

User manual
CYPECAD

CYPECAD

User manual



**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. .

1st Edition (September 2010)

Edited and printed in Alicante (Spain)

Windows is a registered trademark of Microsoft Corporation.

Presentation	9	2.1.1. Template fundamental properties	23
1. Program introduction	11	2.1.2. Template options	23
1.1. General aspects of the working screen	11	2.2. DXF/DWG Object Snap	25
1.1.1. Tabs	11	2.3. Drawings and Reports	25
1.1.2. Main menus	11	3. Groups and Floors	26
1.1.3. First toolbar	12	3.1. General information on groups	26
1.1.4. Second toolbar	12	3.2. Floor and group introduction	26
1.1.5. Background Colours	12	3.2.1. Creation of new floors	26
1.2. On screen help	13	3.2.2. Use categories	27
1.2.1. Pressing F1	13	3.2.3. Elevation of foundation plane	28
1.2.2. Question mark icon	13	3.2.4. Creation, edition and deletion of groups	28
1.2.3. Book icon	13	3.3. Floor and group options	29
1.2.4. Quick guide	13	3.3.1. General Options	29
1.3. Job introduction	14	3.3.2. Views	30
1.3.1. Automatic introduction	14	4. Columns	32
1.3.1.1. Automatic introduction DXF/DWG	15	4.1. Definition of columns, shear walls or starts	32
1.3.1.2. Automatic introduction IFC	16	4.1.1. Column starts	32
1.3.2. Manual introduction	19	4.1.2. Columns	32
1.3.3. File management	20	4.1.2.1. New column	32
1.3.3.1. Recent files	20	4.1.2.2. New column with its start on another column	33
1.3.3.2. ASCII file import to generate the foundations	20	4.1.2.3. Column edition	35
1.3.3.3. Job examples	20	4.1.2.4. Introduction of new columns using data from another column	35
1.3.3.4. Search	20	4.1.3. Shear walls	36
1.3.3.5. Send, Decompress and Share jobs	21	4.1.4. Column minimum geometric ratios	36
2. Templates	23	4.2. Column results	36
2.1. Template properties and options	23		

4.2.1. Column design	36	5.1.7. Design options	46
4.2.1.1. Concrete columns	36	5.1.8. Column disconnection in masonry and concrete block walls	46
4.2.1.2. Steel columns	37	5.2. Drawings and reports	46
4.2.1.3. Column data	38	5.2.1. Reports of block walls	46
4.2.1.4. Column layout	39		
4.2.2. Block column reinforcement	39	6. Beams	47
4.2.2.1. Things to consider when using this option	39	6.1. Beam introduction manager	47
4.2.2.2. Procedure	40	6.1.1. Beam introduction menu	47
4.2.3. Force diagrams	40	6.1.2. Beam types	49
4.3. Drawings and Reports	40	6.1.3. DXF or DWG object snap during beam introduction	50
4.3.1. Drawing options	40	6.1.4. Polybeams	50
4.3.1.1. View of the fixed point in the drawings	40	6.1.5. Predefined reinforcement spans	51
4.3.1.2. Column stirrup options	41	6.1.6. Sloped beams	51
4.3.2. Report options	41	6.1.6.1. Diagonal braces	51
5. Walls	42	6.1.7. Common beam	53
5.1. Wall introduction	42	6.1.7.1. Create common beam	53
5.1.1. Floating menu	42	6.1.7.2. Undo common beam	53
5.1.2. Wall types	42	6.1.8. External edge beam introduction	53
5.1.2.1. Reinforced concrete walls	42	6.1.9. Assign beam and wall options	53
5.1.2.2. Generic reinforced concrete block walls	42	6.1.10. Extend beam	54
5.1.2.3. NORMABLOC walls	43	6.1.11. Corbels	54
5.1.2.4. Masonry walls	43	6.1.11.1. Corbel introduction	54
5.1.3. Wall openings	44	6.1.11.2. Corbel edition	54
5.1.3.1. General	44	6.1.12. Corbel adjustment	54
5.1.3.2. Opening reinforcement	44	6.1.13. Beam reinforcement within walls and crown beams	54
5.1.3.3. Opening reinforcement design	45	6.1.14. Effective slab width when using steel beams	55
5.1.4. Extend walls	45	6.1.15. Beam reinforcement editor configuration	55
5.1.5. Assign walls	45	6.1.16. Beam joint and portal frame base references	55
5.1.6. Edit walls	46	6.1.17. Template object snap for beam introduction	55

6.2. Beam analysis and results	.56	7.1.2.5. Composite slabs	.71
6.2.1. Steel beam design within the analysis menu	.56	7.1.3. Sloped slabs	.72
6.2.2. Beam errors	.56	7.1.3.1. Sloped floor slabs/ EI. Changes Window	.72
6.2.3. Sloped beams	.56	7.1.3.2. Creation	.72
6.2.4. Copy reinforcement between frames at different floors	.57	7.1.3.3. Edit	.73
6.2.5. Design of steel beams	.58	7.1.3.4. Assign	.73
6.2.6. Frame grouping	.58	7.1.3.5. Automatically assign a slope to beams	.73
6.2.7. Block frame reinforcement	.58	7.1.3.6. Advice on sloped panels	.73
6.2.7.1. Things to consider when using this option	.59	7.1.4. Introduce openings	.79
6.2.7.2. How the option works	.59	7.1.5. Floor slab options	.80
6.2.8. Steel beams	.60	7.2. Slab analysis and results	.80
6.2.8.1. Shear stud errors	.60	7.2.1. Joist floor slabs	.80
6.2.9. Corbel checks	.60	7.2.1.1. Views of joist floor slabs	.80
6.2.9.1. Redesign corbel reinforcement	.61	7.2.1.2. Match	.81
6.2.10. Sloped beam reinforcement edition	.61	7.2.1.3. Errors	.82
6.2.11. Configuration of the beam reinforcement editor	.61	7.2.1.4. Information	.83
6.2.12. Force diagrams	.61	7.2.1.5. Assign	.83
6.3. Drawings and reports	.62	7.2.2. Flat and waffle slabs	.84
6.3.1. Sloped beams	.62	7.2.2.1. Reinforcement introduction in flat and waffle slabs without analysing	.84
6.3.2. Corbels	.62	7.2.2.2. Flat/waffle slab options in the Results tab	.85
7. Floor slabs	.63	7.2.2.3. Add additional shear reinforcement around drop panels	.86
7.1. Floor slab introduction	.63	7.2.3. Hollow core slabs	.87
7.1.1. Floor slab floating menu	.63	7.2.3.1. Hollow core slab views	.87
7.1.2. Floor slab types	.64	7.2.3.2. Hollow core plates catalogue	.87
7.1.2.1. Awaiting definition	.64	7.2.4. Composite slabs	.87
7.1.2.2. Joist floor slabs	.65	7.2.4.1. Composite slab design	.87
7.1.2.3. Waffle slabs	.69	7.2.4.2. Composite slab results	.91
7.1.2.4. Hollow core slabs	.71	7.2.5. Force diagrams	.93
		7.3. Drawings and Reports	.93

7.3.1. Composite slabs	93	9. Steel, timber and aluminium structures	108
7.3.1.1. Reports	93	9.1. Integrated 3D structures	108
7.3.1.2. Drawings	94	9.1.1. Integrated 3D structures menu	108
7.3.2. Joist floor slabs	94	9.1.2. Metal 3D program	111
7.3.2.1. Report of forces in joist floor slabs	94	9.2. Joint design	111
7.3.2.2. Joist floor takeoff	94	9.2.1. Types of joint design	112
7.3.3. Sloped panels	94	9.2.2. Joint Design	112
8. Stairs	95	9.2.3. Consulting the designed joints	115
8.1. General information	95	9.2.4. Reasons why a joint has not been designed	116
8.2. Location of Stairs module	95	9.3. Export to other programs	116
8.3. Data introduction in the Stairs module	96	10. Loads	118
8.3.1. Create a new staircase	96	10.1. Loads to be defined in General data	118
8.3.1.1. Staircase common data	96	10.1.1. Actions	118
8.3.1.2. Staircase flight data	97	10.1.1.1. Wind loading	118
8.3.1.3. Introduction of the staircase in the job	99	10.1.1.2. Seismic loading	118
8.3.1.4. Tools to facilitate staircase and flight definition	101	10.1.1.3. Fire resistance	119
8.3.2. Edit existing staircases	103	10.1.2. Additional loadcases	119
8.3.3. Delete a staircase	103	10.1.3. Limit states	120
8.3.4. Move a staircase	104	10.1.3.1. Combination configuration for each limit state	120
8.3.5. Rotate a staircase	104	10.2. Definable live loading in groups	122
8.4. Results, reports and drawings of the Stairs module	104	10.2.1. Group loads	122
8.4.1. View staircase reinforcement details	104	10.2.2. Loads at foundation level	122
8.4.2. View forces and displacements using contour maps	104	10.3. Loads Menu	122
8.4.3. Staircase design	104	10.3.1. Surface loads on slabs	122
8.4.4. Reports	105	10.3.2. Automatic introduction of line loads on beams	123
8.4.5. Stair drawings	105	10.4. Drawings and reports	123
8.5. Stair module configuration (materials, reinforcement tables and stair options)	106		

11. Foundations124

11.1. Baseplates	124
11.2. Foundations	124
11.2.1. Foundation properties	124
11.2.2. Foundation floating menu	124
11.2.3. Further options for footings	126
11.2.4. Wall foundations (Strip footings)	127
11.2.5. Foundation options	128
11.3. Strap and tie beams	129
11.3.1. General information on strap and tie beams	129
11.3.2. Strap and tie beam floating menu	130
11.3.3. Further options for strap and tie beams	132
11.4. Automatic generation of footings and beams	132
11.5. Analysis and results	133
11.5.1. Design	133
11.5.2. Code check errors	133
11.5.3. Eliminate overlaps	133
11.6. Reports and drawings	133
11.6.1. Foundation report	133
11.6.2. Foundation and foundation reinforcement floor plans ..	134

12. Contour maps and deformed shape136

12.1. Contour maps	136
12.1.1. Loadcases and floors window	136
12.1.2. Forces menu	137
12.1.3. Flat/Waffle slabs menu	137
12.1.4. Window menu	138

12.2. Deformed shape138

12.2.1. 3D view of the deformed shape of the structure	138
12.2.2. Indication of the deformation using a colour scale.	139

13. Tables and libraries140

13.1. Reinforcement tables	140
13.1.1. Use of the reinforcement tables	140
13.1.2. Restore default tables	140
13.2. Section library	140
13.2.1. Edition of an individual section	141
13.2.2. Library manager	141
13.2.2.1. Use of existing libraries	141
13.2.2.2. Library edition	141
13.2.2.3. Library creation	141

Presentation

CYPECAD is the software for reinforced concrete and steel buildings 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 and steel 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.

1. Program introduction

1.1. General aspects of the working screen

1.1.1. Tabs

The following tabs can be seen at the bottom left hand corner of the main screen of CYPECAD:

- Column Definition
- Beam Definition
- Results
- Contour Maps
- Deformed shape

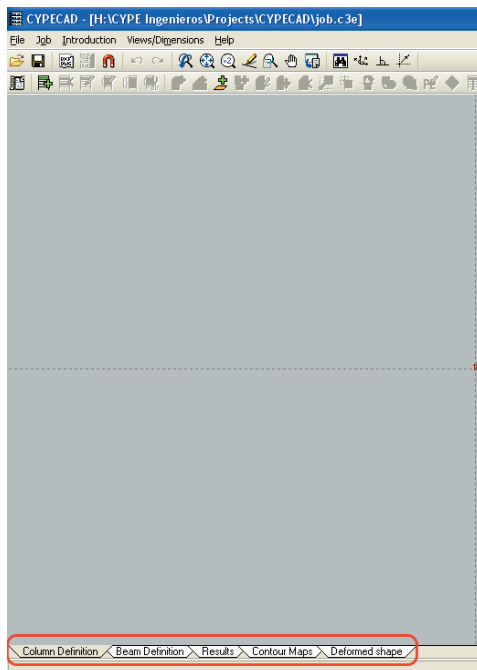


Fig. 1.1

By clicking on these tabs, different menus will appear containing various options for each stage of the building analysis.

1.1.2. Main menus

The main menus are located at the top of each main window of CYPECAD. These will allow you to access the various program options. The menus can also be accessed via the option bar which can be activated upon clicking the folder icon at the top right hand corner of the first toolbar.

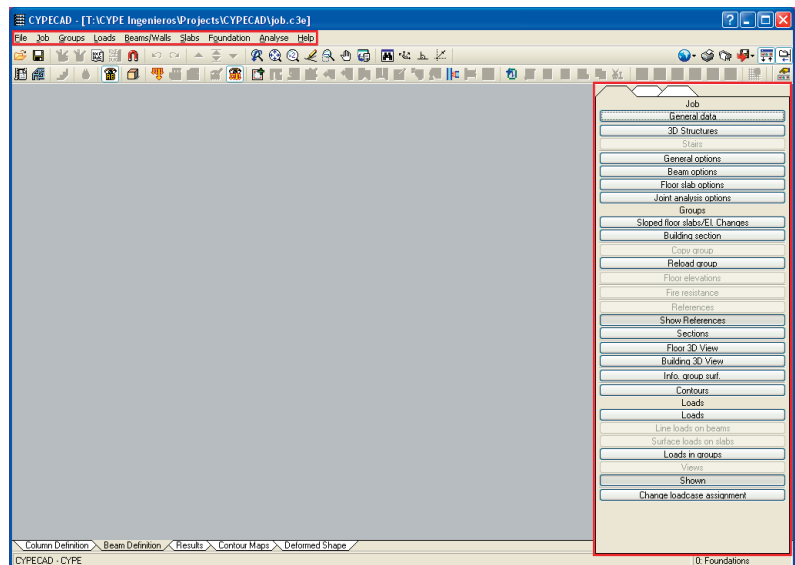


Fig. 1.2

1.1.3. First toolbar

The first toolbar has many elements common to all the tabs and are mainly visualisation and template options.

The **Undo** and **Redo** icons within the **Column Definition**, **Beam Definition** and **Results** tabs, allow to cancel or repeat the last modification.

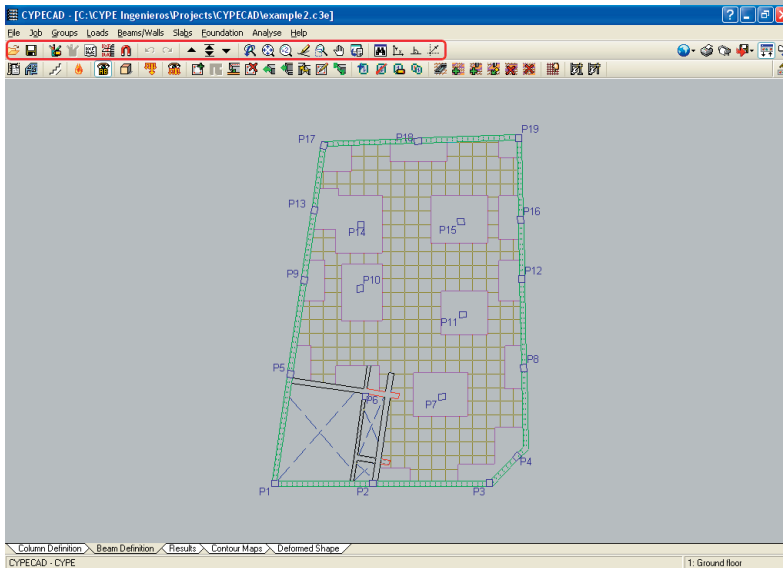


Fig. 1.3

1.1.4. Second toolbar

The program contains a toolbar with shortcuts to the menu options of the program (within the **Column Definition**, **Beam Definition** and **Results** tabs).

These toolbars can be configured by adding or removing options by clicking on the hand icon (toolbar configuration) situated on the right of the toolbar.

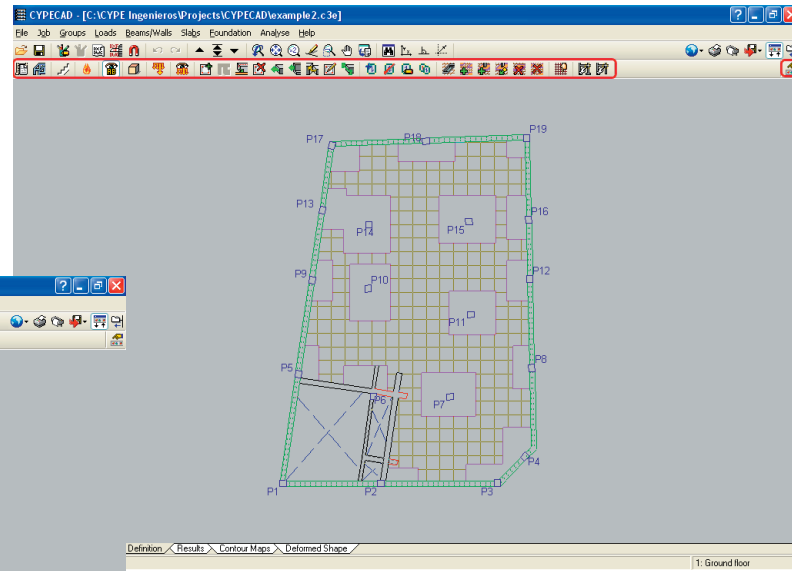


Fig. 1.4

1.1.5. Background Colours

The choice of background colours of CYPECAD is limited to three options: white, grey or black. This way, a greater contrast is created with the colours of the structural elements and hence an optimum visualisation is obtained.

The background colour can be selected from the drop down menu that appears upon clicking on the globe icon located on the right of the first toolbar.

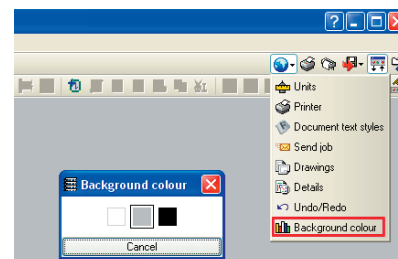


Fig 1.5

1.2. On screen help


CYPE programs have on screen help icons to provide the user with the necessary information on the program menus, dialogue boxes and options.

There are four different ways to access the help options:


1.2.1. Pressing F1

To obtain help on a menu option, press F1. All the icons with help explanations available will be highlighted in blue. If you wish to obtain help on an option within a menu, having pressed F1, unfold the menu and click on the desired option.

1.2.2. Question mark icon


A question mark icon  can be seen at the top right hand corner of the title bar. Specific help on a program option can be obtained the following way: click on the question mark icon then unfold the menu containing the option for which the help is sought and click on the option. A window will appear containing the required information. This option provides the same information as when the **F1** key is pressed.

This help can be deactivated in three ways: by clicking on the right mouse button, by clicking on the question mark icon or by pressing the **Esc** key.

You can equally obtain information on the toolbar icons. Upon selecting the question mark icon , the icons will be highlighted in blue (as with the F1 option). By then clicking on the icon, the help information will be displayed.

Option windows that open within the program also contain the question mark icon in their title bar. By clicking on it, the option or icon help can be accessed.

1.2.3. Book icon

Within the title bar of several windows, an open book icon  can be found which upon clicking on it will display general information on the contents of the window.

1.2.4. Quick guide

The option information displayed using the F1 key can also be consulted and printed out by selecting **Help > Quick guide**. These explanations are those corresponding to the options found within the screen tabs.

The explanations in the dialogue boxes are different to those of the guide.

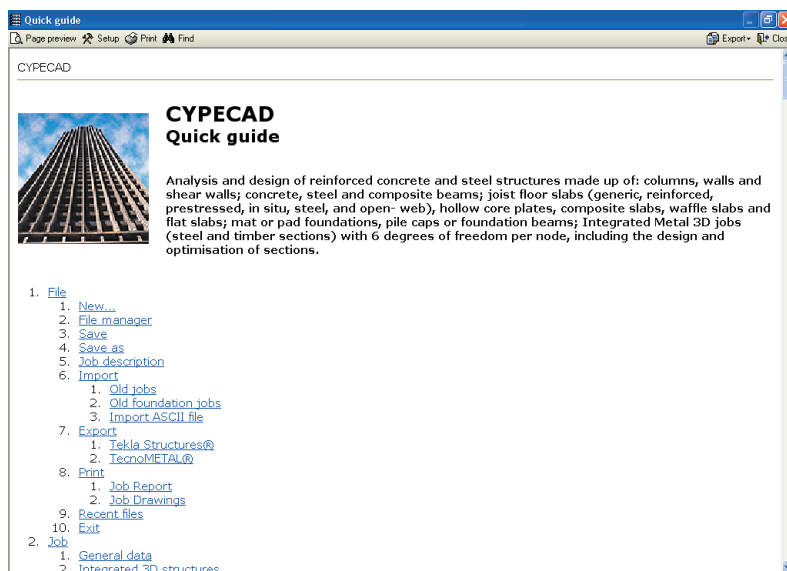


Fig. 1.6

1.3. Job introduction

To create a new job, click on **File > New**. A dialogue box will appear in which the job location, name and description of the job are required.

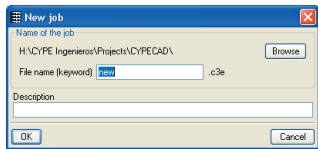


Fig. 1.7

Once filled in, click on **Accept** and a second dialogue box will appear where the program will prompt the user to choose between a manual introduction (**Empty job**), or, if you possess the corresponding license, an **Automatic introduction** using a DXF/DWG file or IFC, or if you wish to view an **Automatic introduction example**.

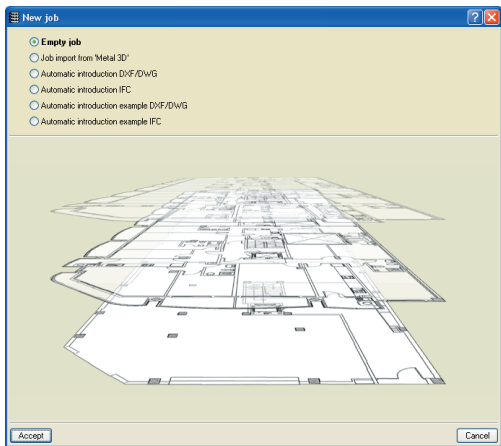


Fig. 1.8

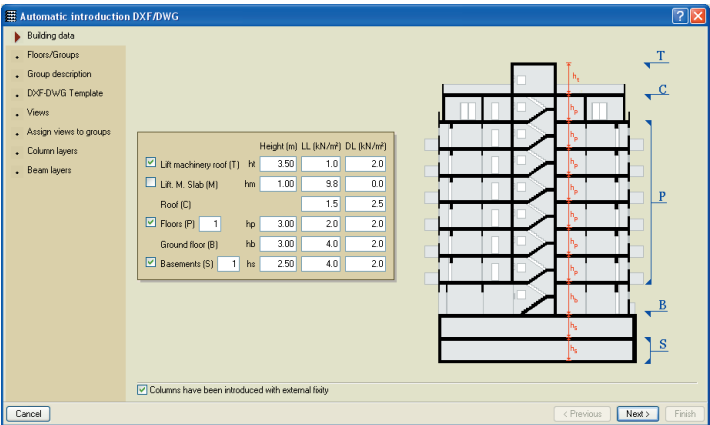


Fig. 1.9

1.3.1. Automatic introduction

The job introduction is carried out in various steps with the help of an assistant which will ask the user to gradually input properties of the job, such as floors, their heights and respective loads. Depending on which type of automatic introduction is selected, be it using a DXF/DWG template or an IFC file, the building outline and support elements can be introduced.

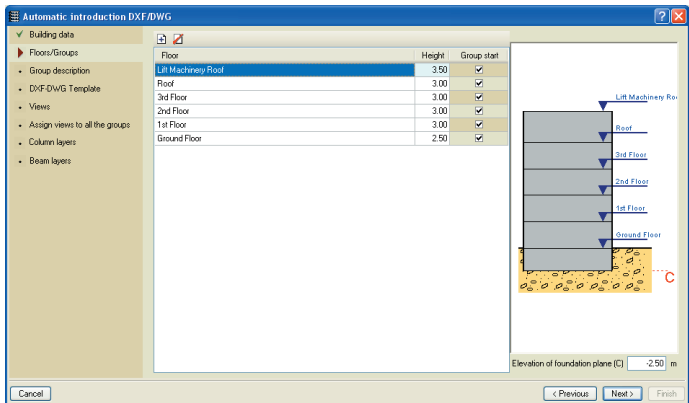


Fig. 1.10

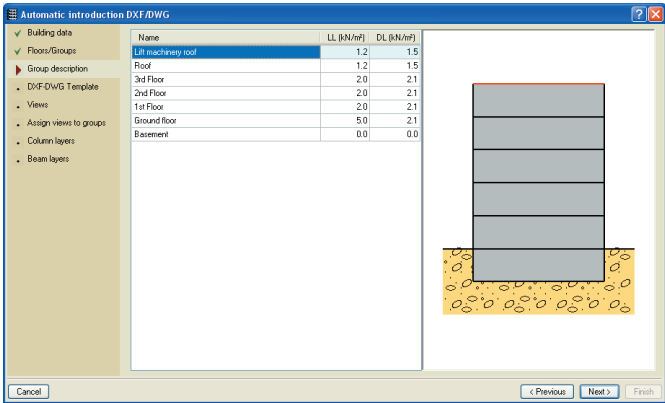


Fig. 1.11

1.3.1.1. Automatic introduction DXF/DWG

Having defined the building and floor data, the program will ask for a DXF or DWG file to be uploaded to use as a template. One or several files can be uploaded.

Their layer colours can be modified and subsequently the user will have to choose which layers are to remain visible. The following window will ask which files are to be applied to the groups, then which layers are to be used to draw the element geometries.

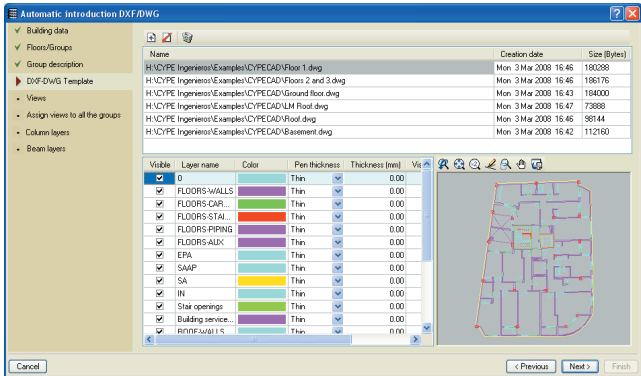


Fig. 1.12

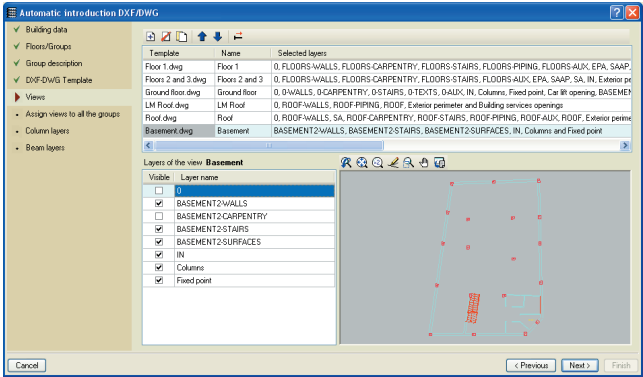


Fig. 1.13

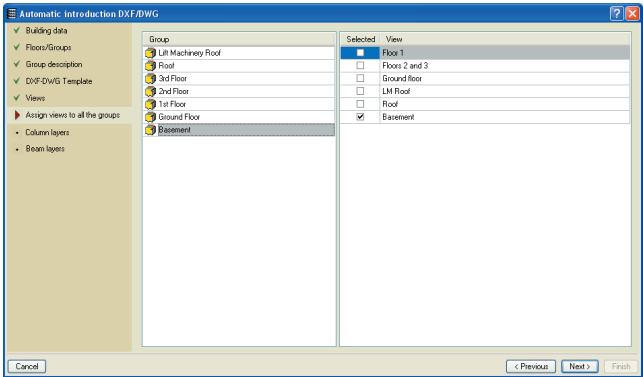


Fig. 1.14

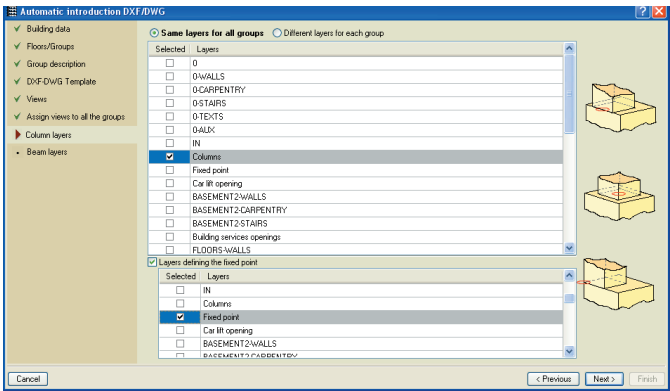


Fig. 1.15

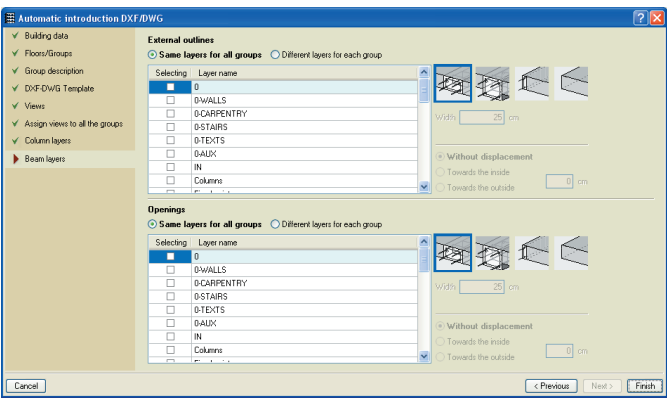


Fig. 1.16

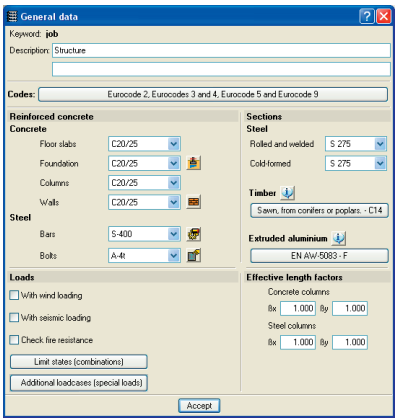


Fig. 1.17

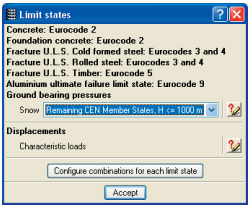


Fig. 1.18

1.3.1.2. Automatic introduction IFC

Upon selecting this option the program will prompt for an IFC file to be uploaded. Then, in a similar way as with the other automatic introductions, the building and floor data will have to be defined. The user will have to select which materials defined within the imported file are to represent the various elements of the job and assign properties to these selections.

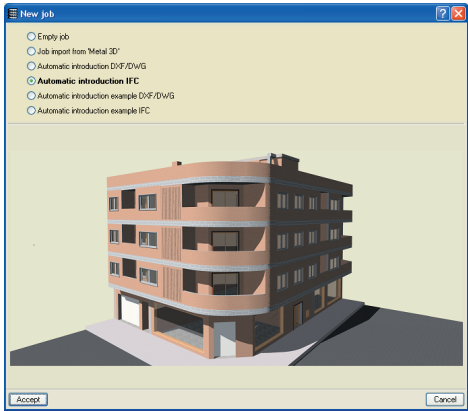


Fig. 1.19

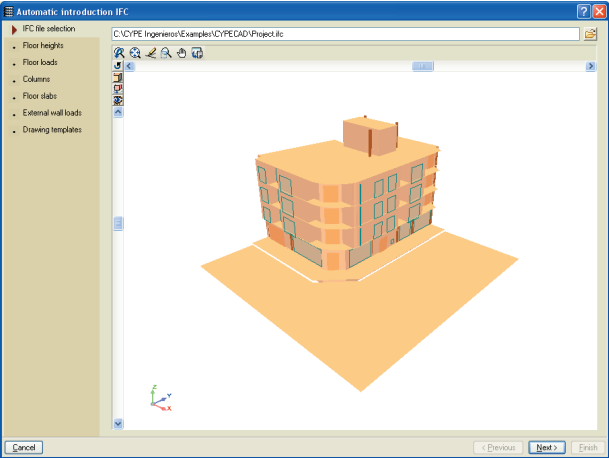


Fig. 1.20

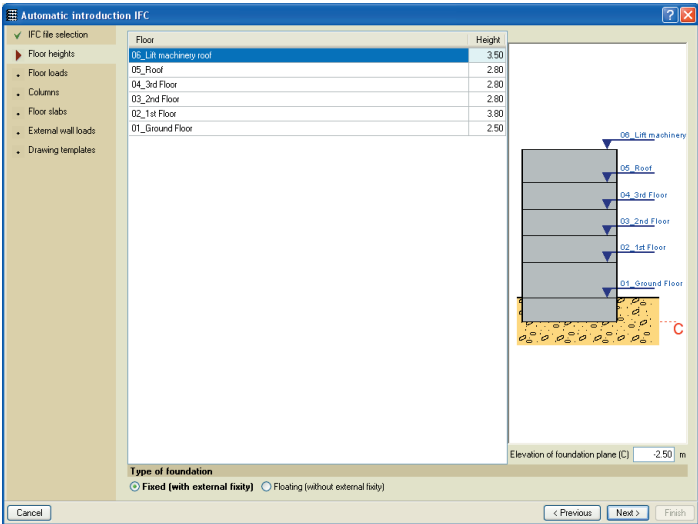


Fig. 1.21

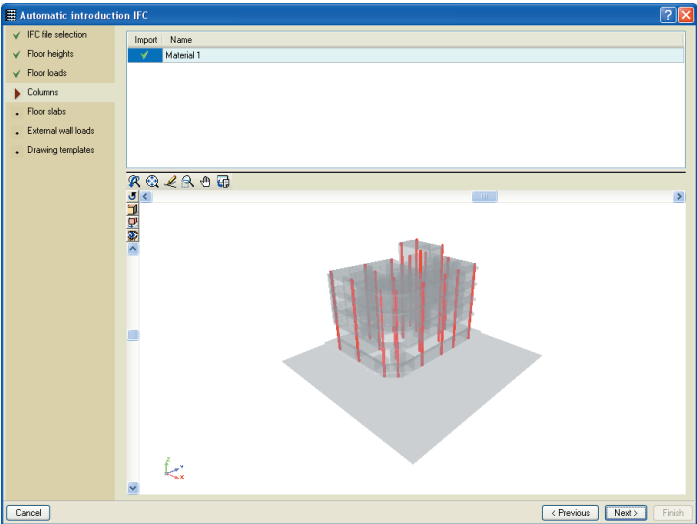


Fig. 1.23

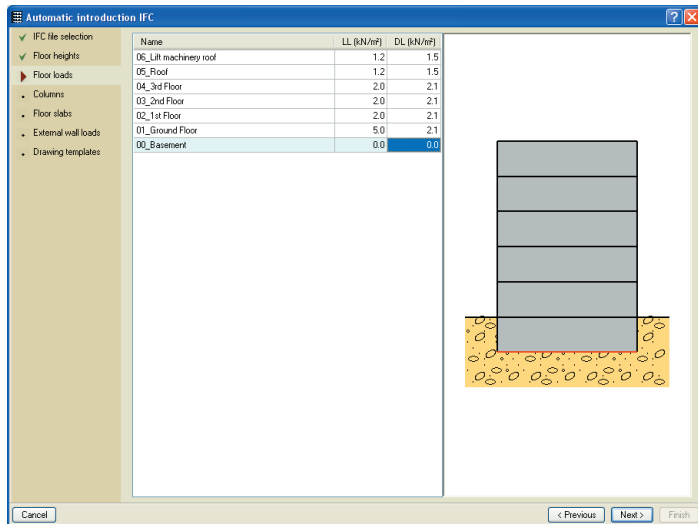


Fig. 1.22

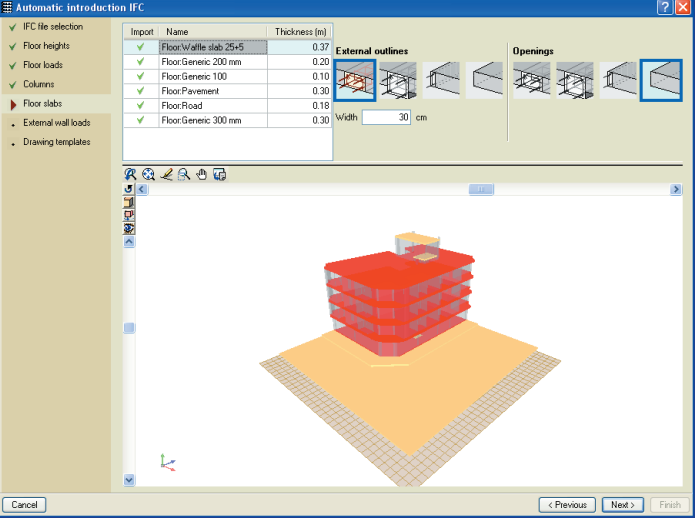


Fig. 1.24

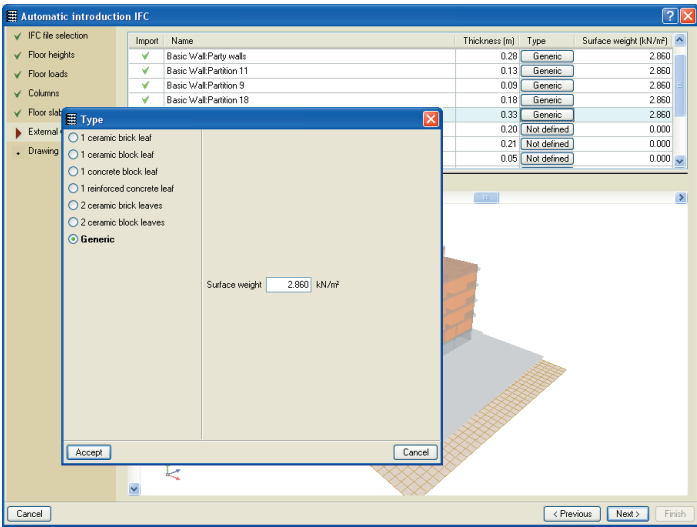


Fig. 1.25

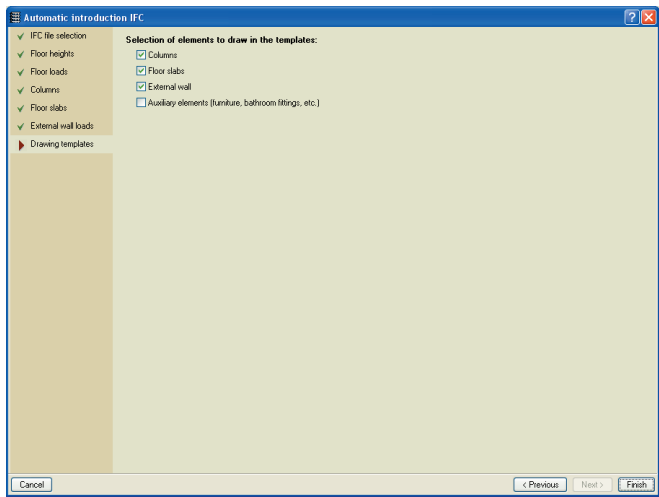


Fig. 1.27

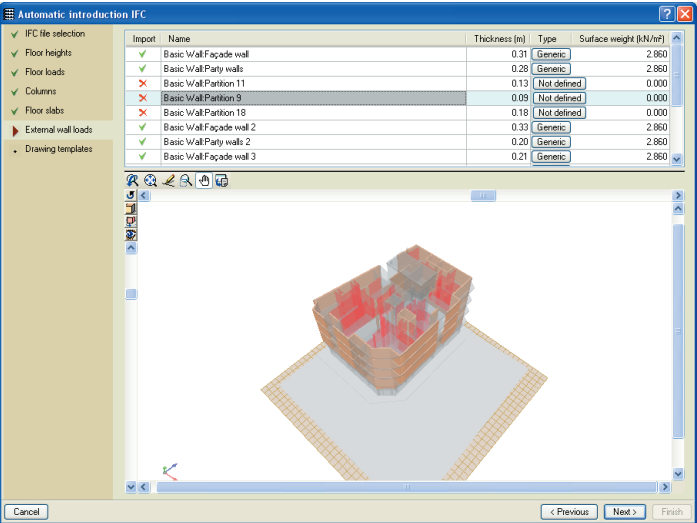


Fig. 1.26

Once the introduction has concluded, the job will be generated and the **General data** window will open to allow for the design codes and job properties to be selected.

Immediately after, the program will display the Column Definition window in CYPECAD where all the introduced data can be modified.

To complete the introduction of the job, the floor slabs, supplementary loads, foundations and any other particular data must be introduced.

