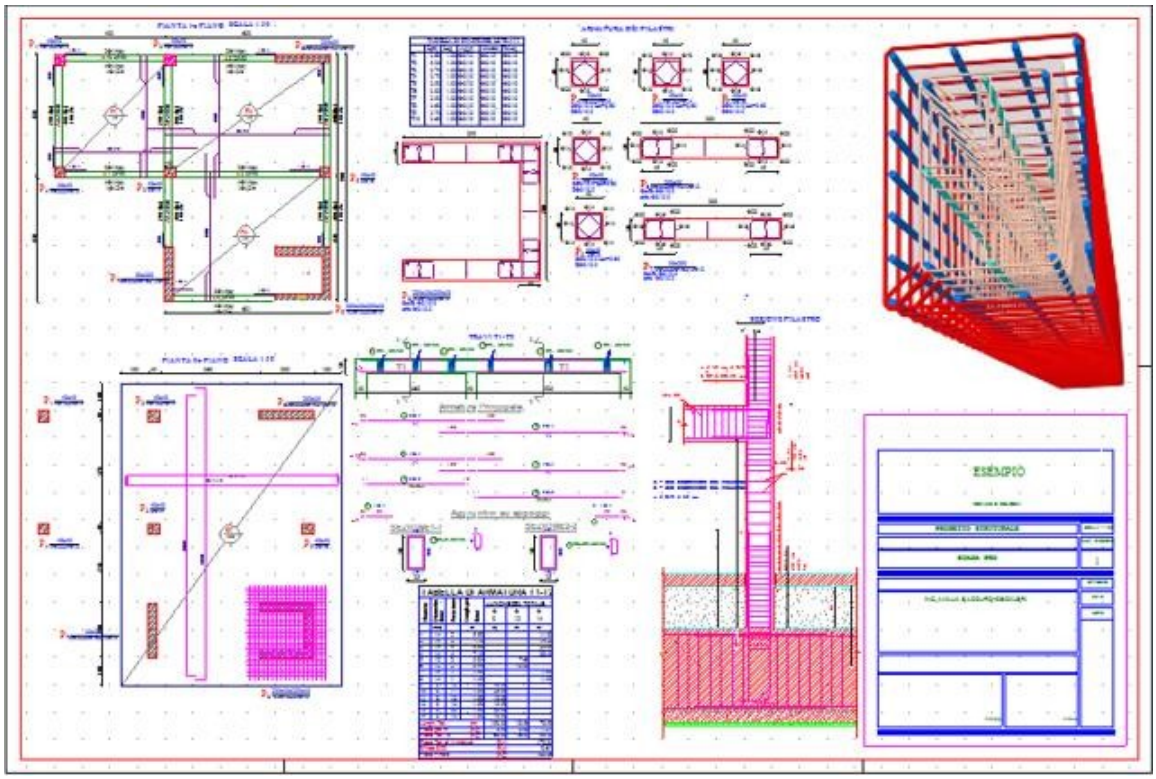
SCADA Pro 25tm

Structural Analysis & Design

User Manual

11.Formwork

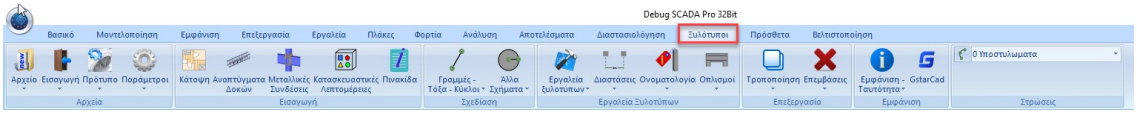


ACE HELLAS

INTEGRATED SOLUTIONS

CONTENTS

Chapter 11: Timber types

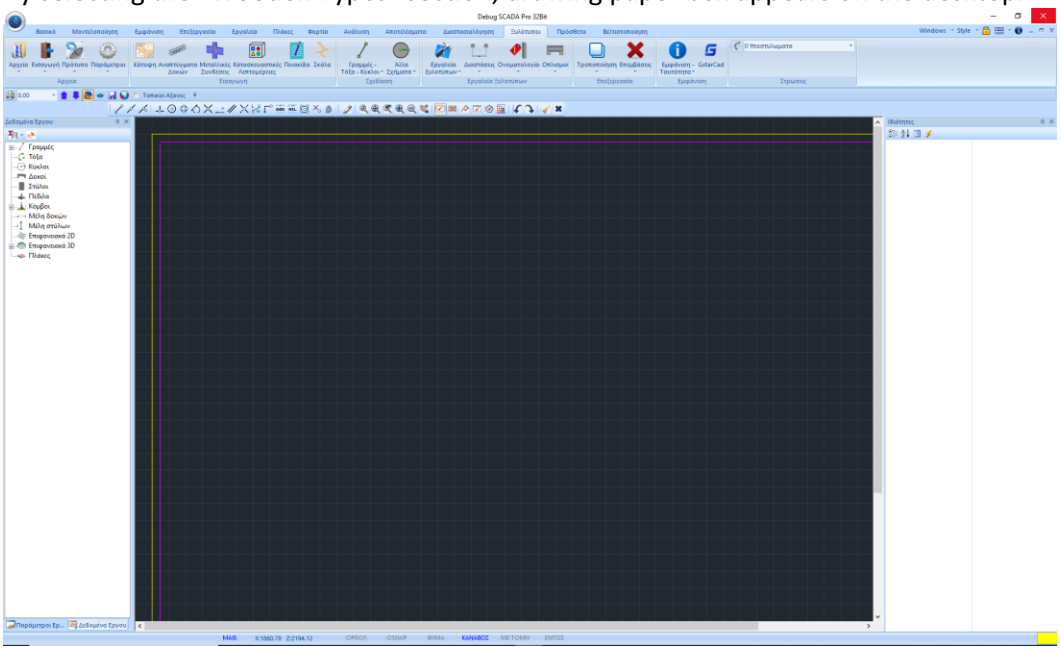


The 11th Module is called "WOODY" and includes the following 6 groups of commands:

- Archive
- Introduction
- Design
- Wood type tools
- Edit
- Show
- Layers

After completing the dimensioning of the structure and the modifications of the reinforcement through "Editor" and "Reinforcement details" commands concrete designs, or the creation of the connections for metal ones, within the Timber Formwork Module you enter, modify and finally create the drawings of the formwork and its details.

