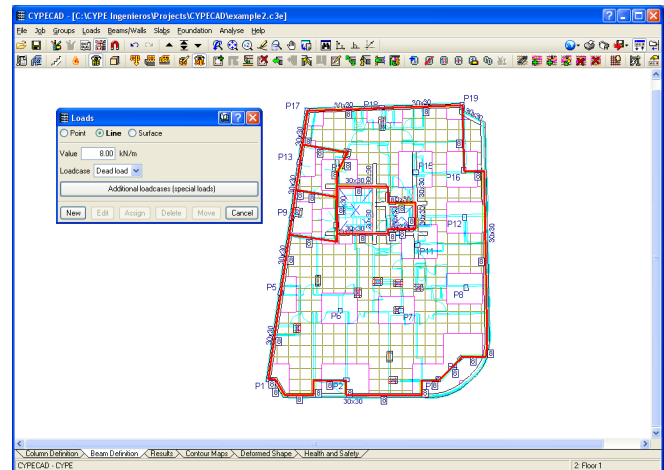


Fig. 15.227

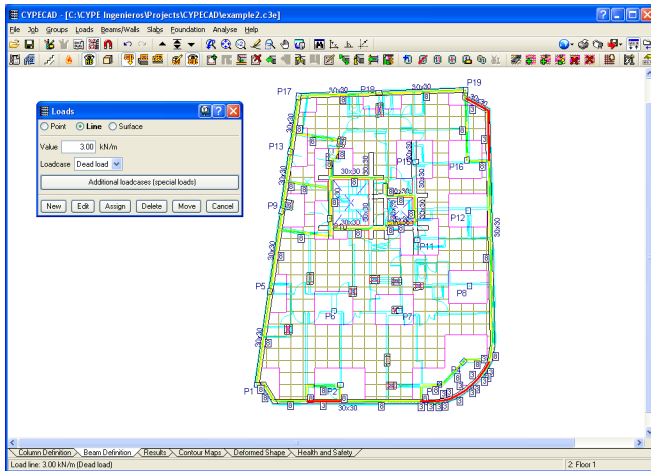


Fig. 15.228

The same loads will be introduced in group 3.

The loads to apply in group 4; Roof, will be as follows:
3 kN/m due to the guardrail around the perimeter of the building,
8 kN/m for the external walls of the lift machinery roof.

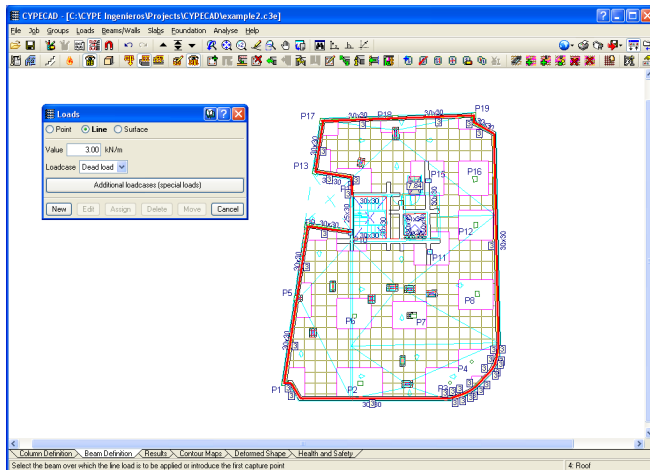


Fig. 15.229

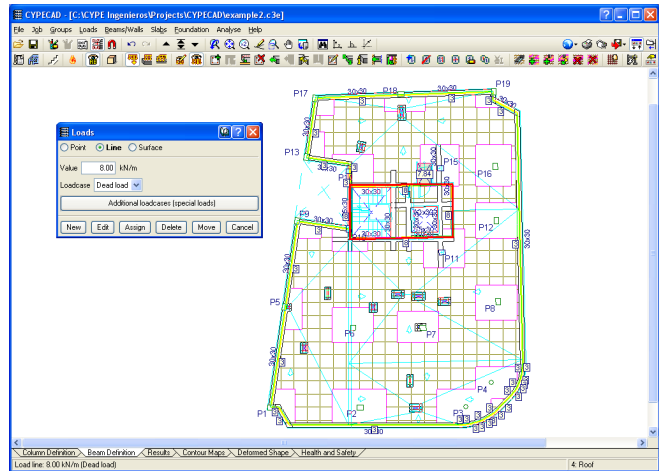


Fig. 15.230

The load transmitted by the machinery room of the lift to the Roof will have to be introduced. This load corresponds to the self weight of the slab 5 kN/m² and a load due to the machinery of 10 kN/m². Due to the supported width of the slab being 0.9 m for the left support, 1.28 for the central support and 0.38 for the right support, the following loads must be introduced respectively: 13.5 kN/m, 19.2 kN/m and 5.7 kN/m.

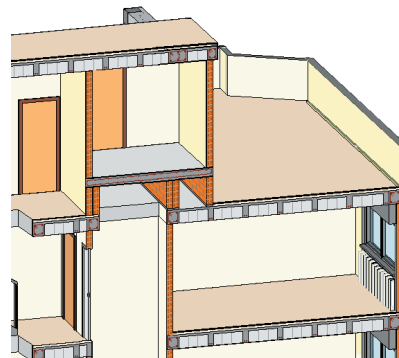


Fig. 15.231

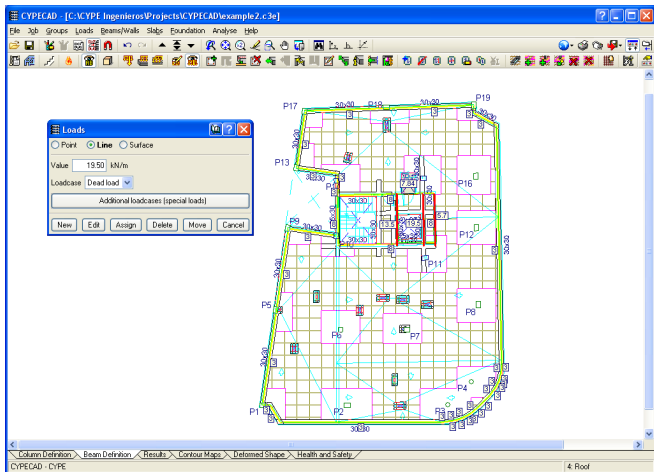


Fig. 15.232

15.3.4.5.3. Load views

For an easier load consultation and introduction, the program has an option to configure the loads which are to be visible.

The view selection can be done depending on their type (point, line or surface) and so allowing to view all the loads of that type belonging to various loadcases or by selecting the specific loadcase to consult.

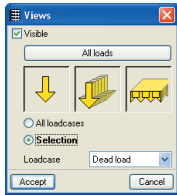


Fig. 15.233

15.3.4.6. Stairs

The first stair to define will be the one providing access to the Basement. This stair consists of a single 1.1 m wide span, spanning across a height difference of 2.50 m by means of 14 steps with an individual height of 17.86 cm.

Position the screen at the Basement and select the following option **Job > Stairs**. A floating menu will open. Click on the **New staircase** option and a window will appear to introduce the parameters of the staircase.

The emerging window contains two tabs. Within the **Staircase data** tab, enter a reference and introduce the data of the staircase: width 1.1 m, tread 0.28 m, riser 0.1786 m and select any of the two rotation options as it will have no effect on the staircase due to it being a single span. The steps are to be built using bricks and as there are no handrails, change the value to 0 kN/m. The floor load is to be assigned a value of 1 kN/m² with an applied live load of 3 kN/m².

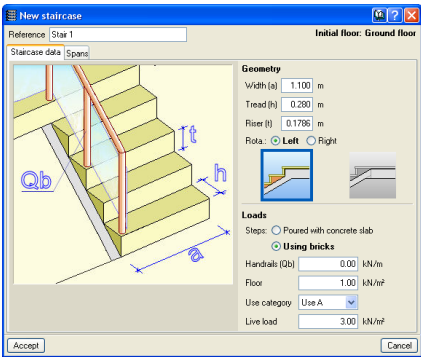



Fig. 15.234

Once the basic data of the staircase has been defined, click on the **Spans** tab to define the geometry of the span. Click on to introduce the new span. In the emerging window, introduce a reference number, indicate there is

only one consecutive flight and add a new type by pressing on the  icon.

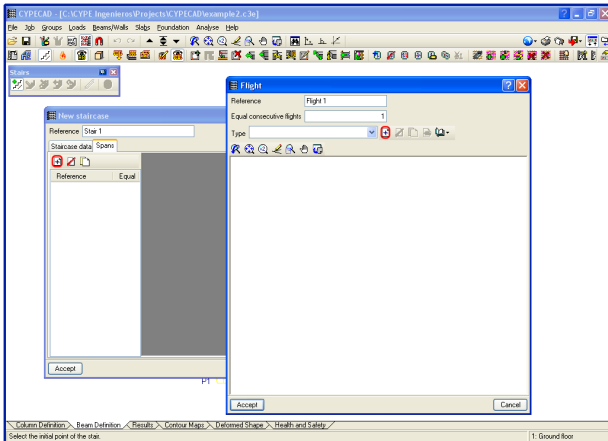


Fig. 15.235

The new staircase will be assigned the reference name **Basement**. Activate the box **Initial steps built on floor** and select, from the options that appear, **Straight** with **4** steps. Having accepted the dialogue box, select the **Straight flight** from the **Predefined shapes** and indicate it is to have **10** steps.