1.3.2. Manual introduction

If an empty job is chosen, the user must manually introduce all the properties of the job.

In order to simplify the introduction of these properties, it is important to be methodical. We suggest the following steps be taken to carry out the project within CYPECAD, or alternatively, any other method can be followed as long as the creation of the information is structured and coherent.

- **General data.** (**Column Definition**, **Beam Definition** and **Results** tabs, **Job** menu). Upon creating a new job, a name and description must be assigned to it, the materials to be used and their respective options must be indicated, as well as the type of control to be undertaken in its construction.
- **Floor definition.** (**Column Definition** tab, **Introduction** menu). Description of the levels of the structure, group organisation, heights between floors and the live and dead loads they must support.
- **Column and shear wall introduction.** (**Column Definition** tab, **Introduction** menu). Description of the column and shear wall geometry and their respective positions.
- **Wall introduction.** (**Beam Definition** tab, **Beams/Walls** menu). Introduction of the walls supporting the structure.
- **Beam introduction.** (**Beam Definition** tab, **Beams/Walls** menu). Introduction of the beams of the floor.
- **Floor slab introduction.** (**Beam Definition** tab, **Panel** menu). Introduction of the floor slabs to be used in the structure.
- **Special load introduction.** (**Beam Definition** tab, **Loads** menu). This consists in introducing loads that have not yet been taken into account, such as those due to internal partition walls. Please recall that the general dead and live loads of the structure have already been introduced when defining the floors and so need not be introduced here. The self weight of the supporting elements (columns, walls, beams and panels) are generated by the program and so do not have to be introduced.
- **Stair introduction.** (**Beam Definition** tab, **Job** menu).
- **Integrated 3D structures introduction.** (**Beam Definition** tab, **Job** menu). Introduction of the 3D structures defined in Metal 3D.
- **Foundation introduction.** (**Beam Definition** tab, **Foundation** menu). Introduction of pad footings, strap and tie beams. For mat foundations, these must be introduced from the Panels menu at foundation level.
- **Analysis of the structure.** (**Beam Definition** tab, **Analyse** menu).
- **Error identification and correction.** (**Results** tab, **Contour Maps** and **Deformed Shape**). Having analysed the structure, the next step is to check the error messages related to the structural elements. All those requiring dimension modifications must be corrected and then the job must be re-analysed so to verify no errors arise and for the changes in self weight of the structure to be readjusted. This must be repeated until those errors implying dimension modifications to be carried out disappear. If no dimension modifications are required, the results may be taken as valid.
- **Preparation of the results.** (**Results** tab). This is the previous step to generating the drawings. Here, the floor slab reinforcement can be edited or made more uniform, column reinforcement can be modified and columns may be grouped, if the user wishes to do so. Text positions can also be corrected here without the need of having to wait to make the corrections in the drawings.
- **Drawings and reports.** (**File** menu, **Print** option or Configuration icons at the top right corner of the main

window). This is the last stage in which the drawings and/or reports are obtained containing all the desired information generated by the program based on the introduced, analysed and edited structure.

1.3.3. File management

1.3.3.1. Recent files

A list of the last six jobs that have been opened can be found by going to **File > Recent files**. By clicking on More..., a list of all the jobs opened with that version of the program will be displayed in chronological order.

1.3.3.2. ASCII file import to generate the foundations

File > Import > Import ASCII file

It is possible to import text files in columns or limited by commas to generate the foundations based on files exported from data sheets or other programs.

From the files, the different loadcases are imported, as are the forces at the column starts for each simple loadcase and the column base dimensions.

The import process is as follows:

Click on **File > Import > Import ASCII file**. A window will emerge asking for the name of the job in which the foundations are to be created.

Subsequently a new window will open from which the file to be imported must be selected.

Having concluded the import, the job will have to be analysed.

1.3.3.3. Job examples

The program provides several examples, accessed by clicking on the **Examples** icon at the top of the **File manager** window.

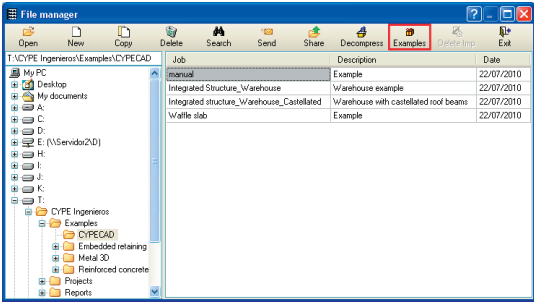


Fig. 1.28

1.3.3.4. Search

Upon clicking on the **Search** button, the **Search for job using thesaurus** dialogue box appears. This options allows the user to search for jobs in the directory selected in the previous dialogue box (File manager) which contain the word indicated by the user in their name or description.

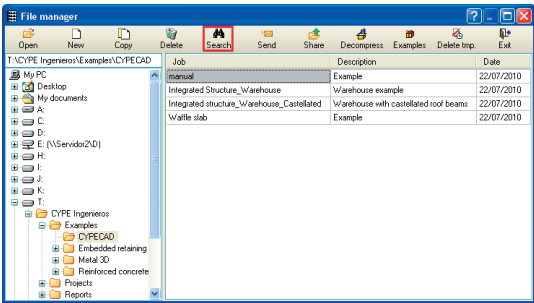


Fig. 1.29

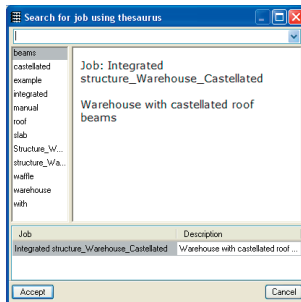


Fig. 1.30

1.3.3.5. Send, Decompress and Share jobs

These options are found in the **File manager** window. All these options deal with compressed jobs i.e. when all the files of a job are compressed into a single file with .CYP format.

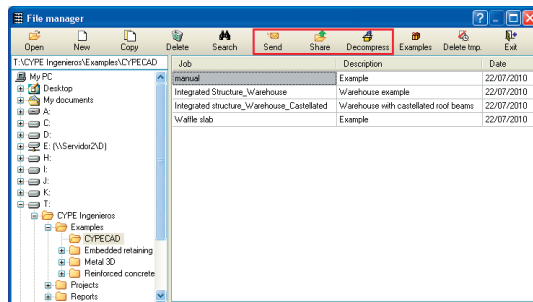


Fig. 1.31

1.3.3.5.1. Send

Using this option, the user can send the job to an email address selected from the scroll menu or save the compressed job to a directory of his/her choice (Only compress job). The user can optionally select whether or not to include safety copies that may have been created from previous versions, or to include the associated files (the results from the last analysis).

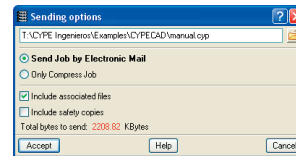


Fig. 1.32

1.3.3.5.2. Decompress

This is the opposite to the Send option; it decompresses a job with .CYP format.

1.3.3.5.3. Share

Using this option, the user can share a job compressed with .CYP format. The job will be published on a server and will be accessible to third parties via a private link. Therefore, only those who know the correct link will have access to the job.

The user can include a title and a comment on the job to be shared. Another dialogue box will then appear which indicates the link where the job is located, which may be copied by selecting it using the left mouse button so it may be provided to the person he/she wishes to see it. This dialogue box also opens the website where the job has been published.

The file **links.txt**, which is generated in the directory `usr/cype/cypecad` contains the links of all the jobs that have been published.

Any CYPE program screen can be captured to then share on internet via a private link which the user can then provide to or publish wherever he/she wishes to. Only those who know this link will have access to the captured image.

The image capture option is activated by pressing the F12 button of the keyboard whilst the active window on screen is that of a CYPE program. A dialogue box will then appear displaying the captured screen image and where the user can assign it a name and add a comment. Within this dialogue box, the user has the option to select only part of the captured image for it to be published. To do so, select the area of the image by opening a selection window over the image using the left mouse button.

Having accepted the dialogue box, the program offers the user in a different window, two types of URL addresses so the user may insert the captures in any forum. The selected address will depend on which forum the URL is to be published at. Within the same window, the link which allows access to the published screen from any internet connection is provided.

The file **links.txt**, which is generated in the directory `usr/cype/cypecad` contains the links of all the published screen images.

2. Templates

2.1. Template properties and options

2.1.1. Template fundamental properties

So to optimise the use of this module, CYPE has established a series of help proposals to define the job.

The elements to be introduced, such as the columns, external outline beams or beams defining openings in panels should be drawn with different layers.

The external outline beams and opening beams should form a closed polygon.

It is possible to use a single layer to define the fixed point of the columns. It will be represented by a closed polygon containing the point of the column to be defined as fixed.

Each group should be defined by different templates or layers when a complete template of the job is provided: columns_grd, columns_flrs, columns_roof, outline_grnd,...

2.1.2. Template options



The Template views manager window can be accessed by clicking on the  icon of the main window toolbar.



Fig. 2.1

To attach the DXF/DWG templates to the project, click on the  button at the top of the window.

The list of imported templates will appear in the dialogue box.

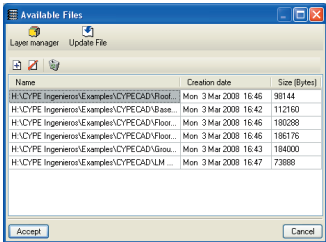


Fig. 2.2

Upon clicking on the Layer manager, the layers corresponding to the selected template will be displayed. These layers can be configured: visible or not visible, name change, colour modification, line thickness edition (if the thickness is established as By size, it is possible to specify its value); it is equally possible to leave layer texts visible and allow object snaps on the entity of a layer.

To the right of the layer list, the results of the layer configuration of the template selected from the above list can be viewed.

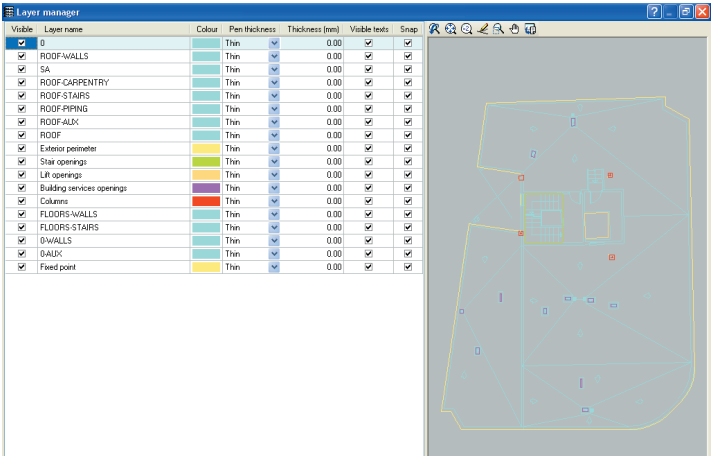


Fig. 2.3

Once the drawing templates have been imported, the program will return to the Template views manager where the templates will be displayed with their respective layers.

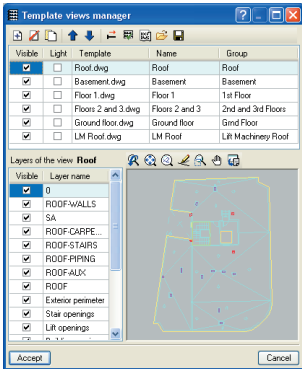








Fig. 2.4


Note: A view is a configuration of layers with the scale and coordinate origin of a DXF/DWG drawing. Generally, it will only be necessary to create those views required as templates for each group making up the project. Each view will be related to a template and will have a unique selection of layers.

Within the top list of the **Template views manager** window, the available views can be seen as well as the **Add**, **Delete**, **Copy**, **Displace**, **Transformation**, **Group views**, **Open** and **Save** options located at the top.

By clicking on Transformation , a window will open allowing displacements and scaled increases or reductions in size to be applied to the template. The scaled and displacement transformations can be carried out in two different ways: manually introducing the value within the Displacement and Scale fields, or graphically by clicking on  to change the coordinates or on  to vary the scale.

Once the origin has been displaced, it is possible to return it to its initial position, in other words, to the original position of the drawing, by clicking on the  icon. This option was brought about to be able to undo all the changes carried out on the coordinates using the **Change the origin of the coordinates of a template** option .

To change the scale of the drawing when the scale value is unknown, use the option **Adjust the scale of a template** . The program will ask to graphically define the first and second points of the segment as well as the distance of the segment with a scale of 1:1.

The option **Group views**  allows to choose for which groups the templates will be visible.

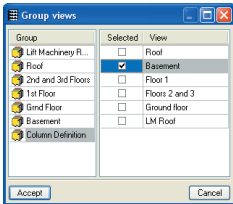



Fig. 2.5

This option is equally accessible later on by clicking on the icon . However, in this case, contrary to the option window **Group views**, the window that opens only allows to activate or deactivate the views per floor.

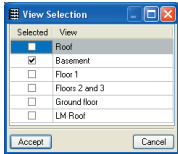



Fig. 2.6

2.2. DXF/DWG Object Snap

Object snaps can be activated within the **Object Snap selection** dialogue box, which appears upon clicking on the  icon.

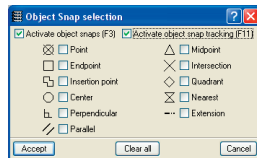


Fig. 2.7

It is equally possible to activate or deactivate the object snaps by pressing the F3 key. If no object snaps have been activated, the object snap window will open automatically upon pressing the key so to select the desired box(es).

If object snaps have already been selected, upon pressing the F3 key, the object snaps will be activated or deactivated.

2.3. Drawings and Reports

Within the drawing editor box, the option exists to draw the DWG or DXF template associated to each floor.

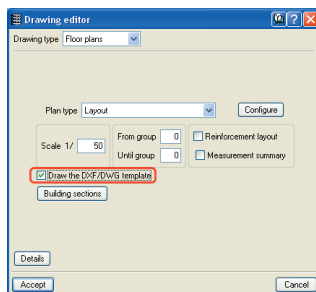


Fig. 2.8

3. Groups and Floors

3.1. General information on groups

The program allows up to 5 floors per group.

A group is defined as an assembly of one or more equal and consecutive floors and therefore, will contain the same reinforcement. Various aspects should be considered for groups containing several floors:

- Data introduction is the same for all the floors of the group.
- The results that appear on screen correspond to the envelope of all the floors belonging to the group.
- The design of all the floors of the group is the same.
- Therefore, a single drawing is obtained for the group, i.e. a single floor plan, beam drawing corresponding to a single group, etc.
- It is not necessary to regroup the floors unless a vertical support (column, shear wall, wall) or sloped beam begins or ends at one of the groups.

3.2. Floor and group introduction

Floor and group introduction is done using the option **Floors/Groups** from the Introduction menu within the **Column Definition** tab.

Unless a floor or group has been defined, the only visible option will be **New Floors**. Later on, other options will become available. These include: **Delete floors**, **Edit floors**, **Edit groups**, **Join groups** and **Divide groups**.



Fig. 3.1

3.2.1. Creation of new floors

Upon creating new floors, the program will ask whether the floors to create are to be Grouped or Single.

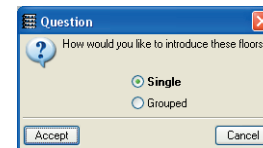


Fig. 3.2

The number of floors to insert must be indicated, as well as their heights, live load and dead load (screed, etc.). A more specific name can also be given to each floor. The use category of the group is also to be indicated. The use categories are defined separately (explained further down). If the user has already defined the use categories of the building, he/she will be able to select the use category for the group from amongst those that had been defined. On the contrary, if no use categories have been defined, only the default option is displayed and the user may modify the use category of each group later on.

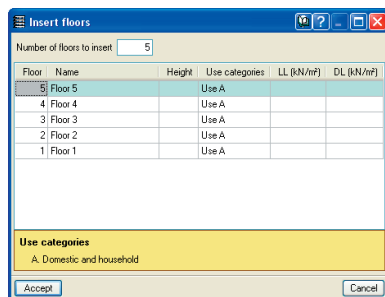


Fig. 3.3

3.2.2. Use categories

Different use categories can be defined for each floor group of the structure. Examples of use categories include: dwellings, shops, warehouses, garages, etc. The number of use categories the program allows to choose amongst depends on the selected code. For example, the use categories for the Eurocode are:

- A. Domestic and household
- B. Offices
- C. Meeting areas
- D. Shops
- E. Warehouses
- F. Vehicle weight ≤ 30 kN
- G. 30 kN $<$ vehicle weight ≤ 160 kN
- H. Roofs

For the program, a use category consists of a group of live loadcases, be they automatic or additional, which combine with the other loadcases that have been defined in the job with the same combination coefficients.

This way, the live load combinations for each use category can be analysed correctly amongst all the use categories and the remaining loadcases of the job for which different use categories exist per floor.

The use categories can be selected in the **Additional loadcases (special loads)** dialogue box within the **General data** window (**Job > General data > Additional loadcases (special loads)**).

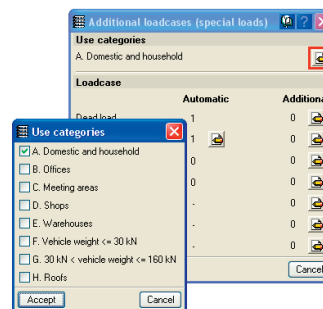


Fig. 3.4

The categories that are selected in this dialogue box are assigned to each floor group in the **Edit groups** dialogue box which opens by selecting the **Loads** menu within the **Beam Definition tab > Loads in groups** menu or in the **Column Definition tab > Floors/Groups > Edit groups**.

If a new job is created using the automatic introduction process, the user will have to define the use categories once the automatic introduction has concluded, then edit the groups and select the use category. If the new job is being defined without the automatic introduction assistant (empty job), the user can choose whether to define the use categories before or after the groups are defined, in which case if these are defined after creating the groups, the user will have to edit them and assign the use categories.

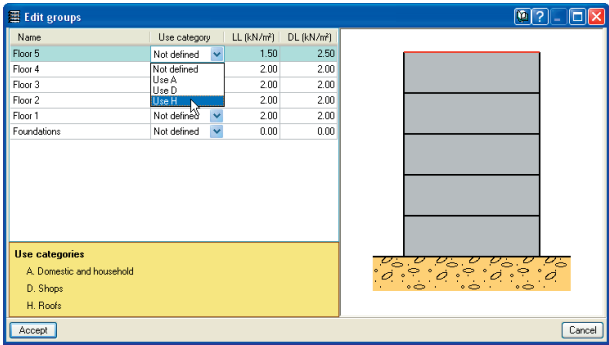


Fig. 3.5

Different use categories can also be defined for a floor group. To do so, simply do not introduce a live load for the group and once the geometry of the floor group has been defined, introduce the live loads at different positions on the group, assigned to the corresponding loadcase and use category.

The user should bear in mind that the more use categories are defined for a job, the more loadcase combinations the program will create, and so, the time taken to analyse the job will be substantially longer.

3.2.3. Elevation of foundation plane

The elevation of the foundation plane can be indicated in the **Edit floors** window. The elevations of all the floors are automatically adjusted to the value given in the foundation. Within this window, it is equally possible to modify the names of the floors.

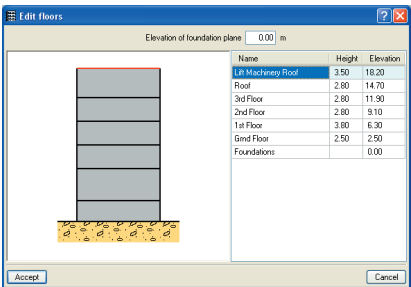


Fig. 3.6

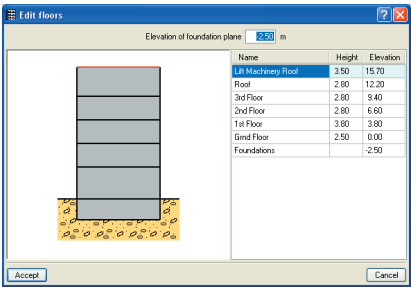


Fig. 3.7

3.2.4. Creation, edition and deletion of groups

To group the floors, click on the option Join groups. This option will allow to select the floors to be grouped.

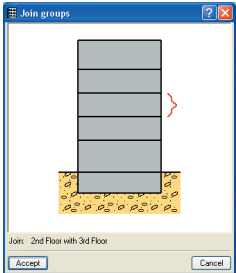


Fig. 3.8

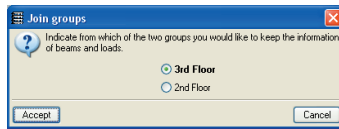


Fig. 3.9

Once this has been done, it is then possible to modify the properties (names, loads and use categories) of the created groups by selecting the option **Edit groups**. Within the window, the name of the floors are displayed with the corresponding selected floors highlighted in red in the diagram.

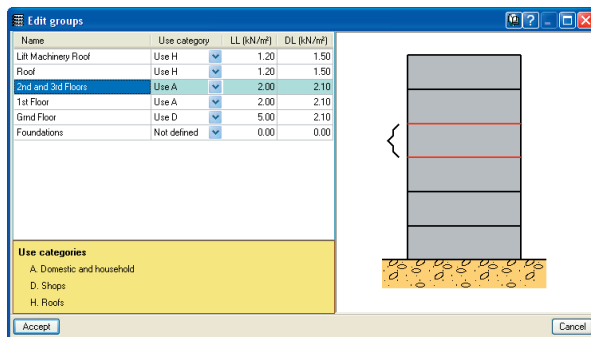


Fig. 3.10

Floors belonging to a groups can equally be separated using the option **Divide group**.

3.3. Floor and group options

These options can be found in the **Groups** menu within the **Beam Definition** tab.

3.3.1. General Options

Copy from another group.

This option allows to copy the properties of a selected group of floors to the floor that is currently being viewed. The destination group can be seen in green and the group to be copied in red (which is selected by moving the cursor over the group diagram).

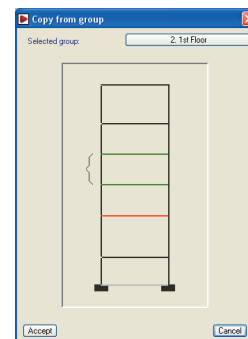


Fig. 3.11

Reload group

This option allows to recover the properties from the last saved copy. Any changes that have been carried out after the save will be lost. The program will ask to confirm the order before its execution.

Group surface information

This option provides information on the total surface area of the current group, its beams and floor slabs. The area taken up by the columns and walls is taken into account within the total surface area.

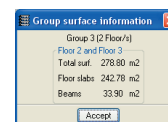


Fig. 3.12

Consult floor elevations

Allows to consult the relative and absolute elevations of each floor.

References

Allows to modify the position and representation of the references of different elements. To do so, click on the reference to be displaced then at its new location.

Show References

Allows to permanently activate or deactivate the references within the floor view. It is possible to select the texts of those elements which are to remain visible.

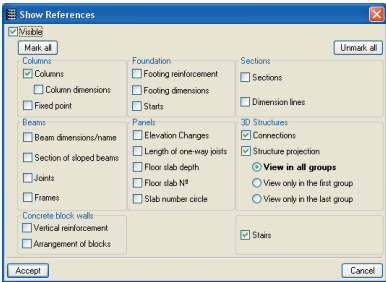


Fig. 3.13

3.3.2. Views

Building section

This option allows for a vertical cut to be made through the structure to obtain a section view at that cut, which can then be viewed on the drawings as well as on screen.

There is no limit as to how many sections can be obtained as long as a name is provided.

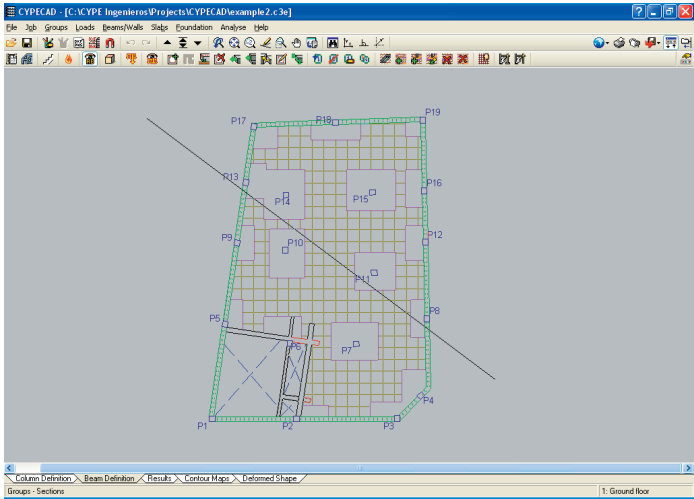


Fig. 3.14

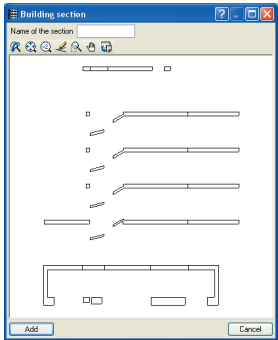


Fig. 3.15

All the created sections can then be edited and/or deleted.

Afterwards, in the floor drawings, click on the Building sections to activate those sections to be visible (the size can be configured by the user) with the chosen floor plan.

Sections

Lines may be introduced in any direction on the floor to obtain section drawings of the floor and visualise a diagram of

these sections. When a section line is introduced within each floor group, a window displaying the section of the building produced by the line will appear automatically.

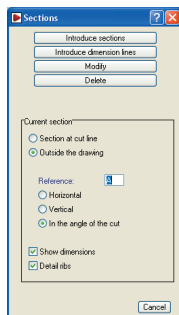


Fig. 3.16

3D View Floors

3D view of the current group of floors or the selected interval of groups/floors.

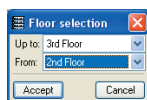


Fig. 3.17

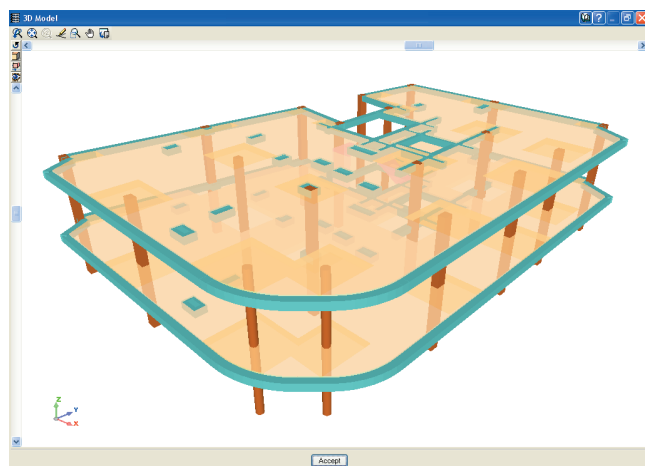




Fig. 3.18

3D View Building

Complete 3D view of the building including, if present, any integrated 3D structures.

Each construction element (columns, beams, floor slabs supported by beams, etc.) appears in a different colour with their respective textures.

The horizontal displacement bar rotates the building about the z-axis, whilst the vertical bar provides rotation about the x-axis.

The 3D view has buttons available which allow to pass from conical projection  to isometric projection  and vice-versa.

Whilst the viewed projection is that of the conical projection, it is possible to move around inside the structure with complete freedom. Within the representation, the floor slabs and walls are drawn with a degree of transparency to facilitate the visualisation of elements that would otherwise be hidden from view.

4. Columns

4.1. Definition of columns, shear walls or starts

To introduce a column, shear wall or start, click on **Columns, shear walls and starts** from the **Introduction** menu within the **Column Definition** tab.

Bear in mind that if no floors have been defined, only starts can be introduced.

Once a column, shear wall or start has been introduced, the **Columns, shear walls and starts** menu will contain more options allowing for them to be edited, displaced, adjusted, modify their fixed point, their angle, introduce fixity coefficients, effective lengths, loads, etc.

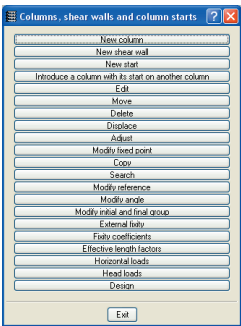


Fig. 4.1

4.1.1. Column starts

Starts are columns with zero height which are used to design the foundations (regardless of them being in the form of a mat foundation, pad foundations or other) without the need of having to introduce the whole structure and therefore, not having to define floor groups.

Having defined the starts, it will be necessary to introduce the loads transmitted to the foundations by the columns using the option **Introduction > Columns, shear walls and starts > Head loads**.

Before proceeding with the foundation design, the option **Analyse > Analyse the job** from the Column Definition tab must be selected.

If the foundation has been introduced, then select **Analyse > Analyse the job (including foundation)**.

4.1.2. Columns

4.1.2.1. New column

When introducing a new column, the initial group, final group, fixity and buckling coefficients, angle, and external fixity must be defined.

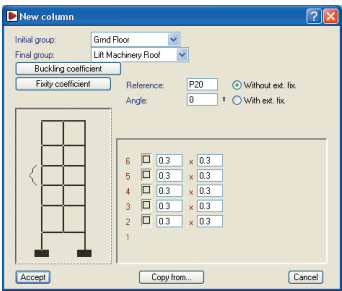


Fig. 4.2

The type of column must also be defined for each floor. To do so, click on the symbol located next to the floor numbers in the bottom section of the window. The program will then ask to choose between a concrete or steel column. Once the type has been selected, its dimensions must be defined.

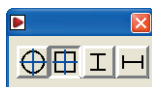


Fig. 4.3

Having defined the properties of the column and clicked on Accept, the window will close and return to the main screen where, the position of the columns can be selected by clicking at the desired position.

4.1.2.2. New column with its start on another column

It is possible to define the start of one or more columns on another column using the option **Introduce a column with its start on another column**. This option can be found within the window that appears upon selecting **Introduction > Columns, shear walls and starts**.

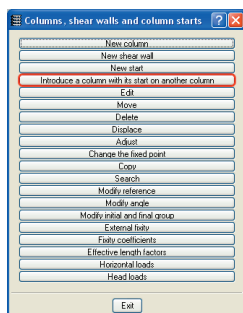


Fig. 4.4

This option provides more possibilities as to the geometrical disposition of the columns within CYPECAD jobs. For example, columns may be defined in such a way that, for different spans, they are adjusted to different fixed points or have different angles or dimensions which cause part of the top span to lie within the projection of the column below.

The program allows for an infinite number of spans to be introduced over each other, in other words, a column can be placed over another which, in turn, is placed over another, etc. Additionally, more than one column can start on the same column.

Procedure

Once the option has been selected and the cursor is then passed over a column, if it changes colour, it is possible to introduce a column on top. This occurs when the column ends at a floor that is not the last floor. To introduce a new column over another, the program draws a concentric circumference with the axis of the column illuminated (this will be the base column) on which it is possible to place the fixed point of the new column.

To position the new column, left click using the mouse. The program will ask for the coordinates of the fixed point of the new column with respect to the fixed point of the base column. Having introduced the coordinates, a new window will appear similar to that used for the introduction of the normal columns, where the new column data will have to be defined.

It is possible to introduce the top column using DXF or DWG object snaps to snap to the axis, corners, or mid-point of the column faces and the faces of the base column. In these cases, the program will not ask for the coordinates of the fixed point of the new column.

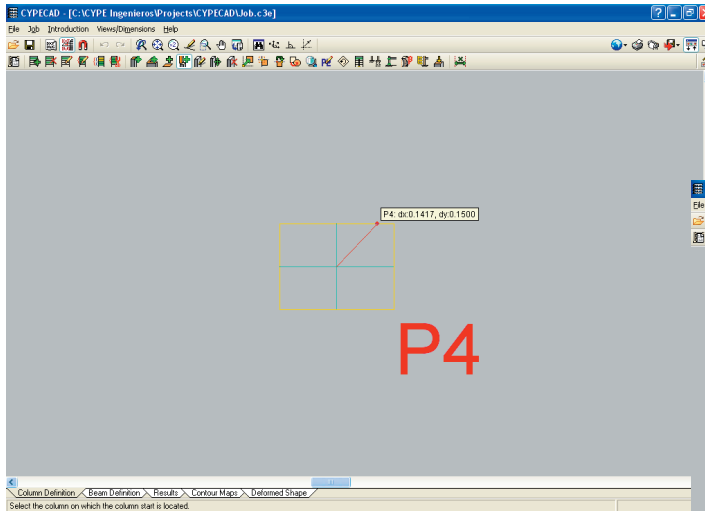


Fig. 4.5

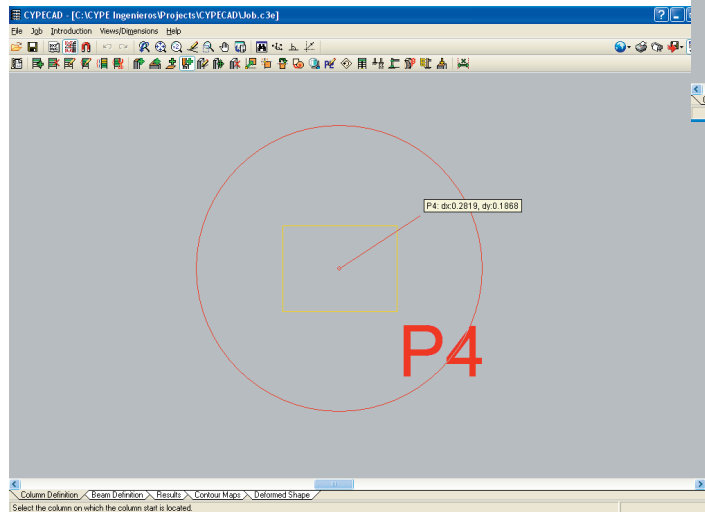


Fig. 4.6

For the reference of the new column or columns that start on a base column, the program uses the same reference as the base column followed by a letter. The first column will have the reference followed by the letter <<a>>, the next with the letter <> etc.

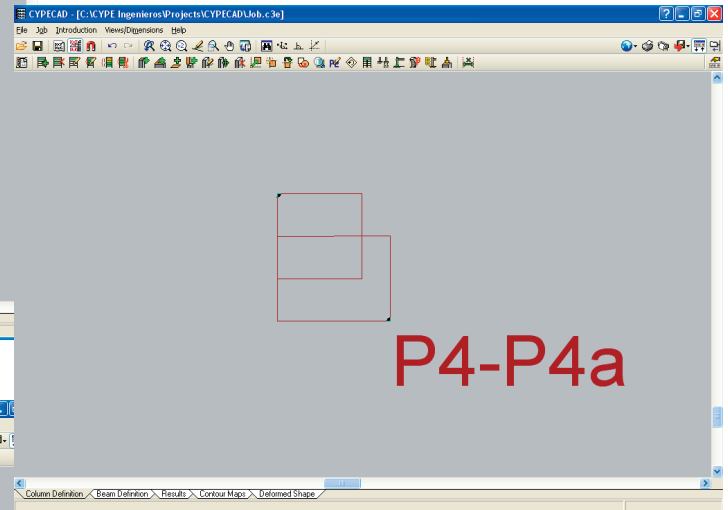


Fig. 4.7

Do not forget that if the columns are not overlapping, the column is to be introduced in the same way as any ordinary column and not with this option as it is not a case of a column starting at the top of another column.

Option specifications

The fixity coefficient of the top end of the column on which the new column begins is not affected by the value in the Fixity coefficient at last floor option (**Job > General data > button By position > button Column options**). All those elements supported by the column (top columns, beams, slabs...) will be considered as being fixed by default, except at the top of the last column span where a fixity coeffi-

cient different to one is applied (menu **Introduction > Columns, shear walls and starts > Fixity coefficients**). It is equally possible to pin a column reaching a beam or introduce a fixity coefficient other than one at the base of a new column.

The transmission of forces from the top column to the bottom column is carried out bearing in mind the existing eccentricity. The user must evaluate the need to provide a construction detail to assure this transmission of forces.

It is not possible to modify the external fixity of a new column starting on another as it is not possible to have external fixity.

4.1.2.3. Column edition

The column options can be modified in the menu **Introduction > Columns, shear walls and starts > Edit**. They can be equally modified independently accessing the options in the Columns, shear walls and starts window.

For example, the fixed point of a column can be modified directly from the **Edit Column** dialogue box or using the option **Modify fixed point** present in the **Column**, shear wall and starts window. This option allows to change the fixed point of the column keeping the column's original position.

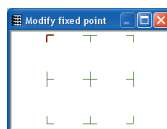


Fig. 4.8

Upon selecting this option, a window will open where the fixed point can be selected. All the columns requiring this fixed point should then have it assigned to them.

The fixed point of all the columns can be visible whilst introducing beams and floors by selecting the **Fixed point** box,

located in the option **Groups > Show references** in the **Beam Definition** tab.

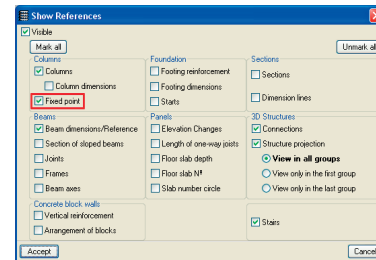


Fig. 4.9

4.1.2.4. Introduction of new columns using data from another column

Sometimes the case may arise that all or many columns of a job possess the same properties as one another. For this case, CYPECAD offers a tool that allows for a new column to be introduced based on the properties of one that has already been introduced.

Within the **New column** window (**Introduction > Columns, shear walls and starts > New column**), a **Copy from** button can be seen. Upon clicking on it, the **New column** window will close and those columns already introduced will appear on screen. Once one column is selected, the **New column** window will reopen with all the data copied from the selected column.

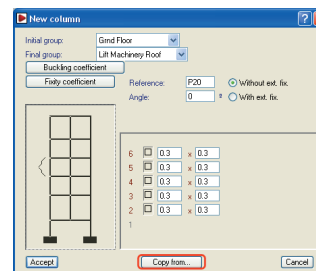


Fig. 4.10

The program keeps the data of the last column introduced when introducing a new column, and so allowing a quicker introduction in the case of similar or equal columns.

Using the **Copy** option from the menu **Introduction > Columns**, shear walls and starts from the **Column Definition** tab, it is possible to copy, as well as the properties that can be copied using the option **Copy from** described previously, the horizontal loads and head loads of the column.

4.1.3. Shear walls

In order to introduce a shear wall, its geometry must first be defined, which must be composed of rectangles. Other data to be indicated includes: a name for the wall, its initial and final group, followed by, in other windows, its thickness, external fixity and angle.

Once all the windows have been accepted, the wall can be introduced at the desired point. The types of shear walls created are saved with the job and so can be introduced at any time with no limit as to how many.

4.1.4. Column minimum geometric ratios

For each code and depending on the selected steel, there are minimum values for the geometric ratios of the columns. These values are usually those applied by the program for column design and verification. Nonetheless, if other values are required, it is possible to introduce them: **Job > General data > By position > Vertical bar options > Minimum geometric ratios** for static and dynamic loading. These user values will substitute the minimum ones from the code.

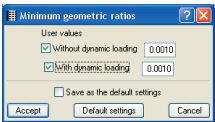


Fig. 4.11

4.2. Column results

4.2.1. Column design

Once the job has been analysed, those columns not passing all the code checks will be displayed in red. They should be verified using the option **Columns > Edit** located in the **Results** tab.

Having selected this option, click, using the left mouse button, on the column whose errors are to be consulted and a window will emerge entitled **Column design**.

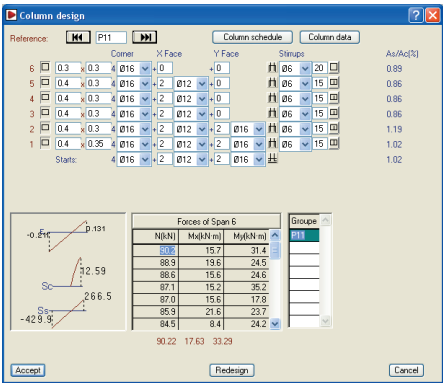


Fig. 4.12

4.2.1.1. Concrete columns

If several columns contain design errors, the following message will appear in the analysis report: "Columns (...) have a design error. They must be reviewed using the option Columns > Edit". The column section is also displayed in red on screen at the floor at which the error is located.

Column errors

Within the Column design window, those column spans containing errors will have a warning symbol displayed on the left. A status column can sometimes be viewed dis-

playing letters, whose meaning is detailed below. Upon clicking on them, details of the check as well as advice on how to solve the errors is displayed.

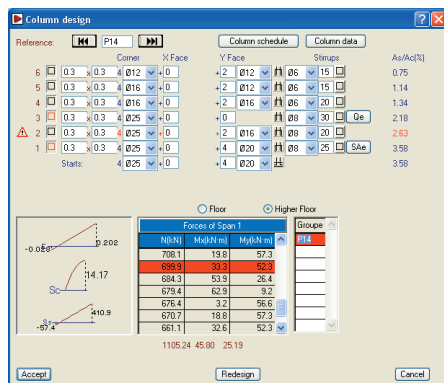


Fig. 4.13

- **SAe: Excessive ratio**

The program considers the mechanical percentage between the concrete and the steel in the columns should not be greater than 100% and that the geometric percentage should not overcome 8%. If these values are exceeded, the letters SAe are displayed.