By selecting the "Wooden Types" section, drawing paper box appears on the desktop.

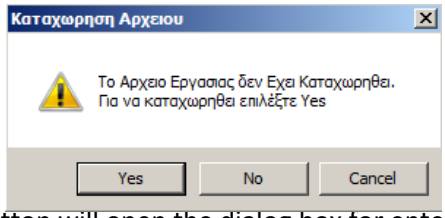


Archive

The commands of the "File" group allow you to import and export the drawings of dimensioned elements, create drawing templates and set drawing parameters.

1.1 Archive at

New : provided you have already imported or created a design, this option "clears" anything that is already designed on the screen, so you can import a new xylotype or start importing entities from scratch. The program displays a dialog box asking for confirmation to exit to a new file, with or without entering the existing drawing.

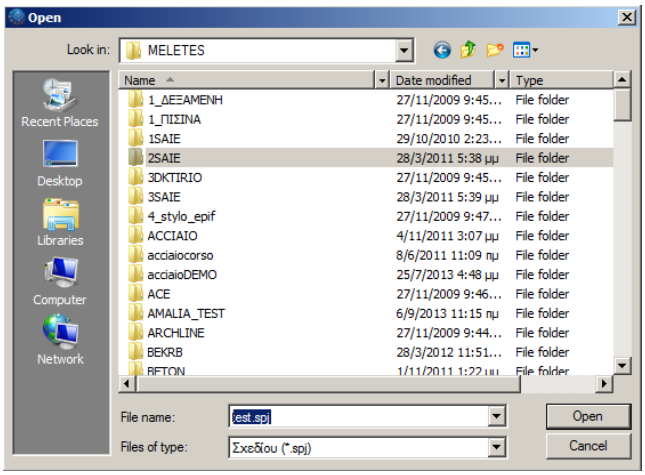


Yes : Pressing the button will open the dialog box for entering design with the name given by the user.