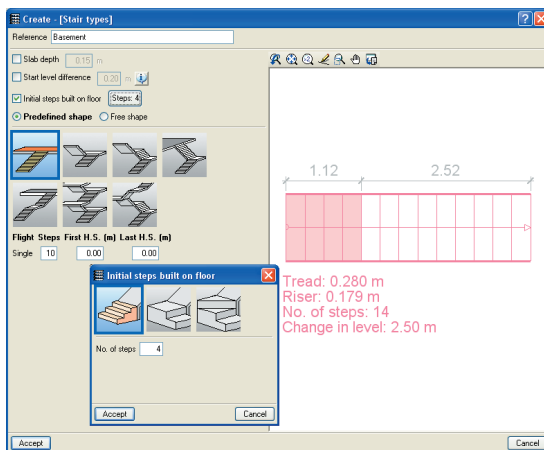



Fig. 15.236

Upon accepting all the windows, the program will return to the floor view where the stair is visible next to the cursor (horizontal). To place it in its correct position, the object snap tools will be used to snap to the CAD template. Click on  to open the object snap selection window and select the intersection and centre object snaps. Bring the cursor close to the start of the stair and click, using the left mouse button, when the centre of the circle is snapped to. Move the cursor towards the end of the flight and click when the intersection symbol appears. The flight has been introduced. To exit the 'enter stair command', right click with the mouse button and click on **Cancel**.

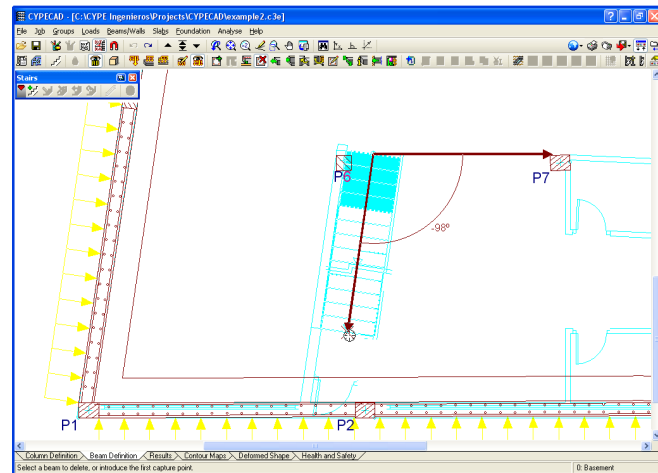


Fig. 15.237

Now the staircase allowing access to the residential area of the building will be defined. The staircase consists of a flight on the ground floor, spanning a height of 3.8 m and 3 identical flights corresponding to the residential floors, each with a height of 2.8 m.

Firstly, situate the main view of the screen at the Ground floor (group 1) and click on the **New staircase** option. Give the staircase a reference and introduce the data of the

staircase: 1m width, 0.28 m tread and 0.175 m riser. Select the rotation of the staircase to be to the Left and mark the option: The intermediate landing or landing form the last step. The steps are to be built using bricks and the hand-rail, floor and live load are 3 kN/m, 1 kN/m² and 3 kN/m² respectively.

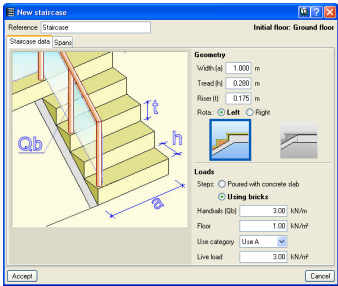


Fig. 15.238

Click on the **Spans** tab. The last defined staircase appears. Click on the icon to add a new staircase type. Give this new staircase a reference e.g. Grnd floor, then click on the icon to define its geometry.

Provide a reference for this staircase geometry (the same reference may be used). As the slab at which the stair flight begins has an infill of 47 cm on it, the option **Start level difference** must be marked and the value of 47 cm introduced. Activate the **Initial steps built on floor** box, and from the emerging window select the second option: **Intermediate landing with a turn, unsplit**. Indicate there are 2 steps and the rotation is the **Same as that of the stairwell (Left)**. Click on **Accept** and select the third option from the predefined shapes: three straight flights with quarter turn landings. Introduce 6, 5 and 6 steps for the first, second and third flights respectively. No steps are to be built on the intermediate landings. To finish off, indicate that the landings are to be supported by lateral 20 cm thick masonry walls.

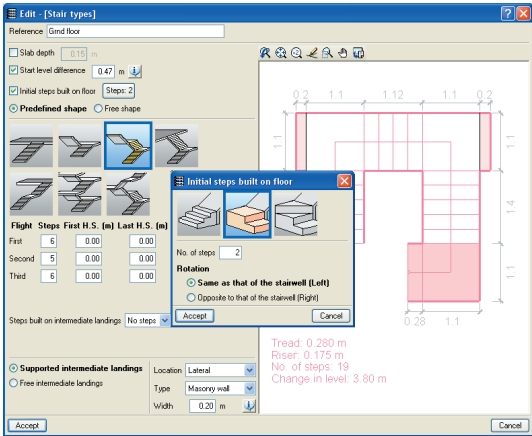


Fig. 15.239

Accept the dialogue box and click on the to add a new set of flights to the staircase. Provide a reference (e.g. Floors) for the stair section, indicate it has 3 equal consecutive flights and click on the icon to define its geometry.

From the predefined shapes, select the third option: three straight flights with quarter turn landings. Introduce 5 steps for the first and second flights and 6 steps for the third flight. Indicate there is a slab length of 30 cm for the first horizontal slab. The supports are the same as those on the Ground floor staircase.

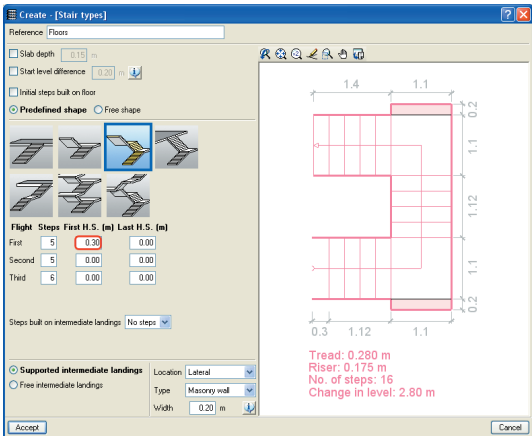


Fig. 15.240

Having defined the spans of the staircase, introduce it the same way the stair accessing the basement was introduced. With the aid of the DXF/DWG template object snaps, select the initial point of the stair and the second point will be adjusted by means of its angle.

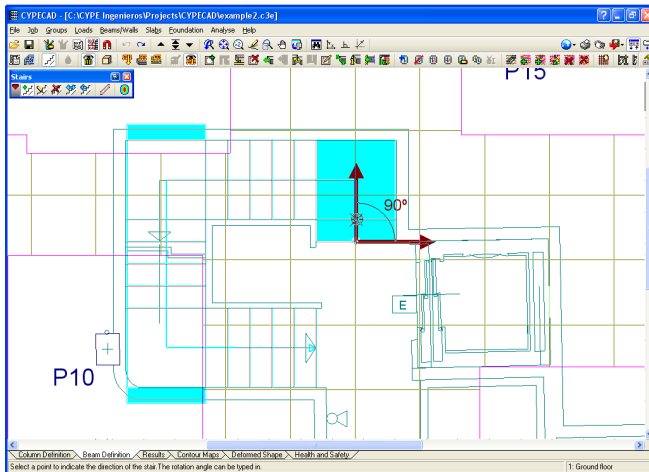


Fig. 15.241

When the program analyses the job, it will apply the corresponding load at the supports which will be transferred to the program. However, the staircase can be analysed and its reinforcement designed without the need of having to analyse the entire job. To do so, click on the View details icon from the stairs floating menu, then on the selected staircase. The program analyses and designs the staircase, and concludes by displaying its reinforcement drawings. If the consulted staircase contains more than one flight, a drop down menu in the top left hand corner of the drawings window allows the user to select the flight to view. The stair slab's forces and displacements, can be seen by clicking on the corresponding icon in the floating menu. As with the previous case, if the stair has not yet been analysed, the program will automatically do so and display its contour maps.

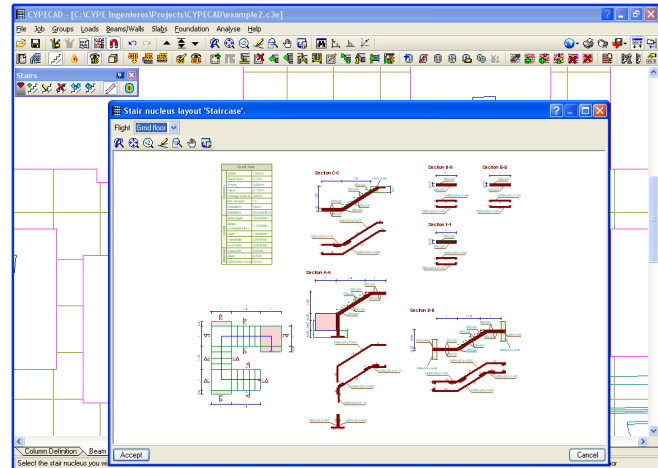


Fig. 15.242

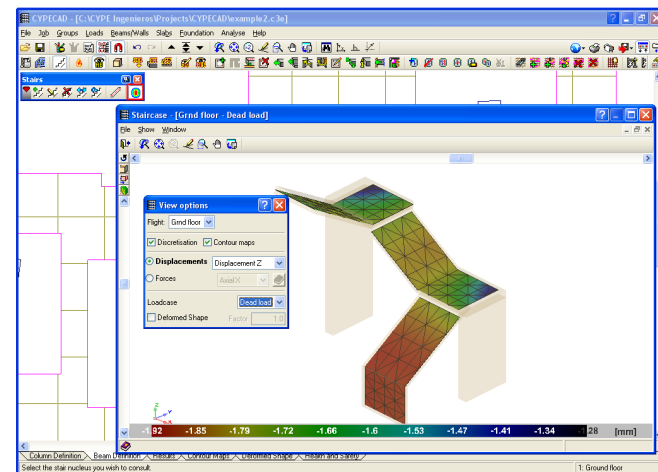


Fig. 15.243

Having analysed the staircase, the loads it generates can be seen at the groups where the flights start, end and have support elements reaching them. Line loads are generated at these points and surface loads are generated where initial steps have been built on the floor slab.