- **Ve: Shear failure**

The program checks that the shear force does not exceed the force that would produce oblique compression failure of the section in reinforced concrete columns and for those cases where it is stated in the selected code. When shear reinforcement is necessary, the program also verifies the spacing required.

Upon clicking on the button, a window appears indicating the longitudinal area in the X and Y directions, the reinforcement area of the stirrup, the forces in both directions as well as the resisted shear force in each direction.

The verification controls that:

$$\left(\left(\frac{V_{xsd}}{V_{xrd}} \right)^2 + \left(\frac{V_{ysd}}{V_{yrd}} \right)^2 \right)^{\frac{1}{2}} \leq 1$$

Where:

V_{sd} = the design shear force to which the columns section is submitted in each direction

V_{rd} = Resisted shear force by the column section in each direction

- **Sot: Stirrups outside table**

Only one stirrup is placed. Review the column reinforcement table and the layout of the stirrups when creating the required types for the reinforcement and dimensions of the existing columns.

- **Vr: Volumetric ratio of stirrups**

In the case of earthquake loading with very high ductility, the confinement geometric percentage is limited. This percentage is the ratio between the mechanical percentage of the stirrups and that of the concrete, per metre length of the column.

- **fr: Fire resistance check**

The program indicates if the section fails the fire check according to what was specified by the user.

- **As/Ac ratio**

Within the Column design window, located in a column on the right is the ratio of the steel section to the concrete section expressed as a percentage.

4.2.1.2. Steel columns

If there are steel columns in the structure, a window offering design options will appear. If the program is to design the steel sections, the manner in which they are to be designed must also be selected.

This includes indicating to the program whether it can design the section using all the sections of the series or using sections greater than the current section. In the first case, if the program finds a smaller section capable of withstanding the forces, this section will substitute the existing section. In the second case, CYPECAD will only use sections that are greater or equal to the chosen section.

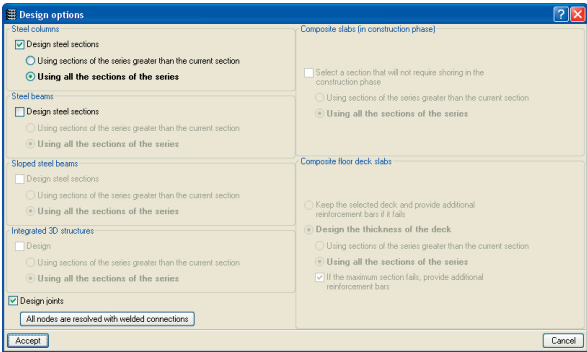


Fig. 4.14

In the **Column design** window and when dealing with steel columns, a new option: **Check series** appears. This option allows to replace the current column with another and carries out an immediate check on the result of the change.

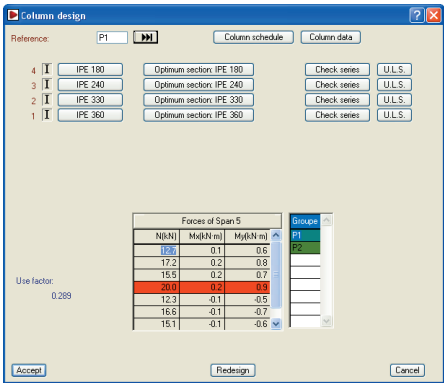


Fig. 4.15

In the windows that opens, the line weight of the sections is displayed as well as the ratio (%) of the resisted force to the design yield strength (elastic limit) and if there are any design errors.

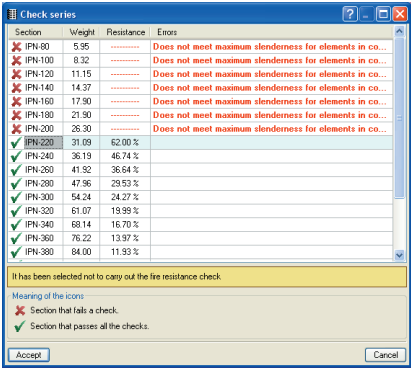


Fig. 4.16

The **Optimum section** button restores the proposed design section if it has been modified by the user.

4.2.1.3. Column data

By clicking on this button (located in the **Column design** window), the following information is displayed: angle, fixity, support elevation change and depth of support.

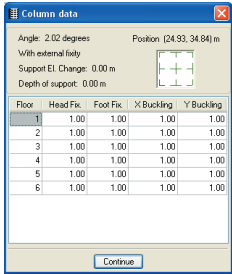


Fig. 4.17

4.2.1.4. Column layout

This button provides a layout of the columns of the selected or selected floors. If there is a column that has not been designed, (manual reinforcement), it will be displayed in red.

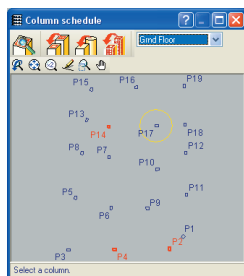


Fig. 4.18

The column selected from the **Column design** window is marked with a yellow circle. By clicking on another column, the yellow circle will appear on that other column and if the **Column layout** window is closed at this point, the column displayed in the **Column design** window will be the newly selected column.

4.2.2. Block column reinforcement

The option **Block reinforcement** allows for the structure to be re-analysed without modifying the column reinforcement. This tool is located in the Column menu within the **Results** tab.

4.2.2.1. Things to consider when using this option

The **Block reinforcement** option has been brought about so not to loose the manual modifications that have been carried out on the column reinforcement upon re-analysing the structure.

When re-analysing the job, the program will carry out a check on all the blocked columns ensuring that all the conditions of the selected code are met. Those failing a check will be displayed in red.

If significant errors appear in a blocked column after a new analysis, they imply that the changes undertaken on the job after the reinforcement has been blocked, substantially affect the columns. It is therefore recommended the column reinforcement be redesigned.

There are two options to redesign the columns: unblock them and then redesign (**Beam Definition** tab, **Analyse > Redesign columns**) or keep the columns blocked and use the **Redesign** option in the **Column design** window of each column (**Results** tab, **Columns > Edit**).

When modifications have been carried out implying an increase or decrease in the number of floors supported by a blocked column, the program will automatically unblock it.

Unlike what occurs with blocked frames, the program undertakes a complete code check on all the blocked columns. Therefore changes made to the structure after blocking the column reinforcement are not as conflictive.

The blocking of columns is a very useful tool to have for different project situations, for example, to study how the column reinforcement varies depending on the applied fixity coefficient; subgrade modulus; jobs with mat foundations; etc.

This option is only applicable to reinforced concrete columns, the equivalent option for steel columns can be produced if, upon selecting to analyse the job, the **Design steel sections** box is left unmarked.

4.2.2.2. Procedure

Once the **Block reinforcement** option from the **Columns** menu is selected, the **Block reinforcement** window will appear.

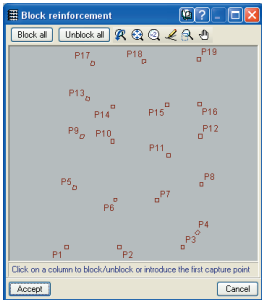


Fig. 4.19

Within this window, it is possible to select the desired columns one by one using the left mouse button. It is equally possible to use a capture window to select several columns at once. At the top left hand corner of the window, there are two buttons allowing for all the column reinforcement to be blocked or unblocked at once. Instructions on how to proceed blocking or unblocking the columns are displayed at the bottom of the window.

Selected columns will appear in green. If there are steel columns, they will appear in grey and cannot be selected.

The dialogue box must be accepted for the selection to be effective.

4.2.3. Force diagrams

Within the **Envelopes** menu in the **Results** tab, the option **Forces in columns and shear walls** can be found. These options allow for force envelopes to be consulted (Moments, Shear and Torsion), by loadcase or loadcase combinations.

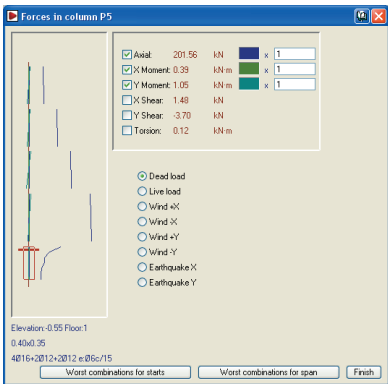


Fig. 4.20

4.3. Drawings and Reports

4.3.1. Drawing options

4.3.1.1. View of the fixed point in the drawings

It is possible to configure drawings to show or hide the fixed points of columns. Upon pressing the **Configure** button in the **Drawing editor** window and then selecting the **Columns** tab, the **Fixed point** box can be selected or unselected.

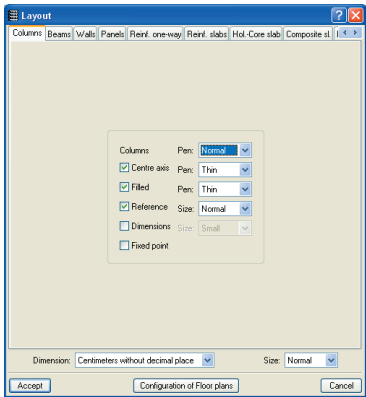



Fig. 4.21

4.3.1.2. Column stirrup options

To take the indicated considerations for the stirrups (**Job > General data > By position**  **> In columns, walls and corbels > Options for stirrup reinforcement**) into account, there is an option, allowing to draw, within the column schedule drawings, a detail displaying these conditions, and therefore avoiding errors onsite.

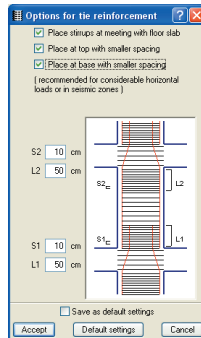




Fig. 4.22

This option can be activated in the Drawing editor dialogue box (**File > Print > Job drawings > Add**  **or Edit**  **Column schedule**).

4.3.2. Report options

Within the report **Forces and reinforcement of columns and shear walls and walls** is the option **Shear resistance check in reinforced concrete columns** in which the shear forces are indicated for the columns of the job.

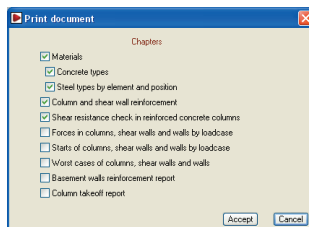


Fig. 4.23

5. Walls

5.1. Wall introduction

Wall introduction is done by selecting **Beams/walls > Define wall** from the **Beam Definition tab**. A floating dialogue box will appear from which the type of wall (a reinforced concrete wall, a NORMABLOC wall, a reinforced concrete block wall or a masonry wall) is to be selected.



Fig. 5.1

5.1.1. Floating menu

Upon selecting the option **Beams/walls > Define wall** from the **Beam Definition tab**, a floating menu will appear in which the introduction options belonging to the different types of walls are located.

For all the types of walls (reinforced concrete, block or masonry walls) a window appears allowing the geometry of the wall to be defined as well as indicating whether or not there are lateral pressures present and what type of foundation it will rest on.

Generic blocks or NORMABLOC blocks can be used for block wall introduction. The dialogue box that is visible on screen contains options that are common to both types of walls and one option exclusive to NORMABLOC walls.

5.1.2. Wall types

5.1.2.1. Reinforced concrete walls

5.1.2.2. Generic reinforced concrete block walls

The block wall dialogue box contains a field in which it is possible to select what type of blocks the wall is to be made out of.

It is possible to choose a block from the program's library or introduce different blocks in the library. These operations can be undertaken using the selection buttons and tools present using the **Block** field.

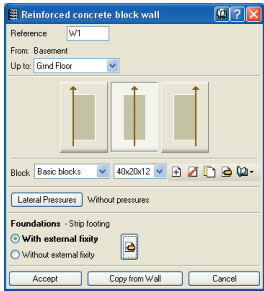


Fig. 5.3

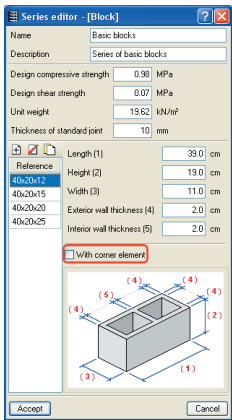


Fig. 5.4

Within the generic block walls, **Series editor** window, it is possible to activate the **With corner element** box. The corner elements are to be used when the width of the block is not equal to half its length. This way, the special corner elements allow for a faster construction of the wall.

5.1.2.3. NORMABLOC walls

Prefabricated concrete walls are a traditional solution due to their properties. They can act as load bearing walls (resisting horizontal and vertical loads) as well as providing partitions or enclosing spaces where required.

It is possible to design and check existing concrete blocks (and those of manufacturer companies as in the case of the Spanish group NORMABLOC: National Association of Masonry and Concrete Block Manufacturers), optionally provide vertical and/or horizontal reinforcement, all this respecting the criteria imposed by Eurocode 6 (Masonry design and construction. Part 1-1. General – Rules for reinforced and unreinforced masonry). Provision of analysis reports, takeoff and floor drawings as well as a diagram on plan of the blocks and their reinforcement.

In the case of NORMABLOC blocks, it is only necessary to select, using the option Block from the dialogue box, the manufacturer and the type of block to be used.

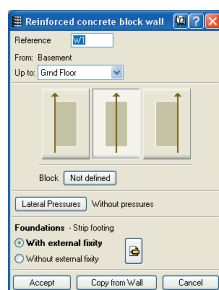


Fig. 5.5




Fig. 5.6

NORMABLOC walls do not have the option **With corner element**, although the program is aware if the manufacturer and selected block model have these special elements available and will use them, if necessary.

5.1.2.4. Masonry walls

This option allows generic walls to be introduced.

Within the **General data** window of the **Job** menu and upon selecting the wall icon , it is possible to define the modulus of elasticity, shear modulus, unit weight, design compressive strength, design tensile strength and whether or not to consider shear stiffness, required in the case of masonry walls resisting horizontal shear (produced by horizontal loads) and tension (in the case of walls supported by floor slabs and connected to higher floors).

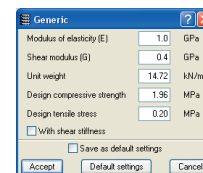


Fig. 5.7

5.1.3. Wall openings

5.1.3.1. General

By selecting the option **Beams/Walls > Wall openings** in the **Beam Definition** tab, a floating menu will appear allowing windows and doors to be introduced in reinforced concrete, generic masonry and reinforced concrete block walls.



Fig. 5.8

To introduce the openings at their correct positions, the floor viewed on screen must be the lower of the two floors between which the window or door is located. For example, if a door is to be introduced in a wall between floors 1 and 2, the introduction floor must be floor 1.

In the case of openings in concrete block walls, when using the Wall openings option, it is possible to adjust the opening within the block distribution of the wall. This way, the dimensions and position of the opening will adjust themselves to the length of the block so to avoid having to provide special pieces.

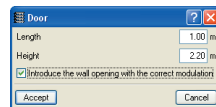


Fig. 5.9

For a better understanding of this option, please read the generic and specific help dialogue boxes that appear in the Wall openings introduction process.

When openings are introduced in basement walls and generic masonry walls, the program does not warn of the excessive constraints in the lintel area. On the other hand, in the case of concrete block walls, this check is carried out and the lintel reinforcement is designed.

5.1.3.2. Opening reinforcement

Upon clicking on the option **Wall openings** from the **Beams/Walls** menu within the **Results** tab, a floating menu with the following options will open:



Fig. 5.10

- **Edit.** Allows to edit the reinforcement of the door or window. If the opening is approached by the cursor, it will be highlighted in yellow and can be edited by clicking on it.

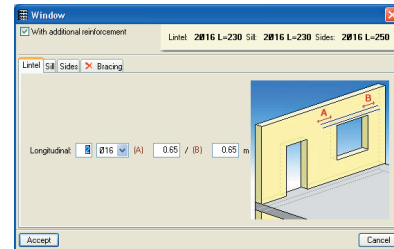


Fig. 5.11

- **Assign door reinforcement.** By clicking on this option, the **Assign reinforcement to wall openings** window appears in which the reinforcement can be defined. Within this window, the **Select** button can be found, which allows to choose the reinforcement of a specific door opening, and the **Assign** button, allowing for all the doors which are to possess this reinforcement configuration to be selected.

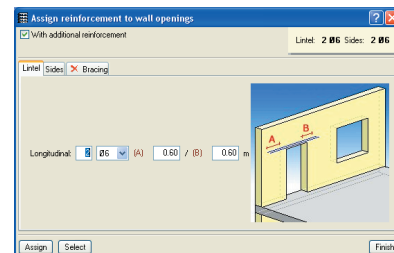


Fig. 5.12

- **Assign window reinforcement.** Same as the previous option except only the window reinforcement can be edited.

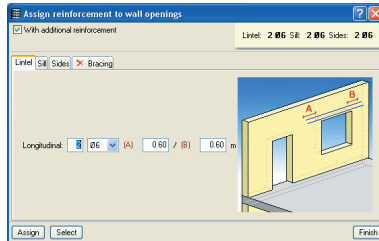


Fig. 5.13

- **Assign lintel reinforcement in concrete block walls.** Works in the same way as the previous options, however in this case, it is only possible to assign reinforcement to concrete block wall lintels.

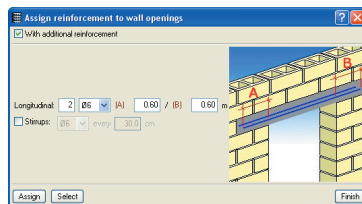


Fig. 5.14

- **Verification of the reinforcement at wall opening level.** Activate this option and select the wall opening whose reinforcement is to be checked. The report of the assessed checks will appear.

Wall opening reinforcement check		
Reference: W7: Window situated at position (-3.73, 0.50, +1.00) (Grnd Floor - 1st Floor)		
Code checks	Values	Status
Minimum reinforcement area:	Calculated: 4.02 cm ²	
- Lintel:	Minimum: 0.09 cm ²	Verified
- Sides:	Minimum: 0 cm ²	Verified
Lintel (Origin):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
Lintel (End):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
Guardrail (Origin):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
Guardrail (End):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
Sides (Origin):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
Sides (End):	Minimum: 70 cm	Verified
Eurocode 2, Article 5.2.2 (pg.150).	Calculated: 70 cm	
All the checks have been verified		

Fig. 5.15

5.1.3.3. Opening reinforcement design

During the reinforcement design process of concrete walls, the reinforcement of the openings (windows and doors) within the walls is designed. This reinforcement is designed to support any lateral pressures, forces at door and window lintels and at window sills.

For those cases where the wall is subject to seismic loading and the acceleration is greater than 0.16 g, inclined reinforcement is laid in the lintel and sill.

If the ratio

$$\frac{\text{Free opening span}}{\text{Lintel width}}$$

is less than 1.5 and the tension in the beam is greater than the tensile resistance of the concrete in compression, the beams containing the frame will constitute the inclined reinforcement.

5.1.4. Extend walls

This option, accessed by clicking on **Beams/Walls > Extend wall** in the **Beam Definition** tab, works in the same way as the option used to extend beams. The wall is extended in all the floor groups in which it has been defined.

5.1.5. Assign walls

This option is located in the **Beams/Walls** menu in the **Beam Definition** tab. This option allows to select a wall and assign its properties to another wall or to a series of walls selected using a capture window.

When executing this option, a type of wall must be chosen from the floating menu. A dialogue box will then appear describing the properties of the type of wall that has been selected.

In this window, the properties to be assigned can be modified or copied from a wall that has already been introduced using the **Select** button. Using this button, it is possible to select walls of a different type to that which was chosen from the floating menu. With the **Assign** button, the properties displayed in the previous window can be assigned to an already existing wall. The assigning of a wall or properties can be done individually or to several walls by means of a capture window.

5.1.6. Edit walls

Beams incorporated in walls and crown beams are visible on plan superposed with the wall. The option Edit from the Beams/walls menu is used to edit the introduced data of a beam or wall.

5.1.7. Design options


Options for wall reinforcement are located in **Job > General data** menu and are accessed by pressing the wall icon .



Fig. 5.16

These options allow to define:

- The type of steel for the horizontal reinforcement in block walls.
- Resistance properties of the mortar and blocks to be used in NORMABLOC walls.
- Reinforced concrete wall reinforcement design options:
 - a. Whether or not to use reinforcement

- a. Always provide reinforcement or only when necessary
- b. Only provide horizontal reinforcement or horizontal and vertical reinforcement.
- c. Type of horizontal reinforcement.

5.1.8. Column disconnection in masonry and concrete block walls

Columns in contact with masonry or block walls are separated from the wall at floor level. This allows the program to obtain a more realistic force transmission for those columns integrated within the walls.

For each column that is contact with a strip footing, it is possible to design an equivalent pad footing.

The forces used to design the pad footing are those reaching the foundation via the columns and the parts of the wall supported by the footing.

5.2. Drawings and reports

5.2.1. Reports of block walls

The block wall takeoffs detail the number of complete elements and special elements such as half blocks or angle elements.

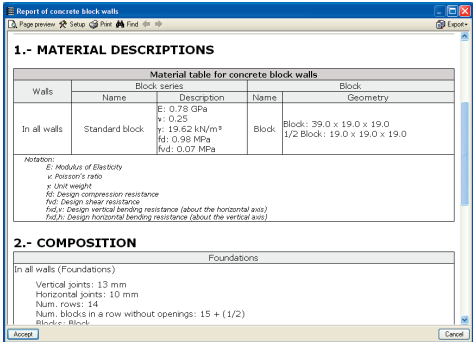


Fig. 5.17

6. Beams

6.1. Beam introduction manager

The introduction of the outline of a floor can be accomplished more quickly using a DXF or DWG template in combination with the various beam introduction tools.

The information indicated in the following pages is available in the generic help windows in the program and the specific help windows that appear during the introduction.

6.1.1. Beam introduction menu

Upon selecting **Define beam** from the **Beams/walls** menu in the **Beam Definition** tab and accepting the dialogue requesting the type of beam section, a floating menu will appear containing the following options:



Fig. 6.1

Current beam selection

The first icon of the dialogue box opens the beam selection window to change the type of beam to introduce.

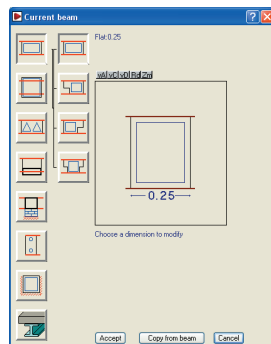


Fig. 6.2

The other icons represent the tools for beam introduction. All are located within this small window and many have direct access icons so to avoid having to displace the mouse in order to use these tools during the beam introduction.

Continuous and discontinuous beams

This option allows for simple or continuous beams to be introduced.



Fig. 6.3

- **Continuous mode**

The final point of one beam is the initial point of the next. To finish off the introduction of a series of beams, click on the right mouse button and select option **Finish introduction** from the emerging menu. While this option is not selected, the introduction of the beam in continuous mode will not be effective.

There are more options in the mentioned menu. Some of these are accessible with a quick key:

- **Finish introduction:** Ctrl + T
- **Delete last point:** Ctrl + B

- **Simple mode**

When the final point the beam is marked, another does not begin. If another beam is to be introduced, its start point will have to be marked, which does not have to coincide with the end point of the previously introduced beam.

Adjustment of the beam

This option allows for the beam to be adjusted to along its central axis or to one of its sides, in other words, change the adjustment line of the beam.



Fig. 6.4

To understand this definition, two concepts used in the program must be explained:

- Adjustment line of the beam. This can be the axis or either edge of the beam. It is the reference line used by the program from which the distance is measured to the introduction line. This distance is indicated in the **Move** option.
- Introduction line of the beam. This is the line that is marked with the mouse pointer during the introduction of the beam. The **Adjust** and **Move** options vary the position of the beam relative to the introduction line.

Changing the adjustment line can be done equally with the shortcut. Similarly, if the adjustment is to be changed during the introduction of the beam, all that has to be done is displace the mouse to the beam introduction menu. By pressing Ctrl + E, the adjustment type will change cyclically.

Beam displacements during the introduction

This option establishes the distance between the introduction line and adjustment line defined in the previous paragraph.

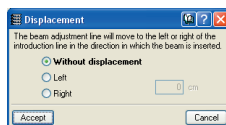


Fig. 6.5

The use of the option is helpful in the following case: the edge of the beam limiting the floor slab is, generally, shifted a few centimetres due to coatings or other elements of the façade. Usually, the lines drawn on the DXF or DWG templates are those representing the façade outline and not the structural floor slab limits. In all cases, the introduction line will not coincide with the adjustment line and will lie outside the width of the beam.

Straight and curved beams

The shape on plan (straight or curved) of the beam to be introduced can be selected here. Upon clicking on this button, the four available options appear (unless the simple beam option is activated, in which case, only the first two options will be visible).

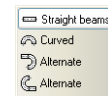


Fig. 6.6

- Straight beams. The initial and final points of the beam have to be marked using the left mouse button.
- Curved beams. Using the left mouse button, three points have to be marked defining the initial, pass-through and final points of the curve. By pressing Ctrl + A, the beam introduction changes from curved to straight and vice versa. What the program is really drawing is a series of straight beams forming a polygon adjusted to the outline of the circumference defined by the three marked points. With regards to the introduction and adjustment lines, these work in the same way as in the case of straight beams, except that in this case, they can be considered as curved lines.
- Alternate introduction of straight and curved beams. When the continuous beam introduction is activated, a beam following the same shape can be introduced or

straight and curved spans can be alternated. Two options appear in the menu: one beginning with a straight span and another with a curved span. This option can also be accessed using Ctrl + A.

6.1.2. Beam types

When introducing a new beam, the Current beam window opens where the type and properties (dimensions, section, shear studs, etc.) of the beam must be chosen. The following beam types are those that are available in CYPECAD:

Flat beams

Flat beams are those possessing a thickness equal to the elevation difference between the highest and lowest points of the floor slab they form the outline of.

Dropped beams

Dropped beams can be rectangular, T-shaped and with a contributing compression head. In the latter case, the flanges contribute to the stiffness of the beam, but, for reinforcement calculations, only the web is used. This is a common way to model flat slabs framed by dropped beams.

Latticed and prestressed beams

The properties of latticed and prestressed beams are defined by clicking on the icon that appears in the top right corner of the selected window.

External fixity beams

External fixity type beams can simulate, for example, the top of a wall. If the path of the walls coincide with the columns, they must be disconnected using the option **Beam/Walls > Pin/Disconnect** so that the vertical movement of the columns is not restrained by the walls. Note that walls may only be disconnected from columns in joist

slab systems since flat slabs or waffle slabs cause the load descending from the column to spread out onto the wall and so not all the load is transferred down the column. If this problem is encountered, it is recommended that the wall support be eliminated and the wall be introduced using the option **Beams/Walls > Define wall**.

The crown beam at the top of the wall cannot be specified with the program. Consequently, the user must provide a construction detail with the dimensions and minimum reinforcement for this element. Consult the construction details library supplied with the program.

Non-structural or limit beams

This option is used to define the outline perimeter for a solid slab panel when the influence of the edge beam with respect to the force resistance is negligible or is not to be considered. Its function is simply to limit the border of the panel. On site, if a beam is to be used to carry out this function, a generic edge beam detail of the beam to be used should be provided.

Foundation beams

Foundation beams can be rectangular, L or T, or flat beams (the latter are used to close off panels in mat foundations).

Steel and composite beams

- **Steel beams.** Steel beams can be introduced as composite beams, welded in a box, separated with batten plates or joined, etc. The same section library is available as for steel columns.

Castellated steel non-composite beams can be designed and verified. Within the **Layout** drop down menu in the **Description** dialogue box, the Castellated (web openings) option can be selected. Upon doing so, a dialogue box will open in which the properties of the beam can be selected as well as the shape of the open-

ning (hexagonal, octagonal or circular), the dimension ratios and the number of filled openings at the ends of the beam.

- **Composite beams.** It is possible to introduce (type 1) steel sections below the floor slab with a contributing concrete head by means of shear studs, whose geometry can be defined.

A partial fixity coefficient of 0.05 is applied to the connection of the composite beam with its support (in the same way as the column heads of the last floor) with the aim to reduce the negative moment and increase the positive moment.

Composite beams are designed so that in the negative moment zone, the steel section resists all the forces and in the positive moment zone, the composite section withstands the forces.

With regards to the moment calculations; it is not necessary to indicate the width of the contributing concrete section. The program takes the minimum between the calculated effective width and the width of the flange plus 10 cm for each side, if it does not consist of an edge beam, otherwise, the program calculates the width of the flange plus 10 cm.

To verify the positive moment sections, the effective width is different to that taken for the bending analysis. Therefore, in the beam editor, the width that appears is that of the effective width in the negative moment zone in case reinforcement is added at the supports, which is not designed by the program in the present version.

6.1.3. DXF or DWG object snap during beam introduction

It is possible to adjust the beam with any of the DXF or DWG object snaps available (nearest, intersection, end, mid-point, etc.) The adjustment is carried out during the introduction of the points making up the simple or conti-

nuous beams and this is why it is not necessary to use a second option (Adjust) to place the beam at its final position.

6.1.4. Polybeams

This option can be found in the **Beams/Walls** menu from the **Beam Definition** tab.

For CYPECAD, a Polybeam is an assembly of consecutive beams grouped in such a manner that if one undergoes a geometrical change, all the others in the group will also change. In a similar way, if one beam of the assembly is deleted, all the others will also be deleted. Nonetheless, the reinforcement of each beam can be different.

This option has great similarity to that of the curved beam introduction option. A curved beam is composed of several straight beams that change direction. If the properties of a curved beam are to be changed, each of the straight beams must be modified individually. To simplify the problem, it is best to group the straight beams and make a **Polybeam**.

Even though it is not always useful, the **Polybeam** option is in most cases. This is because the **Polybeam** option allows for other options to be used to vary the groups regardless of it being straight or curved.

The beams forming a curved beam are by default grouped as a **Polybeam**. Using the Polybeam option from the **Beams/Walls** menu, the beam can be decomposed or modified. The beams forming a curved beam are by default grouped according to a Predefined reinforcement span. This option, present in the Beams/Walls menu, also allows to decompose the group.

6.1.5. Predefined reinforcement spans

A Predefined reinforcement span is composed of an assembly of consecutive beams in which the bottom reinforcement does not overlap at an intermediate point of the group. This will be applied for every group of beams whilst the length of the steel bars do not exceed the maximum bar length established in the program options.

The reason for the existence of Predefined reinforcement spans is the same as that given previously to justify the existence of Polybeams. This allows to simplify beam reinforcement construction.

Even though the Predefined reinforcement spans option is useful in most cases, it is not always. This is because the Predefined reinforcement spans options can be decomposed into various options allowing to vary the groups regardless of the beam being straight or curved.

The composition or decomposition of the Polybeams and Predefined reinforcement spans, which imply a change in the number of beams making up the alignments (frames), is useful to view the designed reinforcement. In order to obtain the new reinforcement, it is only necessary to redesign the frames using the options in the Analyse menu.

6.1.6. Sloped beams

For steel or concrete sloped beams. It is possible to generate a beam for each floor between the selected initial and final floor groups.

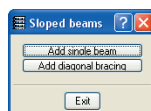


Fig. 6.7

- Simple. The program generates a single sloped beam between the selected initial and final floor groups.
- Multiple. A sloped beam is generated for each floor between the selected initial and final floor groups.

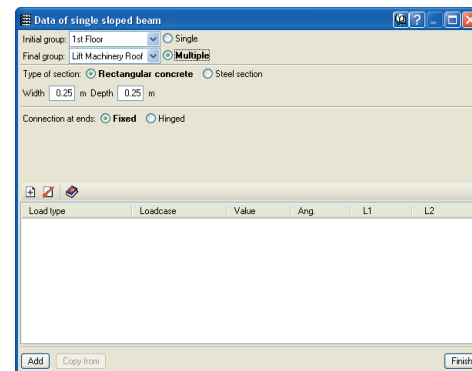


Fig. 6.8

6.1.6.1. Diagonal braces

Using the option **Beams/Walls > Sloped beams > Add diagonal braces**, diagonal braces can be introduced and then carry out other operations with them.

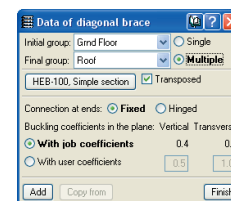


Fig. 6.9

The window contains the following options:

- **Initial and final group.** Floor groups at which the brace begins and ends. These groups should have a single floor. Nonetheless, the diagonal brace can go through intermediate groups containing more floors.

- **Simple.** The program generates a single diagonal brace between the selected initial and final groups.
- **Multiple.** A diagonal brace is generated at each floor between the selected initial and final groups.
- **Type.** If a simple beam is selected, the user must indicate whether it is a concrete or steel beam. If a concrete beam is chosen, the width and depth of the beam must be specified, otherwise, the steel section to be used must be selected from the section library. In the case of a steel beam, a further option is provided whereby it may be placed transposed or upright.
- **Connection at ends.** It is possible to indicate whether the connection at the ends of the brace is a fixed or hinged connection.
- **Buckling coefficients.** These are required for sloped steel beams or for steel diagonal braces. The coefficients introduced are multiplied by the complete length of the beam between nodes. Therefore, in the case of two sloped beams intersecting at a point, the program will not take this intersection into account.

The vertical and transverse buckling coefficients are established with respect to a vertical plane containing the bar and the Z axis of the structure and its transverse plane, not in the local axes of the bar.

- **Copy from.** If the user wishes to introduce diagonal braces based on others that have already been introduced previously on screen, click on this option. Having done so, click on the diagonal brace to be copied and the window containing the properties of the selected brace will open.
- **Add.** Allows to introduce a diagonal brace. The two end points of the diagonal have to be indicated for its introduction. For example, click on a beam from which one of the diagonals starts and then on the opposite beam, where it ends. Upon carrying out this operation, an automatic floor jump is not produced, only the floor

at which the diagonal starts at is left on screen. The braces drawn on screen may look horizontal, but this is not the case. Two arrows are drawn within the bar indicating each diagonal.

As additional information regarding diagonal braces, it is convenient to know that a node is generated at the intersection of the diagonals and that these are designed for tension as well as for compression. For this reason, they generally present a slenderness problem, given that the program verifies that the slenderness of the bar does not overcome the maximum slenderness (defined in the code) for those elements in compression.

Once a diagonal brace has been introduced, new options will appear in the **Sloped beams** dialogue box:

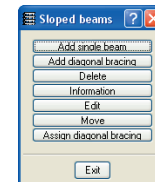


Fig. 6.10

- **Delete.** Deletes the selected diagonal brace. In the case of the generation being multiple, they are deleted in all the groups.
- **Information.** It is possible to know the type of diagonal brace, the initial and final groups, real length, type of section, connections and buckling. The result is also drawn on the beam drawings of the group corresponding to the initial group.
- **Edit.** Allows to modify the data introduced for the diagonal brace. When editing the diagonal brace, regardless of whether or not it was a multiple introduction, it is only edited in the current group.
- **Move.** Allows to displace the ends of a diagonal brace.

When displacing the diagonal brace, regardless of whether or not it was a multiple introduction, it is only displaced in the current group.

- **Assign diagonal braces.** This option allows to copy the properties of a diagonal brace to one or more diagonal braces that have previously been introduced. The window that appears is similar to the **Add diagonal braces** window, except that in this case, as it has been explained previously, its function is to modify the properties of beams that have already been created. Upon assigning a diagonal brace, regardless of whether or not it was a multiple introduction, it is only assigned in the current group.

6.1.7. Common beam

6.1.7.1. Create common beam

This option gives a beam that have been previously introduced in a group of floors, the property to exist in another group. This is the case for example, of a perimeter beam supporting a horizontal slab which meets an inclined slab. The beam supporting the slabs is a **Common beam**. The beam is only introduced in one of the two groups. Select this option and click on a beam; the program will then ask for the common group to be selected. On doing so, the beam will automatically be created in the group. The axis of this type of beam is drawn with a discontinuous dash-dot line. To undo a common beam, it is first recommended it be deleted from the associated floor group, if not from the group in which it was first introduced.

The reinforcement and envelopes must be consulted in the group in which the beam was introduced.

It is possible to create a common beam for half the length of the beam, by selecting either end of the beam. Each part can be common to different groups.

6.1.7.2. Undo common beam

Option **Beams/Walls > Common beam > Undo common beam**. Splits a common beam into the two original beams. This must be carried out on the associated beam.

6.1.8. External edge beam introduction

The option **Beams/Walls > Define rectangular external edge** allows to introduce a beam forming a rectangular outline outside the current perimeter of the floor. The command contains a further two options. Upon choosing either of them, the Current bema window will open to allow the user to select they type of beam to be introduced.

- **Introduce rectangular external edge with 0 angle.** The two opposite corners of the rectangular opening must be selected (the sides of the rectangle must be parallel to the general axes).
- **Introduce rectangular external edge with any angle.** Two corners of one side have to be indicated followed by a point on the opposite side.

6.1.9. Assign beam and wall options

The **Assign beams** option within the **Beams/Walls** menu in the **Beam Definition** tab allows for the properties of the last beam introduced (or that selected as the current beam) to be copied to previously introduced beams. The beam onto which the properties are to be copied can be selected in several ways by means of a capture window.

The same option is available for walls using the option **Assign walls**.

6.1.10. Extend beam

This option allows for the beam to be extended a fixed length or manually at one end. By selecting the Fixed length option box, introducing the length and then accepting, upon approaching the cursor to the end of the beam in question, the extension of the beam will be displayed. By clicking the left mouse button, the extension will be confirmed. Alternatively, the beam end in question can be extended manually by not selecting the Fixed length option box, accepting, then clicking on the end and dragging it using the mouse to the desired position, finally confirming with a click of the left mouse button.

6.1.11. Corbels

These allow for a beam to be supported on a column indirectly to represent expansion joints. They should not be used as column supports.

6.1.11.1. Corbel introduction

The following steps should be followed in order to introduce a corbel.

1. First introduce a regular beam up to the face of the column.
2. Select the corbel to be introduced and define its type and geometry (the dimensions of the support area must also be specified here).
3. Click on the face of the column or columns in question.

At the centre of the support, the program will internally place a fixed support in such a way that the beam continues up to the support and only the vertical load is transmitted to the corbel. The beam is automatically disconnected from the face of the column (any shear or moment is not transmitted).

6.1.11.2. Corbel edition

This option allows for the corbels to be modified.

6.1.12. Corbel adjustment

By clicking on the right mouse button, if sloped planes have been introduced previously, an option appears allowing for the beams to be adjusted to the intersection of the sloped planes or the maximum slope line. The adjustment of the planes to the intersection is necessary so to ensure the axis of the beam separating two floor panels, of which at least one is a sloped plane, coincides with the intersection of the planes of these floor panels. This intersection is represented by a continuous magenta line if it does not find any beam axes and a discontinuous magenta line in the opposite case. The axis can be adjusted to the maximum slope line, in which case, it will be represented by a continuous magenta line on the beams belonging to the sloped panels where the direction is not parallel to the maximum slope line. This line passes exactly through the centre of the beam (relative to its thickness and width). To adjust the beam, it must be clicked on.

6.1.13. Beam reinforcement within walls and crown beams

The possibility exists to optionally reinforce the beams at floor panel level and generic masonry walls and concrete block wall crown beams. In the case of basement walls, the beam is reinforced at the top of the wall.

In the window **Beam reinforcement within walls and crown beams** located by clicking on **Job > General data > By position > Beam options**, it is possible to indicate whether or not to reinforce the beams for the different types of walls. The minimum thicknesses of these beams should also be indicated.

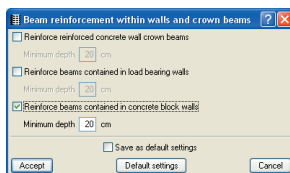


Fig. 6.11

To understand how the option works and know in advance the dimensions of the beams that are going to be placed, consult the help options available in the window.

6.1.14. Effective slab width when using steel beams

The **Information** option from the **Beams/Walls** menu in the **Beam Definition** tab allows for, amongst other things, to know the effective width of the slab supported by the steel beams.

6.1.15. Beam reinforcement editor configuration

It is possible to define the covers within the confinement zones (only when the analysis has been carried out with earthquake loading and all the reinforcement has been placed on site). In this case, vertical blue lines appear.

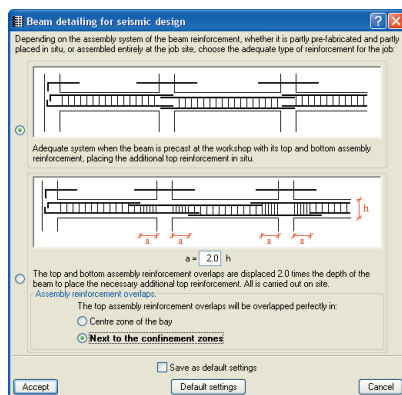


Fig. 6.12

6.1.16. Beam joint and portal frame base references

Allows the user to save the particular base references to be used for future jobs.