No : The existing plan will be deleted without being registered. All previous work will be lost.

Cancel : Cancel the "New" option and continue editing the existing design on the screen.

Existing : With this option you call an existing formwork file (a drawing that you have already edited and entered as a formwork drawing file in the program). These files for ScadaPro have an extension (*.spj). The *.scj extension is no longer valid.



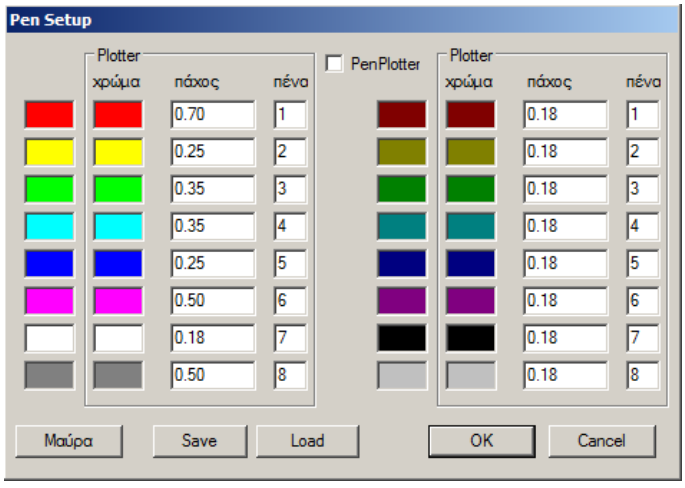
Entry : Activating this command opens a dialog box similar to the one shown above, where the wood type you are editing is entered as a drawing file of the program.

⚠ The first entry, even if made with the "Entry" option, is made as ¨San entry¨ (see next paragraph).

Listing as: You register the project with a new name. This option is useful in case you have made modifications to the xylotype and you want to register the modified drawing with a different name than the one it already has.

SetUp printer: Enabling this option opens the Setup of the default printer from the Windows Control Panel to confirm or modify its parameters.

Print: this option prints your designs on the printing medium installed as the default in Windows. In the dialog box that opens, you specify parameters concerning the thicknesses of the lines to be printed, depending the color of these lines on your screen.



The group colors to the left and outside the box titled "plotter" refer to the colors you see on the screen. These will match the colors and thickness will be printed on the paper and are shown in the corresponding columns of the box titled "plotter".

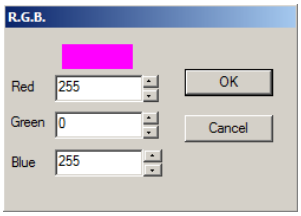
CHAPTER 11 'FORMWORK'

α. Configuration for printing on an inkjet .

1. You disable 

option, thereby informing the program that your rasterizer is sprayed.

2. You select for each colour of your design (screen colour):
2α. The colour which will be printed in its place. Initially the program assumes that the colors on the screen will be printed in same color as your paper. Changing the color (which will be drawn) is done by pressing twice (double click) inside the box of the corresponding color from the column located in the box titled "plotter". A new dialog box entitled "R.G.B." opens immediately. where the resolution of the given color is shown in the three primary colors (RED-RED, GREEN-GREEN, BLUE-BLUE).



Changes to color resolutions can be made with  or by typing a value directly into the corresponding field.
After selecting the appropriate color, press the 

button and return to the previous dialog box to continue with the definition of the remaining parameters.

2β. The thickness of the lines for each colour. This definition is made for each colour, in the column entitled "thickness", where you enter the corresponding value in each position.

⚠ For inkjet plotters the column entitled "pen" is inactive. b. Configuration for printing on pen plotters (pen plotters).

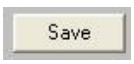
1. You activate 

option by pressing the mouse once on  .
2. For each colour of your screen (left column of colours) you select the number = position of the pen that will be selected from the available ones of the radiograph. The determination is made in the corresponding boxes in the column titled "pen".