15.3.4.7. Analysis

Having introduced the two stairs in the project, the job can be analysed. To do so, click on **Analyse > Analyse the job except foundation**. Before beginning its calculation, the program will first analyse the geometry of the structure and will emit a warning if any incidences are found.

Once the analysis has finished, the program displays any problems that have been incurred as well as a list of the elements that are to be checked.

15.3.4.7.1. “Distributed analysis: 2 and 4 processor” modules

The program has two modules which allow for parallel processing using 2 or 4 processors. If the user has acquired any of these modules, the time taken for the program to complete the analysis can be reduced significantly as the processes are shared amongst the two or four processors. If the module has been acquired, when analysing, the window displays the number of processors available and how many are used.

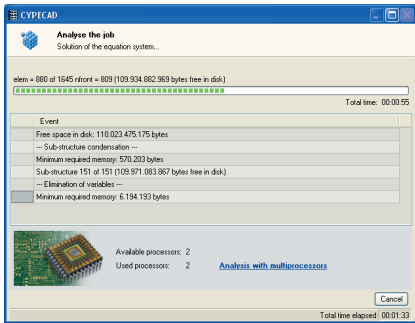


Fig. 15.244

15.3.5. Foundations

The foundations of the job will now be introduced. To do so, first, place the main view of the screen at Group 0. The wall strip footings can already be seen (these were defined

when the wall was introduced). The introduction will be completed by entering the pad footings and strap and tie beams.

15.3.5.1. Pad footings

To enter the pad footings, use the option **Foundation > Foundation elements > New**. The New element definition window will open. Here, select the **One column elements** option and click on the first button on the left (reinforced concrete footing). Click on the Selection of type button and choose the eccentric rectangular footing (third option from the left).

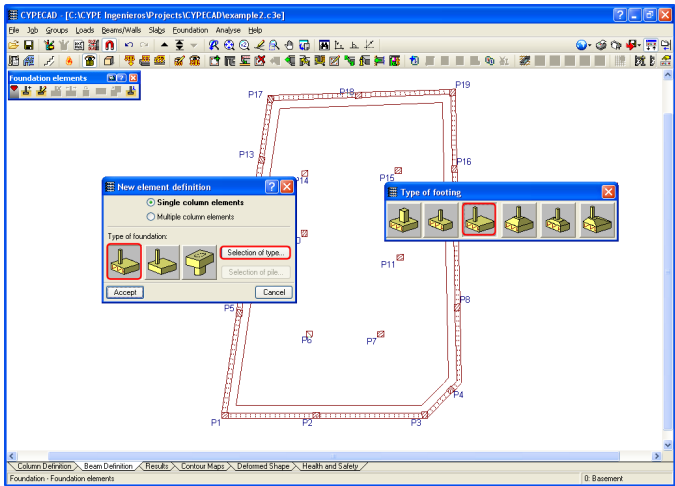


Fig. 15.245

Having accepted the windows, the cursor's appearance will change and will adopt the image of a footing. When brought close to a column, depending on where it is placed near it, the footing will change to become a corner, edge or centred footing.

Place the cursor over columns P7, P10, P11, P14 and P15 to introduce the centred footings and to the right of P6 to introduce an edge footing, so not to invade the car lift pit.

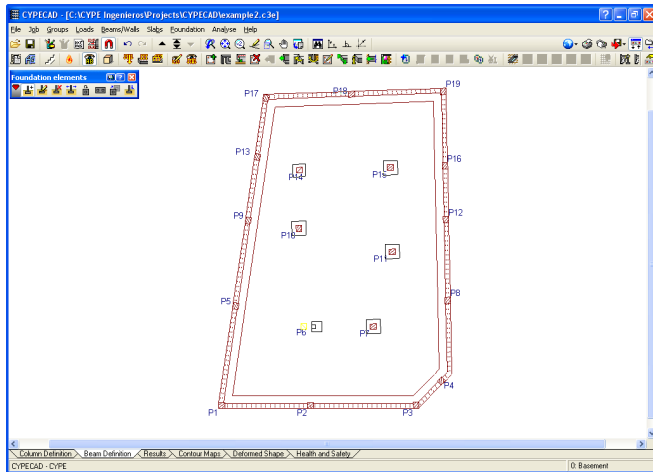




Fig. 15.246

Using the Rotate option , modify the angle of the footing to adapt it to the edge of the car lift pit. Upon activating this option, a further three options are available to modify the angle: Manual introduction, two point introduction or angle introduction. The two point introduction will be used . Using the car lift pit outlined on the DXF/DWG template outline, mark the two points defining the angle (first the bottom point then the top), then click on the footing which is to have this assigned to it.

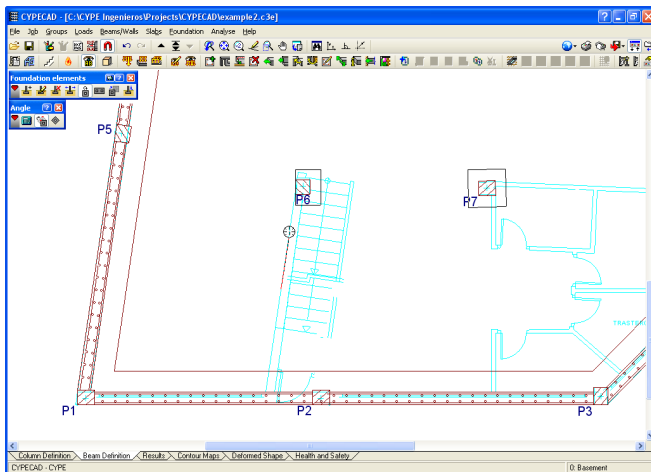


Fig. 15.247

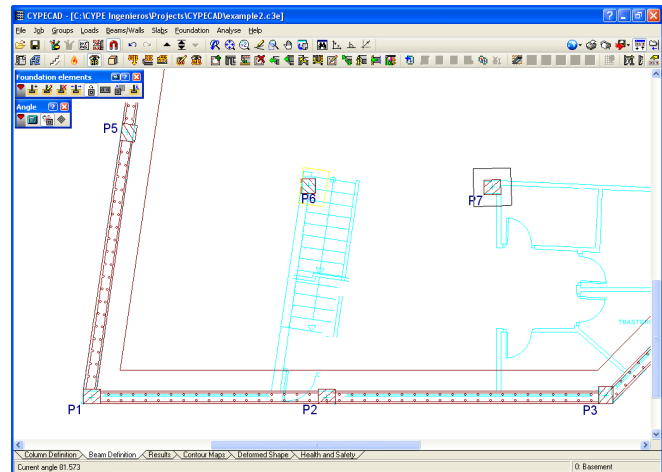


Fig. 15.248

15.3.5.2. Strap and tie beams

The strap and tie beams will now be introduced between the footings. Using the option: **Foundation > Strap and tie beams > Beam with automatic balancing at the ends** (this way, the program automatically detects if the footing is an edge or corner footing and introduces a strap beam, otherwise a tie beam is defined), introduce the beams.

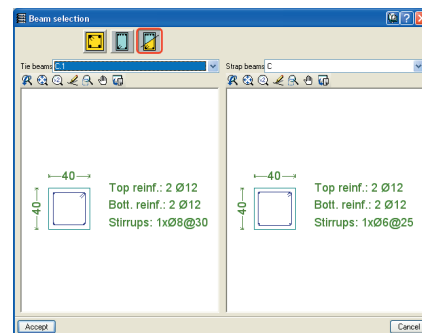


Fig. 15.249

Click, first of all, on a pad footing and then on the wall strip footing to introduce the strap beam that will balance the eccentricity of the wall footing. Do so with all the pad footings.

Introduce the beams between the pad footings in such a way that all the footings remain tied in the two orthogonal directions (required as the job is located in a seismic zone).

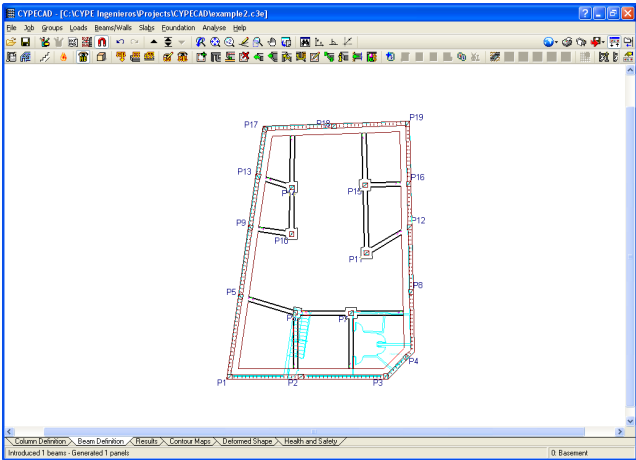


Fig. 15.250

The beams defining the perimeter of the lift pit must now be adjusted: **Beams/Walls > Adjust**. They are adjusted in the same way as were the walls in point 13.3.4.1. of this example.