An option exists for the beam joints and another for portal frames. These are located within the menu **Job > General options > By position > Beam options**.

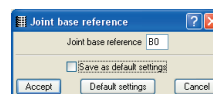


Fig. 6.13

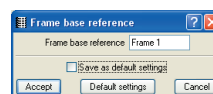


Fig. 6.14

6.1.17. Template object snap for beam introduction

This option is only available to those users possessing the Automatic job introduction module. This allows, based on a single click, to introduce a series of beams adjusted to a template entity.

Once this option has been selected, if the cursor of an entity (line, arch or polygon) is placed close to the template, the beam to be introduced will appear automatically. This is then adjusted at its face or axis. If the cursor is situated to one side of the entity which is going to be snapped to, the beam is displaced in the direction of the cursor, that is to say, the adjusted side is that opposite to the cursor. If the cursor is situated exactly on the entity to be snapped to, it is adjusted to its axis.

In both cases, to validate the introduction, click on the selected position.

6.2. Beam analysis and results

6.2.1. Steel beam design within the analysis menu

If there are steel beams within the structure, a window appears displaying the design options available. If one or more of the options are chosen, the design process to be followed must be selected.

An initial section size will be selected from the tables. The program will prompt whether the design is to be carried out using sections greater than that initially used in the job or from the first section size in the table. In the first case, CYPECAD will only design those sections greater or equal to that selected. In the second case, if a smaller section exists that can resist all the forces, this will be used instead.

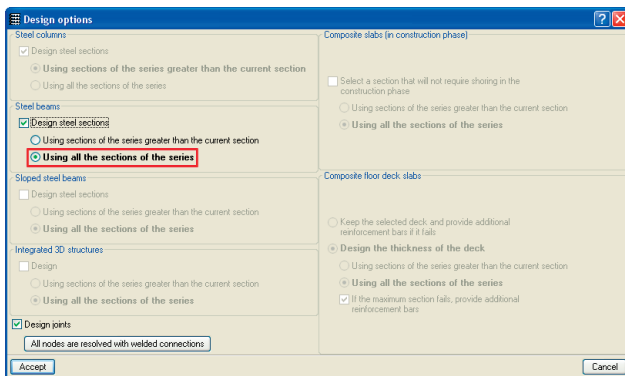


Fig. 6.15

6.2.2. Beam errors

Stirrup diameter greater than the thickness/10. In the NB-1-200 code the maximum diameter of the stirrups is limited. If the result exceeds this limit, the program will emit a warning.

Change in elevation requiring special detail. For those beams with a longitudinal elevation change, it is necessary to provide a construction detail indicating the additional

stirrups of the elevation change beam to ensure continuity of the longitudinal reinforcement of the beam for both elevations.

Length of the bar greater than the maximum length.

This error arises when the program cannot divide the bar and the resulting lengths exceed 12 m.

6.2.3. Sloped beams

Having analysed, a report appears indicating the errors incurred upon analysing the sloped beams.

The following options are located in the **Beams/Walls** menu in the **Results** tab.



Fig. 6.16

Information. This option allows to obtain the properties (types, initial and final floors, section or dimensions and reinforcement, fixity, buckling) of the selected beam.

Efforts. Provides a list of materials, descriptions and forces by loadcase, combination forces, force envelopes, usage coefficient in the case of steel beams, design results (or reinforcement in the case of concrete beams) and lastly, any design errors.

Errors. Sloped beams containing errors or problems are displayed in red. To view the specific error, click on the beam and the program will provide more information.

Edit. In the case of a sloped concrete beam, the Results single sloped beam window can be accessed.

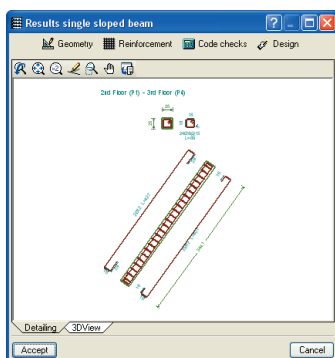


Fig. 6.17

In the case of concrete beams, the window contains the following options:

Geometry. Allows to modify the width and thickness of the beam.

Reinforcement. Allows to edit the stirrups, the top, bottom and skin reinforcements.

Code check, design, detailing and 3D view options available.

In the case of sloped steel beams, the same dialogue appears as with the option **Beams > Beam errors > Check series** for horizontal steel beams.

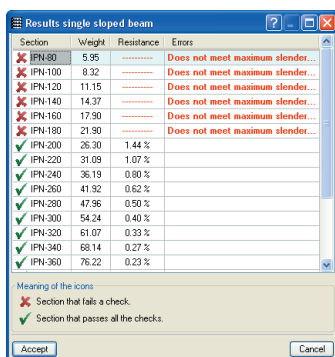


Fig. 6.18

Design concrete beams. Redesigns the reinforcement of all the sloped concrete beams of the job using the forces obtained from the last analysis.

Design steel beams. Redesigns the sections of all the sloped steel beams of the job using the forces obtained from the last analysis. Here, two options appear: Using sections of the series greater than the current section and using all the sections of the series.

6.2.4. Copy reinforcement between frames at different floors

It is possible to copy the reinforcement of one frame to another similar frame regardless of its floor level. This can be done using the option **Copy reinforcement between frames** located in the **Beams/Walls** menu in the **Results** tab. The explanation on how to use this option is displayed upon pressing F1.

- Copy reinforcement between frames of the same group of floors.

Situate the main view at the floor at which the frames are to be copied and select the frame to copy followed by the frame or frames to which the selected reinforcement is to be assigned.

- Copy reinforcement between frames on different groups.

Situate the main view at the floor on which the frame to be copied is located. Click on the right mouse button and choose from the window that appears, the floor to which the selected frame is to be copied. Upon doing so, the selected floor will appear where the frames can have the reinforcement assigned to them. Immediately afterwards, the screen will return to the initial view; the floor where the 'master' frame was selected.

Be it frames of the same group or frames on different floor groups, the reinforcement is copied in the following way: in

the case of assembly reinforcement and the first layer of bottom reinforcement, the diameter, number of bars and anchorage length. For the remaining longitudinal bars, the diameter, number and length of the bars are copied. In the case of stirrups, their diameter, separation and reinforcement length are copied. Having copied the frame, it is recommended the reinforcement be checked in case the reinforcement laid out has been reduced with respect to what was required, as the program does not warn of this when it is copied.

The frame onto which the reinforcement is to be assigned, must comply with the following conditions:

- All spans must be of reinforced concrete.
- All spans must be of the same type. For example, it is not possible to copy the reinforcement of a flat beam to a lattice beam, however it is possible to copy the reinforcement from a flat beam to a dropped beam.
- The beams must contain the same number of spans.
- Any difference in span length must be less than 25 cm.
- If there are any elevation changes, their elevations and positions must coincide.

If for one of the previous reasons, it is not possible to copy the reinforcement of a frame onto another, the program displays information explaining the cause of the failure.

On the other hand, copying of reinforcement between frames does not imply that the beam drawings will all be the same; this will only be the case for those whose geometry is exactly the same.

6.2.5. Design of steel beams

Once the structure has been completely analysed, the option is available to design the floor steel beams.

Additionally, once the job has been analysed and the analysis results have been saved, the option **Check series** is displayed in the **Beam errors** menu. The design can be carried out at any moment, for example if the design option was not selected at the time of analysing the structure or if the beam section has been modified and the user wishes to carry out a check without having to reanalyse the whole structure.

6.2.6. Frame grouping

Within the **Beams/Walls** menu in the **Results** tab, the option **Group frames** can be found, which allows for frames to be grouped amongst themselves. This is done after the job has been analysed. If the grouping is not possible, the program will warn of this and provide a reason.

This option does not undertake any checks regarding the conditions of the grouped frames.

Any modifications carried out on one frame will affect all the others in the group.

In the beam drawings, a single frame will appear per group, indicating all the references belonging to the grouped frames. The option, **Ungroup frames**, performs the opposite process.

Finally, the option **Add frame** to grouping allows for new frames to be added to a previously established group.

6.2.7. Block frame reinforcement

The option **Block frame reinforcement** allows for the structure to be analysed maintaining the reinforcement of those frames that are selected before the analysis. This tool can be found in the **Beam/Walls** menu in the **Results** tab.

6.2.7.1. Things to consider when using this option

The option **Block frame reinforcement** has been brought about so not to loose the manual modifications of the frames due to the job being reanalysed or due to minor changes. This option does not intend to verify the frame reinforcement.

If, once the reinforcement of the frames has been modified manually, it is necessary to carry out small changes on the structure and therefore re-launch an analysis, any manual modifications made to blocked frames will not be lost.

The option **Block frame reinforcement** can be used to maintain the reinforcement of those frames whose groups have not undergone any changes or whose modifications are insignificant.

Those frames that are not blocked will be undergo the reinforcement design process each time the job is reanalysed.

If the changes in the job imply a change in the number of spans of a frame, it will automatically be unblocked.

Warning

Be aware that after a new job analysis, the program will only check if the reinforcement provided (blocked reinforcement) in the blocked frames possesses the required steel area and if the deflection conditions are verified. The remaining checks are carried out in the regular design process (as in the case of unblocked frames). After a new analysis, any blocked frames not verifying the aforementioned conditions will be displayed in the colour configured for beam errors. The user should consult the reinforcement containing errors (**Results** tab, **Beams/Walls > Edit beams**) and decide whether, to resolve the problem, a small modification to the reinforcement is sufficient or if the frame should be unblocked and the reinforcement redesigned.

If significant errors appear in a blocked frame after a new analysis, it implies that the changes undertaken in the job after the blocking of the frame reinforcement substantially affect it and therefore it is recommended it be unblocked and reanalysed. When the program is reinforcing an unblocked frame, the program carries out all the checks stated in the selected code (bar spacing, anchorage length, etc) using the forces obtained in the last analysis.

The program offers two options to redesign the blocked frames: Redesign all the frames or Redesign frames with changes (available in the Beam Definition tab, Analyse menu). This last option proves to be a useful tool as the program only redesigns the reinforcement of those frames that have been unblocked and those whose section has been modified.

All these considerations should be taken into account by the user to decide which is the best solution.

6.2.7.2. How the option works

Having selected the **Block frame reinforcement** option, the **Frame selection** window will appear.

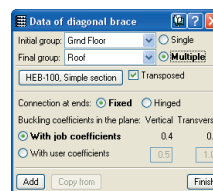


Fig. 6.19

Here, all the frames of a group of floors can be blocked by selecting the box situated in the **All** column and in the row of the corresponding group. In the same way, by selecting the box in the **None** column, it is equally possible to unblock all the frames in a group of floors. If only a few frames

in the group are to be selected, click on the **Selection** button in the row of the desired group. The selected group will appear on screen where the frames to be blocked can be selected by clicking on them with the left mouse button. The outline of the selected frames will be drawn in red so to be able to view those selected and those that are unblocked. To undo a selection, re-select the frame. By clicking on the right mouse button the **Frame selection** window will reappear where the chosen frames will be displayed in the **Selected** column in the row of the chosen floor group.

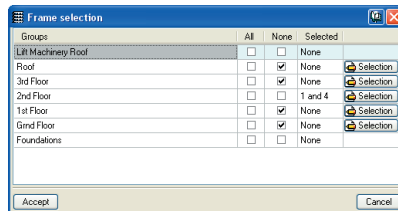


Fig. 6.20

6.2.8. Steel beams

6.2.8.1. Shear stud errors

If a condition is not verified, it can be consulted using the option **Beams > Beam errors**.

Errors that can arise when using shear studs include the following:

- The nominal diameter of the shear stud is too small. Its diameter must be increased.
- The total length of the shear stud is insufficient. Increase the length of the bar or head thickness.
- The shear stud is not anchored in the slab. Reduce the length of the bar or the thickness of the head.
- The total length of the shear stud is insufficient for its nominal diameter. Increase the length of the bar.

- The head diameter is insufficient for its nominal diameter. Increase the head diameter.
- The head thickness is insufficient for its nominal diameter. Increase the head thickness.
- The nominal diameter of the shear stud does not assure the ductility of the connection. Increase the diameter of the bar.
- The length of the shear stud is insufficient to assure the ductility of the connection. Increase the length of the bar.
- The number of shear studs is excessive for the length of the beam. Increase the diameter of the bar or provide a greater section in order to reduce the shear force.
- The thickness of the flange is insufficient to place the shear stud. The thickness of the flange onto which the shear stud is welded must have a minimum value. This thickness depends on the diameter of the shear stud.
- The width of the flange is insufficient for two shear studs to be placed. Bear in mind that there are minimum distances to maintain between the faces of the studs and between the stud and the edge of the flange. A section with a greater flange width must be chosen.

6.2.9. Corbel checks

Once the structure has been analysed, it is possible to carry out a check on the corbels introduced in the job by clicking on **Beams/Walls > Corbels > Check corbels** from the **Results** tab. Any corbels displaying design errors will be drawn in red. A report of the checks performed on a corbel (regardless of whether designed correctly or not) can be obtained by clicking on it.

6.2.9.1. Redesign corbel reinforcement

Redesigns the reinforcement of all the corbels introduced in the job using the forces obtained from the last analysis.

6.2.10. Sloped beam reinforcement edition

It is possible to modify sloped beam reinforcement. By selecting **Edit** from the **Sloped beam** window from the **Results** tab. The reinforcement can be edited and modified upon selecting the **Reinforcement** button.

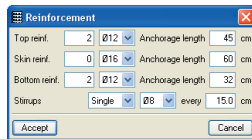


Fig. 6.21

6.2.11. Configuration of the beam reinforcement editor

In the **Results** tab, upon selecting **Beams/Walls > Edit beams > Configuration of the beam reinforcement editor**, a window will appear containing the following options:

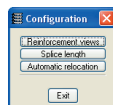


Fig. 6.22

- **Reinforcement views.** The types of reinforcement to view on screen can be chosen so to modify or consult the reinforcement more comfortably. It is possible to consult:
 - Top reinforcement
 - Bottom reinforcement

- Skin reinforcement
- Stirrup intervals
- Beam section
- Confinement zones

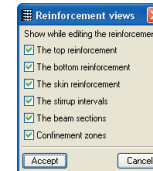


Fig. 6.23

- Splice lengths (l_b). This can be modified by multiplying it by a chosen factor.

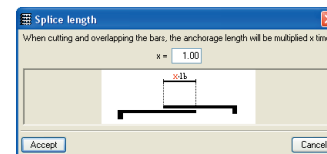


Fig. 6.24

6.2.12. Force diagrams

Within the **Envelopes** menu in the **Results** tab, the option **Forces in beams** can be found. These options allow to consult the force envelopes (Moments, Shears and Torsion), for simple or combined load cases.

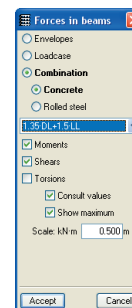


Fig. 6.25

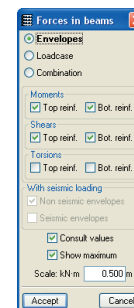


Fig. 6.26

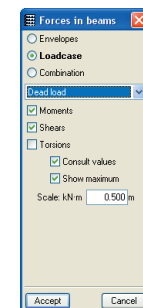


Fig. 6.27

If the option **Loadcase** or **Combination** is activated, a scrolling menu appears in which one may select the desired loadcase or combination. If a dynamic earthquake loadcase is selected, another scrolling menu will appear in which the **Vibration mode** of the structure can be selected.

Having accepted the dialogue box, by clicking on the beam alignments, hollow core slabs or composite slabs with the left mouse button, their force diagrams may be consulted.

6.3. Drawings and reports

6.3.1. Sloped beams

A drawing can be obtained of the sloped beam reinforcement.

6.3.2. Corbels

Corbel drawings display the dimensioned geometry of the front and side views of the element, the main reinforcement and stirrups, the measurements and references of the frame and the column on which it is situated.

7. Floor slabs

7.1. Floor slab introduction

7.1.1. Floor slab floating menu

To open this menu, click on **Slabs > Panel manager** in the **Beam Definition** tab. This menu contains the following options:



Fig. 7.1

- **Define panel.** Upon clicking on this option, the **Panel manager** dialogue box will appear, in which the available slab types are displayed: **Joist floor slabs**, **Hollow core slabs**, **Composite slabs**, **Waffle slabs**, **Flat slabs**, **Mat foundations** and **Awaiting definition**.

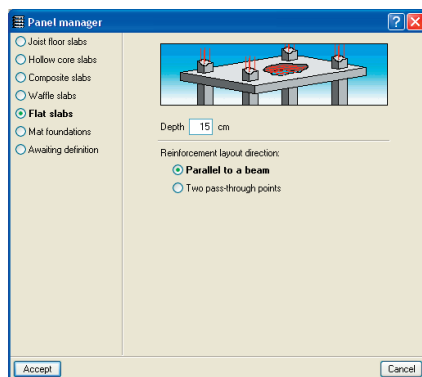


Fig. 7.2

When a structural perimeter (area surrounded by beams) has been created, the program automatically defines it as a panel 'Awaiting definition'. It is displayed on screen with a question mark, to remind the user that it must be defined.

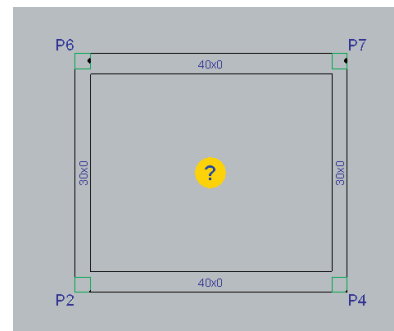


Fig. 7.3

When introducing the panel, the direction of the reinforcement, ribs or joists must be indicated. Once this has been accepted, place the mouse cursor over a space closed off by beams so to select it and then click on an element parallel or perpendicular (depending on the selected introduction configuration) to finish off the introduction and establish its layout.

- **Delete panel (introduce opening).** Place the cursor on the selected slab. Once this has been clicked on, the program will delete it.
- **Change insertion point.**
- **Change layout.** Upon choosing this option, a dialogue appears allowing to select the way in which the slab is to be modified. Once selected, upon placing the cursor on the slab, it will be highlighted and if it is clicked on using the left mouse button, it will be selected. The program will then prompt the user to select the beam or the two pass-through points that will define its new direction.

- **Panel data.** Displays the data of the panel and allows for it to be edited.
- **Copy panel.** Copies the data of a previously selected slab by clicking on it. The window that appears depends on the type of slab. It is possible to select what data is to be copied (Plane, Fixity coefficient, etc). Attention: It is very important to deactivate the Plane option whilst copying slabs with different slopes. If the panel from which the data is to be copied is located on another floor, by clicking on the right mouse button, a floor group selection window will appear.

Select the group in which the panel is located and click so to display the group. Select the panel and the program will automatically return to the group where the panel is to be copied to.

- **Detail forms.** Activating this option when waffle slabs have been introduced will display the forms making up the lightweight elements between the ribs; at the same time, the ribs are drawn with their real width.
- **Minimum moments.** This option is only available if the following types of floor panels have been introduced in the group: joist floor, hollow core or composite.

Upon selecting this option, a dialogue will appear allowing for the minimum moment values to be assigned to the different slabs.

- **Environment.** This option is only available if prestressed joist or hollow core slabs have been introduced in the group.

Upon selecting this option, a dialogue box will appear in which the environment may be selected. Using the buttons Assign or Assign all, it is possible to assign it individually or to all the panels of the group.

- **Fixity coefficient.** A dialogue appears allowing to select the edge fixity coefficient. Using the buttons Assign or Assign all, it is possible to assign the selected coefficient to each individual panel or to all the panels of the

same group. This option is not available for waffle slabs. If an edge fixity coefficient less than one is to be assigned to this type of panel, it must be done using the End fixity option from the Beam/Walls.

- **Construction process.** This option is only available for hollow core slabs. Allows to specify if the construction process of the hollow core slab is to be carried out with Shored construction or Unshored construction.
- **Enter double joist.** This option is only available for joist floor slabs. Creates a second or triple joist next to a selected joist in the panel.
- **Delete double joist.** In the case where there are double or triple joists in a panel, it is possible to delete these by clicking on them, having selected this option.

7.1.2. Floor slab types

7.1.2.1. Awaiting definition

When a structural perimeter is defined by means of beams or walls, the program automatically assigns a panel awaiting definition to the inside of this perimeter. Using the option **Awaiting definition** from the **Panel manager** dialogue box (Slabs > Panel manager > Define panel) any opening or previously defined panel can be changed to one awaiting definition. Panels awaiting definition can be substituted by openings (Slabs > Panel manager > Delete panel) or by any other type of panel available in CYPECAD (Slabs > Panel manager > Define panel). A panel awaiting definition can be assigned an elevation change, a slope and a depth. If the depth is assigned, any flat beams around its perimeter will adopt this depth.

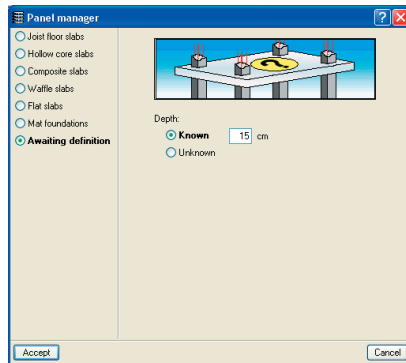


Fig. 7.4

Jobs containing panels awaiting definition cannot be analysed and so there is less probability of the user forgetting to define a panel before the analysis.

The panel awaiting definition is displayed in the 3D views. This allows the user to have a 3D view in which openings and slabs can be seen before the final type of panel has been introduced.

7.1.2.2. Joist floor slabs

Depending on the selected code, various types of joist floor slabs can be introduced.

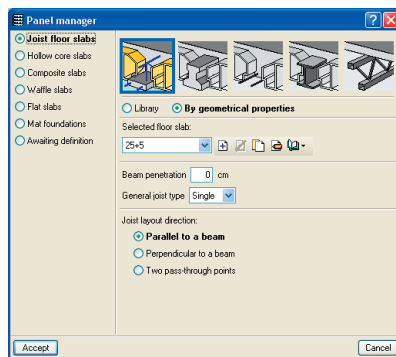


Fig. 7.5

7.1.2.2.1. Reinforced / Prestressed concrete joist floor slabs

The user can choose between using a predefined joist floor slab from the **Library** (catalogue of manufacturers) or to define a new floor slab by defining its **geometrical properties**. Depending on the selected code, the user may only have the option of creating a new slab by defining its geometrical properties, in which case the dialogue box will not display the two options.

By geometrical properties

The user can create, copy, delete, etc., generic panel types, for use when the panel that will finally be used in the job is unknown, as well as being a practical aid as to establish whether the joists to be used will be reinforced or prestressed joists due to their deflection.

When creating or editing a concrete joist floor slab, the following data will have to be entered:

- Reference. Panel description.
- Compression layer thickness. Thickness of the compression layer above the form.
- Form depth.
- Rib spacing. Distance between the axes of adjacent joists.
- Rib width. The minimum thickness of the joist is to be indicated here.
- Rib width increment. This increment refers exclusively to the thickness of the intermediate walls of the forms for the calculation of the stiffness and cracking moment when calculating the deflection. This is only considered when using concrete forms.
- Concrete volume. Depending on the data introduced previously, the program calculates the volume of

concrete per square metre. Nonetheless, the user can modify this value by activating the box and introducing the corresponding value.

- Type of form. The program allows to select between a concrete, terra-cotta, polystyrene or generic form to be used. Based on all the parameters introduced at this stage, the program calculates the total weight of the panel per square metre (volume of the concrete plus the forms). When generic forms have been used, their surface weight must be indicated by the user.
- Deflection Code check. Depending on whether the slab built on site is composed of reinforced or prestressed joists, the type of joist has to be indicated so the program may calculate the deflection accordingly, as the cracked stiffness in these cases varies greatly (the case may occur where the cracked stiffness is practically equal to the gross stiffness, due to the prestressed effect, and so, the cracked stiffness/gross stiffness ratio should be consulted with the manufacturer).

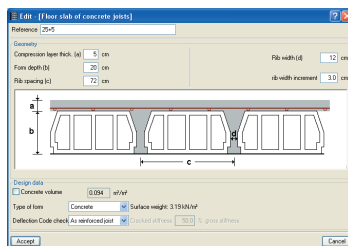


Fig. 7.6

Once the geometrical properties of the slab have been defined, the user can opt to export the defined slab to an internal library, so the slab may be used in future jobs without the need of having to define the properties sheet again.

Library

These are prefabricated joists whose data sheets or authorisations have been provided by various manufacturers. CYPE does not assume any responsibility for the data provided and so, it is recommended these be verified before use. Nonetheless, a logical control has been carried out on the data introduced. On the other hand, in those cases where the user wishes to introduce data belonging to another manufacturer, it can be forwarded to us and we can include it in the program, or, alternatively, a new data sheet can be created using the Panel data sheet editor program.

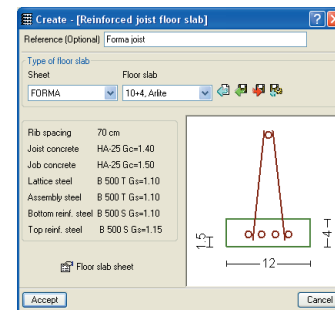


Fig. 7.7

The data to select is as follows:

- Reference (optional). To be introduced only when the user wishes to assign a reference different to that proposed by the program, which is the name of the sheet plus the name of the panel, e.g.: FORMA, 25+5, Arlite.
- Sheet. Select the manufacturer. It may occur that the same manufacture may possess various joists models (different steel properties, rib spacing, etc.).
- Floor slab. Depending on the previously sheet selected, a floor slab may be chosen amongst those available in the series with varying thickness and rib spacing.

Using the previously selected data, a floor slab is obtained whereby the properties are summarised as information for the user: joist concrete, job concrete, prestressed steel, etc.

Floor slab sheet: this option attaches the technical properties of the slab for single as well as double joists. The contents of the properties sheet are those provided by the manufacturers. It contains data that must be provided and other that is recommended be introduced:

Reinforced joists

- Positive flexure of the slab. The data corresponds to that at mid-span:
 - Type of joist. Established name of the joist.
 - Reinforcement per joist. This is the reinforcement defined by the: Base reinforcement + additional reinforcement of the whole length of the joist + additional reinforcement of a smaller length.
 - Rib area. Steel section area in cm².
 - Ultimate moment.
 - Cracked moment. To calculate the deflection using the Branson method.
 - Total stiffness. This is used to compose the stiffness matrix of the concrete joist.
 - Cracked stiffness. To calculate the deflection using the Branson method.
- Negative bending of the floor slab. The data refers to the support or end of the joist:
 - Additional top reinforcement per rib. The reinforcement is defined by reinforcement 1 + reinforcement 2 of a smaller length. Both lengths are calculated by the program.
 - Rib area. Steel section in cm².
 - Typical section ultimate moment. Maximum resisted moment (ultimate).

- Infill section ultimate moment. Maximum resisted moment (ultimate) in the case where there is no form (infill).
- Cracked moment, total stiffness and cracked stiffness. Same as for positive flexure.
- Ultimate shear force of the section. Ultimate shear force resisted by the section. There may be several values depending on the type of truss, for example. For this reason, the type of truss to use will be displayed on screen and in the drawings.

Prestressed joists

- There are two differences between the prestressed joist sheets and those belonging to reinforced joists:
 1. Service moments are displayed. These are the resisted moments depending on the type of prestressed concrete and the exposure class. The equivalence is as follows: Environment I = Exposure class III (structures inside buildings or external structures with little humidity exposure); Environment II = Exposure class II (structures with normal, non-aggressive exposure, or ordinary terrain); Environment III = Exposure class I (structures exposed to aggressive industrial or marine atmosphere, or in contact with aggressive soil, salt water or slightly acid water according to the outdated EH-91 and EP-93 codes). The resulting design service moment is compared, depending on the environment defined for the joist (using the option **Slabs > Panel manager > Environment**) to that of the sheet, and if it is smaller, it is accepted. On the contrary, a joist is found verifying all the conditions and if non are found, an error message is emitted at the end of the analysis.
 2. No shear reinforcement exists.

7.1.2.2.2. In situ joist floor slab

This consists of beams completely built on site. These are built analogically to concrete beams. Therefore, a base reinforcement can be assigned to them using the option **Slabs > Base reinforcement**. Upon creating or editing an in situ floor slab, the following data must be indicated:

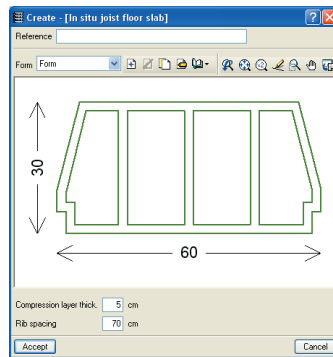


Fig. 7.8

- Reference. A description should be indicated here for the type of slab.
- Form. Choose the type of form to use: concrete, terra-cotta, etc. Possibility to create, copy, delete, etc. the different types of form. The forms are generated using the library (for more information, consult the element Library manager).
- Compression layer thickness. Thickness of the compression layer above the form.
- Rib spacing. Distance between joist axes.

7.1.2.2.3. Steel joist floor slab

This consists of rolled I-section joists. Upon creating or editing a slab composed of joists of this type, the following data should be defined:

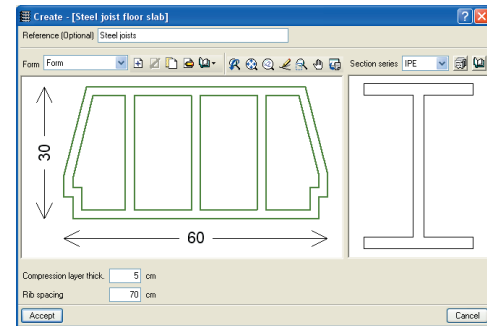


Fig. 7.9

- Reference (optional). To be introduced only when a different reference to that proposed by the program is to be displayed, which is that of the section series + the rib spacing.
- Rib spacing. The type of form is to be chosen: concrete, terra-cotta, etc. as well as its properties (dimensions, weight, etc.). All form types can be created, copied, deleted, etc. The forms are generated using the library.
- Compression layer thickness. Thickness of the compression layer above the form.
- Rib spacing. Distance between joist axes.
- Section series. The series of the section is selected but not the exact section within the series. The advantage here is that the program automatically calculates the required beam following the resistance criteria each time the joists are simply supported between the supports (the program automatically provides the slab a fixity coefficient of 0, which cannot be modified).

7.1.2.2.4. Open web joist floor slab

This consists of simply supported steel truss joists. Upon creating or editing a slab using this type of joist, the following data should be entered:

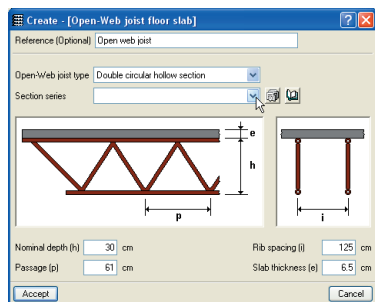


Fig. 7.10

- Reference (optional). To be introduced only when a different reference to that proposed by the program is to be displayed, which is that of the number of sections of the two chords, the section series and the rib spacing.
- Open web joist type. Select the type of joist. Each of the chords shall be composed of one or two identical sections depending on the selected type. The diagonals of the truss will be a single section, belonging to the series selected subsequently.
- Section series. The selected section series should be composed of rolled or laminated steel sections. In a similar way as with steel joists, the series of the section is selected but not the exact section.
- Nominal depth of the truss. Distance from the top surface of the chord to the bottom surface of the chord (not the distance between chord axes).
- Passage of the truss. The angles of the truss should be between 30° and 60°.
- Rib spacing. Distance between the joist axes.
- Slab thickness. Thickness of the concrete layer (non-contributing) above the top chord.

Option **Job > General options > Torsional stiffness reduction coefficients of a joist floor slab**.

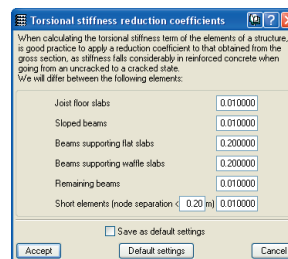


Fig. 7.11

It is possible to modify or consult the value of a torsional stiffness reduction coefficient of a joist floor slab.

7.1.2.3. Waffle slabs

Slabs > Panel manager > Define panel > Waffle slabs. Waffle slabs can be divided into two types: those with a lost form or those with a removable form. When defining a new waffle slab, previously defined forms can be imported from the library (will open automatically upon entering the option for the first time) or alternatively, defined by the user.

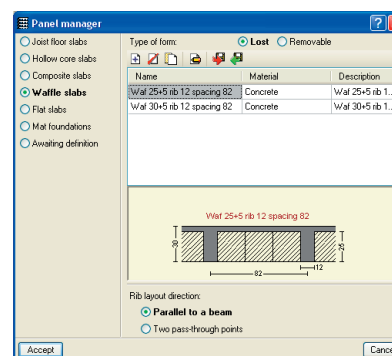



Fig. 7.12

Once a form type has been selected, this can be edited in case the data provided needs modifying to adapt it to the needs of the user. Additionally, the layout of the forms may be viewed on screen by selecting the  icon from the Panel manager menu.

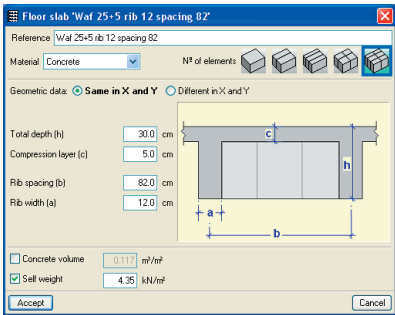


Fig. 7.13

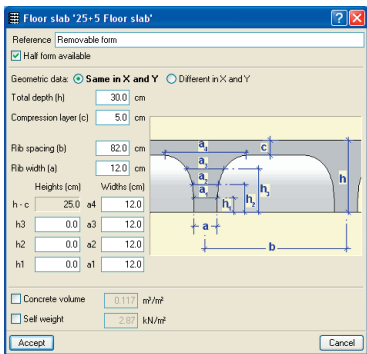


Fig. 7.14

When entering the panel, the user may choose to enter it parallel to a beam or by means of two pass through points (in case the are no beams in close proximity with the sought layout).

Waffle slab panels defined in one group may be copied to other panels in the same group. Similarly, once a base reinforcement has been defined, it may be assigned to the waffle slabs of the group.

7.1.2.3.1. Drop panels

The configuration options for drop panels can be located in **Slabs > Drop panels**. A dialogue box appears displaying the options (the amount of options available depending on whether or not drop panels have been introduced or not). By clicking on the **Configuration of drop panel generation** button, the following parameters can be defined:

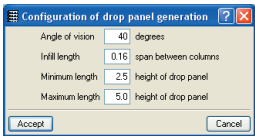


Fig. 7.15

Angle of vision. From the centre, for each column face, the program sends a vision cone in the directions of the grid. Each grid is established with the angle provided in this option. If another column falls within the angle of vision, the distance to this column will be taken as the design span length used in the generation of the drop panel. If there are various columns within the angle of vision, the program will take the nearest one. If there are no columns in sight, the minimum length will be assigned.

Infill length. The infill is measured from the column face to the drop panel perimeter edge, and such length is calculated as a percentage of the distance between the column considered and the nearest column within the angle of vision.

Minimum length. Minimum length from the column face to drop panel perimeter edge with respect to the drop panel depth, in case of there not being any other columns within the angle of vision.

Maximum length. Maximum length from column face to drop panel perimeter edge with respect to the drop panel depth.

7.1.2.4. Hollow core slabs

Hollow core slabs may be imported from CYPE's library or created by the user. Upon creating a new slab the program will require an amount of data to be introduced. To aid the user in its introduction, a book icon can be located in the top right hand corner of the dialogue box, providing information on what is to be defined.

Edition of hollow core plates

Manufacturer: ALVISA
 Reference: a14cc5
 Description: ALVISA: PP-14+ 5/80

Total slab depth (D): 19 cm
 Plate width (W): 600 mm
 Compression layer thick. (T): 5 cm
 Minimum plate width: 300 mm
 Dead load: 0.37 Ton/m²
 Concrete volume: 0.05 m³/m²
 Bearing (min): 8 cm (max): 20 cm
 Lateral bearing: 5 cm

Plate concrete: HA-40, Y_c=1.5
 Layer and joint concrete: HA-25, Y_c=1.5
 Top reinf. steel: B 500 S, Y_s=1.15

Reference	Ult. M	Clk. M	Tot. stl.	Clk. stl.	Ser.M.1	Ser.M.2	Ser.I
14-A	4324.0		1638.0		1827.0	2901.0	345
14-B	5561.0		1648.0		2413.0	3457.0	405
14-C	6736.0		1656.0		3029.0	4123.0	465
14-D	7897.0		1663.0		3726.0	4621.0	535
14-E	8910.0		1679.0		4282.0	5357.0	595
14-F	9968.0		1689.0		5106.0	6233.0	685
14-G	10751.0		1699.0		5153.0	6288.0	695

<14-A> Reference
 Slab negative flexure
 Finish

Fig. 7.16

7.1.2.5. Composite slabs

Panel manager

☐ Joist floor slabs
☐ Hollow core slabs
☒ Composite slabs
☐ Waffle slabs
☐ Flat slabs
☐ Mat foundations
☐ Awaiting definition

Form deck ☒ Composite deck

EUROMODUL44 U position 0.75mm

Total depth: 15.0 cm

Rib direction:
☒ Parallel to a beam
☐ Perpendicular to a beam
☐ Two pass-through points

Accept Cancel

Fig. 7.17

Composite slabs consist in a steel deck profile collaborating with the concrete slab and with top and bottom reinforcement. The steel deck profile can be used to work in one of the following ways:

- **Form deck.** During the construction phase, the deck alone resists its dead load, the weight of the fresh concrete and the construction live loads. During the service phase, it is only the reinforced concrete slab that has a resisting function.

Some manufacturers propose various ways to position the deck: 'n' or 'u', depending on the properties sought by the user. 'n' represents the normal position whilst 'u' the inverse position. Once the composite slab has been introduced on screen, a discontinuous line will be drawn at the deck panel overlap axis, where the width is that indicated on the data sheet.

- **Composite deck.** During the construction phase the deck works as lost formwork. During the service phase, the deck is considered to combine structurally with the hardened concrete, acting as reinforcement in tension, resisting the positive moments in the finished floor slab. The deck is capable of transmitting shear stresses at its interface with the concrete as long as a mechanical connection is provided by deformations in the deck (embossments).

The analysis and design of the steel deck are carried out according to the code: UNE ENV 1994-1-1: June 1995-Eurocode 4: Composite steel and concrete construction design. Part 1-1: General – common rules and rules for buildings, which is the official version of the UNE ENV 1994-1-1 dated October 1992.

Composite slabs are applicable in structural projects in which the applied loads are predominately static loads, including industrial buildings where the slabs can be submitted to mobile loads.

The total thickness of a composite slab, the thickness of the concrete on the ribs of the deck and the minimum height of the shear studs on the deck (in the case of composite beams) are limited.

7.1.3. Sloped slabs

Defining and assigning sloped and different horizontal slabs on the same floor.

Firstly, all the elements of the floor should be introduced with the same horizontal elevation. Having done this, the various slopes or horizontal elevation changes can be assigned to them.

This option is located in **Groups > Sloped floor slabs/ El. Changes** in the **Beam Definition** tab.

7.1.3.1. Sloped floor slabs/ El. Changes Window

The following fields can be found in the dialogue box:

- **Name.** Reference given to the defined plane. This field can be edited. If a yellow arrow appears to the left of the name, it represents that the plane in question has been assigned to a floor.
- **Colour.** Displays the colour assigned to each plane to recognise where they have been assigned on the floor.
- **Data.** Field in which the value of the elevation plane is to be entered, in the case of horizontal planes, or where the elevation values can be modified.
- **Assign.** Assigns the defined plane to panels defined in the floor group. All those to which the plane is assigned will adopt the colour of the new plane.

7.1.3.2. Creation

An inclined plane can be defined in three different ways: 3 points with el. change, horizontal line with slope, and maximum slope.

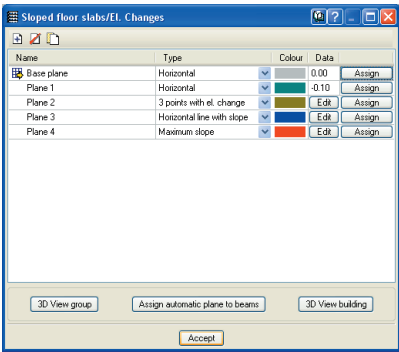


Fig. 7.18

For each one, the definition can be done analytically (**Edit**) or graphically on the floor (**Define on floor**). Once the plane has been defined, it should be assigned to a previously defined slab. Using the **Define on floor** option, the window will close and the floor to which the plane is to be assigned will appear. The plane is then defined by graphically introducing the elevations relative to that of the floor at the specified locations. Having a DXF template of the floor on screen can prove to be a practical tool as it can aid in snapping to beam axes, corners, beam edges, etc. to designate the point locations on the floor. In all cases, if beams have been introduced, the mouse cursor will be displayed as a red point when over a beam's axis or edge, otherwise it will be displayed in black.