⚠ For pen plotters the columns titled "color" and "thickness" are inactive.

: By pressing with the mouse inside the field, all the colors to be drawn on the paper become black (column titled "color" inside the "plotter" box). In way we can achieve monochrome prints from a color plotter.

⚠ The program allows you to register in "palette" files all settings you make so that when you want to use the same ones for designing different studies, you can "load" the registered "palette" file, avoiding each time to repeat the same procedure.



: After you have specified all the color and thickness parameters, press the button and a dialog box opens with the title "Enter Color Palette" where you give the name

of the "palette" file without extension. The files created are of the format { "*.pal".)

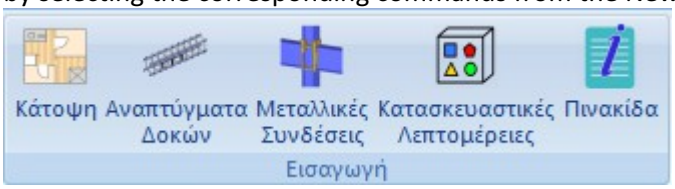
: Select one of the available files (*.pal) to update the colour and thickness parameters of the wood type to be designed according to those of the palette file.

1.2 Introduction

You are creating for the first time a study form that has been dimensioned by ScadaPro. The program reads geometry data from "Modeling" and Dimensioning results from the corresponding module.

OBSERVATION

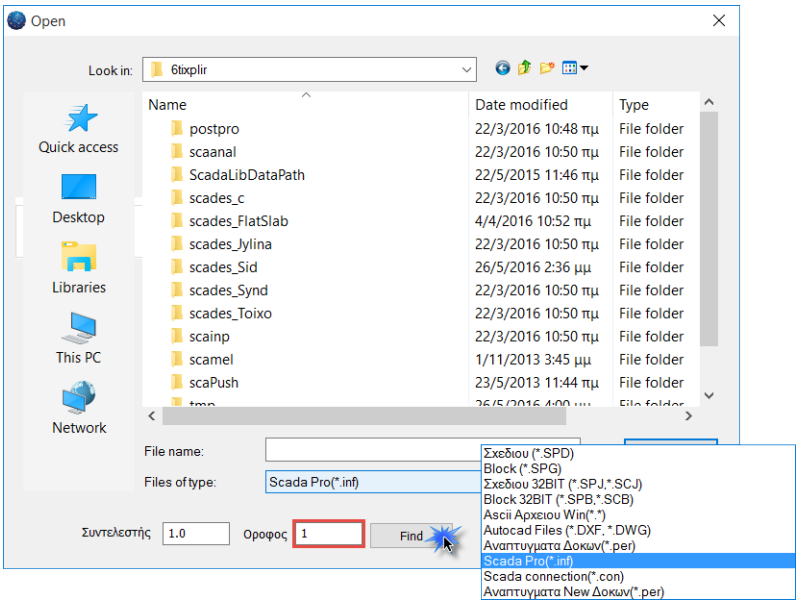
With the new version of SCADA Pro you have the possibility to import all the plans for each floor by selecting the corresponding commands from the New group "Import".



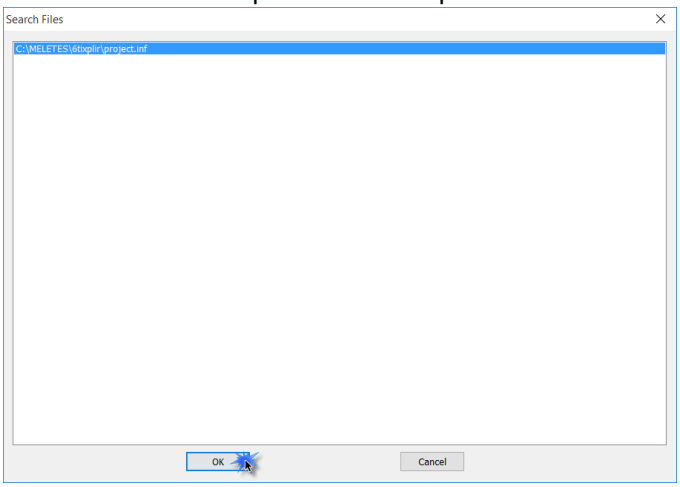
The Import command opens the window for selecting the study folder. Select:

- the type of design from the **Files of Type**
- the number of the floor and
- the coefficient

you press the **Find** command.