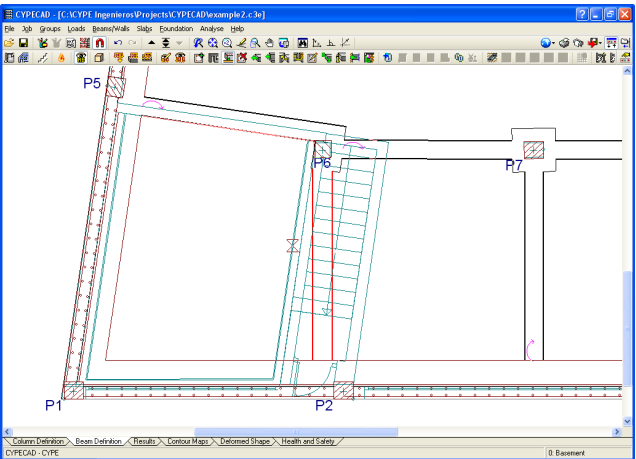


Fig. 15.251

Having introduced the foundation and if the job has been previously analysed, the foundation elements can be designed. This must be done once the results of the job have been consulted and modified, as required. If not, problems can arise. For example if the diameter of the starter bars of walls and/or columns has been modified, the depth of the footings may no longer be enough to guarantee the correct anchorage of the modified reinforcement.

15.3.6. Results

Once the structure has been analysed, any errors that may have arisen can be verified, and if necessary, the required modifications can be carried out to amend them.

15.3.6.1. Deformed shape of the structure

It is recommended that before any individual element of the job is verified, the deformed shape of the structure be consulted to ensure its displacements are in accordance with what is to be expected. To do so, click on the **Deformed Shape** tab which will display a wire frame model of the structure whereby the elements will be drawn in a series of colours corresponding to displacement values, indicated on the scale at the bottom of the screen.

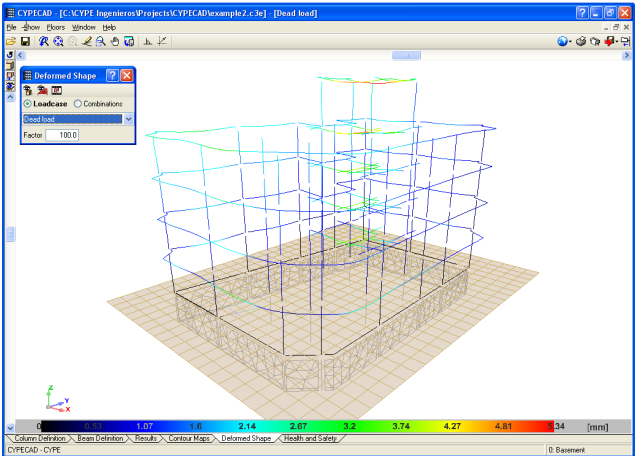



Fig. 15.252

Here, the displacement values can also be checked to comply with the selected code. The program also provides a report where the distortions of the columns of the job are indicated. By clicking on the Reports button  at the top of the screen, a list of the available reports is shown. Select **Column distortions** and a list will appear indicating the absolute and relative distortions of each column.

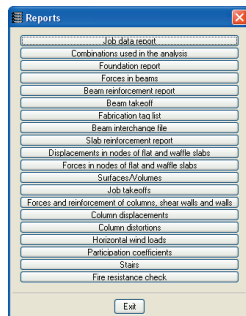


Fig. 15.253

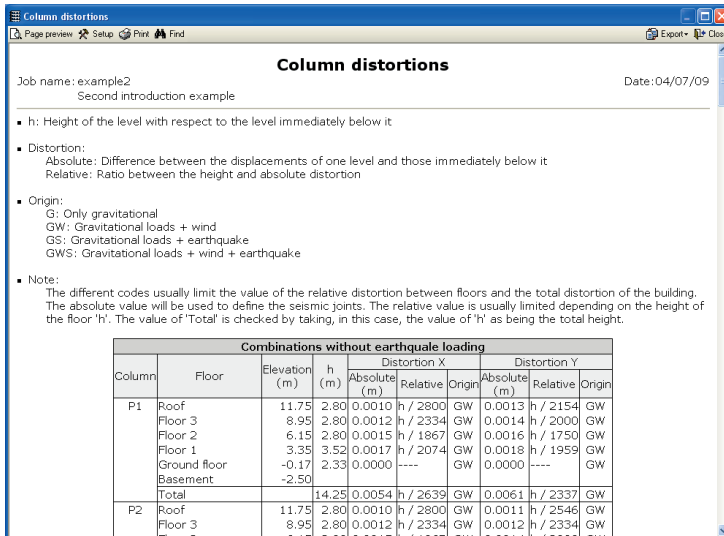


Fig. 15.254

If the distortions are not verified, the structure's stiffness must be increased and then re-launch an analysis.

To consult the displacements in a specific floor, the program contains a Contour maps tab allowing the user to view the displacements for each loadcase or combination loadcase by means of contour lines with values or contour maps.

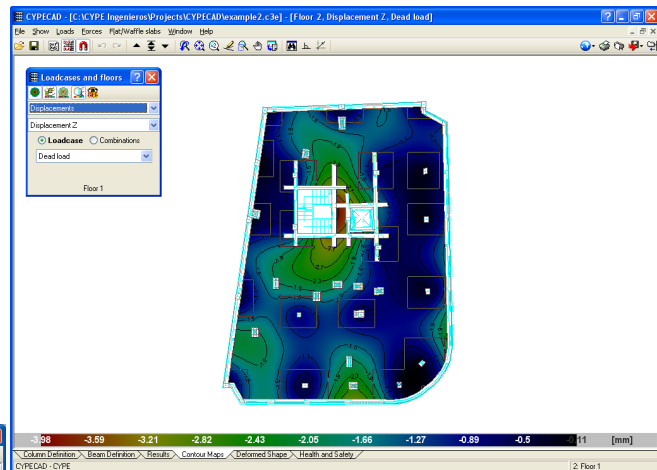


Fig. 15.255

15.3.6.2. Columns

Having checked the deformed shape of the structure, its columns are next in line to be checked. Any columns not verifying a check are displayed in red. To consult these errors, click on the **Results** tab, select **Columns > Edit**, then click on a column.

The Column editor window opens listing all the columns on the left hand side. Columns with design errors have a red cross displayed next to them.

In this example P2 is displayed in red. Click on it in the **Column edition** section. The **Reinforcement edition** section indicates that the ground floor span fails.

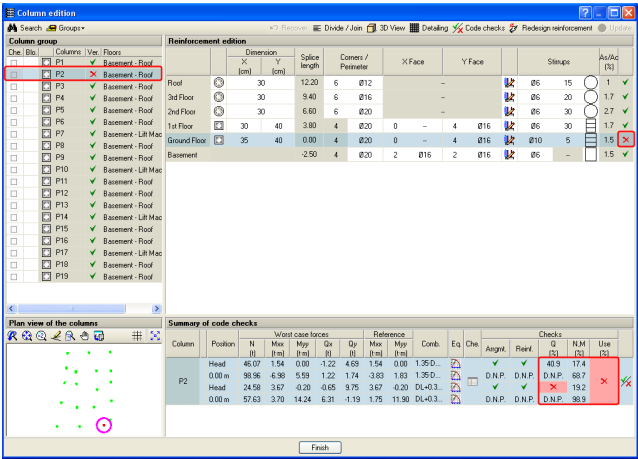


Fig. 15.256

By clicking on the **Code checks** option at the top of the **Column edition** window, the program displays a report detailing all the checks which are carried out and whether the column in question passes or fails the check. This way, the reason why the column fails can be consulted. In this case it is because of the stirrups. Change the column section to 40x40cm at that span, by changing the X dimension value in the table. Click on **Redesign reinforcement**, and the column will then be displayed in green to indicate it no longer fails.

Now we can group the columns. The dimensions of columns P3, P4, P5, P8, P9, P13, P15 and P16 match at all their spans. They all require similar reinforcement except P3 and P4 which require slightly more. So not to penalise all the columns, P3 and P4 will be grouped separately. Click on P9 (which is the column which requires the most reinforcement) in the **Column group** section then on Groups > Copy reinforcement to other groups. Within the window which opens, click on the columns to which the reinforcement is to be matched, or alternatively, mark them on the column on the left.

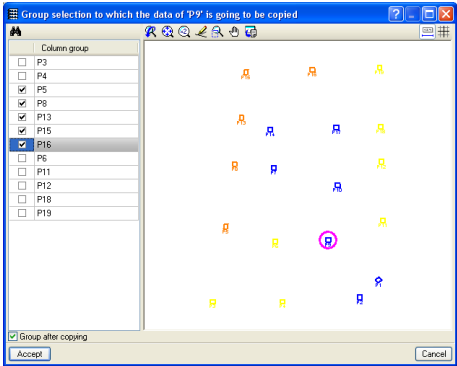


Fig. 15.257

The selected columns will be highlighted in orange. The remaining columns will either be blue, which means the reinforcement cannot be copied to those columns, or yellow implying the reinforcement can be copied although it is not guaranteed to work (the program would warn of this). Click on Accept.

The Column edition window now displays the columns as a group. However, P15 fails with this newly assigned reinforcement.

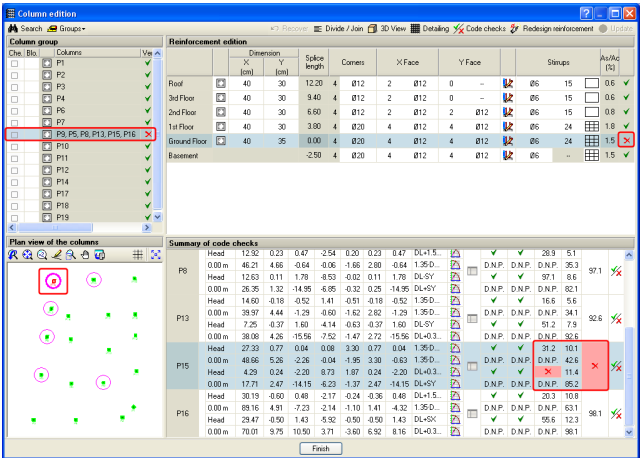


Fig. 15.258

Select the Ground floor span in the table within the Reinforcement edition section and then P15 in the summary of Code checks section. Click on the Code checks option at the top of the window and a report will appear displaying the checks carried out on the column. Here, we can see the column fails due to stirrup spacing. Close the report and change the 24cm spacing to 15cm spacing. At the same time change the 24cm spacing of the span above to 15cm; this way all the spans have a 15cm stirrup spacing. Upon changing the values, the column and column group no longer fail.