Using the **Edit** option, the required data will have to be introduced manually and the position of the points will be placed automatically. The program will position the elevation changes graphically on the floor in the same way as with the **Define on floor** option.

- 3 points with el. change. Select three points on the floor and introduce the relative elevation (el. change) of each with respect to the floor.
- Horizontal line with slope. Select two points on the floor (the direction of the maximum slope will be perpendicular to the line joining the two points), and introduce a single elevation for the two points as well as the slope.
- Maximum slope. Two points indicating the maximum slope. Introduce the elevation change of the first point relative to that of the current floor and the slope.

7.1.3.3. Edit

Upon clicking on this option, the window closes and the definition points appear on screen displaying the elevation data. Another window also appears, displaying the introduced data analytically.

- Edit plane on floor. If the cursor is placed on a definition point, it will be highlighted in yellow, indicating it can be displaced. If it is placed on an elevation or slope text, the value can be modified and the new value typed in. Once this has been done, click on accept.
- Edit a slope analytically. The values can be modified in the window. Having done so, click on accept.

7.1.3.4. Assign

Once the slope has been defined, it can be assigned to previously introduced panels. After assigning a slope, a blue arrow will appear on the panel indicating the direction of the maximum slope in the downwards sense. The direction of the ribs of a panel should always be parallel or perpendicular to the maximum plane. If a plane is not defined in this way, when it is assigned, the program will emit a warning.

7.1.3.5. Automatically assign a slope to beams

If a slope has been assigned to a beam and a panel exists with another beam with a different slope, the program detects this incongruence and displays the affected beams in red. It is possible to then assign the slope of the adjacent panel to the beams by simply clicking on them.

7.1.3.6. Advice on sloped panels

Preliminary concepts

To introduce a sloped slab, its horizontal projection should first be drawn. This can be done on a higher group (towards the top) or lower group (towards the bottom).

The user should have a clear image of the appearance of the building to provide a precise definition of the floor groups.

Below are some examples which should be taken into account when defining sloped planes.

Current cases

Case 1. There are no horizontal panels supported by the beams defining the sloped panel. In this case, a group is defined with the elevation of the small columns of the roof and positive elevation changes are assigned to the inclined planes.

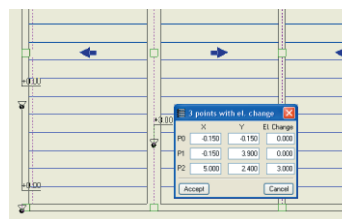


Fig. 7.19

Case 2. There is a horizontal panel at one of the ends of the inclined plane. In this case, the inclined plane cannot be defined in the lower group and then assign positive elevation changes as it would overlap the horizontal panel.

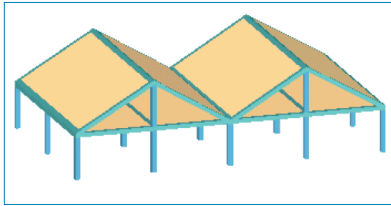


Fig. 7.20

Here, the horizontal panel has to be defined in the lower group and then on the higher group, another horizontal slab onto which the elevation changes will be assigned. All the elevation changes in this higher floor will be negative. The edge beams of the two groups are shared between the two groups (**common beam**); the beam need only be entered once, for the lower group, for example. Before moving up to the higher floor, edit the beams and make them common with the higher group and they will automatically appear in that group.

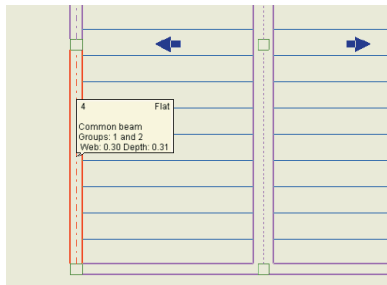


Fig. 7.21

For more information, consult the option **Beams/Walls > Common beam > Create common beam**.

Case 3. Horizontal planes at either end of an inclined plane. This can be the case of a ramp where common beams have been used at its ends. An intermediate group between the two horizontal groups should be defined to allow for the preliminary horizontal introduction of the ramp. Once introduced, the defined slope can be assigned to the ramp.

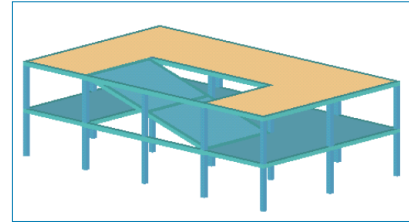


Fig. 7.22

Case 4. There are beams that when projected vertically cause overlaps to occur. If projected vertically downwards on the floor group where the smallest roof column ends, a beam or panel overlap arises. In this case, a floor group must be defined at the elevation of each beam or panel producing this overlap.

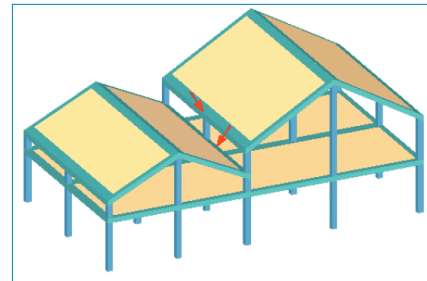


Fig. 7.23

Creation and introduction process of a sloped slab

It is important not to proceed to the next stage without having first completed the previous step. The following is a summary of the process:

- Define the group of floors, depending on which of the previously described cases best describes what is to be introduced. It is very important to draw the building beforehand with all its elevations.
- Introduce all the horizontal panels, except those that will eventually be assigned a slope.
- Assign any common beams, if any.
- Introduce all the panels that will be assigned a slope. It is advisable to use a DXF or DWG template or any other template displaying the intersection lines of the various sloped planes. Separation or transition beams of these planes shall be entered with their axes adjusted at the intersection line of the template.
- Create all the sloped panels.
- Assign the slope plane to the panels.

Other aspects to take into account

- **Horizontal beam projection.** When a sloped plane is assigned to a beam, the program vertically projects the axis or the edge of the beam, depending on whether the previous adjustment is to the axes or edge of the beam. Once this projection has been carried out, the width of the projected beam is drawn in such a way that the top face of the beam is seen with a continuous line. Therefore, a ridge beam, will always appear to have a greater width whereas a valley beam will be narrower.
- **Beam rotation.** If the user wishes for those beams with a horizontal longitudinal axis, such as ridge beams, roof edge beams etc., to rotate their transverse section so to have the same angle as the adjacent panel, the beams must be defined as flat beams. If they are defined as dropped beams, they will not rotate.

It may occur that the encounter between two sloped planes may not coincide with the ridge or valley beam. In this case, the beam will not rotate, even though it has

been defined as a flat beam, and, additionally will have enough depth to support both panels.

- **Limitations.** Several limitations exist regarding the used of sloped planes with walls:
 - A sloped panel cannot be supported by a wall.
 - A common beam cannot be created with the crown of a wall.
- **Drawings.** Drawings can be obtained by clicking on: **File > Print > Job drawings > Floor plans**. The value of the slope is drawn next to the elevation change text at a point of the sloped plane. It is also possible to draw a sketch of the floor. In the **Others** tab within the **Configure** button, the **Floor layout** option is available (only if there are sloped panels) where the size of the sketch box can be indicated in which the columns, beam axes and slope arrows are drawn.

Examples

• Example 1

In the case of a roof with two slopes (mansard roof). An overlap occurs if one of the edges of the roof is projected downwards towards the horizontal plane below it.

This example can be seen in Case 1, explained previously. Therefore, the slopes will have to be defined in the roof group. The groups are defined as follows:

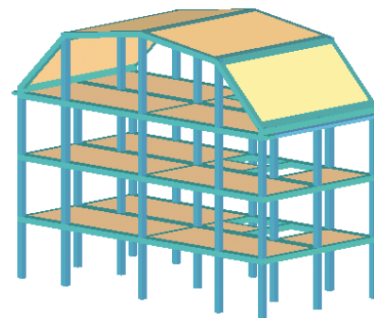


Fig. 7.24

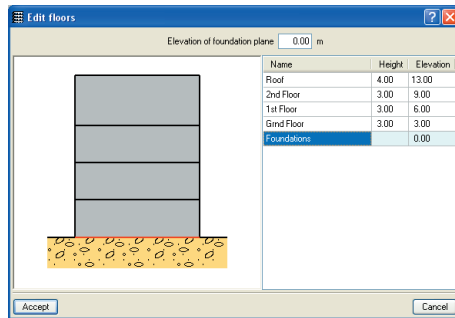


Fig. 7.25

The beams which have been designated as common beams can be seen in the group below the roof group. This has been done using the option **Beams/Walls > Common beam > Create common beam** whereby the beam has been selected to be common with the roof group. The figure displays these beams with a dash-dot axis.

The common beams appear automatically in the roof group and so it is not necessary to redefine the beams. The sloped roof, even though it contains various slopes, can be defined in a single floor group. Once this has been done, the remaining roof beams and panels can be introduced. Since the sloping panels have been defined in the roof group, all the sloped panel elevations will be negative.

Observe layouts 1 and 2.

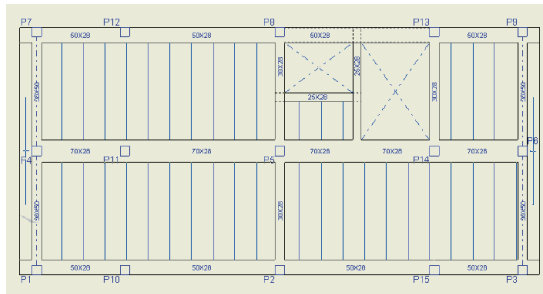


Fig. 7.26

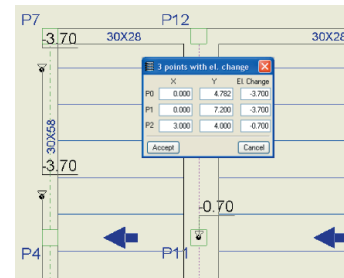


Fig. 7.27

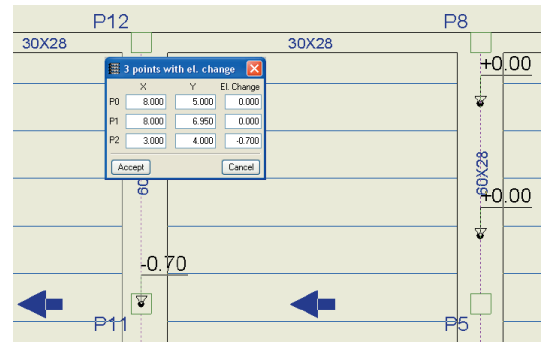


Fig. 7.28

Layouts 3 and 4 are symmetrical to 1 and 2.

• Example 2

This mansard roof contains two dormer windows, but these are only taken into account to simplify the analysis. The sloped roof is laterally supported by dropped beams parallel to the façade.

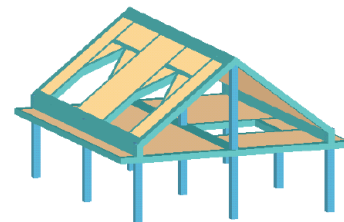


Fig. 7.29

A cantilever roof is not defined as a cantilever as such, but is built on the horizontal floor below. The groups are defined as follows:

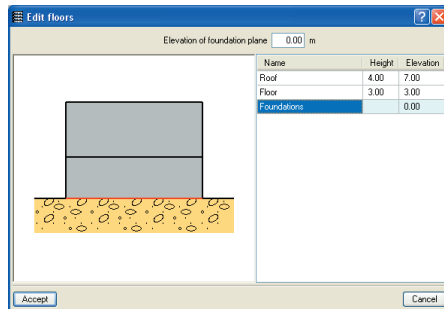


Fig. 7.30

The common beams are then assigned to the horizontal and sloped panels. This is done using the Common beam option where the roof is selected as the common group. The axes of the beams are displayed with a dash-dot line.

The common beams appear automatically in the roof group, and so these beams do not have to be redefined. The remaining roof beams and panels are inserted and the line loads representing the loads of the mansard roof and beams making up the attic are introduced.

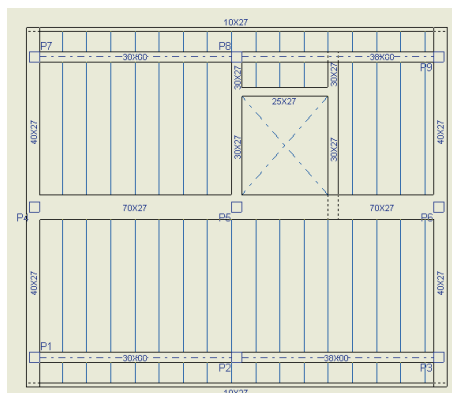


Fig. 7.31

Since the sloping panels have been defined in the roof group, all the sloped panel elevations will be negative. Check to view the layout of the first panel. The second panel is symmetrical.

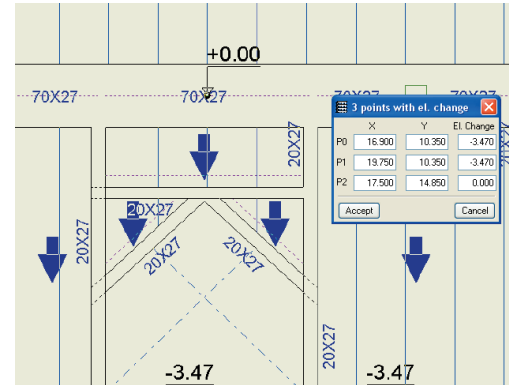


Fig. 7.32

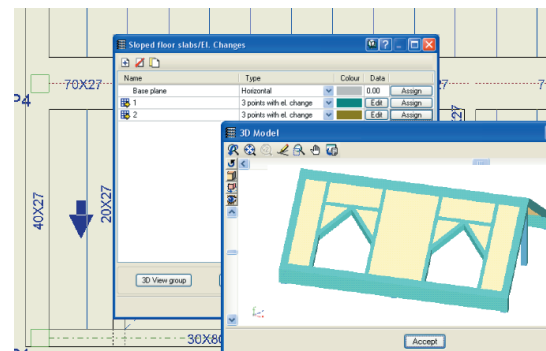


Fig. 7.33

• Example 3

A ramp descending to a garage basement is to be introduced.

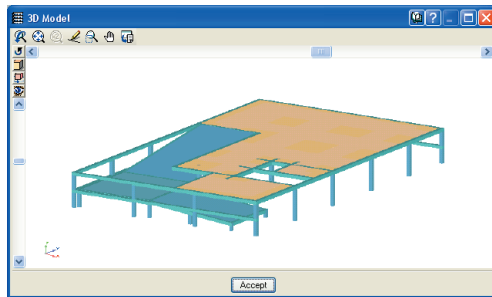


Fig. 7.34

An intermediate group of floors between the foundations and the first slab is created to build the ramp, for example, halfway between both groups.

In the Column definition tab, several independent columns are introduced to sustain the ramp. These columns are to span between groups 0 and 1 (that of the ramp).

The ramp is introduced in group 1. This is defined using a flat slab and flat beams spanning between columns. A masonry wall is introduced at the bottom end of the ramp. The top end will be defined as a common beam with the perimeter beam of the opening in group two. Three panels are defined; two sloping and one horizontal for the curve.

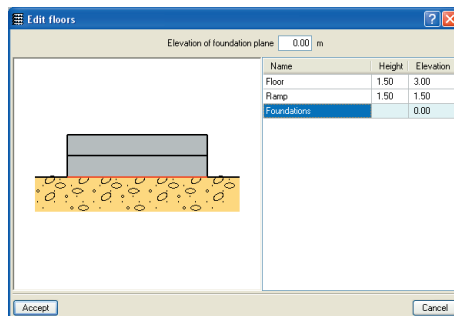


Fig. 7.35

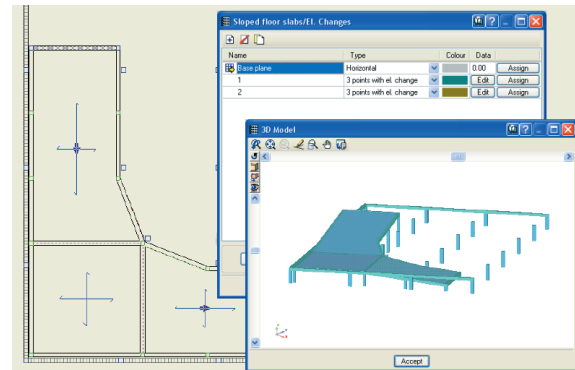


Fig. 7.36

Group 2 contains the ramp opening in the slab and the common beam with group 1.

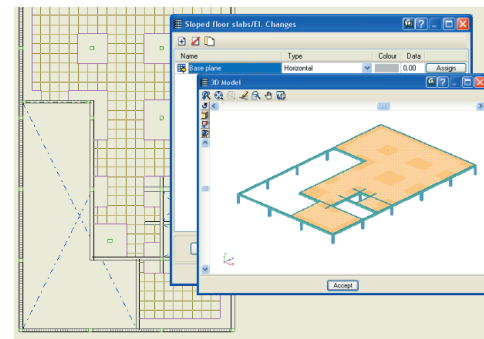


Fig. 7.37

Remember:

- A sloped panel cannot be adjacent to a wall
- A common beam cannot be created using the crown beam of a wall
- It is preferable that curves be defined on a horizontal plane

• Example 4

Introduction of an access ramp between car park floors.

An intermediate floor group is introduced for each ramp between floors. This is defined with a flat slab and flat beams spanning between columns. At the bottom end of the ramp, a common beam is created with the beam directly underneath it in the floor below. At the top end of the ramp, a common beam will be created with the beam above it.

Five floors are defined, three sloping and two horizontal for the curves.

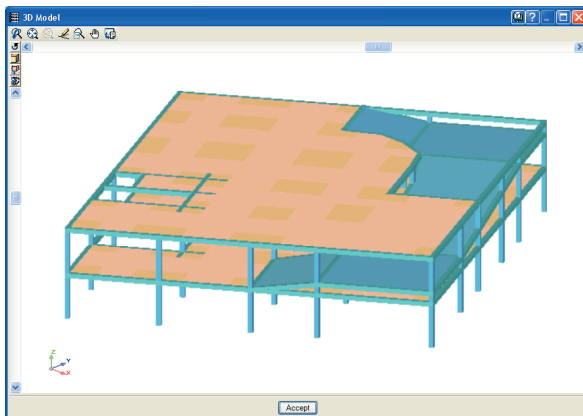


Fig. 7.38

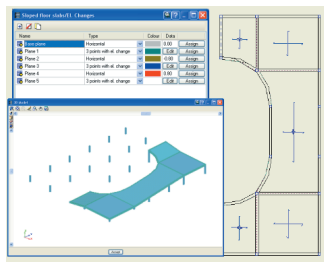


Fig. 7.39

The top group will contain the opening of the ramp as well as the common beam with the group below.

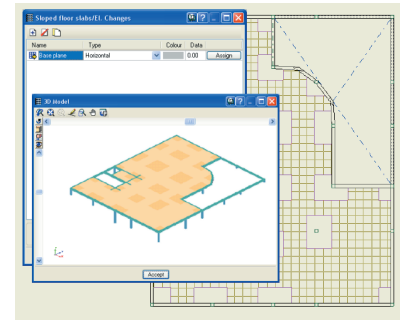


Fig. 7.40

Remember:

It is preferable that curves be defined on a horizontal plane.

7.1.4. Introduce openings

Within the **Slabs** menu, the **Introduce opening** option is located, containing the options detailed below. For each of the options, the **Current beam** dialogue box will open so the user may select the type of beam to be introduced.

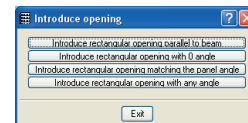


Fig. 7.41

- **Introduce rectangular opening parallel to beam.** A beam parallel to one of the edges of the opening is to be selected. To define the opening, select the points where the two opposite internal corners will be located.
- **Introduce rectangular opening with 0 angle.** The opening that will be introduced will have an angle of 0°. To define it, select the points where the two opposite internal corners will be located.

- **Introduce rectangular opening matching the panel angle.** The two opposite internal corners of the opening are to be selected, bearing in mind that the angle of the opening will be that of the ribs of the slab in which it is located. If the panel where it is introduced is empty, the new opening will be generated with its sides parallel to the general axes.
- **Introduce rectangular opening with any angle.** In this case, the user must select two points defining one of the internal edges of the opening and then drag the cursor to the point of the opposite internal edge.

7.1.5. Floor slab options

Slab options can be located within the menu: **Job > Floor slab options** in the **Beam Definition** and **Results** tab.

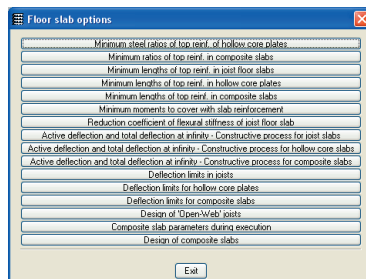


Fig. 7.42

- Active deflection of joist slabs. The default values are different, this option is similar to that of concrete beams.
- Total long term deflection of joist slabs. This option is analogous to that of concrete beams.
- Shear in floor slabs of in-situ joists. This option only appears if the concrete analysis is to be done according to the Spanish EHE code. The program will verify whether it is necessary or not to provide shear reinforcement in accordance with the code. If it is necessary, if

this option is activated, the EF formula is applied which does not depend on the area of longitudinal steel in tension and considers that the shear resistance is a function of the concrete strength used. Therefore, this alternative is applicable based on what is stated in article 1 of EHE.

- Design of open web joists. Here, three options appear providing design alternatives:
 - Design all sections equal
 - Design chords equal
 - Design sections individually

7.2. Slab analysis and results

7.2.1. Joist floor slabs

7.2.1.1. Views of joist floor slabs

These options are located in the menu: **Joists > Views** in the **Results** tab.

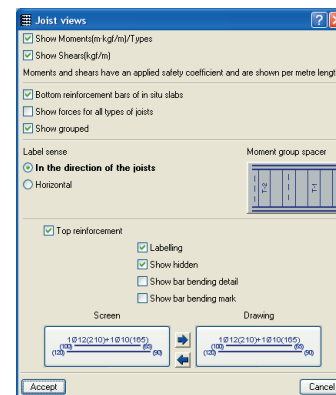


Fig. 7.43

- **Show Moments/ Types**

This option works in combination with **Show forces for all types of joists**. If this last option is activated, the

bending moments (with the applied safety factor and per metre length) are visible. On the contrary, the following can be seen:

The type of joist in the case of prestressed and reinforced joists.

The section in the case of steel joists and open web joists.

The diameter and length of the bottom reinforcement in the case of in-situ joists.

- **Show Shears**

This option works in combination with **Show forces for all types of joists**. If this last option is activated, the shear forces (with the applied safety factor and per metre length) are visible. On the contrary the following can be seen:

The type of truss in the case of prefabricated reinforced joists.

The diameter and separation of the shear reinforcement in the case of in-situ joists.

- **Bottom reinforcement bars of in-situ slabs**

Allows to view the bottom reinforcement of in-situ slabs. This option is not available if the option **Show forces for all types of joists** is activated.

- **Show forces for all types of joists**

If this option is activated, the positive bending moments and shears appear next to each joist (as long as the **Show Moments/Types** and **Show Shears** options are activated).

- **Show grouped**

If this option is activated, the forces/types of joists for each group of identical joists are displayed.

- **Top reinforcement**

- Show bar bending detail. Only for sloped slabs. A bar bending detail is drawn next to the bar.
- Show bar bending mark. The point at which the bar is to be bent is indicated by a small line perpendicular to the axis of the bar.

7.2.1.2. Match

This option can be found in the **Joists** menu in the Results tab.

The joists are grouped in such a way that the following statement is confirmed:

$$(\text{Maximum moment of the group} - \text{Minimum moment of the group}) < (\%) \times \text{Maximum moment of the group}$$

Where (%) is the percentage indicated by the user.

Once the joists that make up each group are known, the program carries out the matching of the different combinations of each one. The matching of the combinations is carried out at each of the points at which the joist has been discretised, according to what has been defined by the user and the type of force:

- **Bending Moments:**

- If the user has selected matching the bending moments to the maximum moment, the program will take the maximum value for each point from all the combinations of each of the joists that make up the group.
- If the user has selected matching the bending moments to the average moment, the program will take the average value for each point from all the combinations of each of the joists that make up the group.

- **Shear:**

The program will take the maximum shear value amongst all the combinations for each joist making up the group.

To match the top reinforcement, a fixed percentage difference and a matching criteria (maximum or average) have to be defined by the user. Once the parameters in the dialogue box have been defined and accepted, the program compares the steel percentages of the top reinforcement of the first joist with that of the adjacent joist. If the percentage difference is less than that selected, both are matched to the greater area of the two (in the case when this option is selected), or the average of the two (in the other case). The steel area obtained in this first step is then compared to that of the next adjacent joist, and the process is repeated. This is done until all the joists of the panel have been analysed and for each individual panel of the job to obtain a more even reinforcement distribution.

7.2.1.3. Errors

Joists containing deflection, shear or any other design error are displayed in red. To know the exact error, click on **Joists > Errors** then on the joist in question and the program will indicate the error that has been encountered.

Possible joist errors include the following:

- **Bottom reinforcement:**
 - Deflection limits. When the instantaneous, long term or active deflection limit has been surpassed (due to dead loading, live loading or assembly load). It is recommended the thickness of the slab be increased.
 - Positive moment at joist support. Change the fixity coefficient of the panel or study the continuity of the bottom reinforcement.
- Insufficient floor slab depth to resist flexure. It is recommended the thickness of the panel be increased.
- No joist has been found to cover the maximum positive moment in the case of prestressed or reinforced joists. It is recommended the thickness of the panel be increased.
- The width of the rib does not allow for the reinforcement due to compatibility problems with the bar spacing and cover. For in situ joists, the bottom reinforcement does not fit in the rib width. Solution: increase the slab depth or increase the thickness of the rib.
- There are no sections in the series that do not fail for this floor slab and form geometry. This occurs in steel joists probably due to having introduced an excessive rib spacing with respect to the width of the form, in such a way that it is not possible to support it on the flanges of the section. An appropriate rib spacing for the form must be placed or vice-versa.
- No section has been found in the series that does not fail for the calculated forces. Applicable in the case of steel joists and open web joists. In this case, select a different section series.
- **Top reinforcement:**
 - Insufficient section for negative moment. The compressed zone does not resist the stresses produced by the negative moment. Not even infilling will solve this problem. The floor slab must be increased.
 - Excessive negative moment. The forces exceed the maximum that can be resisted by the manufacturer's joist sheet. The value of the moment is shown. Solution: increase the floor slab depth or place a double joist.
 - Required top reinforcement area. The top reinforcement has not been designed due to the excessive

steel area required at the joist support (the program designs the top reinforcement in this area based on the moment at the support, in other words at the internal edge of the supporting beam). It is recommended the thickness of the slab be increased.

- Top reinforcement outside of table. Applicable in the case of generic joist floor slabs and in situ slabs. The negative reinforcement defined in the table is not sufficient, but the program will calculate the required reinforcement regardless. However, it is recommended the floor slab depth be increased or a double joist be placed instead of modifying the reinforcement table.
- Infill greater than twice the depth. To avoid the bottom reinforcement working in compression, since the concrete is at its limit, an infill is provided, but the program emits a warning when the infill exceeds this length. Solution: increase the floor slab depth.
- Infill greater than 20% of the clear span. Same as the previous case.
- Fixed or continuous ends in this type of joists are not designed. Although steel joists are automatically pinned at the edges, in the case of cantilevered joists, the program gives continuity to the interior panel to achieve local equilibrium. The negative moment is not designed for.
- The calculated shear cannot be resisted. The calculated shear exceeds the capacity of the joist in the sheet. Solution: increase the floor slab depth or place a double joist.

7.2.1.4. Information

Click on the option **Joists > Information** then on the chosen joist. The deflection and other information of the joist will be displayed.

7.2.1.5. Assign

Allows to modify the designed bottom reinforcement of the joist. Upon clicking on the **Select** button, the window will close and the properties can be assigned to the selected joist. To assign, click on **Assign** followed by the first joist and then the last. The selected joists will be displayed in yellow.

- **Generic concrete joists.** Allows to assign the bending moment and the end shear forces to the selected joist to, for example, to carry out a manual match of the moments and shears of the joists.

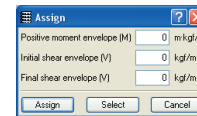


Fig. 7.44

- **Reinforced concrete joists.** Allows to modify:
 - The type of joist within the series.
 - The shear reinforcement.
 - The partial forces. In three different ways: complete length, length to be specified or by percentage length. On screen and on drawings, these are always displayed as a percentage, regardless of the way in which the reinforcement has been specified.

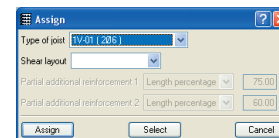


Fig. 7.45

- **Prestressed joists.** Allows to change the type of joist within the series.



Fig. 7.46

- **In situ joists.** Allows to change the bottom and shear reinforcement. The length of the bars can be modified as long as this modification does not imply that the bar will extend past the external face of the beams. Up to two longitudinal reinforcement bars can be specified. In the case of shear reinforcement, the length of the bar in the reinforced zone is converted into a number of bars depending on their separation.

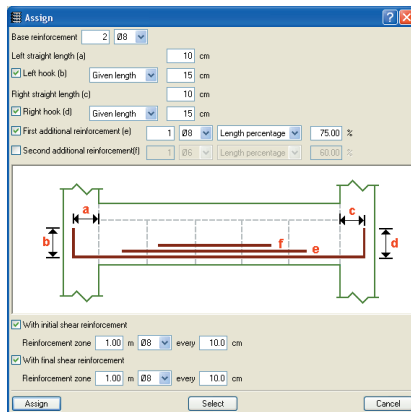


Fig. 7.47

- **Steel joists.** Allows to change the section within the section series.

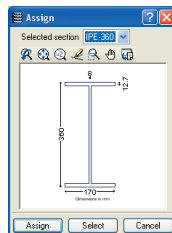


Fig. 7.48

- **Open web joists.** Allows to change the section to one within the selected series for the chords or diagonals.

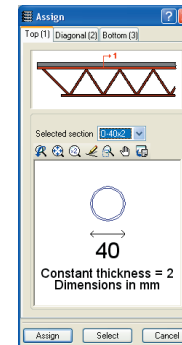


Fig. 7.49

7.2.2. Flat and waffle slabs

When flat slabs or waffle slabs have been used in the structure, a note will appear having analysed the job emitting the following warning: 'The program does not automatically check the deflection of waffle and flat slabs. For these floor slabs, the values of the elastic deflection can be consulted between any two points indicated by the user. The limits in accordance with the code selected for the job should be consulted and estimate the corresponding deflections.' See **chapter 12. Contour maps and deformed shape**, for more information.

Additionally, if there are steel sections greater than those introduced, a message will appear recommending the job be recalculated.

7.2.2.1. Reinforcement introduction in flat and waffle slabs without analysing

This option appears in the Analyse menu. Using this option, the program generates all the data required to introduce, with the current geometry of the job, reinforcement in flat slabs and waffle slabs without having to run a full analysis. If the user proceeds with this option, all the reinforcement of the job will be lost. Once the process has concluded, the reinforcement can be modified.

7.2.2.2. Flat/waffle slab options in the Results tab

• Reinforcement views

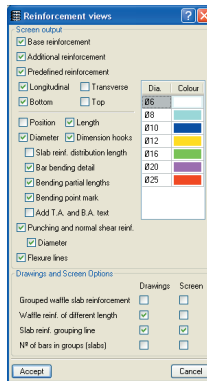


Fig. 7.50

- Bar bending details. Only for sloped slabs. A bending diagram is drawn with the bar.
- Bending partial lengths. Only for sloped slabs. The lengths of the bar on either side of the bending point are detailed.
- Bending point mark. Only for sloped slabs. The bend point is marked with a line perpendicular to the bar axis.
- Add T.A. and B.A. text. These initials are added (top and bottom additional) next to the corresponding reinforcement.
- Flexure lines. Flexure lines that have been introduced are visible using the option **Flat/Waffle Slabs > Modify reinforcement > Define flexure lines**.

• Modify reinforcement

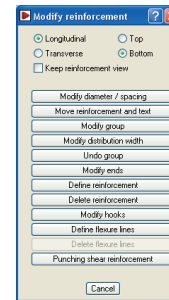


Fig. 7.51

Modify diameter/spacing. In the window that appears, select Table and accept. A table will appear displaying all the reinforcement combination packets defined within the table. A reinforcement package can be selected and then assigned to the slabs. The contents of the table may also be modified.

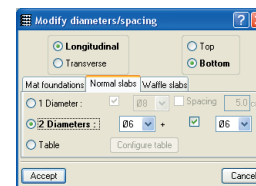


Fig. 7.52

It is not necessary to close the reinforcement window whilst the modifications are being carried out, providing a more comfortable and faster reinforcement solution.

Undo group. Bar grouping can be undone by clicking on the bar group. Upon doing this, all the bars belonging to the group are displayed.

Keep reinforcement view. This option causes the reinforcement activated in the Flat/Waffle slabs > Reinforcement views menu to be visible and appear together with the reinforcement selected to be modified. This

way, for example, the bottom longitudinal reinforcement can be seen whilst the top longitudinal reinforcement can be modified.

- **Save reinforcement copy of all groups**

Using this option, a copy of the reinforcement of all the groups can be saved. This is useful, for example, when an unforeseen opening in the slab has to be introduced and the user does not wish for the previously obtained reinforcement to be lost. In this case, the reinforcement must first be saved, the opening introduced, and then, restore the reinforcement.

- **Recover reinforcement copy of all groups**

This option allows the user to recover reinforcement copied using the option **Save reinforcement copy of all groups**. Note that the reinforcement is only restored in those areas where no geometrical changes have occurred. The reinforcement is cut at the edge of new openings, however anchorage hooks are not provided.

- **Copy reinforcement from another group**

Copies the reinforcement from one group of floors to another. The user must first be located in the destination group. If the destination floor has a different geometry to the master floor, the reinforcement shall only be copied in the areas in which there is slab. The reinforcement is cut at the edge of new openings, but the hooks are not provided. The angle and insertion point of the grid must coincide. For this reason, it is recommended that when defining a group of floors, to always copy the group directly below it. Do not forget to check the bars that have been cut: provide hooks, etc.

7.2.2.3. Add additional shear reinforcement around drop panels

The new option **Add additional shear reinforcement around drop panels (Results tab, Flat/Waffle slabs >**

Modify reinforcement > Punching shear reinforcement > Add additional shear reinforcement around drop panels) allows for groups of shear reinforcement bars to be added in the ribs reaching a drop panel.

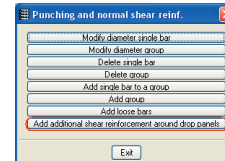


Fig. 7.53

When this option is selected, a dialogue box will appear in which the number, diameter and spacing of the bars are to be introduced.

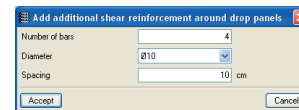


Fig. 7.54

Having accepted the dialogue, the drop panel over which the mouse cursor is placed will darken. Having made the selection using the left mouse button, the chosen reinforcement will be placed in the ribs surrounding the drop panel.

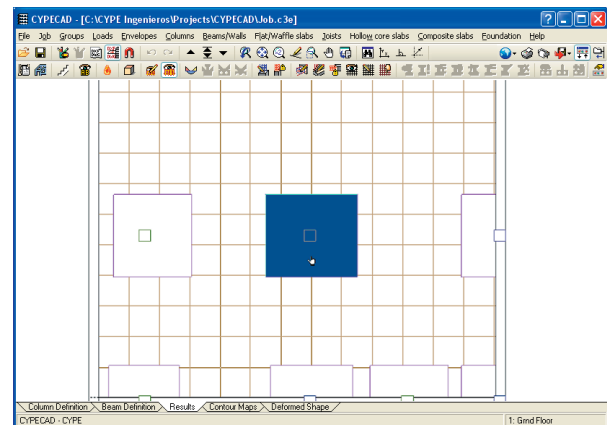


Fig. 7.55

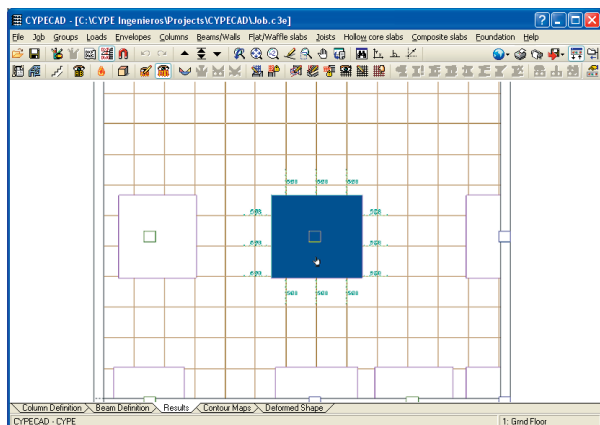


Fig. 7.56

The new bars will be added even if there are already shear bars present in the ribs reaching the drop panel. If you wish to substitute the existing bars for other bars, these must first be deleted. To delete the existing reinforcement, use the following option:

- **Delete punching shear using a capture window**

The option **Delete group (Results tab, Flat/Waffle slabs > Modify reinforcement > Punching shear reinforcement)** allows to delete the punching shear reinforcement groups in flat, solid and waffle slabs one by one or several at once by means of a capture window.

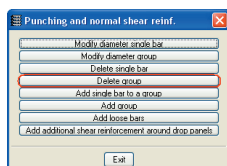


Fig. 7.57

7.2.3. Hollow core slabs

7.2.3.1. Hollow core slab views

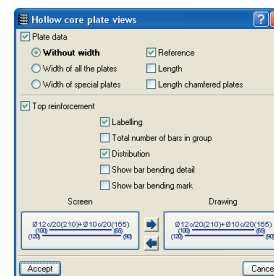


Fig. 7.58

- **Show bar bending detail.** Only applicable to sloped panels; the program draws a bending detail next to the bar.
- **Show bar bending mark.** Only applicable to sloped panels; the point at which the bar is bent is marked with a dash perpendicular to the axis of the bar.

7.2.3.2. Hollow core plates catalogue

The program contains a manufacturer catalogue with property data sheets of the hollow core plates. A table summarising the properties is displayed in the job data report.

7.2.4. Composite slabs

7.2.4.1. Composite slab design

The design and analysis process is carried out in two stages:

a) Construction phase

- To calculate the resistance of the profiled steel deck, the following loads are taken into account: the weight of

the concrete, weight of the deck, and the construction loads. The construction loads represent the weight of the workmen and equipment and any other loading that may arise during the construction phase.

- To calculate the deflection, the construction loads are not taken into account.
- A fixity coefficient of 0 between the panel and its surrounding beams is considered internally (simply supported).
- An option is available to design the deck in case an ultimate limit state or deflection limit is not verified or to calculate the spacing of the shoring without having to design the profiled deck.

b) Service phase

- In the service phase, the profiled steel deck has supposedly been designed in the preceding phase.
- The program assigns a fixity coefficient of 0 to all the panels by default. Therefore with this load distribution on the steel beams supporting the panel, no positive moments arise in the intermediate supports. Once the first analysis has concluded and the beams have been designed, the user can modify the fixity coefficient (between 0 and 1) and re-launch the analysis. If the user assigns a fixity coefficient other than 0, there can be two possible outcomes:
 1. In the previous phase, a slab was designed which did not require shoring and a deck verified all the checks. In this case, the panel must only be analysed with the additional load to be applied after the construction phase, composed of dead and live loads, since the steel deck supports the self weight of the slab.
The way in which the program can take into account, in an approximate manner, only these loads

is by applying fixity coefficients, which the program calculates and applies internally to adjacent panels. As a guideline, the value of the fixity coefficient to assign to the panels depends on the ratio between the self weight of the slab and the total load, supposing the load distribution is uniform. The value of the fixity coefficient would be:

$$\text{Fixity coeff.} = \text{User's fixity coeff} \times (1 - (\text{slab self weight} / \text{total load})).$$

2. In the previous phase, a slab requiring shoring was obtained. In this case the program considers the total load in the service phase (permanent load + surcharge)

- There is an option available allowing the user to optionally design the deck. The bottom reinforcement can also be designed regardless of whether the deck has been designed or not, or whether a suitable decking profile has been found within those available. Either way, if bottom reinforcement is used the collaborating role of the deck is ignored.
- Whenever it is necessary to place reinforcement within the depth of the concrete slab, at least one bar per rib will be laid.