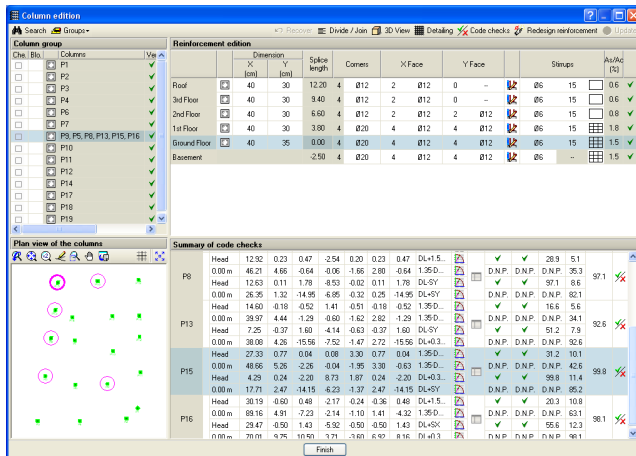




Fig. 15.259

The fire resistance check of the columns has also been carried out. Go to Group 1: Ground Floor. Click on the fire resistance icon  and from the floating menu that appears, select the **fire check of the current group** option . No columns are displayed in red, implying all of them pass the fire check. However, the floor slab does appear in red. If it is approached by the cursor, the information box that appears indicates the ribs should be at least 16mm thick to pass the check.

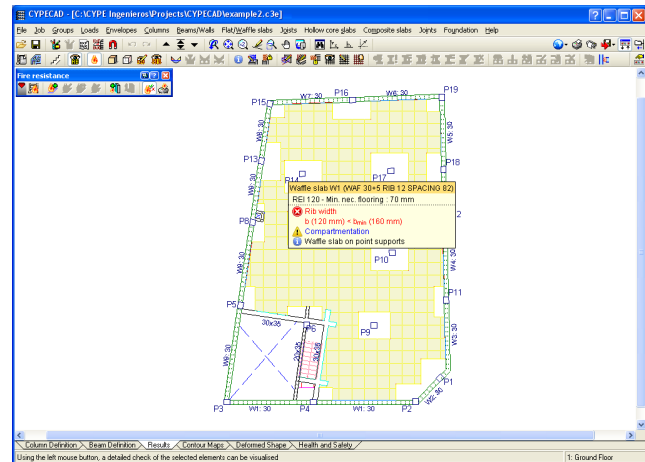


Fig. 15.260

By then selecting it using the left mouse button and clicking with the right, a window appears with more details of the fire resistance check.

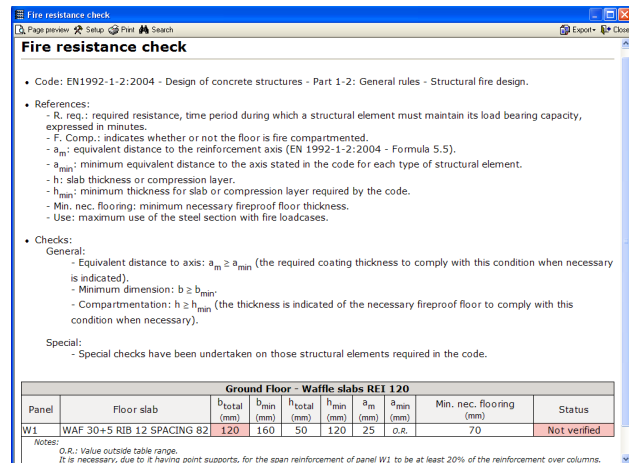



Fig. 15.261

Therefore, the waffle slab must be edited and the job reanalysed. Select the **Beam Definition** tab and click on **Slabs > Panel manager** and from the floating menu select **Define panel**. Select Waffle slabs. From the emerging dialogue box click on the Waf. 30+5 rib 12 spacing 82 slab defined in the table, followed by a click on the edit icon . Edit the geometry of the slab so its rib width is 16mm and its spacing 78cm (so the rib distribution remains the same).

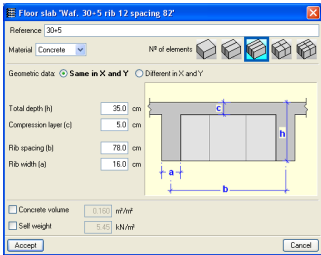


Fig. 15.262

The job must be reanalysed due to the slab changes. However, if these had not been present, as the dimensions of the columns have changed, it is recommended the job be re-analysed anyway to take into account the increase in stiffness of these columns.

As the columns have been grouped and modified, and the job has to be re-analysed, the user has an option available so that the column modifications are not lost; the program allows the reinforcement to be blocked. The option can be accessed by clicking on **Columns > Block reinforcement**. Upon activating this option, a window opens displaying all the columns of the job. By clicking on the columns or by selecting them using a capture window, the columns whose reinforcement is to be blocked can be chosen.

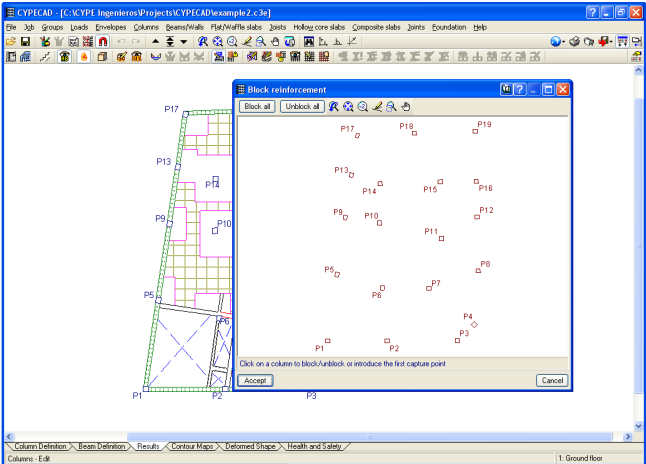


Fig 15.263

15.3.6.3. Beams

15.3.6.3.1. Errors

The next step consists in checking the incidents and errors that arise in the beams during the analysis phase. Using the option **Beams/Walls > Beam errors**, in the **Results** tab, beams displaying errors can be consulted (these are displayed in red or yellow after the analysis).

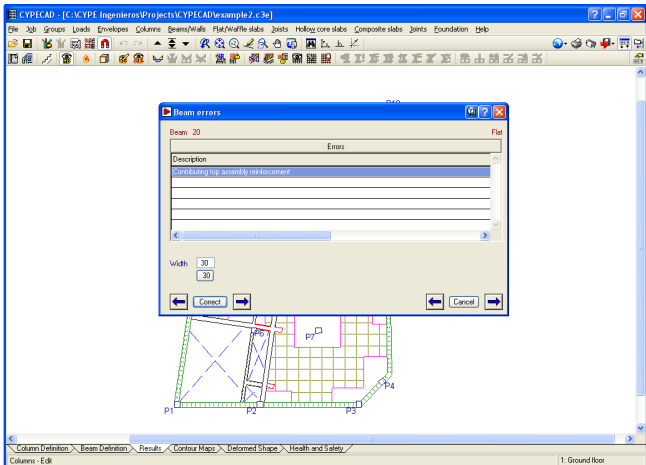



Fig. 15.264

The errors window displays the errors the consulted beam contains. A button in the bottom left hand corner displays the current dimensions of the beam and below it, the proposed dimensions the beam is to have in order to correct the error. The window contains a help button  explaining all the possible errors a beam can have.

The colour in which the beam is displayed is related to the **Error evaluation** option located in the beam options section in the **General Data > By position**. Here, all the errors that may arise in a beam are displayed, organised in tabs and by element, (stirrups, concrete, etc.) with an associated colour depending on its importance.

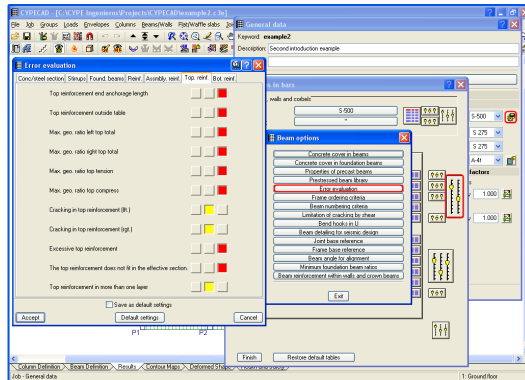


Fig. 15.265

15.3.6.3.2. Reinforcement modification

The program contains an option to consult and modify the beam reinforcement: Beams/Walls > Edit beams in the Results tab. Using this option, the reinforcement of the various beam alignments (continuous beams) that have been analysed can be edited.

15.3.6.3.2.1. Alignments

The program generates an alignment between two beams if they contain a common node and if the deviation angle of

one with respect to the other does not exceed the value indicated in the option **General data > By position > Beam options > Beam angle for alignment**.

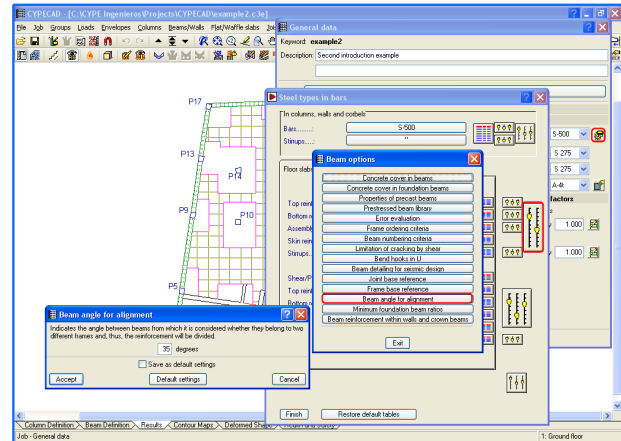





Fig. 15.266

As well as this automatic option, the program contains options to divide or join alignments generated by the program: **Beams/Walls > Beam alignments in the Beam Definition** tab.

15.3.6.3.2.2. Edit beams

When this option is selected, and then a beam is clicked on, a window appears displaying the alignment. For this example, select the beam spanning between columns P11 and P15 in group 2.

Click on the **Longitudinal reinforcement button**  at the top left of the screen so it may be modified. Select the **Join** option  and click on the 3ø10 (bottom reinforcement) to the left of column and once selected, click on the 3ø10 at the centre of the span. A window asking how many bars are to be joined appears. Indicate 3. The bars will be

combined to form three individual bars. Repeat the process with the reinforcement to the right of column P15. The reinforcement has to be resituated within the beam. To do so, click on the **Resituate** button , which repositions the modified bars. If the required steel area is verified, the bars will be displayed in green, otherwise in red.

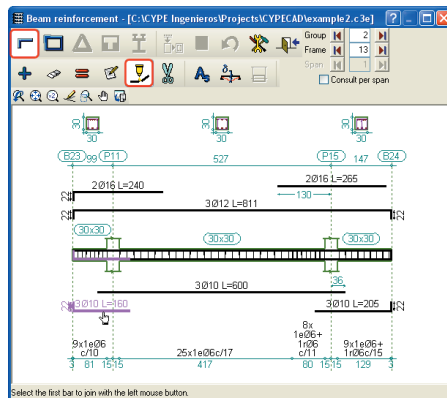


Fig. 15.267

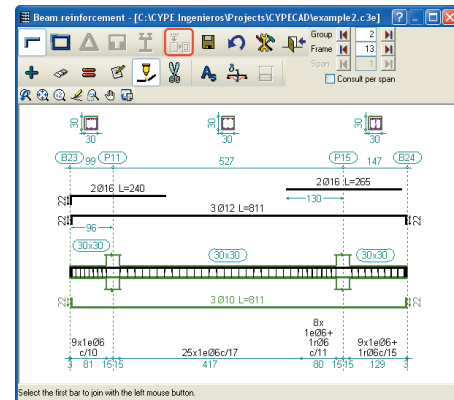



Fig. 15.269

Once the reinforcement has been resituated, click on **Save**  to save the modifications. Using the previous or next Frame or Group buttons, the user can view all the alignments (frames) of the job.

15.3.6.4. Floor slabs

15.3.6.4.1. Reinforcement modifications

The program designs the slab reinforcement using the forces obtained from the analysis. In the case of waffle and flat slabs, the reinforcement may not be very uniform due to these forces. To avoid this situation and allow for more homogenous results, the program has a series of tools available allowing for the reinforcement to be edited or modified. These options are located in **Flat/Waffle slabs > Modify reinforcement** in the **Results** tab.

First, the reinforcement to modify must be selected. In this case, the reinforcement will be matched in the support area and in the intermediate zones. To do so, the sum of the steel areas of the ribs is to be found and then divided by the number of ribs, resulting in a mean steel area which is

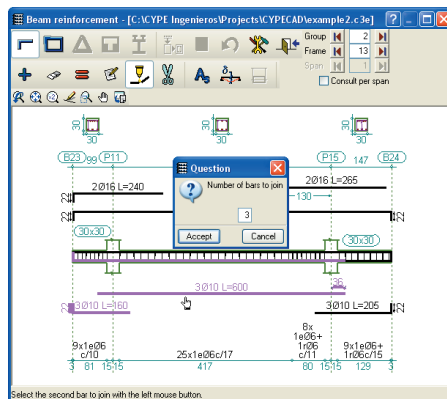


Fig. 15.268

what is to be left. As an example, the process will be carried out in the area supported by columns P5-P6-P9-P10-P13-P14 on the Ground floor.

The reinforcement modifications can also be carried out within the **Contour Maps** tab. Here, the forces with which the reinforcement has been designed can be seen whilst modifying the steel areas and lengths. This can result to be very helpful as the force distribution can be used as a guide as to the layout of the reinforcement.

Click on the **Results** tab and select **Flat/Waffle slabs > Views** and mark the following boxes: Additional reinforcement, Predefined reinforcement, Longitudinal and Bottom.

The required reinforcement for the area between P5 and P6 is of 2ø10 bars per rib. The next rib up requires 1ø12 + 1ø10, then the next two 1ø16 + 1ø10. The 1ø12 + 1ø10 bars will be matched with the 1ø16 + 1ø10 bars.

+ 1ø10 reinforcement (just above the 1ø12 + 1ø10) and drag the cursor to the rib above it and click with the mouse. Click on the same rib again and drag the mouse cursor downwards to the where the 1ø12 + 1ø10 is, finishing off with a click.

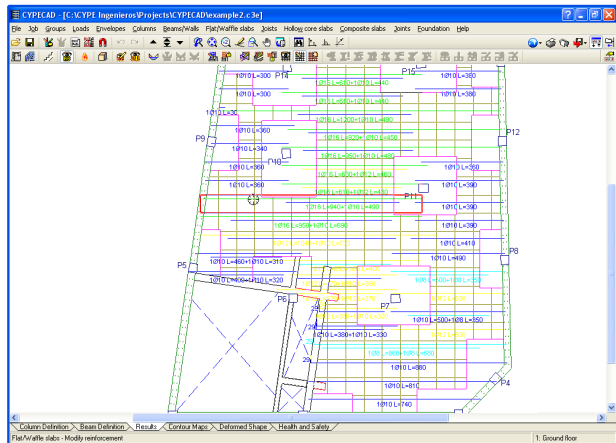


Fig. 15.271

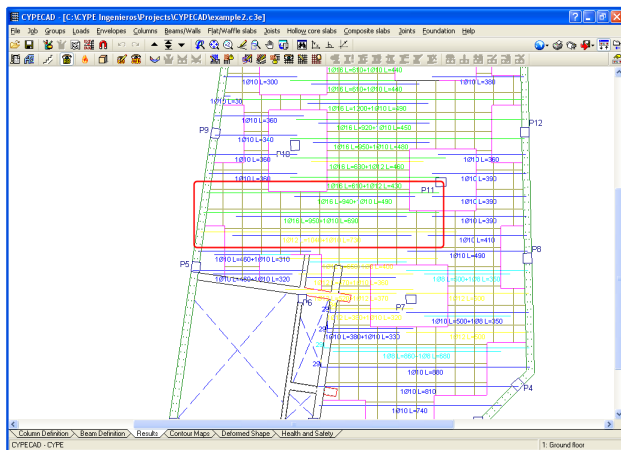


Fig. 15.270

Go to option **Flat/Waffle slabs > Modify reinforcement > Modify distribution width** and mark the **Eliminate overlapped reinforcement box**. Click on the lower 1ø16

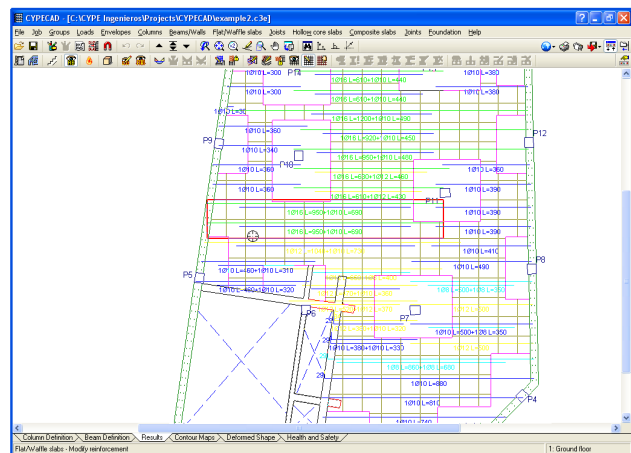


Fig. 15.272

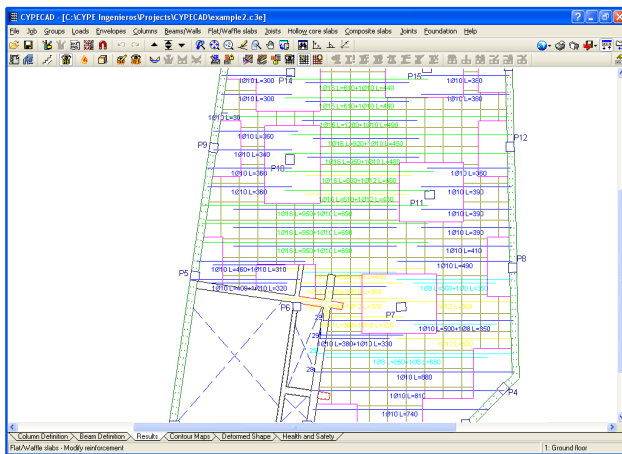


Fig. 15.273

The reinforcement has been matched. The total steel area is now greater than what was originally provided by the program, which implies the required area is present. If it were to be the other way around and less steel area is provided, if there is an adjacent longitudinal beam, it can be edited and its bottom reinforcement modified to include the necessary extra steel area.