The resistance of a composite slab will be sufficient to support the design loads and assure that no failure limits are reached. Failure methods include:

- **Critical section I:** Flexure: design value of the ultimate bending moment at mid-span. This section can be critical if there is a complete connection at the interface between the deck and the concrete.
- **Critical section II:** Longitudinal shear: the resistance of the connection is decisive. The design value of the ultimate bending moment in Section I cannot be reached. This situation is defined as a partial connection.

- **Critical section III:** Vertical shear and punching shear: design value of the ultimate shear force next to the support. This section will be critical only in special cases, for example, in the case of slabs with a large depth and small spans and relatively large loads.

The design value of a section's bending moment resistance is established using the plastic moment theory of a section with a complete connection.

To obtain the effective area of the steel deck, the deck is assumed to be smooth and uniform. This is the value indicated in the properties sheet of the deck and validated by the user.

The program calculates the design value of the resisted positive moment of a composite slab depending on whether the neutral axis is situated above or within the steel deck.

To calculate the longitudinal shear force, the design value of the ultimate shear force, which is , partly, a function of the 'm-k' coefficients, provided by the manufacturers of the deck. This analysis is that which corresponds to slabs without an end anchorage, i.e., shear studs on a composite beam.

The design value of the resistant shear force of a composite slab is found.

Punching shear is not analysed when concentrated loads are applied.

Fissures in areas of negative moment are not analysed.

The Branson method is used to calculate the deflections, as both the top and bottom reinforcements (be it the deck or reinforcement) are known.

As indicated previously, the program checks and carries out the design so that the deflection limits defined for the

construction phase are not overcome. This is done by increasing the thickness of the deck or by shoring. In the service phase, only the deflection is checked, CYPECAD does not design the deck, and so, in this way, the deflection limits defined for the service phase are complied with as what avoids overcoming the limits is increasing the total depth of the slab.

The specific reinforcement tables for top and bottom reinforcement are created. These can be accessed by clicking on the bar options within the **General data** window in the Job menu.

Within the menu **Job > Floor slab options**, there various specific composite slab options similar to those of joist floor slabs:

- Minimum ratios of top reinforcement in composite slabs.

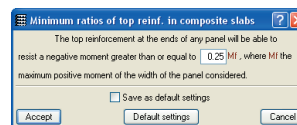


Fig. 7.59

- Minimum lengths of top reinforcement in composite slabs.

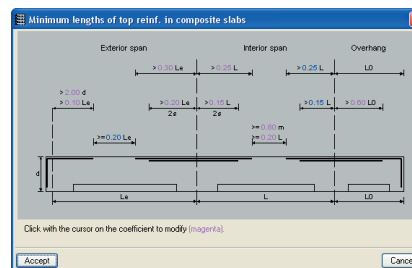


Fig. 7.60

- Deflection coefficients - active deflection and total long term deflection for composite slabs.

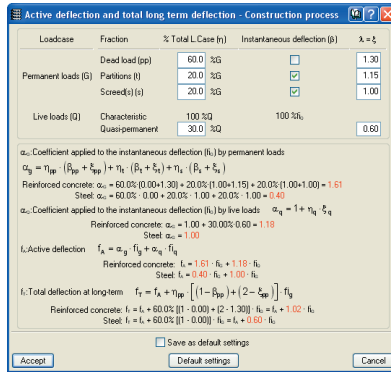


Fig. 7.61

- Deflection limits for composite slabs.

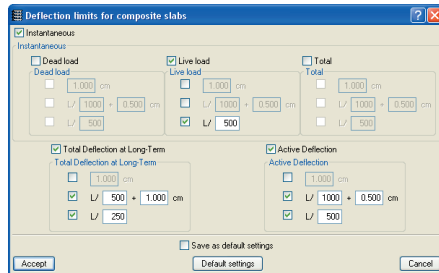


Fig. 7.62

- Composite slab parameters during execution. The following options are available:

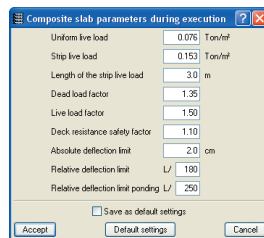


Fig. 7.63

- Uniform live load.** The construction loads represent the weight of the workers and concrete equipment, and take into account any impact or vibration that may occur during construction. Over any rib, and in a length of 3 m (or the span of the floor slab if smaller), a load of 0.150 t/m² is added to the weight of the concrete (with the corresponding extra thickness of concrete if ponding occurs), which represents the typical construction load and possible extra thickness (local accumulation). This load is placed at midspan to produce the maximum bending moment, since the bay is simply supported during construction. Over the rest of the rib, a typical load of 0.075 t/m² is added to the weight of the concrete.
- Dead and Live load factors.** The partial safety factors for loads in building structures are: 1.35 for permanent loads and 1.50 for variable loads.
- Deck resistance safety factor.** The partial safety factor for the resistance of the ribbed deck must be 1.10 for the construction phase. Even if this coefficient is modified, the program will continue to apply a value of 1.10 in the construction phase.
- Absolute deflection limit.** Maximum deflection of the deck its dead weight and the weight of the fresh concrete (with the corresponding extra thickness of concrete if ponding occurs). The program checks and designs so that these deflection limits defined for the construction phase are not exceeded, increasing the deck thickness or placing shoring. In the service phase, the program only checks deflection; it does not design the deck to meet these deflection limits, since this problem is solved by increasing the total floor slab depth.
- Relative deflection limit ponding.** Maximum relative deflection of the deck under its self weight and the weight of the fresh concrete. If this limit, or the 'Absolute deflection limit', is exceeded, the program recalculates the concrete weight, adding to the thickness the deflection obtained multiplied by 0.7.

- Design of composite slabs. The following options are available:

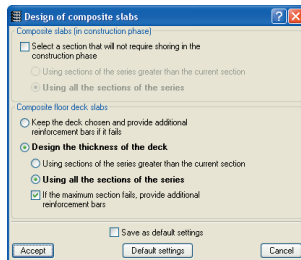


Fig. 7.64

- **Composite slabs (in construction phase).** This option refers to the analysis of the deck in the construction phase, when it supports the load of fresh concrete plus the execution live load by itself. **Select a section that will not require shoring in the construction phase.** If the option is not activated, the program does not design the deck in the construction phase. If the deck placed by the user does not meet an ultimate limit or service state, the distance between shoring is calculated so that the deck works. If the option is activated, the program will design the deck in the construction phase. If the deck placed by the user does not meet an ultimate limit or service state, a deck that does will be selected. If the program cannot find a deck that works, the deck placed by the user will be kept and the distance between shoring will be calculated.
- **Composite floor deck slabs.** These options refer to the analysis of a contributing deck, in which it combines with the concrete to form a composite deck:

Keep the selected deck and provide additional reinforcement bars if it fails. The program checks the deck resulting from the construction phase. If it fails, the necessary bottom reinforcement is designed.

Design the thickness of the deck. The program checks the deck resulting from the construction phase. If it fails, the program looks in the series for a deck with sufficient thickness. If the option: **If the maximum section fails, provide additional reinforcement bars** is activated, in case no valid thickness is found in the series, the necessary positive reinforcement is designed, keeping the deck resulting from the construction phase. If the option is not activated and no valid thickness is found in the series, a warning is emitted of the failure with the **Composite slabs > Errors** option.

7.2.4.2. Composite slab results

- **Views.** Using this option, the data for the top and bottom reinforcement can be configured on screen, as well as the labels that will appear on screen and in the drawings. Note that the bottom reinforcement is laid out by rib, and the top reinforcement is distributed uniformly over the whole width.

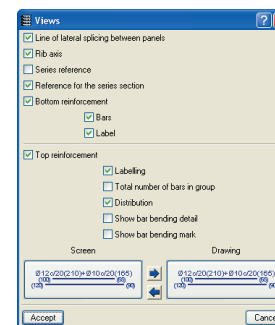


Fig. 7.65

- **Errors.** If there is an error in any section of the composite slab, it is shown in red. To view the error message, click on the marked section. A window is also available with two buttons (Previous and Next) to sequentially consult the panel errors.

- Error in execution phase:
Maximum clear span of panel = 2.89524 m.
The thickness of the deck is not sufficient to
avoid shoring.
Maximum distance between shores = 2 m.

Fig. 7.66

- Information.** The following information appears:
 Name of the manufacturer, type of deck and total depth of the slab.
 If, having analysed, the panel requires shoring in its construction phase, their maximum separation is indicated.
 The design span for the construction phase (if shoring is required, this will be the shoring separation).
 The deflection in the construction phase.
 The total span in service phase (distance between supports).
 Deflection in service phase. If this is too large, it is displayed in red. Recall that the program designs during the construction phase so that the deflection limits and strength of the deck are not exceeded (increasing thickness of the deck or providing shoring), but in the service phase, the deflection problem can only be solved by increasing the depth of the slab.

HLM-60/220, 1.20mm, 15.0 cm
Requires shoring every 2.50 m
Construction deflection:
Span: 2.50 m
Instantaneous: 0.768cm (L/326)
Service deflections: Span (secant)
Total span.: 3.76 m
Tot. l. term.: 1.311 cm (L/287)
Active 0.858cm (L/439)

Fig. 7.67

- Assign deck** (bottom reinforcement). Allows for a type of selected deck to be assigned to one or various panels to, for example, carry out a manual match once the job has been analysed.



Fig. 7.68

- Assign reinforcement** (bottom reinforcement). Allows for bottom reinforcement to be assigned to one or more ribs to, for example, manually match or make the reinforcement uniform, once the job has been analysed.

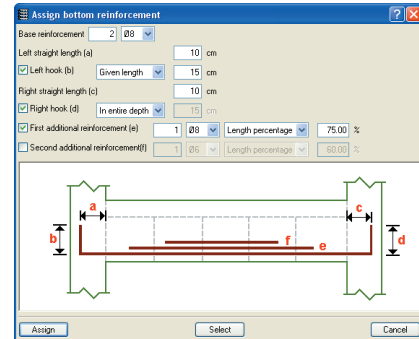


Fig. 7.69

- Modify position** (bottom reinforcement). The representation of the reinforcement of a slab width can be moved to another point within the same width. This can be useful when the reinforcement label interferes with another text.
- Delete reinforcement** (bottom reinforcement). Deletes bottom reinforcement.
- Modify** (top reinforcement). This option allows to change the spacing, diameter and length of bars. Reinforcement layers or groups can also be added or removed.

Once the bars to be modified have been selected, a window opens indicating the different values depending on the properties of the selected top reinforcement.

The lengths of the bars anchored as straight bars are measured from the axis of the nearest beam. The lengths of the bars with a hook anchorage indicate the length from the bending point.

- **Assign** (top reinforcement). Assigns the type of selected top reinforcement to one or more panels.
- **Introduce** (top reinforcement). Using this option, the **Current reinforcement** window will open. If it does not, click on the right mouse button and it will appear. The button **Deflt. Lgths.** Allows for the lengths of the bars to be introduced to be defined. Having selected the reinforcement and accepted the dialogue box, select the initial reinforcement introduction point on the panel and drag to the final point.
- **Delete** (top reinforcement). Deletes the selected top reinforcement.
- **Modify position** (top reinforcement). Allows to move the representation of the reinforcement of a slab width to another point within the same width. This can be useful when the reinforcement label interferes with another text.
- **Panel data.** This option is exactly the same as the Panel data option in the Beam Definition tab.

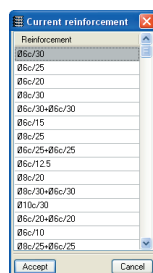


Fig. 7.70

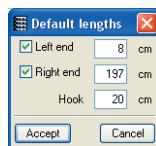


Fig. 7.71

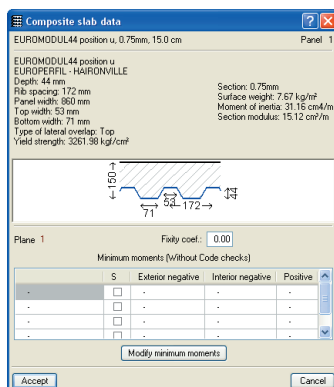


Fig. 7.72

7.2.5. Force diagrams

In the **Envelopes** menu from the **Results** tab, the options **Forces in joists**, **Forces in hollow core plates** and **Forces in composite slabs**. These options allow for the force envelopes (Moment, Shear and Torsion), loadcases or combinations to be consulted.

If the **Loadcase** or **Combination** option is selected, a scrolling menu appears from which the desired loadcase or combination can be selected. If a dynamic earthquake loadcase is selected, another scrolling menu will appear from which the **Vibration modes** of the structure can be selected.

Having accepted the dialogue box, by clicking with the left mouse button on the beam alignments, joists, hollow core slabs or composite slabs, the forces or force diagrams can be consulted.

7.3. Drawings and Reports

7.3.1. Composite slabs

7.3.1.1. Reports

The following composite slab reports may be obtained:

Forces in composite slabs

Composite slab takeoff

Composite slab reinforcement takeoff

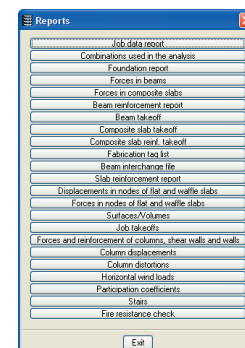


Fig. 7.73

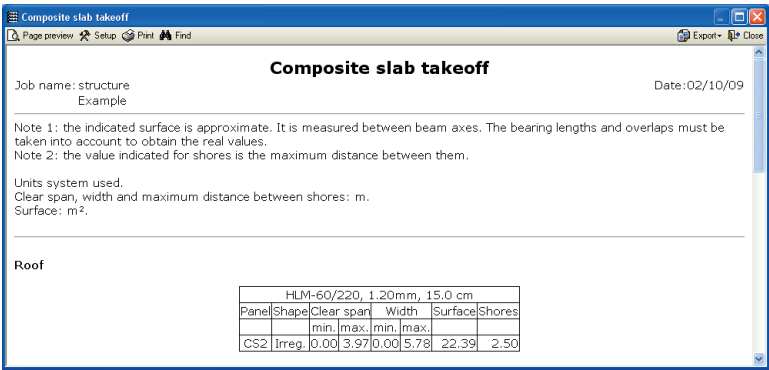


Fig. 7.74

The slab data may be viewed in the **Job data report** and the material quantities in the **Job takeoff**.

7.3.1.2. Drawings

Within the **configure** option of the **Floor plan**, a tab labeled **composite slabs** can be selected and within which all the elements relative to the composite slabs can be found.

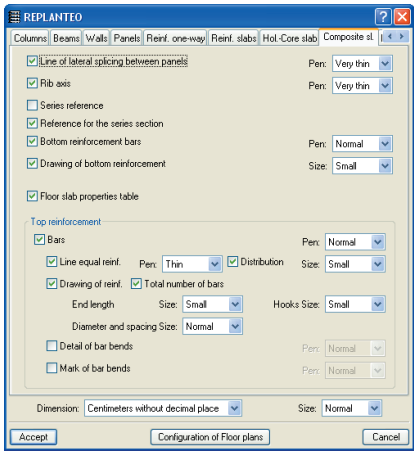


Fig. 7.75

7.3.2. Joist floor slabs

7.3.2.1. Report of forces in joist floor slabs

Draws the moment, shear and torsion envelopes of any joist alignment. As well as the force envelope drawn in red, a diagram of the floor group and the joist in question are provided for each floor group.

7.3.2.2. Joist floor takeoff

The joist reinforcement weight and takeoff per metre length (including fabrication loss) can be obtained. The top, bottom and shear reinforcement in the case of insitu joists is measured (regarding the shear force, the total length or the omega shaped reinforcement is provided).

7.3.3. Sloped panels

The drawings are obtained via the following route: **File > Print > Job drawings > Floor plans**.

The value of the tangent of the slope is attached to the elevation text on the inclined slope. A diagram of the floor can also be provided. Within the **Others** tab in the **Drawing configuration** option, the option **Floor layout** can be found (only available if there are sloped slabs) where the size of the diagram in which the columns, beam axes and slope arrows are displayed can be established.

8. Stairs

8.1. General information

The **Stairs** module analyses and designs the reinforcement of stair slabs as isolated elements of the structure. Depending on the geometry, the type and the layout of the supports and the applied gravitational loads, the program establishes the reactions on the main structure which are transmitted as line and/or surface loads as dead and live loads.

The program considers the staircase to be composed of spans between floors containing landings of the following types:



Straight flight



Two straight flights with half turn landing



Three straight flights with quarter turn landings



Two straight flights with full turn landing



Two consecutive flights with intermediate landing



'n' straight flights with half turn landings



'n' straight flights with quarter turn landings

The program interprets the “staircase” as the group of stairs spanning between floors defining the vertical circulation of a specific area of the building. The “flight” is the inclined part of a stair formed by a continuous succession of steps covering the elevation change between two horizontal planes.

The intermediate landing is the intermediate horizontal plane between two consecutive flights.

8.2. Location of Stairs module

The options relative to stairs are located in the **Stairs** floating menu, accessed by clicking on **Job > Stairs** in the **Beam Definition** tab.



Fig. 8.1

These allow the user to:



Create a new staircase



Edit existing staircases



Delete staircases



Move a staircase



Rotate a staircase




View reinforcement details of all the spans of a staircase



View forces and displacements using contour diagrams

8.3. Data introduction in the Stairs module

8.3.1. Create a new staircase

By clicking on the  icon from the **Stairs** floating menu, the **New Staircase** dialogue box appears.

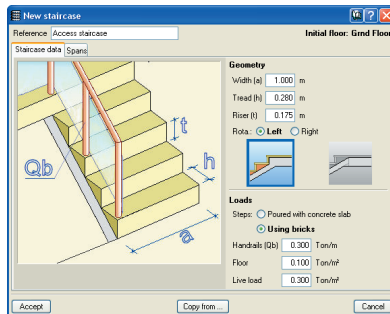


Fig. 8.2

The window contains two tabs. In the **Staircase data** tab, the common properties of the staircase assembly can be defined and in the **Spans** tab, the properties of each span between floors.

The program considers the new staircase to start at the current group, the name of which appears in the top right hand corner of the dialogue box.

8.3.1.1. Staircase common data

In the Staircase data tab, the geometric properties and loads of a new staircase are defined.

These properties are common to all the spans belonging to the staircase. If the case arises that part of a staircase contains particular data or possesses different values (different loading, for example), then it must be introduced as two separate staircases.

Geometric properties

- **Width**

The width of the stair or length of the risers.

- **Tread and Riser**

The tread is the width of the step measure on plan and the riser the vertical distance between two consecutive treads (height of the step).

- **Rotation**

Left or **Right** may be selected depending on the rotation direction the person takes when going up the stair.

- **Layout of last step**

The last step of an ascending flight can have one of two different layouts:



The intermediate landing or landing form the last step.



The last step is situated on the inclined slab.

Loads

- **Steps**

The step construction method (**Poured with concrete slab** or **Using bricks**). This information affects the calculation of the dead loads applied to the stair and when measuring the quantity of concrete used for its construction.

- **Handrail, floor and live loading**

The required weight of the handrails is the total weight, therefore the user has to introduce the data taking into account if there are one or two handrails.

8.3.1.2. Staircase flight data

The flights of the staircase are the panels that join one floor with another and can contain one or more spans. The properties defined for the flights can be different from each other (slab thickness, initial elevation, layout, number of steps etc.).

In the program, the flights of a staircase are defined by selecting the **Spans** tab in the **New staircase** window. Having selected the tab, the following window will appear:

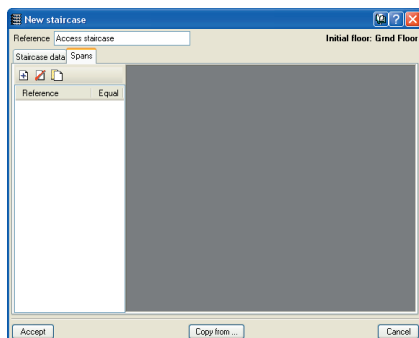


Fig. 8.3

In this window, flights may be attached by clicking on the corresponding icon, which will open the **Flights** dialogue box:

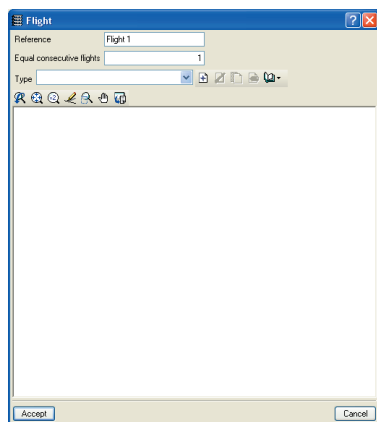



Fig. 8.4

New flight types may be added by choosing amongst those predefined by the program from the **Type** scrolling menu or, alternatively, a new flight can be created by clicking on the  icon. Upon clicking on this icon, a new window **Create [Stair types]** window appears in which the properties of the new type are defined.

The group of types forms part of the job library, and therefore a type of flight can be used more than once in a staircase and in more than one staircase of the job.

The program also allows for staircases that have been defined in one job to be used in another using the library manager, which allows for types to be imported and exported.

In the **Create [Stair types]** dialogue box, the following data is specified:

- **Reference of the stair type**

Identification of the defined flight

- **Slab depth**

The depth of the slab can be proposed by the program or set by the user.

For the program to be able to pre-design the depth of the slab, do not activate the **Slab depth** box; the program will then assign a depth whose value is 1/30 of the true maximum span between supports.

When the **Slab depth** box is activated, the user has to introduce the value for the depth of the stair slab.

- **Start level difference**

It is possible to indicate an elevation change (greater or equal to 10 cm) at the start of each staircase flight due to the eventual depth of the slab.

An example of the where this value can be applied, is in the case of a staircase starting at the foundations and must cross a raised floor slab or screed.

- **Initial steps built on floor**

If there are steps built on the floor at the start of a span of stairs, the number of steps must be indicated. Once the stair has been calculated, the program applies a uniform surface reaction to the main structure corresponding to the mean load of the steps built on the floor and introduces it as a dead load. The program does not generate the live load that may occur in the area taken up by the built steps as this element is located on a structural element (for example, a slab) that already has a live load applied to it.

- **Span and landing layout**

Select one of the types to be introduced.

- **Number of steps**

Depending on the type of stair that has been selected, the program will ask for the number of steps of each slab making up the flight.

- **Stairwell width**

This is only required if there are half turn landings.

- **Steps built on intermediate landing**

If there are any, the number of steps is required.

- **Intermediate landing supports**

Intermediate landings can be supported or free.

If supported, the location of the supports must be specified: Front, Lateral, Both

Type and width of support. The program allows the user to choose amongst the following supports:

- **Hanging bar:** these are bars that hold the edge of the landing by means of a structural element (usually a beam) situated on the floor above the landing. The program requires the width to be able to consider the weight of the brick wall situated between the landing and the floor above.

Masonry wall (*)

Concrete wall (*)

(*) Its load is applied on the structural element situated under the supported edge of the landing on the floor below. The self weight is taken as the load of a wall of a given width and of height equal to the level difference between the intermediate landing and the floor below. A specific weight of 15.70 kN/m³ is taken for the case of a masonry wall and of 24.53 kN/m³ for the case of a concrete wall. If there were to be a wall spanning between the intermediate landing and the floor above, or any other type of partition or external wall, the user must apply the line load on the floor below corresponding to the load of the aforementioned element.

- **Connectors:** these are shear connectors joining the intermediate landings to structural elements such as a beam or wall, to which the end reaction is transmitted.

Having accepted the **Create [Stair types]** dialogue box, the program returns to the **Flight** window where the number of **Equal consecutive flights** is required, that is, the number of consecutive floors of the staircase that contains this type of flight.

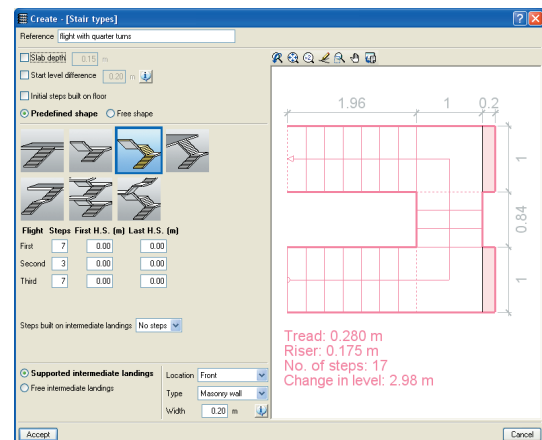


Fig. 8.5

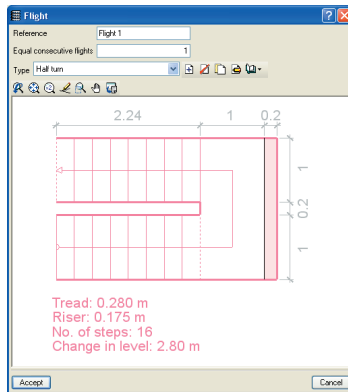


Fig. 8.6

Once these values have been defined, the program returns to the **New staircase** window, which will display the new flight that has been defined.

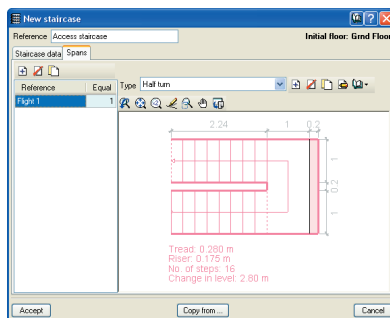



Fig. 8.7

Within this window, the number of consecutive equal flights can be defined.

Any number of flights can be defined following the same procedure. Additionally, a new flight can be defined by copying the data of a previously defined flight using the  icon at the top of the list.

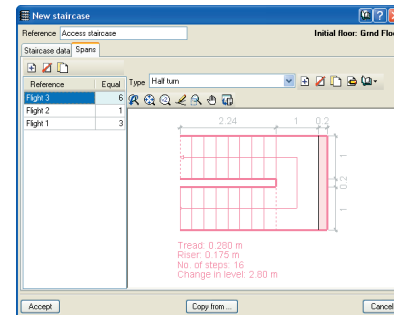


Fig. 8.8

In section **8.3.1.4. Tools to facilitate staircase and flight definition**, there are several interesting indications which may result useful when defining new staircases and flights.

8.3.1.3. Introduction of the staircase in the job

Once all the data (common and specific to each flight) of the staircase has been defined, all that is required is to position the staircase on the current floor.

To introduce the staircase, click on **Accept** in the **New staircase** dialogue box. The program will display the current floor on screen and will draw the first flight of the staircase to be introduced next to the mouse cursor.

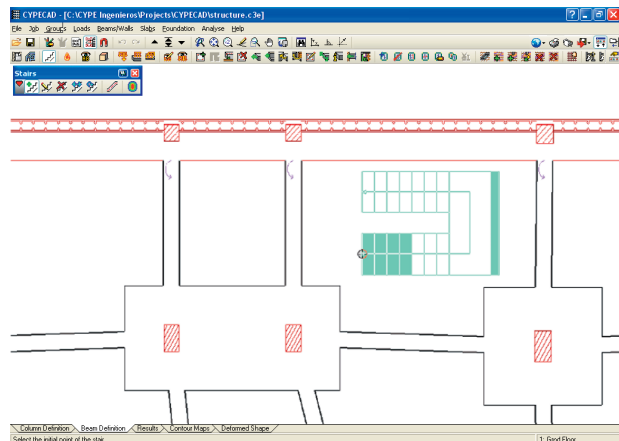


Fig. 8.9

The staircase moves with the cursor and will remain at a fixed position upon clicking on the left mouse button. To help situate the start of the staircase, object snaps can be used with a DXF or DWG template.

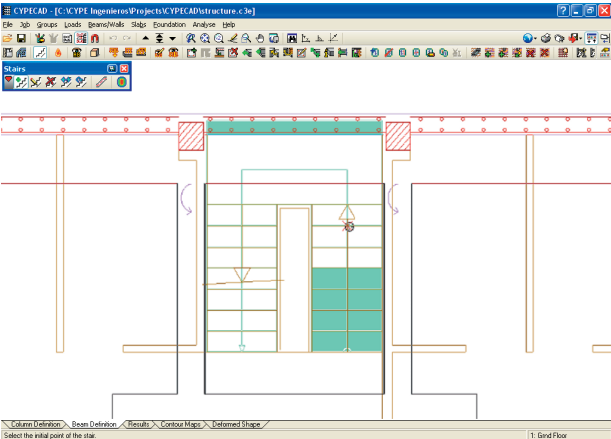


Fig. 8.10

Having positioned the start of the staircase with the left mouse button, the staircase rotates around the selected point.

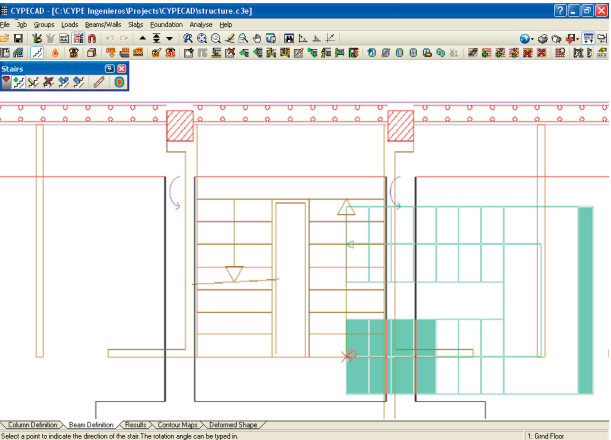


Fig. 8.11

Upon clicking, once again, on the left mouse button, at another point that will define its direction on the floor, the staircase will be introduced.

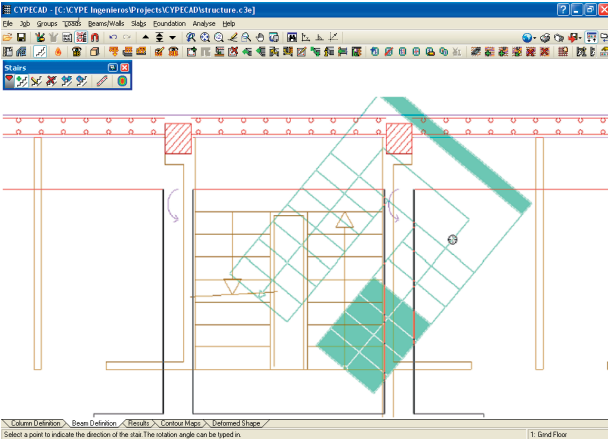


Fig. 8.12

Object snaps to an imported DXF or DWG template or introduction using the orthogonal mode, are very helpful tools when introducing the staircase.

Once the staircase has been introduced, the program allows for new staircases to be introduced identical to that previously introduced, whose references are numbered consecutively. Click on the right mouse button to finish introducing the staircases.

If a geometric error arises when introducing the staircase, the program will display an error message on screen.

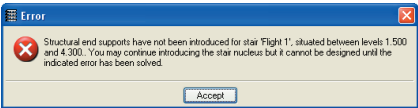


Fig. 8.13


The user can visualise the staircases introduced in the job using the 3D view (**Groups > Building 3D View** or **Floor 3D view**), allowing the user to check the data introduction, shape, if there is any interference with surrounding elements, etc.

8.3.1.4. Tools to facilitate staircase and flight definition

Copy staircase

There are structures which contain the same or similar staircases, therefore, it is very useful to be able to use the data of an already introduced staircase to create a new element.

For this reason, CYPECAD retains the data of the last introduced staircase whilst another job is not selected or the user exits the program.

Additionally, the program allows for, upon introducing a new staircase, to copy the data of a previously introduced staircase which is visible on the current floor. To do so, having clicked on the  icon from the **Stairs** floating menu, select the **Copy from** button.

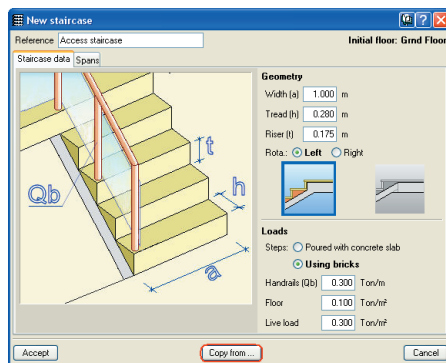




Fig. 8.14

Upon clicking on **Copy from** button, the current floor is displayed so the user can select the staircase to be copied using the left mouse button. The program will then return to the dialogue box where the user can check that the data of the selected staircase has been copied, including all that belonging to its flights. This data can then be modified, if necessary, to adapt it to the new staircase to be introduced.

An example highlighting the advantage of the **Copy from** option can be seen in the case of structures with symmetrical staircases. In this case, one of the staircases is defined and introduced. The second staircase is copied from the first and the ascending rotation is modified; with this the staircase is completely defined and all that is left to do is introduce it in the structure.

Predefined types of stair flights

When a new flight is introduced in a staircase (**Spans** tab, clicking on the  button), the Flight dialogue box appears and whereby pressing the  button, new types of flights can be added. Alternatively, instead of defining a new flight, one may be selected amongst those listed in the **Type** scrolling menu.

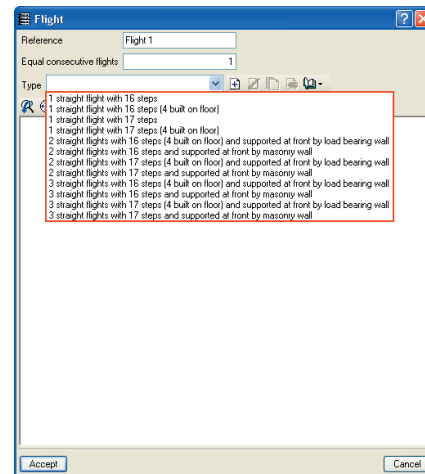



Fig. 8.15

The types displayed in the list are the predefined types as well as those defined by the user.

The type of flight can also be selected from the scrolling menu in the **New staircase** dialogue box.

Flight copy within the same staircase

A new flight can be introduced by copying a previously introduced flight belonging to the same staircase. This is done using the **Copy selected element and add it to the list** button  located at the top of the **New staircase** dialogue box.

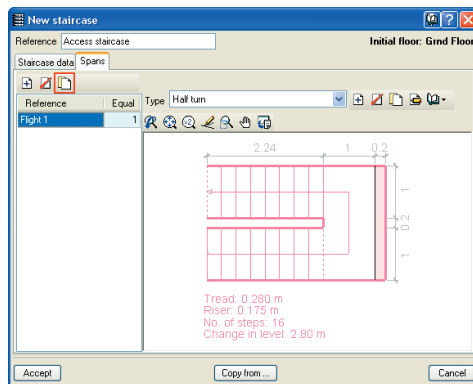




Fig. 8.16

When this order is executed, a new flight with the same properties as that which was selected is created at the top of the list. This implies that physically, the new flight is situated above those flights previously defined in the staircase.

This tool is very useful to add flights with similar properties to the staircase.

Copy and edit types of stair flights

To manage the flight types, the **Copy**  and **Edit**  tools can be used. These are located at the top right hand corner of the **New staircase** dialogue box and at the top of the **Flight** dialogue box.

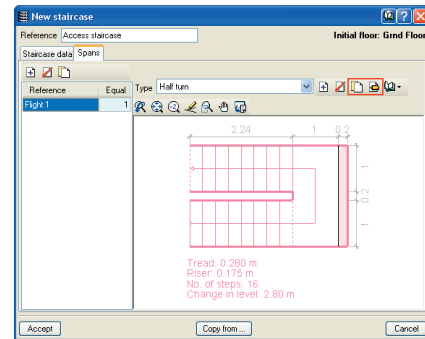




Fig. 8.17

The **Copy** button  repeats the properties of the type displayed in the drawing in another new type which the user must assign a new reference to. This allows for similar flight types to be created quickly.

The **Edit** button  allows for the selected type to be modified. If the type to be edited is being used in another stair span, the edit function is blocked to avoid an involuntary modification from being carried out on other flights. If the type is not used in any other span, there are no modification restrictions.

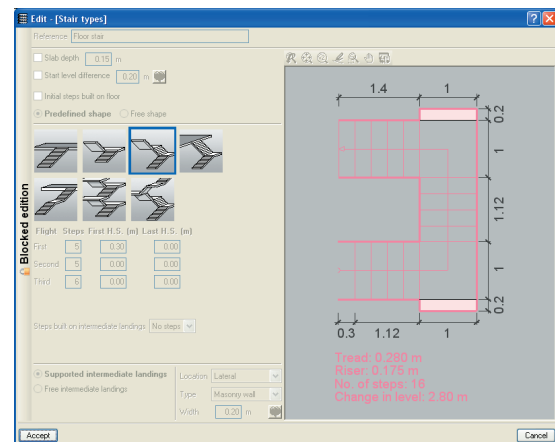


Fig. 8.18

If the user wishes to edit the type, and therefore, modify other stair types, the lateral button **Blocked edition** must be clicked on, so to unblock it.

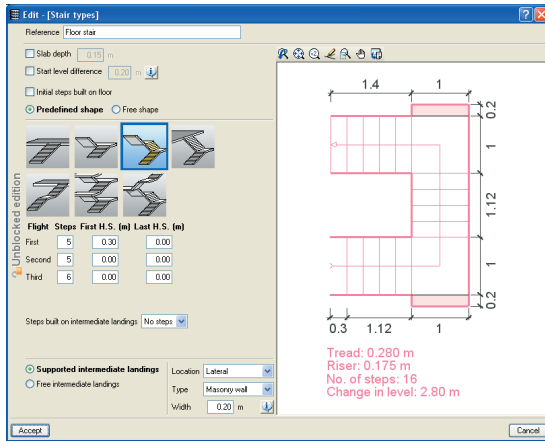



Fig. 8.19

Stair flight library

The types of stair flights can be saved in a library and subsequently used in other jobs. This way, jobs with similar stair flights can share data without the need of having to reintroduce them. The tool that manages the types library is activated by clicking on the **Library Manager** button , situated at the top right hand corner of the **Flight** dialogue box.

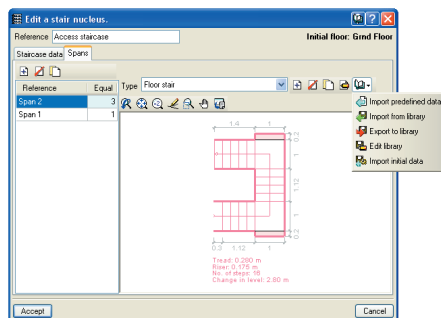





Fig. 8.20

The library can be managed in such a way that flight types defined in the job can be exported  to the user library so that these can then be imported  later on in other jobs. The use of the library management buttons is described by means of a tool tip when the cursor is placed over them.

8.3.2. Edit existing staircases

To edit the data of existing staircases in the job, click on the  button of the **Stairs** floating menu, followed by a click on the selected staircase. The program will display the **Edit staircase** dialogue box.

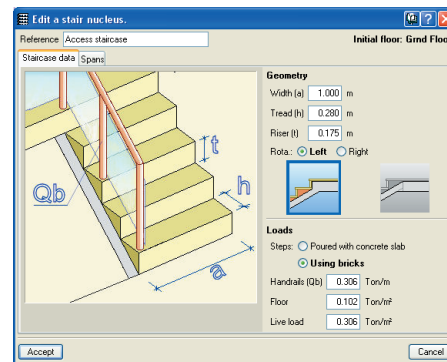




Fig. 8.21


8.3.3. Delete a staircase

To delete one or several staircases click on the  button from the **Stairs** floating menu and then click on the staircase to be deleted, or by means of a capture window, select the staircase or staircases to be eliminated.

8.3.4. Move a staircase

To move a staircase, click on the  button from the **Stairs** floating menu followed by a click with the left mouse button on the staircase to be moved, marking a point which will be taken as the reference point for its new position. Use of object snaps to DXF or DWG drawings may be useful when establishing the new position of the staircase.

8.3.5. Rotate a staircase

To rotate a staircase, click on the  button from the **Stairs** floating menu. Select the staircase to rotate using the left mouse button. The program will display the staircase pivoting around the start point of the flight in the current group. There are two ways to define its rotation: by selecting a point on screen defining its new direction or by clicking on the right mouse button and introducing the angle in the dialogue box that appears on screen.

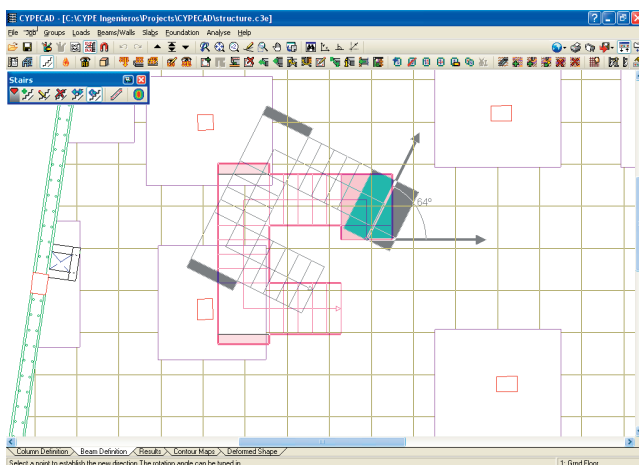



Fig. 8.22

8.4. Results, reports and drawings of the Stairs module


8.4.1. View staircase reinforcement details

Click on the  button from the **Stairs** floating menu, followed by a click with the left mouse button on the staircase to view its reinforcement. If it is the first time the staircase has been selected or modifications have been carried out since the last analysis, the program will proceed to design and reinforce it. Once the staircase has been designed, a window will appear displaying the reinforcement of each flight making up the staircase.