The process can be repeated for the reinforcement next to column P13: change the **1Ø12 + 1Ø10 to 1Ø12 + 1Ø12 as in the rib above and the 1Ø8 + 1Ø8 and 1Ø10 + 1Ø10 to 1Ø10 + 1Ø8** at both ribs. If no options have been selected since the last step, the reinforcement can be modified without having to click on the option again, as the command is still active. Click on the reinforcement in the rib next to P13 and drag the mouse cursor to the next. Click on the **1Ø10 + 1Ø10** rib and drag the cursor to the rib above it. As this is to be changed to 1Ø10 + 1Ø8, right click and from the menu select **Modify group**. In the emerging window, change the reinforcement of the second layer to Ø8 and Accept.

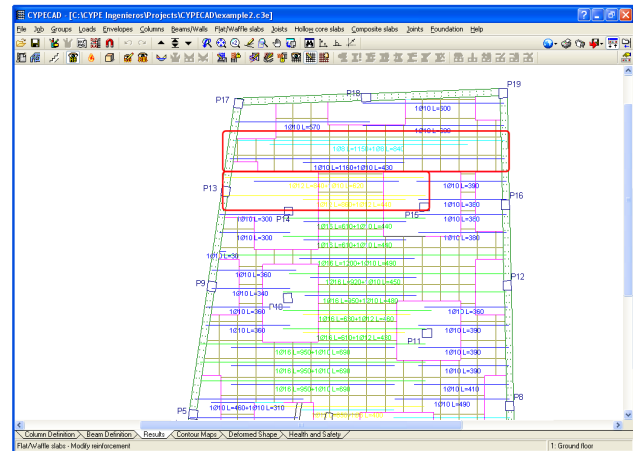


Fig. 15.274

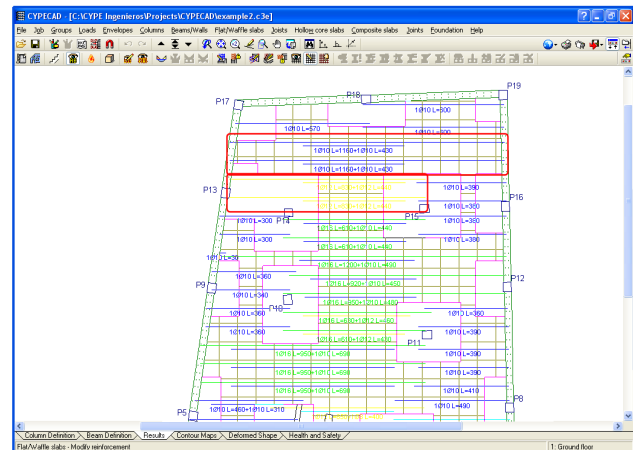


Fig. 15.275

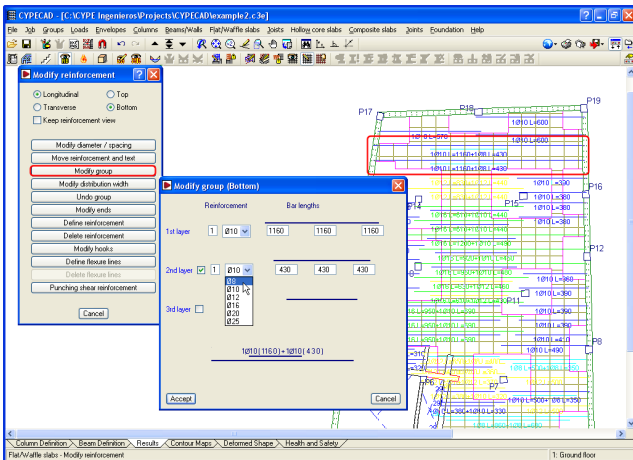


Fig. 15.276

Carry out the same process in the central area and in the area to the right so to match the reinforcement.

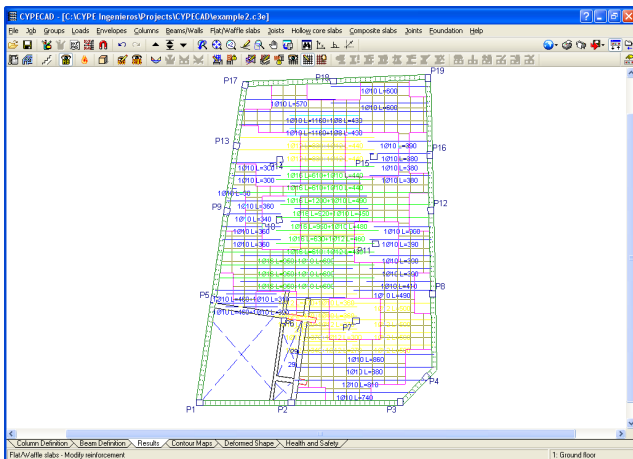


Fig. 15.277

The remaining reinforcement: longitudinal top reinforcement and transverse top and bottom reinforcement of the floor and remaining floors can be modified in the same way.

15.3.6.4.2. Copy reinforcement from another group

If the case arises where the floor geometry and loading is the same for one or more groups, the modifications only need to be carried out in one group and then using the option **Flat/Waffle slabs > Copy reinforcement from another group** from the **Results** tab, the reinforcement can be copied. To do so, place the main view of the screen at the floor to which the reinforcement is to be copied, click on the option and in the floor group window that opens, select the floor from which the reinforcement is to be copied.

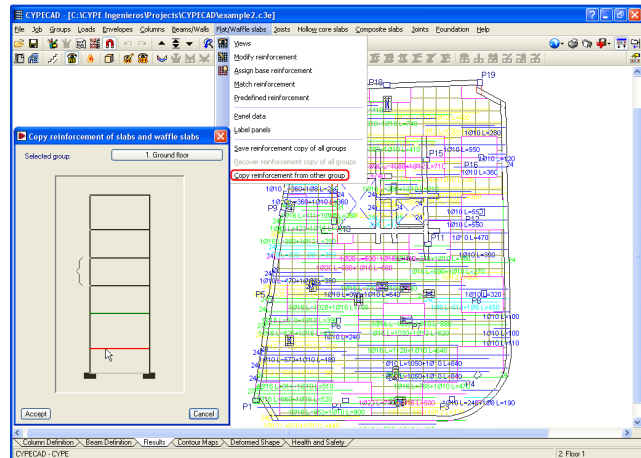


Fig. 15.278

This option shall not be carried out for this examples as the two floors with equal geometry and loading have already been grouped.


15.3.6.4.3. Save reinforcement copy of all groups

If a modification has to be undertaken on the job which implies having to re-launch an analysis, so not to loose any reinforcement modifications, these may be saved (otherwise, the program will provide the required reinforcement, regardless of any previous modifications that were carried

out). This option can be accessed by clicking on Flat/Waffle slabs > Save reinforcement copy of all groups in the Results tab.

15.3.6.5. Foundations

15.3.6.5.1. Soil data

Before designing the foundations, the properties of the soil and concrete to be used for the foundations must be specified in **Job > General data**. Having selected the General data option, click on the Foundation data button . Here, the allowable bearing pressure for persistent and accidental situations can be defined. To the right of the allowable bearing pressure in persistent situations box, a blue arrow icon is displayed. Upon clicking on it, a table will appear indicating the most common types of soil and their corresponding bearing pressures. In this case select **Gravel**. Mark the **Consider wind and earthquake combinations**. As the foundations are connected to one another by strap and tie beams, the **Verify footing sliding** box need not be selected.

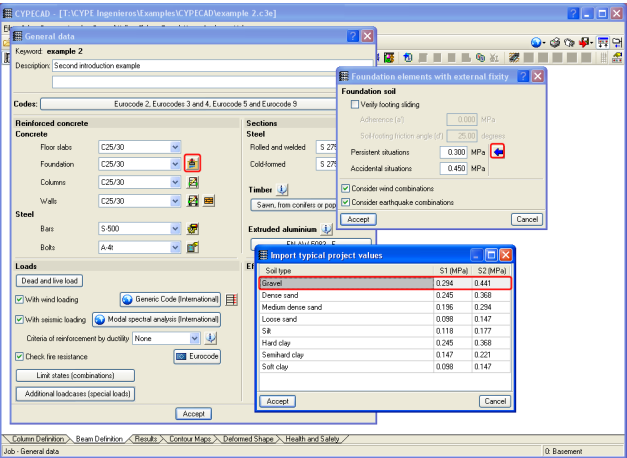


Fig. 15.279

15.3.6.5.2. Design

The foundations are designed using the option: **Foundation > Design**. A window will open in which the elements to design are to be indicated. In this case, select all the elements and the **Iterative design** method.

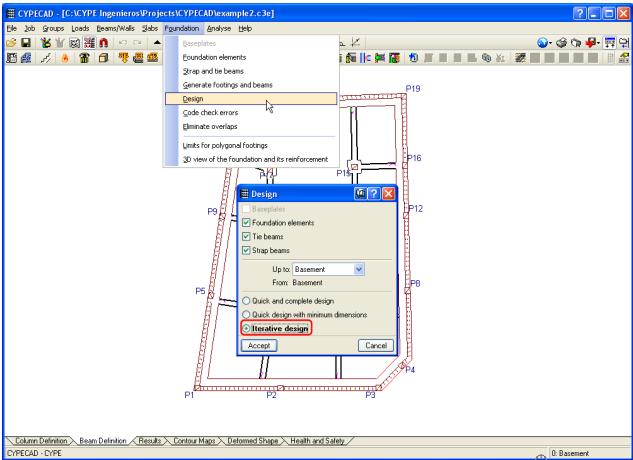



Fig. 15.280

15.3.6.5.3. Footing edition

Having designed the foundations, the elements must be edited to match the results obtained by the program.

First of all, the strip footing below the wall at the top is to be edited. Click on **Foundation > Foundation elements > Edit**  and click on the strip footing. In the emerging window, click on **Geometry** and modify the depth of the footing to 60 cm and the overhang to 49.0 cm (as the right overhang is 1.0 cm).

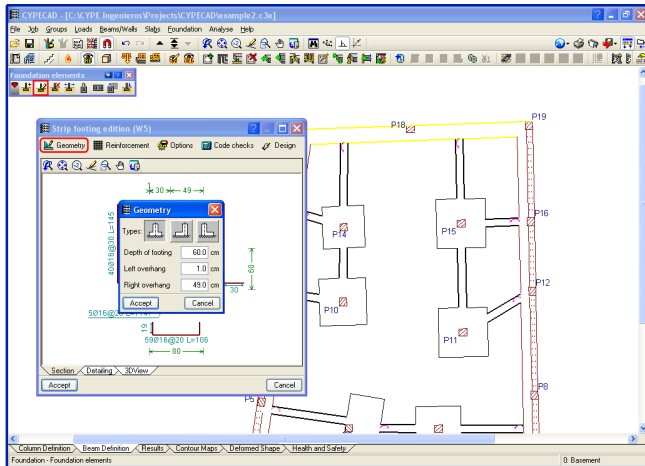


Fig. 15.281

Click on **Design > Redesign**, to redesign the reinforcement without changing the geometry of the footing. The program will emit a message indicating the checks have been verified.

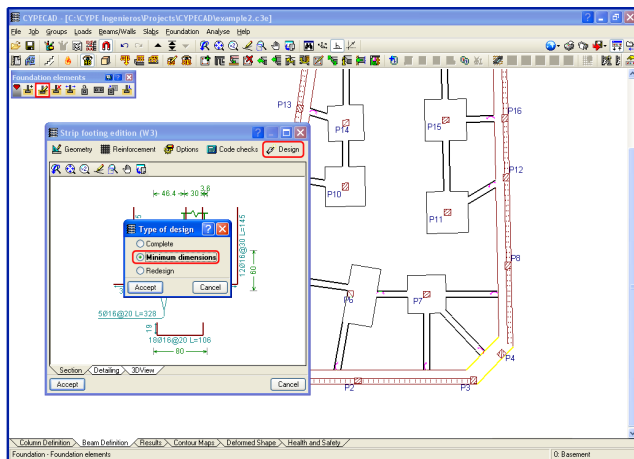


Fig. 15.282

Repeat the process with all the strip footings, giving them a depth of 60 cm and a total overhang of 50 cm.

The strip footing in the bottom right hand corner has not been designed due to it being in an area where forces concentrate as well as not having any strap beams reaching it to balance out the forces. The walls on either side of it are probably capable of balancing it, but as they do not reach it at right angles, the program does not bear them in mind. To avoid this situation, introduce a strap beam from the footing below column P7 to the strip footing. Edit the footing geometry and assign the previous dimensions to it. Click on **Design > Minimum dimensions**, this way the initial proposed geometry is maintained and the dimensions of the footing can only increase. The program proposes a footing with an overhang of 70 cm and a depth of 60 cm. Having designed the footing, the strap beam can be eliminated, confiding in the adjacent walls to balance out the moment.

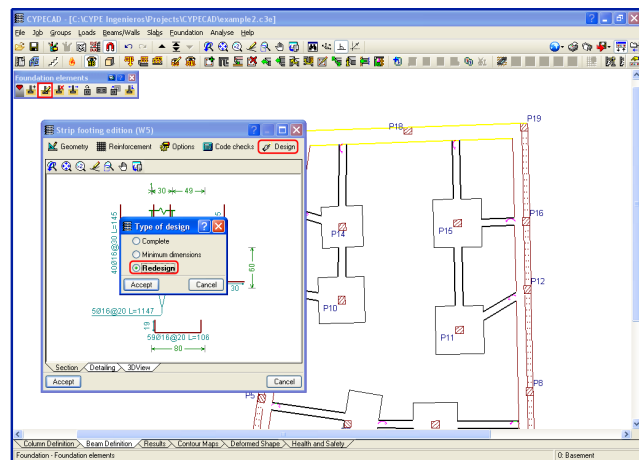




Fig. 15.283

15.3.7. Edition resources

The program has a series of graphics tools available to draw lines, circles, arcs and dimensions, add texts and even insert CYPE construction details or from other CAD programs. The edition resources can be accessed by clicking on the  button in the top toolbar. A window will open upon its selection where all the drawing options are listed. These options can be used together with the object snap options  to snap to elements of the program or entities of the DXF/DWG file that has been imported in the job.

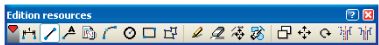




Fig. 15.284

15.3.8. Drawings

Having checked all the results, the drawings of the project can be obtained. To do so, click on the Job drawings button in the top toolbar . The Drawing selection window will open. The drawings to be obtained are added  one by one, selecting the type of drawing from the scroll menu.

The configuration of the drawings can be modified as well as being able to add construction details to the type of selected drawing by clicking on the Details button.

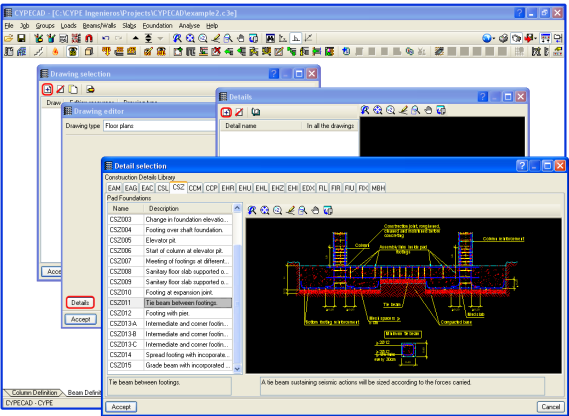


Fig. 15.285

The peripherals to be used must be defined for each drawing as well as specifying whether or not the edition resources used on the floors of the job are to be drawn.

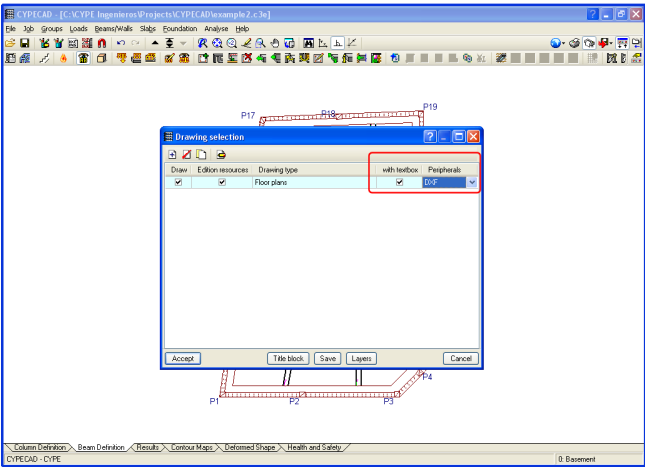


Fig. 15.286

The option of introducing a title block is also available. The Title block option is located at the bottom of the Drawing selection window. Upon clicking on it, a file with the title block can be imported, a default title block can be selected or, alternatively, the option **None** can be selected and so the program will respect the area marked for the title block when distributing the layout of the drawings.

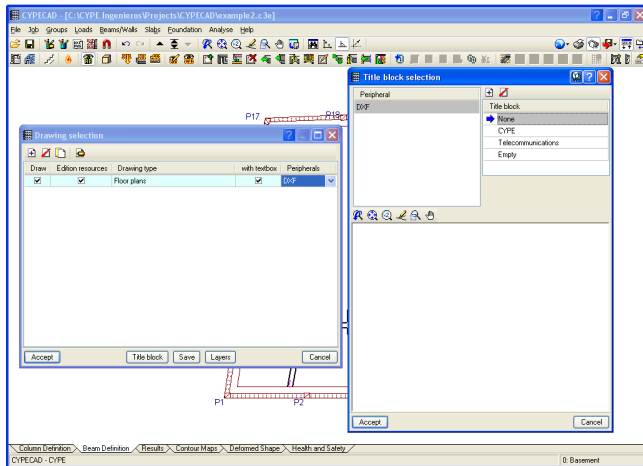


Fig. 15.287

Having concluded the drawing selection and clicking on Accept, the Drawing composition window appears in which a previous view of the drawings is displayed. The user can move the elements, modify their scale, as well as move the element texts before finally printing out the drawings.

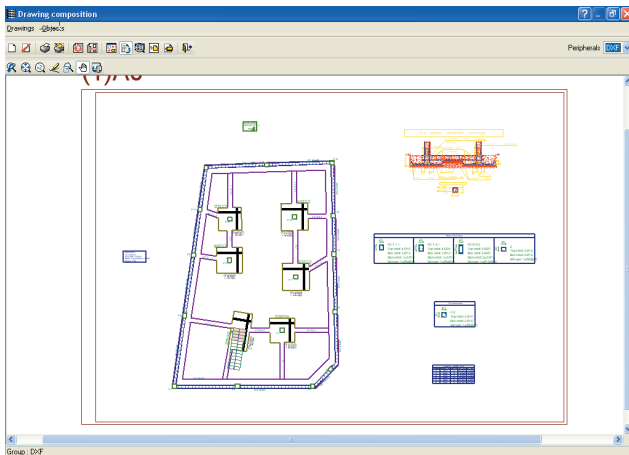



Fig. 15.288

15.3.9. Reports

Finally, the program contains a series of reports to aid in the completion of the Project report. To obtain a list of the available reports click on Reports , located to the left of the drawings button, seen in the previous section. Upon selecting it, a window appears from which the required document can be selected.

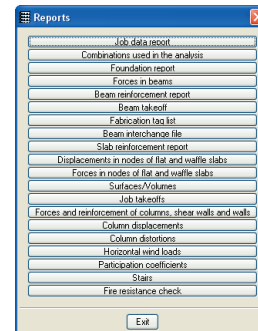


Fig. 15.289