To view the reinforcement of other flights of the staircase, click on the scrolling menu situated at the top left hand corner of the window.

8.4.2. View forces and displacements using contour maps

The program designs the stairs in an isolated manner and by finite element method, taking into account the usual loadcases in the design of staircases: dead and live loads.



To view the forces and displacements of a staircase click on the  button from the **Stairs** floating menu, followed by a click with the left mouse button on the staircase whose results you wish to consult. If it is the first time the staircase is selected or if changes have been carried out since the last analysis, the program will design it. Once the staircase has been designed, a window displaying a three dimensional view of the stair flight will open. The displacements and forces of the selected flight can then be consulted.



8.4.3. Staircase design

Staircases can be designed using any of the following methods:

- When analysing the job:

When the job is being analysed, all the staircases are also designed, so that its reactions can be applied to the main structure. Therefore, the first thing the program analyses are the stairs.

- Using the  or  icons from the **Stairs** floating menu:

If the job has not been analysed, each staircase can be designed individually upon selecting the  or  buttons and clicking on the staircase for the first time.

If the user modifies the staircase once the job has been analysed and these changes affect the reactions of the staircase on the structure, a new analysis should be launched so that these modifications are taken into account. The program warns of this situation.

8.4.4. Reports

Added to the list of job reports, is the option to generate a report of all the staircases introduced in the job.

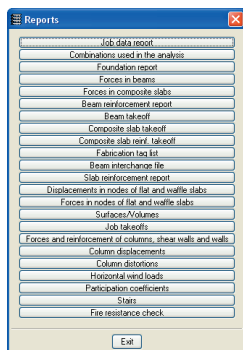


Fig. 8.23

The stair reports contain the general data of all the staircases of the job (materials and code used), common data (geometry, loading, etc.) and specific data (reactions on

the main structure, reinforcement, resultant forces in each span section) of each flight.

8.4.5. Stair drawings

Added to the list of job drawings (**Beam Definition, File > Print > Job drawings**) is the option to create stair detail drawings of each of the staircases of the job.

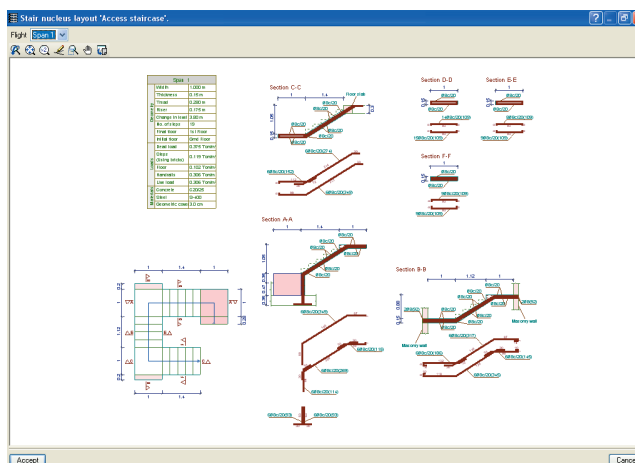


Fig. 8.24

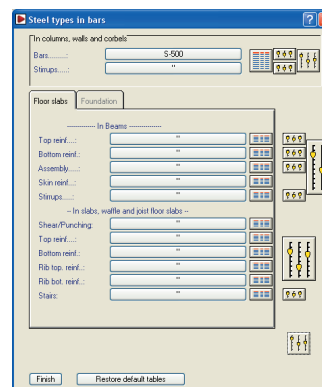



Fig. 8.25

The drawings display all the required information to define the stair layout: longitudinal and transverse sections, property tables of each span with its geometric data, loads and materials. The reinforcement takeoff tables are also included (per staircase, flight and total steel summaries).

8.5. Stair module configuration (materials, reinforcement tables and stair options)

The concrete used for the stairs is the same as that defined for floors and is defined in the **General data** dialogue box (**General data** found in the **Column Definition, Beam Definition** or **Results tab**).

By pressing on the **By position** button  of this dialogue box, another dialogue box opens (**Steel types in bars**) from which the steel to be used can be selected and the reinforcement tables to be used to reinforce the staircases can be managed.

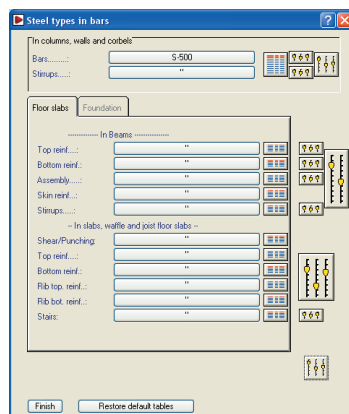


Fig. 8.26

Similarly, the Stair options button is added whereby the following options and data is configured:

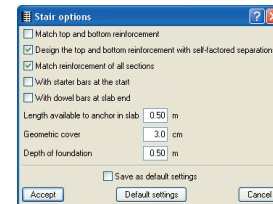


Fig. 8.27

Options

- **Match top and bottom reinforcement**

Having analysed the job, the program assigns the maximum reinforcement value to both reinforcements.

- **Design the top and bottom reinforcement with self factored separations**

Activating this options implies the program will lay the reinforcement in such a way that their separations will be multiples of each other. For example, one at 10 cm and the other every 10 cm or 20 cm.

- **Match reinforcement of all sections**

If the option is activated, the program assigns the maximum reinforcement value to all the sections (longitudinal and transverse) of the same flight of stairs. All the top reinforcement is matched and then the bottom.

- **With starter bars at the start**

- **With dowel bars at slab end**

Data

- **Length available to anchor in slab**

This value indicates the available room to anchor the anchorage bars of the first flight using a flat beam. If a greater anchorage length is required, the anchorage bars will be bent. If the width of the flat beam is greater than that indicated in this option, the width of the beam will prevail as the space available for anchorage.

- **Geometric cover (also referred to as nominal cover)**

The cover of the reinforcement closest to the external edges of the slabs should be indicated depending on the aggressiveness of the type of environment where the stairs are situated.

- **Foundation depth**

This value is used to define the anchorage of flight starts at foundation level when a foundation element does not exist below the stair start.

9. Steel, timber and aluminium structures

As well as steel beams and columns being introduced in the job, CYPECAD offers further options regarding non-concrete structures. These include the possibility of incorporating Integrated 3D structures to the job and steel joint design.

9.1. Integrated 3D structures

The option **Integrated 3D structures** has been included in the **Job** menu, which allows for three dimensional steel, timber or aluminium structures made up of nodes and bars with six degrees of freedom to be defined and connected within the project being edited at the time. This option allows for several 3D structures to be added to the same project.

Upon activating this option, a dialogue box appears containing the following options to introduce 3D structures.



Fig. 9.1

9.1.1. Integrated 3D structures menu

Upon selecting any of the options within the menu, the program displays the steps that are to be followed in the bottom part of the screen (below the Column Definition tab, Beam Definition tab...). The user also has help available within the dialogue box where the options appear.

New Structure

The connections that are going to be defined of the 3D structures have to be located and marked in the project.

The connections are the points where the integrated structure is 'attached' to the structure in CYPECAD and they should be situated at columns, and if no column is used as a connection point as in the case of beams or a floor slab, a start should previously be introduced at that point.

Once the option has been activated, upon approaching a column, it can be snapped to and having done so, a red point will appear with floating message. The message indicates the reference of the snapped column and the X and Y displacements with respect to the centre of the column as well as its elevation change with respect to the level of the floor group, whose default value is 0.

If the elevation of the connection compared to that of the group is to be modified, click on the right mouse button and the following dialogue box will appear:

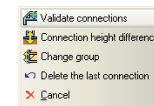


Fig. 9.2

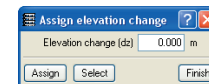


Fig. 9.3

The previous figure displays the options to modify or change the group if the 3D structure is connected to more than one group. Upon clicking on the left mouse button the connection will be introduced.

If all the connections have been defined, click on the right mouse button and select the option Validate connections, the connections will be saved and Metal 3D will open to introduce the structures.

Edit structure

Select the structure to be edited, upon approaching the cursor to a structure, it will be displayed in yellow and by clicking on it with the left mouse button, Metal 3D will open for its edition.

Delete structure

Works in a similar way as the previous option. Deletes the selected structure.

Move structure

The user can change the position of an integrated 3D structure by selecting this option, clicking on the structure and moving it to its new position.

Copy structure

This option allows users to create a new integrated 3D structure in the current group based on one which has previously been introduced in another group.

Rotate structure

This option allows users to rotate an integrated 3D structure. It can be rotated in two ways:

- **By indicating the numerical value of the angle**
 - Select the integrated 3D structure to rotate
 - Select the rotation point
 - Select the rotation angle
- **Graphically using reference points**
 - Select the integrated 3D structure to rotate
 - Select the rotation point
 - Select a reference point
 - Select any point of the straight line joining the rotation point with the future position of the selected reference point.

The rotation point and reference point do not have to be points of the structure. The program provides a provisional display of the integrated 3D structure as the mouse cursor is moved after the reference point has been selected and before selecting its future position.

If horizontal loads are present, such as wind loads, the program analyses and manages the rotation of these loads together with the rotation of the integrated 3D structure. The program will ask the user to confirm different actions, depending on whether the structure has or has not got any loads included in a loadcase used by other integrated 3D structures or in the job in CYPECAD:

- **The horizontal loads of the integrated 3D structure to be rotated act in a loadcase not used by other integrated 3D structures or by the job in CYPECAD**

In this case, the program asks the user to define how these loadcase combine with one another and with other horizontal load loadcases of the same type (wind, for example) that may exist in the structure of CYPECAD or in other integrated 3D structures. The program's default option is set so these loadcases are analysed as independent loadcases, i.e. they do not act simultaneously with the others.

- **The horizontal loads of the integrated 3D structure to be rotated act in a loadcase are also used by other integrated 3D structures or by the job in CYPECAD**

The program asks the user to choose between the following:

- Keep the horizontal loads of the integrated 3D structure to be rotated in their current loadcase and check whether they are valid later on.
- Apply the horizontal loads of the integrated 3D structure to be rotated in new loadcase combinations created automatically by the program.

In either case, the program then displays a dialogue box where the loadcases can be combined, similar to when the loads of an integrated 3D structure to be rotated belong to a loadcase not used by other integrated 3D structures or by the job in CYPECAD.

Align structure

This option allows users to align an integrated 3D structure. The aligning of a structure consists of a displacement and a rotation. An integrated 3D structure can be aligned by selecting two reference points and indicating its new position. The reference points do not necessarily have to be points of the structure. The procedure is as follows:

- Select the integrated 3D structure to align
- Select the first reference point
- Select the new position of the first reference point
- Select the second reference point
- Select any point of the straight line which will join the new positions of the reference points.

The program provides a provisional display of the integrated 3D structure as the mouse cursor is moved after the reference point has been selected and before selecting its future position.

If horizontal loads are present, these are dealt with in the same manner as in the case of the Rotate structure option (described in more detail above).

New connection

This option allows for connections to be added to a previously introduced 3D structure. To do so select the structure, followed by the connections on the columns or starts. Once all the new connections have been introduced, click

on the right mouse button and select the option Add connections.

Delete connection

Allows to delete connections of a previously introduced 3D structure. To do so, select the connections to be deleted. When connections are deleted, they are transformed into nodes of the 3D structure, which should be eliminated if they are not required.

Move connection

Allows for previously introduced 3D structure connections to be moved. To do so, select the connection to move and click on its new position. Be aware that when a connection is moved, a node will be generated at its original position.

Connection information

Using this option, information of the connection is displayed: absolute elevation and elevation change with respect to the floor group to which it connects.

Assign elevation change

Upon selecting this option, a dialogue box appears in which the elevation change to be assigned to other connections is to be introduced. Using the Select button, the same elevation change assigned to another connection can be assigned to other connections.

List of integrated 3D structures

This option allows to view and edit all the 3D structures without the need of having to find them in the CYPECAD job. Upon activating it, a new dialogue box opens displaying a list with the structures integrated in the job. To the right of the dialogue, a representation of the selected structure can be seen.

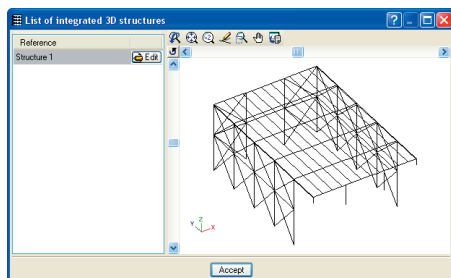


Fig. 9.4

9.1.2. Metal 3D program

External fixity

The External fixity option in Metal 3D allows to specify if the bars reaching the introduced connections in CYPECAD are 'fixed', 'pinned' or '?'.

Introduction of load states

Loadcases are defined using the option Additional loadcases (special loads) in the General Data dialogue box of CYPECAD. If a new loadcase is to be defined, the user must return to CYPECAD to define it.

Predesign the structure

It is necessary for the structure to be predesigned before carrying out a joint analysis of the CYPECAD project. If this is not done, the force results could vary greatly due to considerable changes in the inertias of the bars of the 3D structures compared to what was obtained in a previous analysis, if they are not correctly predesigned.

- Analyse the isolated structure. This option allows to carry out a quick analysis of the forces in the isolated structure. This analysis will allow to carry out a fast design of the structure. In the analysis, any interactions with CYPECAD are not taken into account, this is why once

the preliminary design has been carried out, the whole project has to be analysed from the Beam Definition tab in CYPECAD.

- Design with current forces and using all the sections of the series. This and the following options allow to design the structure using the forces obtained in the option Analyse the isolated structure or with those obtained from analysing the entire CYPECAD project. These allow to consult the connection reactions.

Once the structure has been predesigned, the user returns to CYPECAD where a global analysis of the project will be carried out. Then, once it has been analysed, the user is to return to Metal 3D to verify the bars using the forces obtained from the global analysis of both structures. In those cases where the inertias of the bars vary greatly, the structure will have to be reanalysed using CYPECAD.

Baseplates

- **Generate.** This option generates the baseplates of the connections defined in the **Beam Definition** tab of CYPECAD on concrete columns or starts, which can be steel or concrete.
- **Edit.** Within the baseplate dialogue box which appears during the edition, the option **Design** can be found. This option satisfies the parameters indicated in the baseplate option within the **General data** dialogue box in CYPECAD, which will propose a modifiable solution.

9.2. Joint design

The program incorporates the analysis and design of connections for rolled and welded steel I sections for various design codes (please consult our website for the available design codes. If the design code with which the job is

being designed is not yet available with the Joints module, the user can opt to copy the job and design them using a different code).

9.2.1. Types of joint design

The program offers several options for connection design: the Joints modules.

The following Joints modules are currently available with the program:

- Joints I. Welded. Warehouses with rolled and welded steel I sections
- Joints II. Bolted. Warehouses with rolled and welded steel I sections
- Joints III. Welded. Building frames with rolled and welded steel I sections
- Joints IV. Bolted. Building frames with rolled and welded steel I sections
- Joints V. Flat trusses with hollow structural steel sections

New modules will be coming soon; check our website for new implementations.

9.2.2. Joint Design

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

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

Welded, e.g.:

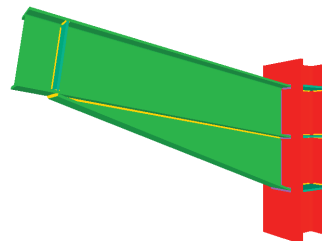


Fig. 9.5



Fig. 9.6

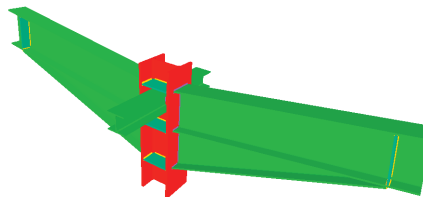


Fig. 9.7

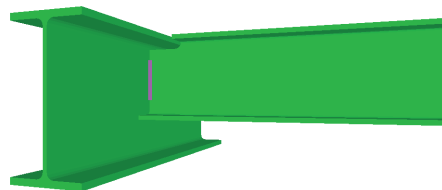


Fig. 9.8

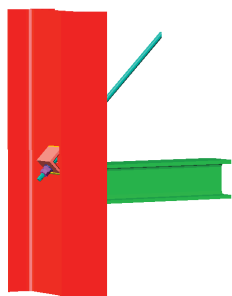


Fig. 9.9

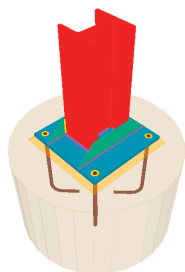


Fig. 9.10

For a more complete list of available welded connections, please consult the corresponding website at www.cypecad.en.cype.com/joints_welded.htm and www.cypecad.en.cype.com/joints_welded_building.htm

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

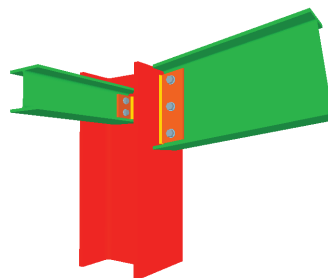


Fig. 9.12

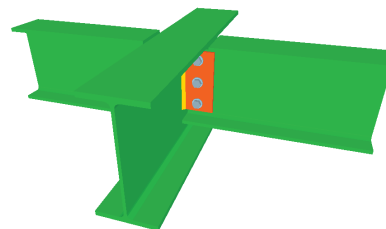


Fig. 9.13



Fig. 9.14

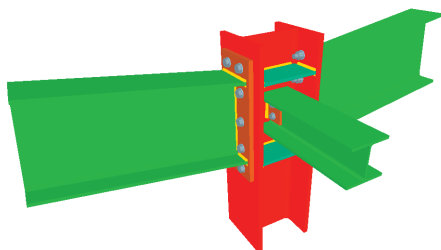


Fig. 9.11

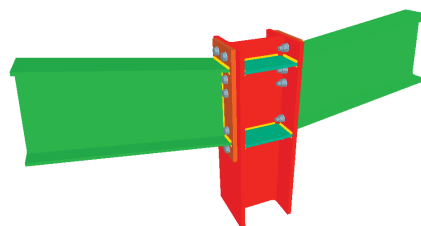


Fig. 9.15

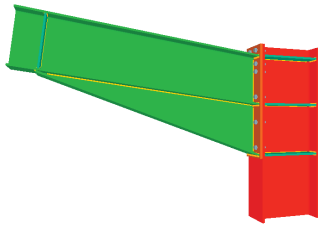


Fig. 9.16

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

www.cypecad.en.cype.com/joints_bolted.htm

www.cypecad.en.cype.com/joints_bolted_building.htm

The program will design the required weld thicknesses and incorporates stiffeners if they are required for the correct transmission of forces. Additionally, in the case of bolted connections, the optimum layout of prestressed or ordinary bolts is provided.

- **For flat trusses with hollow structural sections:**

The hollow structural sections must consist of welded sections with the additional condition that the bar acting as the truss chord must be a continuous bar, in the case of an intermediate node. The hollow structural sections are designed so to meet all the requirements indicated in the selected code. The program represents the edge preparation of the tube ends to be able to weld the sections correctly.

e.g.:

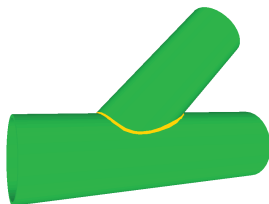


Fig. 9.17



Fig. 9.18

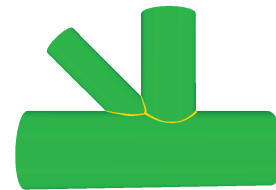


Fig. 9.19

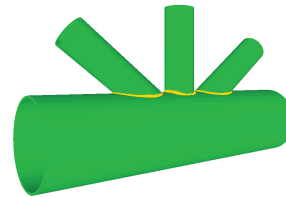


Fig. 9.20

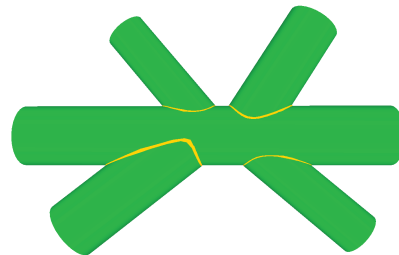


Fig. 9.21



Fig. 9.22

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

www.cypecad.en.cype.com/joints_flat_trusses_hollow_structural_sections.htm

9.2.3. Consulting the designed joints

The joints can be designed at the same time as the general analysis (by marking the Design joints box in the **Analysis** dialogue box).

To then consult the designed joints, click on **Joints > Consult**. Upon activating this option, all the joints which have been designed by the program will be marked with a green circle. Those that have not been resolved will be marked with a red circle. Joints which are partially resolved i.e. containing connections that have been designed and others that have not will be marked in an orange circle.

If the mouse cursor is brought close to a node in which there is a designed joint, the joint will be highlighted in blue and an information window will appear indicating the type of connections associated to that node that are present. By then clicking on the joint, the detail drawings of the connections associated to that node are displayed.



Fig. 9.23

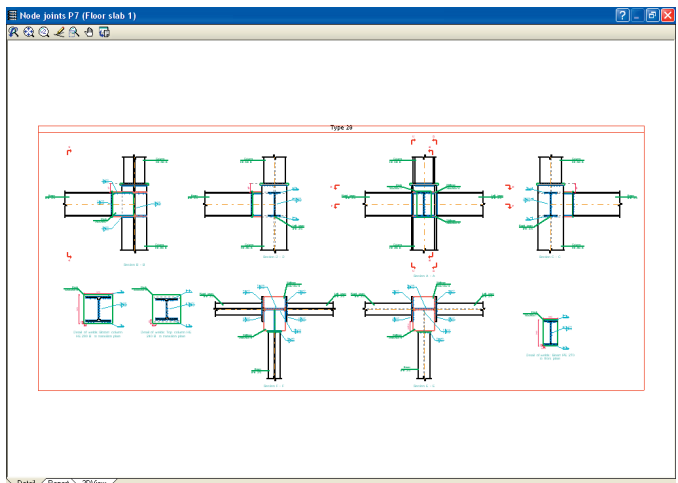


Fig. 9.24

If the cursor is brought close to a node in which there are connections that have not been designed but belong to one of the types recognised by the program, an information window is displayed providing the reasons why the joint has not been designed.

9.2.4. Reasons why a joint has not been designed

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

a. Section fixed to the web of another section

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

b. Interference between sections and stiffeners

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

c. Thickness of the elements

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

d. Orthogonal elements

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

e. Angle

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

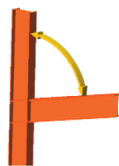


Fig. 9.25

9.3. Export to other programs

CYPECAD allows for the designed steel structure to be exported so it may be viewed and modified in other programs. The Export options are located in **File>Export**. The programs include Tekla Structures and TecnoMETAL 4D.

Additionally, the option exists to export in CSI/2 format. CIS/2 (CIMsteel Integration Standards) is a **standard interchange format** to facilitate data flow between applications which intervene in the analysis, design and fabrication process of **steel structures**. CYPECAD and the Integrated 3D structures of CYPECAD can export the introduced and designed structure in CIS/2 format, advised by the NIST (National Institute of Standards and Technology) so it may later be use in programs such as Graitec Advance Steel.

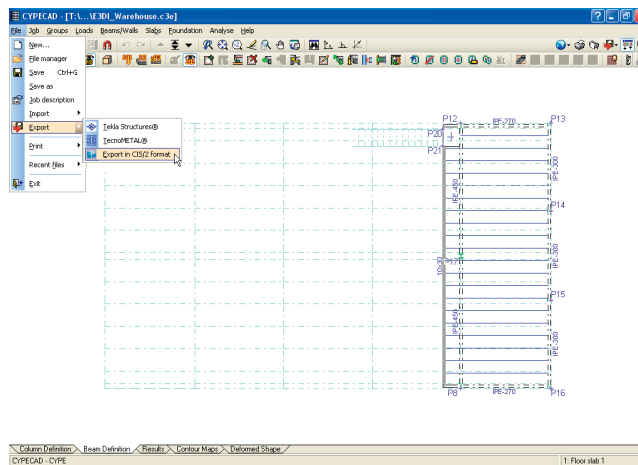


Fig. 9.26

External loads acting on the structure can be introduced in CYPECAD or the Integrated 3D structures, so to obtain its optimum design in accordance with the selected code. Additionally, the construction details of bolted and welded connections of rolled and welded I sections that are generated can also be exported to other programs and obtain a 3D model of the whole in that program.

There are two options when exporting:

- **Only geometry and properties of nodes and bars.**

Exports the geometry and steel sections recognised by the program it is being exported to. The structure does not have to be analysed or designed before it is exported. It does not include the export of joints and so these do not have to be designed before exporting either.

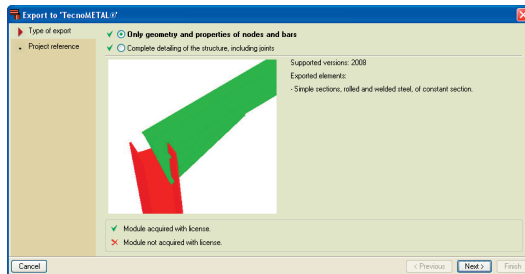


Fig. 9.27

- **Complete detailing of the structure, including joints.**

Exports the geometry of the job, the steel sections recognised by the program it is being exported to and the joint detailing corresponding to the steel elements. The job and joints must be analysed before exporting.

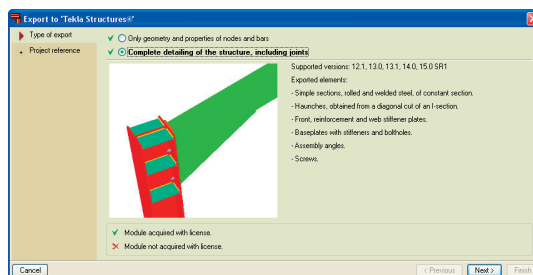


Fig. 9.28

Please consult our web-site for the minimum program requirements to use these options.

10. Loads

10.1. Loads to be defined in General data

10.1.1. Actions

10.1.1.1. Wind loading

Wind action can be specified in the General data dialogue box by selecting the **With wind box** in the **Loads** section. When selected, a new dialogue box will appear where the code to follow and properties can be specified. The data to be entered will vary depending on the chosen code, however, all require the tributary widths and load coefficients in the X and Y directions to establish the correct wind coefficient. The analysis may be carried out with or without second order effects. If the code that is sought is not available, the user has the option to use a **Generic Code**.

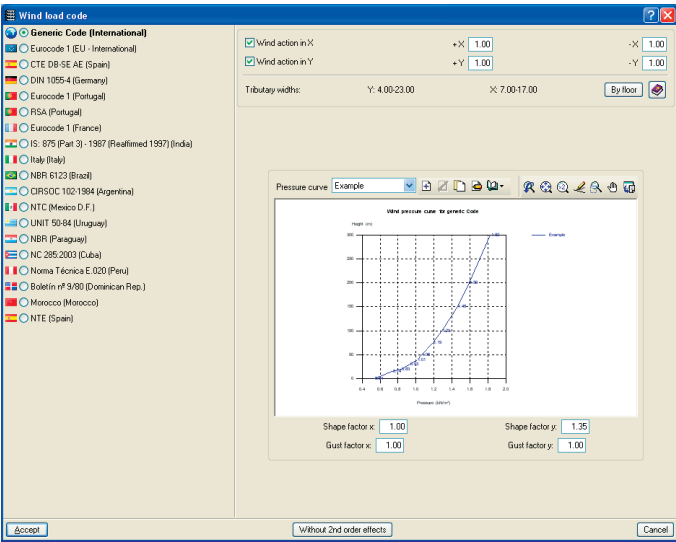


Fig. 10.1

10.1.1.2. Seismic loading

Earthquake loading can be specified by selecting the **With seismic loading** box. The dialogue box that appears is similar to that of the wind action whereby a list of codes is displayed on the left hand side of the box and their specific data, which will vary depending on the selected code, is entered on the right. A generic code is also available, only here, it is presented as a **Modal spectral analysis** or by specifying the load **Coefficients per floor** that have to be applied. The option to analyse with or without second order effects is also available.

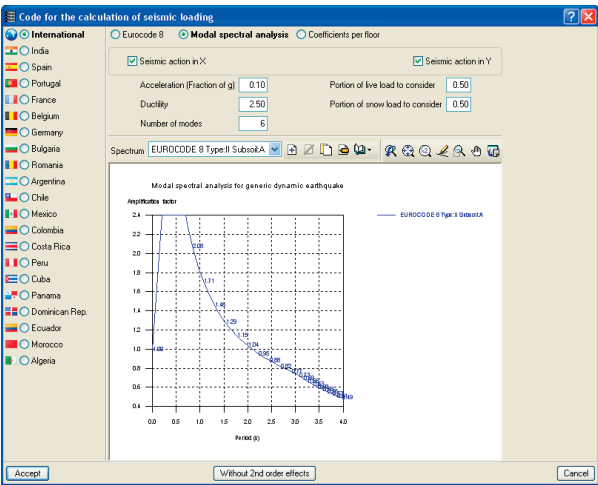


Fig. 10.2

10.1.1.3. Fire resistance

A fire resistance check is available, however this is undertaken in accordance with the Spanish code (CTE DBSE-SI). When selected, the **Fire resistance check: General data** dialogue box appears where the properties of the coatings to be applied to the construction elements are indicated.

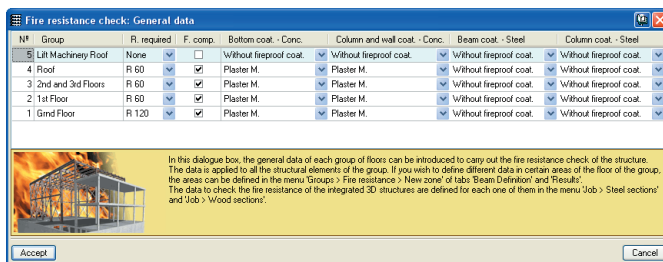


Fig. 10.3

10.1.2. Additional loadcases

Additional loadcases may be defined by clicking the option in the General data dialogue box. The option displays all the loadcases of the job and allows for them to be edited as well as allowing for other loadcases to be added (dead, live, wind, earthquake and snow loads) and establish their compatibility conditions.

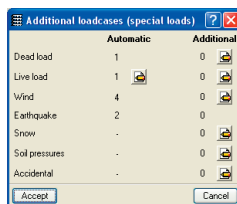


Fig. 10.4

To create a new loadcase, click on the **Edit** icon to the right of the name of the loadcase. A dialogue box opens which will display any additional loadcases that have been defined. A new loadcase can be added by selecting the **New additional loadcase** button in the top left hand corner.

Another dialogue box will appear where the reference and description can be defined. An extra option appears: **With different load dispositions**. By activating this option, the compatibility of the loadcases can be defined (compatible, incompatible or simultaneous). This is useful when defining alternating loads, mobile loads etc.

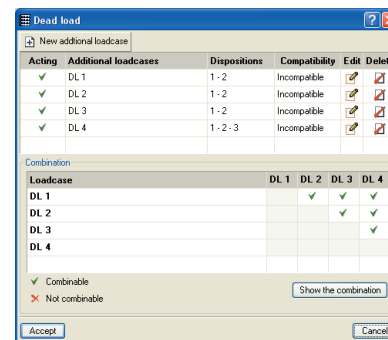


Fig. 10.5

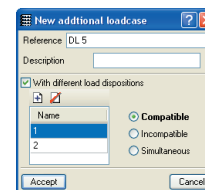


Fig. 10.6

What are the different load dispositions?

The load dispositions are a group of special loads which belong to the same loadcase. These can be:

- **Compatible:** The load groups can be applied on their own or combined with other load groups belonging to the same loadcase.
- **Incompatible:** The load groups do not combine with other load groups of the same loadcase (for example in

the case of wind loading where the wind is blowing in a given direction and will not be combined with wind blowing in another direction).

- **Simultaneous:** The combinations of the defined additional dispositions are generated taking into account that they all act simultaneously. This type can be used when wishing to define a group of loadcases divided into load groups and so make them easier to edit, visualise, etc.

Once the dialogue box has been accepted, and having defined at least two additional loadcases, the program will display a table at the bottom of the dialogue box where the combination between loadcases of the same nature can be combined amongst themselves.

10.1.3. Limit states

Once this option has been selected, the **Limit states** dialogue box appears allowing the user to define the execution control, use category and snow level. By clicking on the pencil and question mark icon on the right, the combination generated by the program can be consulted and where the partial safety coefficients γ and combination coefficients γ are indicated.

10.1.3.1. Combination configuration for each limit state

This option can be found in the bottom part of the **Limit states** dialogue box and allows to specify the limit states for concrete, foundations, cold formed steel, rolled steel, and wood. For any of these, the option **Project situations introduced by the user** can be selected. This option allows the user to specify the particular project combinations to be used situations to in the calculations corresponding to the structural elements of the job.

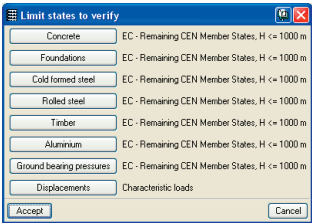


Fig. 10.7

Project situations introduced by the user

This option allows to configure particular project situations. It is carried out in the same manner for all the structural element types (Concrete, Foundations, Steel, Timber, Ground bearing pressure and Displacements).

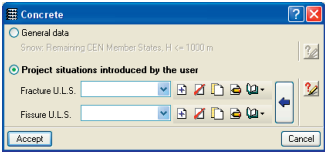


Fig. 10.8

The following options are available to define the project situations introduced by the user:



Create new situations



Delete previously introduced situations



Copy the selected situation



Edit the selected situation



Library manager to use the situations in other jobs



Create based on data of situations stated in the selected code

- **Create new situations**

By clicking on the  button, the **Create – [Situations]** window will open:

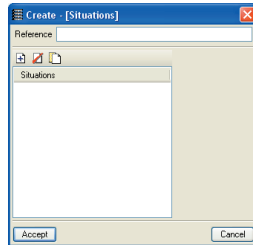



Fig. 10.9

A reference must be introduced to indicate the situation that is going to be defined. Having done so, click on the **Add new element to the list** button  and the **Combination coefficients** window will open.

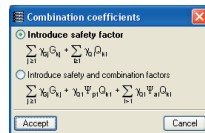


Fig. 10.10

Here, the user can define the combinations in two different ways: by safety factors or by safety and combination factors.

For the first case, the data to introduce is as follows:

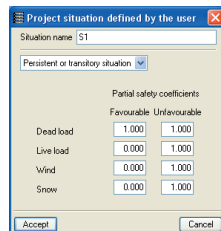


Fig. 10.11

For the second case:

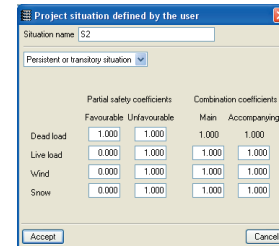



Fig. 10.12

- **Edit situations**

If this option is selected , dialogue boxes similar to those displayed when creating the situations appear:

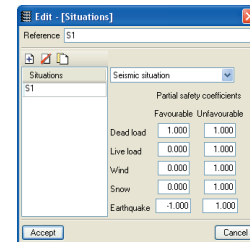



Fig. 10.13

All the previously defined situation parameters can be edited in this dialogue box.

- **Create based on the code situations**

By selecting this option ( button), code project situations of the selected material can be imported and modified to the user's needs. This reduces the possibility of errors occurring when introducing the coefficients.

Warning

By offering the user the option to create the project situations, the user is given a certain degree of freedom for the analysis of the job. Therefore special attention must be paid to not create project situations which may imply that the fi-

nal results have an applied safety factor less than that required by the selected code.

10.2. Definable live loading in groups

10.2.1. Group loads

When introducing new groups (**Column Definition tab, Introduction > Floors/Groups**), a live load must be introduced which will be applied on the floors that will subsequently be defined. These loads will not be taken into at openings.

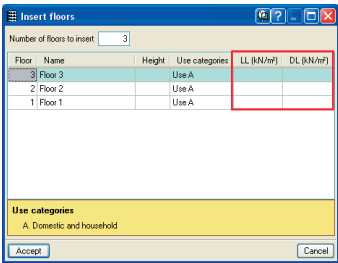


Fig. 10.14

10.2.2. Loads at foundation level

Loads can also be applied at foundation level by selecting the option **Introduction > Floors/Groups > Edit groups** from the **Column Definition** tab. This load will only be taken into account by mat foundations.

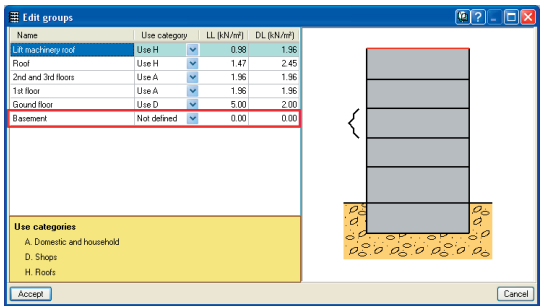


Fig. 10.15

If there is a mat foundation and no loads have been defined for the foundation group (0), when launching the analysis, the program will emit a warning indicating that no loads have been introduced. The loads may be established at that point without having to return to the Column Definition tab. Once the loads dialogue has been accepted the job analysis will commence.

10.3. Loads Menu

10.3.1. Surface loads on slabs

Within the **Loads** menu in the **Beam Definition** tab the option **Surface loads on slabs** may be found.

This option allows for a surface load to be applied on a slab by means of a single click.

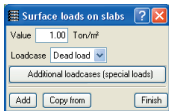


Fig. 10.16

Once the load has been selected, a dialogue box will appear where the value of the load and loadcase to which it belongs must be defined. The button **Copy from** may be used to copy the load from another existing load.

By clicking on **Add** and displacing the cursor over a slab will cause the slab to be highlighted in yellow. By then clicking on the mouse button, the load will be applied on the highlighted slab.

10.3.2. Automatic introduction of line loads on beams

The option **Line loads on beams** is located in the **Loads** menu in the **Column Definition** tab. Upon selecting this option, a dialogue box will appear which, once accepted, will allow for line loads to be introduced along the axis of the selected beams. The introduction may be carried out on a beam to beam basis or by means of a capture window.

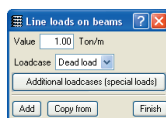


Fig. 10.17

10.4. Drawings and reports

The dead and live loads used in the analysis for each group can be displayed in the job drawings. These loads appear in the Special loads drawing (**File > Print > Job drawings > Load distribution, Configure** button, **Special loads**). Loads do not appear by default in any other plans. When selecting the Configure button in the sequence described above, a list of other default drawings will be displayed as well as the **Special loads** drawing. These may be edited (**Edit, Others** tab, **Live and dead loads**) to display the applied loads.

11. Foundations

11.1. Baseplates

The program designs baseplates regardless of the layout of the steel columns and for all types of materials.

The floating menu containing the options relative to baseplates opens by clicking on **Foundations > Baseplates** within the **Beam Definition** or **Results** tab.

The menu contains the following options:



Fig. 11.1

- **Generate**

To be used when there are steel columns or steel bases without a baseplate. The option generates all the baseplates at the base of steel columns.

- **Delete**

Deletes a previously introduced baseplate.

- **Edit**

Allows for the baseplate to be designed automatically and verify or consult the layout and arrangement of the baseplate.

- **Match**

Copies the properties of one baseplate to another, once the foundations have been designed.

Using this option the baseplate design proposed by the program can be modified. The program proposes the optimum baseplate for each individual column, however, it may be useful to reduce the number of different baseplates used in the job and hence providing a more uniform result.

Once a baseplate has been copied onto another baseplate, the program carries out a check to verify that it does not fail. If any of the conditions are not complied with, the baseplate is displayed in red.

11.2. Foundations

11.2.1. Foundation properties

The number of columns, walls or shear walls bearing on a foundation or pile cap is unlimited. The analysis used by the program is that used for rigid foundations. The user must ensure this is the case.

Columns, walls and shear walls may be placed in any position relative to the axis of the foundation.

11.2.2. Foundation floating menu

The floating menu containing the foundation options is opened by clicking on **Foundations > Foundation elements** in the **Beam Definition** or **Results** tab.



Fig. 11.2

The menu contains the following options:

- **New**

Selection of the type of pad footing: reinforced concrete footing, mass concrete footing or pile cap.

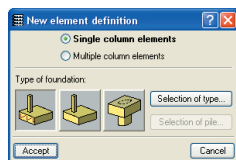


Fig. 11.3

If the pile cap option is selected, the type of pile cap must be selected. This will establish the shape of the cap and the number of piles it will rest on. The piles must then be defined. This is done by selecting the **Selection of pile**. Here, a description of the pile can be introduced as well as its shape (circular or square), its bearing capacity (unfactored) and separation between pile axes if there is more than one pile for that cap.

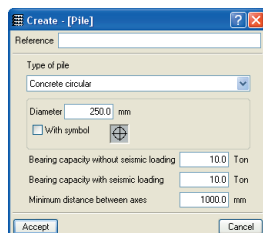


Fig. 11.4

Single column elements: having selected this option, and the type of foundation to be introduced, a column approached by the mouse cursor will be highlighted in yellow.

When the cursor is brought close to the column displayed in yellow, a square will appear containing a smaller square whose position will vary depending on where the mouse cursor is placed relative to the column. The smaller square indicates the location the column will hold relative to that of the footing if clicked on at that position.

The default value of the angle of the footing will be the same as that of the column bearing on it.

Multiple column elements (or combined footing): by selecting this option, various columns or shear walls

can be selected to bear on the same pad footing (click, using the left mouse button, on the columns and right click to finish the selection). The cursor will then adopt the same shape as in the case of a pad footing with a single column and the insertion point of the footing will be displayed (a red target symbol).

The insertion point used by the program is established by dividing the sum of all the coordinates in each direction and dividing by the number of columns in each direction, hence obtaining a mean x and y coordinate. This position may later be modified by the user using the Move option.

The program uses the angle of the first column introduced as the default angle of the combined pad footing.

• Edit

Allows for the foundation element to be designed automatically or verified if the user wishes to carry out any modifications on that proposed by the program or use a specific reinforcement layout and/or dimensions.

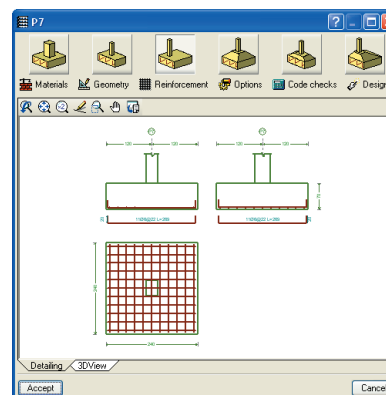


Fig. 11.5

• Delete

Deletes a previously introduced pad footing or pile cap.

- **Move**

Allows for the position of the foundation elements to be modified.

- **Rotate**

Rotates the pad footing or pile cap about its centre axis. The possibilities are as follows:

- Manual introduction of the angle
- Two point introduction. Select two points on the screen defining the direction vector and click on the foundation element.
- Angle of the column. Click on the foundation element followed by the column bearing on the foundation whose angle is to be adopted. In the case of several columns bearing on the same foundation, the angle taken by the program is that of the first column introduced.

- **Join**

Allows for several footings to be joined and combined to make a single footing. Not applicable to pile caps.

The selection of the footings is carried out using the left mouse button and concluded with a right click.

- **Match**

Matches the geometry, type and reinforcement of the pad footings.

Click on the master pad footing from which the properties are copied, followed by the footing onto which the properties are to be assigned.

- **Information**

Displays information on the materials used (including the allowable bearing pressure of the soil in the case of footings), the geometry and analysis results (with warnings and design errors).

Foundation element P17	
Materials	
Concrete:	C25/30
Footing reinforcement:	S-500
Pile cap reinforcement:	S-500
Allowable bearing pressure in persistent situations:	3.06 kgf/cm ²
Allowable bearing pressure in accidental situations:	4.59 kgf/cm ²
Geometry	
Eccentric rectangular footing	
Initial width X:	100.0 cm
Initial width Y:	100.0 cm
Final width X:	100.0 cm
Final width Y:	100.0 cm
Footing width X:	200.0 cm
Footing width Y:	200.0 cm
Depth:	60.0 cm
Reinforcement	
X:	10Ø12@20
Y:	10Ø12@20
Analysis	
Ground bearing pressures	
Mean stress in persistent situations:	1.17 kgf/cm ²
Maximum stress in persistent situations:	1.82 kgf/cm ²
Analysis forces	
Moment X/Y:	12.64 / 11.99 Ton m
Shear X/Y:	9.41 / 9.58 Ton

Fig. 11.6

11.2.3. Further options for footings

If the user has acquired the **Advanced design of surface foundations** module, then further options are available:

- **Polygonal limits for footings**

Using this option, the user can define limits or boundaries which cannot be invaded by reinforced concrete or mass concrete footings. Examples of these limits or boundaries may include property limits or zones reserved for other uses such as deposits, lift shafts etc. These limits may also be defined for the program to define irregular polygonal footings for any other reason, regardless of whether there are real physical limits.

To define the limits, polylines have to be introduced, which the program then uses to automatically trim the footings which pass or invade the established limits in that group. A footing can be trimmed by more than one limit. The fraction of the footing in which the column starts or columns remain is conserved and is, therefore, what is considered in the analysis. During the design of the footings, the program does not allow the established limits to be exceeded.

The limits are introduced in CYPECAD using the **Limits for polygonal footings** option located in the **Foundations** menu within the Column definition or Results tab.

The limits may be introduced manually by defining each edge or by snapping to a DXF/DWG template. In CYPECAD, the limits that have been introduced in a group, only affect the footings of that group.



Fig 11.7

Once the polylines defining the limits have been introduced, the user may delete or move them, insert a new vertex or eliminate a section of the polyline.

These polylines have certain logical restrictions regarding their position:

- They cannot intersect a column
- They must cut the outlines of the footings before and after their design
- Footings with multiple columns cannot be trimmed in such a way as to leave one column isolated from the rest of the columns or column starts sharing the footing.

If a polyline does not comply with one of these conditions, the program will not consider the limit generated by the polyline.

The defined limits have no effect on strap and tie beams, pile caps, tapered footings or strip footings. They only affect reinforced concrete and mass concrete footings of constant depth.

- **Loads applied on footings, pile caps and, strap and tie beams**

Point, line and surface loads can be applied on footings, pile caps and, strap and tie beams, and are taken into account in the design. Using this module, CY-

PECAD also recognises the loads transmitted by the stair supports resting on the aforementioned foundation elements.

CYPECAD assumes that the soil below the strap and tie beams does not intervene in the force equilibrium or distribution to which they are submitted. Therefore, strap and tie beams are designed to also support any loads that may be present acting on them and will transmit the forces to the footings or pile caps at their ends, or other strap and tie beams they may intersect.

11.2.4. Wall foundations (Strip footings)

When introducing a wall (option **Beams/Walls > Define wall** within the **Beam Definition** tab) regardless of which type of wall is selected, a dialogue box will appear where the properties of the new wall are to be defined as well as its foundation. The wall may be introduced with or without external fixity. In the case of with external fixity, the wall may be considered to be **Fixed** or to bear on a **Strip footing** (left or right overhang, or both).

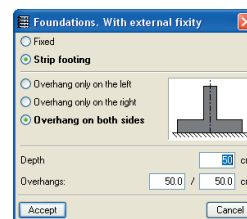


Fig. 11.8

Strip footings are designed automatically.

Strap beams will be required to connect the strip footing to internal pad footings in order to centre the moment that arises due to a non-centred strip footing (for example: a footing with an overhang only on one side).

If a wall is introduced without a footing, it can easily be assigned one later using the **Foundation elements** option from the **Foundations** menu explained above.

11.2.5. Foundation options

Various foundation options can be found having clicked on the **Options** button in the **General data** window. Some of those options are described below.

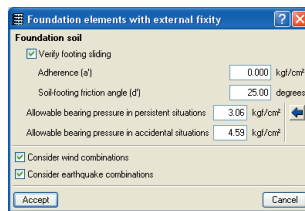


Fig. 11.9

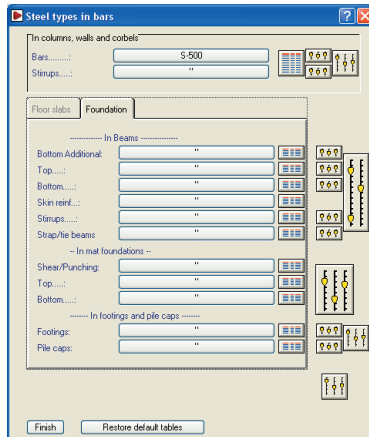


Fig. 11.10

- **Reduction of minimum mechanical ratio**

In the case where the selected code allows for this reduction, the minimum mechanical ratio required due to bending can be applied. This reduction is always applied during the verification.

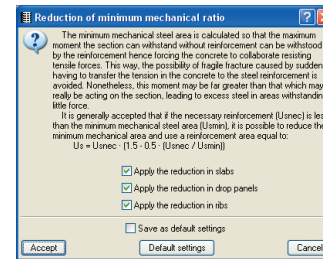


Fig. 11.11

- **Allow uplift in the footing**

If this option is deactivated, uplift will not be allowed in the footings, that is, the resultant force must pass through the central nucleus of the footing.

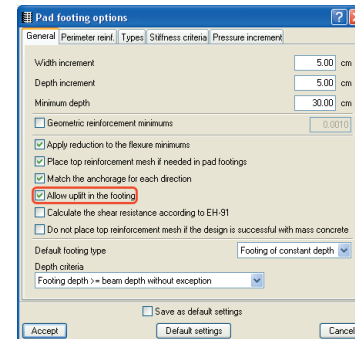


Fig. 11.12

- **Do not place top reinforcement mesh if the design is successful with mass concrete**

When the design is being carried out, a top reinforcement mesh will be placed if the mass concrete section fails for flexure, i.e. a tensile resistance of the concrete is assumed.

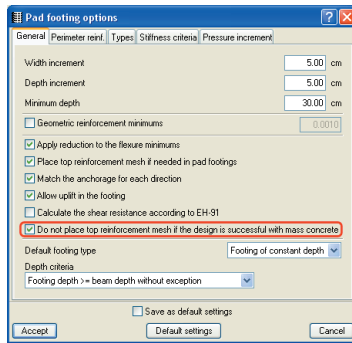


Fig. 11.13

• Balancing type

This option allows to select the type of pressure distribution acting under the footing as a response of the soil. (A) Rectangular pressure distribution under the footing: If this option is selected, the program assumes that the balancing of the strap beam is perfect, resulting in a rectangular soil pressure distribution. (B) Trapezoidal pressure distribution under the footing: The program admits that the balancing of the strap beam is not perfect, that is, a certain rotation of the footing occurs, which leads to a trapezoidal pressure distribution and therefore, a relaxation of the forces acting on the beam. This displacement of the resultant force produces a reduction of the moment on the beam of approximately 10%.

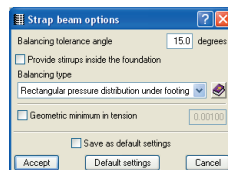


Fig. 11.14

- **Maximum ratio of maximum span-depth.** It is the maximum permissible ratio between the maximum span amongst all the columns of the footing and the footing depth to be able to consider the footing as rigid in its design.
- **Maximum ratio of maximum overhang-depth.** It is the maximum permissible ratio between the maximum overhang and the depth in the design.

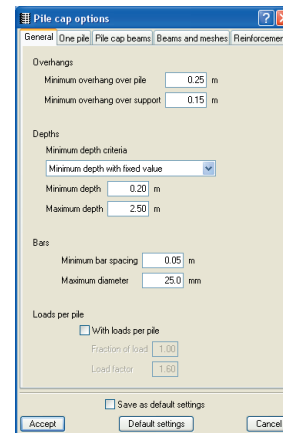


Fig. 11.15

11.3. Strap and tie beams

11.3.1. General information on strap and tie beams

Rigid bending is not considered in the case of tie beams or strip footings however it is in the case of strap beams which balance the load and act as tie beams (they absorb the moments produced at the base of the columns, due to the eccentricity of the force normal to the columns with respect to the axes of the footings and transferred to the axes of the footing).

The program does not consider the beams to be supported by the ground.

The program allows the user to assign the 'balancing' to one or both ends. For example, in the case of a beam running from a centred pad footing (with respect to the column) to an eccentric pad footing, the user can balance the end reaching the eccentric footing and leave the other as a tie beam.

11.3.2. Strap and tie beam floating menu

The strap and tie beam floating menu which opens upon clicking on **Foundations > Strap and tie beams** in the **Beam Definition** and **Results** tabs contains the various options related to the beams:



Fig. 11.16

- **New beam**

When introducing a new beam, the user can choose between:

- Tie beam
- Strap beam. These balance the moments transmitted by the columns, shear walls and walls to the foundations relative to their axes. Any type of footing can be balanced.
- Beam with automatic balancing at ends. Using this option, the user can introduce a beam between two previously introduced footings and the program will automatically balance the ends which require balancing because the footing they reach is eccentric with respect to the supported element(s) (e.g. edge footing, corner footing, etc.).

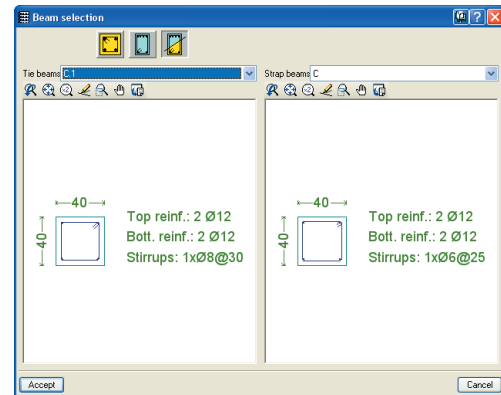


Fig. 11.17

- **Edit beam design**

After the general analysis, the result obtained for the beam can be modified. The footing can be designed independently and later on, design the beam.

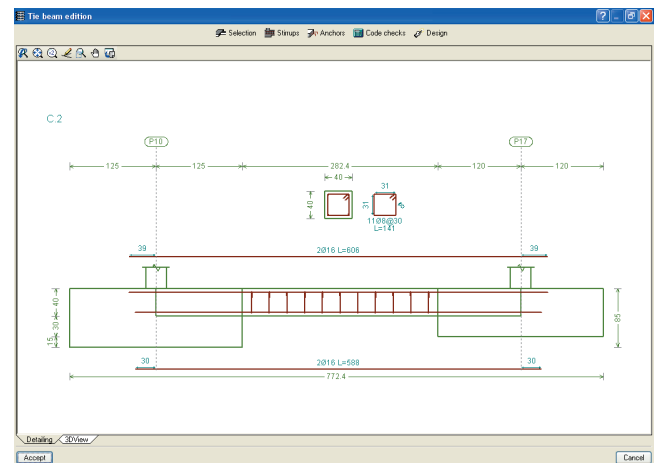


Fig. 11.18

To do so, select a beam, and click on the Edit button. A window containing the following options will open:

- **Selection.** This allows for the type of beam to be modified. The new beam (of the same type) can be selected from the scroll list.

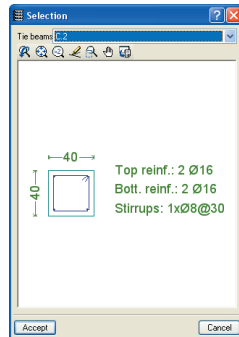


Fig. 11.19

- **Stirrups.** If the options are activated, the stirrups of the strap beam will be extended into the foundation up to the column starter bars.

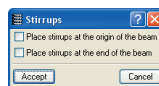


Fig. 11.20

- **Anchorage.** Modifies the anchorage lengths.

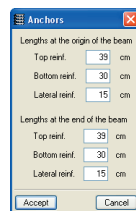


Fig. 11.21

- **Code checks.** Verifies the current beam. Once the process has concluded, a list of all the verifications that have been carried out is available.

Code checks	Values	Status
Minimum beam width recommendation: 1. Calavera, 'Cálculo de Estructuras de Cementación' 4ª edición, INTEMAC, Section 3.15 (pg. 126).	Minimum: 6.9 cm Calculated: 40 cm	Verified
Minimum beam depth recommendation: 1. Calavera, 'Cálculo de Estructuras de Cementación' 4ª edición, INTEMAC, Section 3.15 (pg. 126).	Minimum: 6.9 cm Calculated: 40 cm	Verified
Minimum diameter of stirrups:	Minimum: 6 mm Calculated: 8 mm	Verified
Minimum spacing of stirrups: Eurocode 2, Article 5.2.1.1.	Minimum: 2 cm Calculated: 29.2 cm	Verified
Minimum spacing longitudinal reinf: Eurocode 2, Article 5.2.1.1. -Top reinf: -Bottom reinf:	Minimum: 2 cm Calculated: 27.2 cm Calculated: 27.2 cm	Verified Verified
Maximum stirrup spacing: -Without shears: Article 44.2.3.4.1 (EN19-98 Code)	Maximum: 30 cm Calculated: 30 cm	Verified
Maximum stirrup separation recommendation for beams resisting compression due to seismic loads (1)		Not applicable
(1) The stirrup separation requirement for bars in compression will not be applied due to the longitudinal compression reinforcement not being necessary.		
Maximum spacing longitudinal reinf: Article 42.3.1 (EN19-98 Code) -Top reinf: -Bottom reinf:	Maximum: 30 cm Calculated: 27.2 cm Calculated: 27.2 cm	Verified Verified

Fig. 11.22

- **Design.** Automatically designs the beam and applies all the conditions stated in the code selected by the user. Once the process has concluded, a list of all the verifications that have been carried out is available.
- **Detailing.** A transverse and longitudinal section of the beam is displayed, with its reinforcement and dimensions.
- **3D view.** Provides a 3D view of the beam.

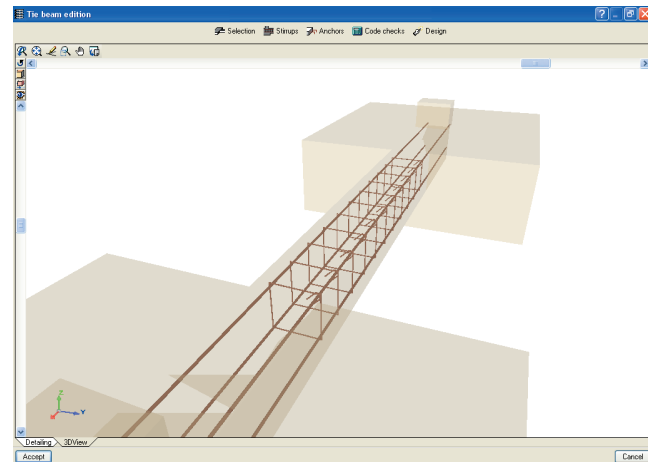


Fig. 11.23

- **Delete beam**

Deletes a previously introduced beam.

- **Balance ends**

Allows for the balancing action to be activated or deactivated upon clicking at the end in question. If the strap beam balances the load at a specific end, a curved arrow will appear indicating that the balancing is active. If the arrow does not appear, the balancing at that end is not active.

- **Match**

This option allows to copy the selected strap beam type from the foundations of the analysed job to other strap beams of the same job. It also allows to copy tie beams in a similar manner.

Using this tool, it is possible to match the strap beam sections proposed by the program having analysed the job. This way, the number of different beams of the job can be reduced, obtaining more uniform results.

A strap or tie beam can be copied onto another strap or tie beam with a larger section. In this case, the beam will be drawn in red to indicate that it has an insufficient section.

The anchorage lengths of the copied beams are redesigned after the assigning has taken place. The master beam will remain unchanged.

11.3.3. Further options for strap and tie beams

If the user has acquired the **Advanced design of surface foundations** module, then further options are available:

- **Loads applied on footings, pile caps and, strap and tie beams**

Point, line and surface loads can be applied on footings, pile caps and, strap and tie beams, and are taken into account in the design. Using this module, CYPECAD also recognises the loads transmitted by the stair supports resting on the aforementioned foundation elements.

CYPECAD assumes that the soil below the strap and tie beams does not intervene in the force equilibrium or distribution to which they are submitted. Therefore, strap and tie beams are designed to also support any loads that may be present acting on them and will transmit the forces to the footings or pile caps at their ends, or other strap and tie beams they may intersect.

- **Strap and tie beam intersections**

Using the **Advanced design of surface foundations** module, CYPECAD can design foundations consisting of footings or pile caps with intersecting strap and tie beams.

11.4. Automatic generation of footings and beams

This option will automatically generate footings and beams according to the established parameters.

A window will open where the parameters to be used in the generating process are to be introduced.

The overhang will be equal to zero if, in one direction, there is a column directly in front but not on the other side.

Vision distance (D) and angle (A). Any facing columns whose axis alignment differs from that of the generating column by a distance less than 'D', as long as the contained angle is within the angle of vision 'A'.

Maximum separation between column axes to create footing. The maximum distance, by default, between the column axes to create a combined footing is 1 m (limit maximum value: 3 m).

Having generated the foundation, the footings and beams can be modified, using the **Edit** option from the **Foundation elements** or **Strap and tie beam floating** menus.

11.5. Analysis and results

11.5.1. Design

This option analyses and designs the foundations which can be composed of pad footings, strip footings, pile caps, baseplates, and strap and tie beams.

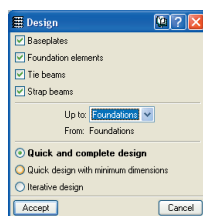


Fig. 11.24

For those cases where errors or warnings appear when designing certain elements, a warning will be emitted at the time of executing the option, after which, the option Code check errors should be selected.

There may be more than one foundation level. Here, it is equally possible to select the floor groups for which the foundation is to be designed.

There are three design options explained in the on screen help of the program.

Bear in mind that the complete structure has to be analysed to obtain the forces at the base of the columns.

11.5.2. Code check errors

After the analysis, if this option is activated, the foundation elements with design errors will be displayed in red. By clicking on the element in question, a message will appear informing the user of the errors that have occurred during the design.

11.5.3. Eliminate overlaps

Once the foundation has been analysed, if there are overlapping footings, a warning will be emitted and this option can be used to modify the layout of the elements.

For those cases where all the overlaps have been eliminated, the program will place a combined footing under the columns.

11.6. Reports and drawings

11.6.1. Foundation report

A report is provided by the program containing the data introduced: footings, pile caps, strap and tie beams and baseplates displaying their dimensions and the verifications that have been undertaken.

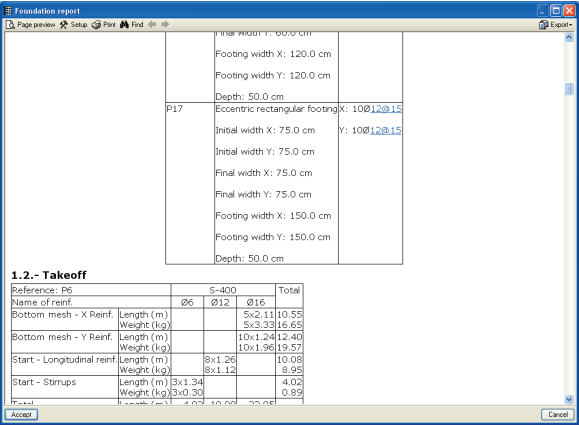


Fig. 11.25

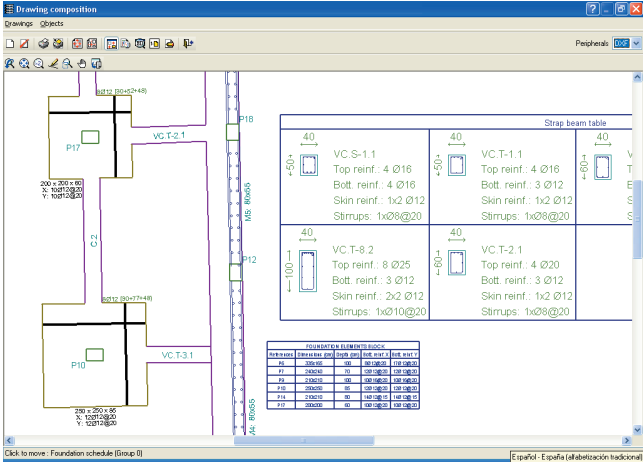


Fig. 11.27

11.6.2. Foundation and foundation reinforcement floor plans

Within the selection of floor plans, there are two types of layouts available for the foundations: Foundation and Foundation reinforcement.

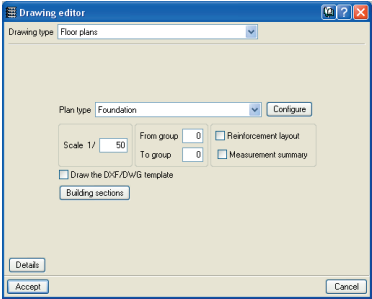


Fig. 11.26

In the first case, the foundation layout on the floor is drawn, as well as the column starts, reinforcement and dimensions of the foundations and the foundation and strap and tie beam schedules.

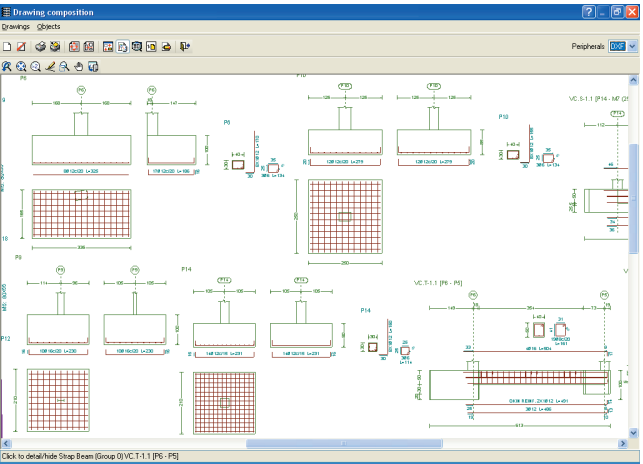


Fig. 11.28

The floor layout drawing configuration tab contains two options: **Footings/ Pile caps** and **Foundation**.

The first option contains all the data regarding the layout on the floor and the column starts. The second the drawings and reinforcement schedules.

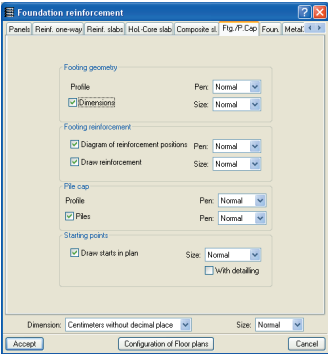


Fig. 11.29

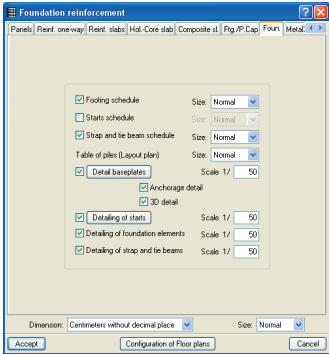


Fig. 11.30

In the layout drawing the footings, pile caps, piles, pile tables, etc. are drawn.

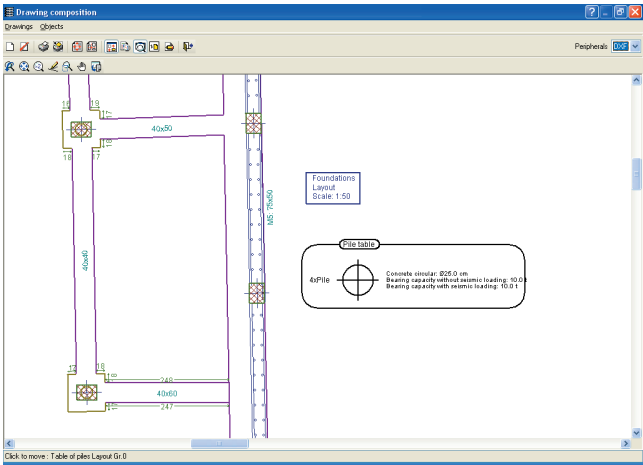


Fig. 11.31

12. Contour maps and deformed shape

12.1. Contour maps

Contour maps can be accessed by selecting the **Contour maps** tab at the bottom of the CYPECAD window. The force and displacement graphics are displayed in colours whereby each tone possess a value detailed on a scale in a floating menu. The **contour lines** can equally be represented.

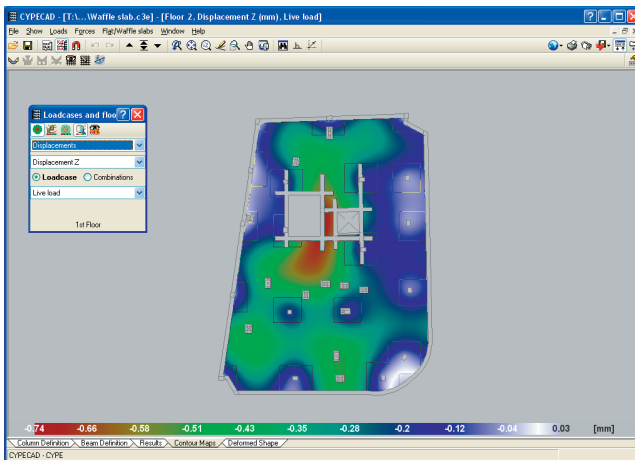


Fig. 12.1

A colour key is displayed at the bottom of the screen indicating the respective values associated to the colours as well as the maximum and minimum values at either end of the key scale.

The contour lines are graphics similar to those of elevation levels in topographic maps, except in this case, each line represents the geometrical position of the points with the same force or displacement.

The contour maps or contour lines can be attached and printed out.

12.1.1. Loadcases and floors window

The **Contour map** window contains a floating menu called **Loadcases and floors**. Here, the visibility of the contour lines and their values can be activated or deactivated. The loadcase or combination loadcase the contour map refers to can be selected, as well as what is to be represented in the contour map (force, displacements, etc.).

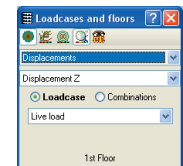


Fig. 12.2

The displacement diagrams, force diagrams and steel area diagrams for waffle, solid and flat slabs are represented per floor and in colour. They represent the loadcase selected at that moment.

Displacements

The contour maps and lines of the floors can be viewed per loadcase or combination loadcase.

- Displacement z. Vertical displacement. Units in mm. Negative values represent a vertical descent.
- Rotation x. Rotation about the x axis of the mesh.
- Rotation y. Rotation about the y axis of the mesh.

Forces

The forces can be selected for a given loadcase or combination loadcase.

- Total shear. Square root of the sum of the shear in X squared and the shear in Y squared.
- Shear X. Shear force in the X axis of the mesh.

- Shear Y. shear force in the Y axis of the mesh.
- Moment X. Bending moment in the X axis of the mesh.
- Moment Y. Bending moment in the Y axis of the mesh.
- Moment XY. Torsion in the floor plan.

Design stresses

The design stresses can be selected. The design stress at a point is the value of the force envelope at that point which results in having to provide the worst case reinforcement.

- Total shear. Square root of the sum of the shear in X squared and the shear in Y squared.
- Shear X. Shear force in the X axis of the mesh.
- Shear Y. Shear force in the Y axis of the mesh.
- Moment X, bottom steel. Bending moment in the X axis of the mesh used for the design of the bottom steel.
- Moment Y, bottom steel. Bending moment in the Y axis of the mesh used for the design of the bottom steel.
- Moment X, top steel. Bending moment in the X axis of the mesh used for the design of the top steel.
- Moment Y, top steel. Bending moment in the Y axis of the mesh used for the design of the top steel.

Steel areas

The steel areas displayed in the contour maps represent the steel area required for the design stress at that point i.e. the worst case reinforcement.

The steel to be seen can be that of the top or bottom in the X or Y direction.

Ground bearing pressures

This option is available in the case of mat foundations.

- Maximum bearing pressure. Worst case bearing pressure of the slab on the soil.
- Uplift. Value of the uplift in mm.
- By loadcase. Bearing pressure by loadcase or combination loadcase.

Some of the options present a Maximum value and/or Minimum value option. These allow to limit the values displayed. The colour order does not vary upon modifying the range, only the values they represent change. These will be displayed in the colour key at the bottom of the screen. This option is very useful to obtain a more precise view of the values.

It is equally possible to select a single limit to locate the point at which the maximum or minimum value occurs.

These options allow, for example, to view which areas of a slab require additional reinforcement to that of the base reinforcement, in which areas can a certain reinforcement diameter be used, etc.

In the case of the bearing pressure option, it is useful to view if there are areas which are greater than the allowable bearing pressure, by introducing the value in the minimum value option.

12.1.2. Forces menu

This option allows to view the forces in beams, joists, hollow core slabs and composite slabs. The results can be displayed by loadcase, combination loadcase or envelope.

12.1.3. Flat/Waffle slabs menu

This menu contains the same options that can be found in the **Flat/Waffle slabs** menu within the **Results** tab; whereby the user can view and modify the reinforcement.

A third option is available: **Deflection between two points**. The user can visualise the secant or tangential de-

flection at a point, and for each of these, their respective maximum relative, maximum absolute or the deflection between the two points. Additionally, the user may select the simple loadcase, combination loadcase, worst case combination and amplification factor for which the deflection values are going to be calculated.

This last option can also be accessed in the **Envelopes** menu within the **Results** tab.

12.1.4. Window menu

The contour maps and force envelopes can be consulted simultaneously in several windows, in such a way that, for example, the floor displacement values can be seen for the dead load loadcase and the live load loadcase at the same time. In a similar way, the corresponding windows can be obtained for different floors. The definition of the window remains whilst the user is not in the Contour map tab but not if the program is closed.

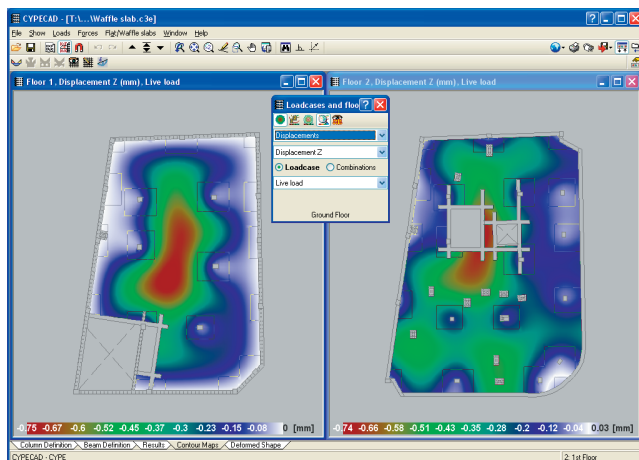


Fig. 12.3

12.2. Deformed shape

12.2.1. 3D view of the deformed shape of the structure

Once the job has been analysed, the deformed shape of the structure can be seen. It is possible to view the maximum deformation caused by a simple loadcase or combination loadcase.

This option appears in the **Deformed Shape** tab.

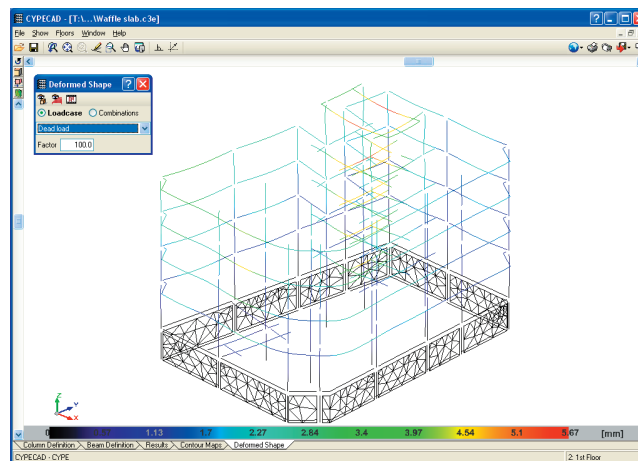


Fig. 12.4

Within this window, the **Deformed shape** dialogue box appears in which a series of options can be defined to configure how the deformed shape is viewed. The following options are available:

- View the deformed shape due to a simple loadcase or combination loadcase.
- Selection of the loadcase or combination loadcase.
- Factor
- View the structure (display structural elements or display simplified structure)

- View waffle and flat slabs
- Deformed shape animation

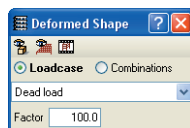


Fig. 12.5

Each of the options described above contains a help on screen describing their use.

Viewing the deformed shape and its animation are efficient tools to identify if there is any unusual behaviour in the structure and any issues due to its design or external actions.

12.2.2. Indication of the deformation using a colour scale.

The structural elements (in their deformed positions) are drawn with an applied colour scale, depending on their displacement when a loadcase or combination loadcase is selected. This way, it is easy to distinguish the way in which the individual elements of the structure are deformed resulting in the overall deformation of the structure.

The key to the colour scale is indicated at the bottom of the screen. The value of the displacement represented on screen is the vector result of the three components: Δx , Δy and Δz .

In a similar way as with the Contour maps, several windows can be opened simultaneously to be able to view and compare the deformed shape when different loadcases or combination loadcases are applied.

13. Tables and libraries

13.1. Reinforcement tables

The program contains predefined reinforcement tables for all the reinforced concrete elements in CYPECAD. All the tables can be edited and modified. It is equally possible to create new, completely independent tables. The tables are managed in **Steel types in bars** window which opens by clicking on the **By position** option in the **General data** window.

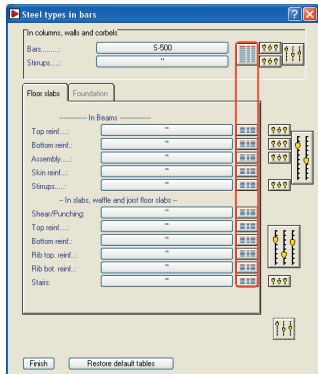



Fig. 13.1

Using the  icons, the user can access the chosen table.

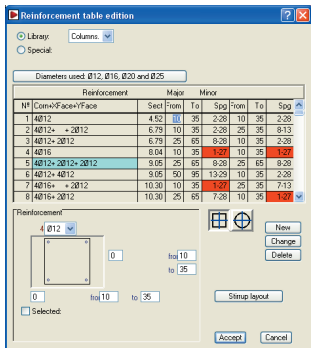


Fig. 13.2

13.1.1. Use of the reinforcement tables

During the analysis, the program chooses the required reinforcement to resist the applied forces and moments from the selected table.

If no suitable reinforcement is found, the program emits a warning and the user should revise the reinforcement table.

When a table is created or modified, verify the coherence of the data introduced.

13.1.2. Restore default tables

This button, located at the bottom of the **Steel types in bars** window, allows to restore the original or predefined (in case they have been modified) reinforcement tables. When restoring the default tables, if they are different to those modified by the user, the program will ask which tables are to be saved.

In any case, tables added by the user to the predefined tables are saved.

13.2. Section library

CYPECAD contains predefined section libraries. These libraries are provided by the manufacturers and allow for the sections to be used directly as their dimensions and properties have already been defined.

It is equally possible to create new sections and new libraries.

13.2.1. Edition of an individual section

A new section can be created by selecting the **Editable** option in the **Section data** field from the **Section description** window. All the dimensions and properties can be modified.

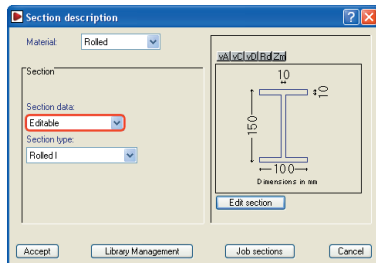


Fig. 13.3

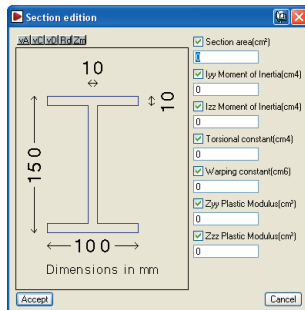


Fig. 13.4

13.2.2. Library manager

When sections from a library are to be used, the library must first be imported into the job section library.

13.2.2.1. Use of existing libraries

The libraries are managed in the **Section library** which opens by clicking the **Job sections** button in the **Section description** window.

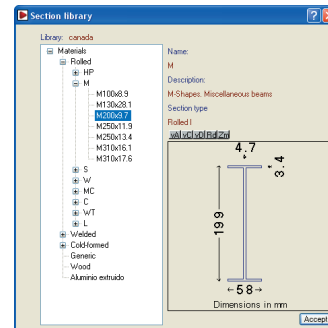


Fig. 13.5

The library which appears by default is that defined in the **Section library** which opens by clicking the **Job sections** button.

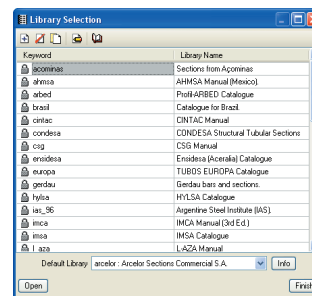







Fig. 13.6

Other libraries can be imported by clicking the import icon .

13.2.2.2. Library edition

It is possible to attach sections to the job library by clicking on the  icon. Sections can be copied by clicking on the  icon, edited by clicking  and deleted using the  icon.

13.2.2.3. Library creation

A new library can be created by either directly creating a new library in the **Library selection** window (accessed by clicking the **Library management** button) and importing the selected sections, or by exporting specific sections of the job using the  icon from the **Section library** window.