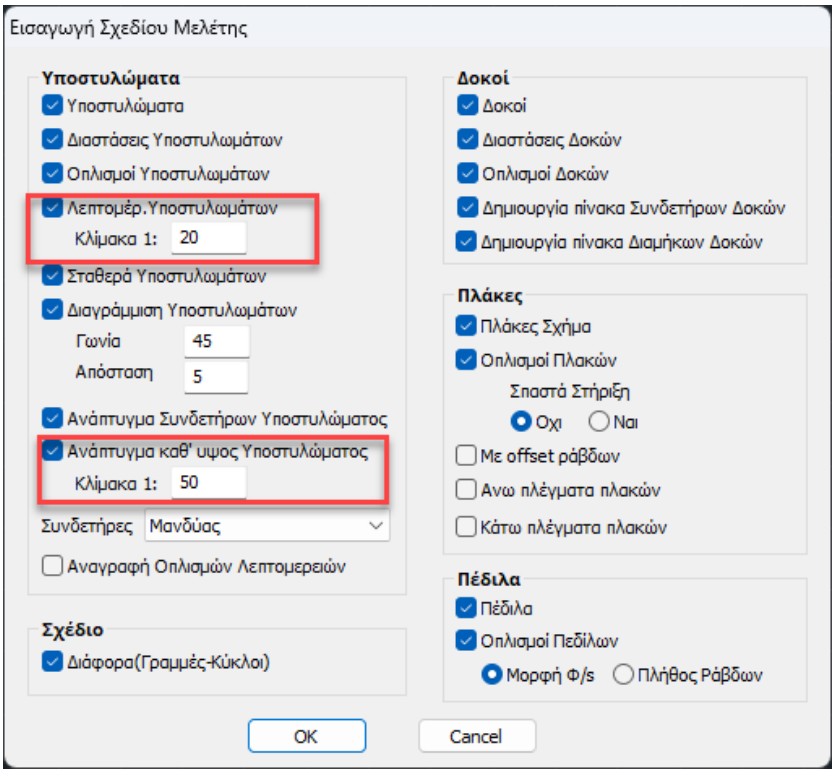
In the window that opens select the path and OK



Coefficient: Enter the scale factor by which the dimensions (drawing size) and text size of the files to be imported will be multiplied with the "FILE >> Import¨.
1. To import a file (*.dxf) whose units of measurement are in m or cm or mm you will enter coefficient values of 100 or 1 or 0.1 respectively.
2. To import a file into your drawing that has been edited in 1:50 or 1:100 scale, you will enter a factor of 1, in case you are working in the same scale (see "PLANNING >> Drawing Paper >> Scale") or 0.5/2 if you are working in 1:100/50 scale respectively.

Floor : Enter the number of the level of which you want to bring the xylotype on the screen. This option only applies to the entry of a design in the "From Study" (*.sca) format and from a file
*.inf.

The following dialog box appears on the screen from which:



- You select the entities to be inserted in your design by activating with "v" the corresponding checkboxes.
- In "**Plate reinforcements**" you will select whether the additional bars of the plate supports are to be designed broken or not.
- In the "**Details - Scale**" field you will enter the scale factor for the details of the columns to be inserted on your paper.

With the new dialog box with the parameters for the input of the floor plan design in the formwork that became more user-friendly and functional, the parameters were grouped and the scale of the input of the expansion across the height of the column is now separate.

- Example : If you are drawing a 1:50 scale x-frame and 1:20 scale post details, you would enter a factor of 50/20 = 2.5.

- In plates that bend only in one main direction, the distribution and separation reinforcement is now automatically designed in the other direction

Activating with "v" the indicator:

- "**With offset bars**" it is possible to separate the straight bars from the broken bars of the plates.

- "**Number of rods**" is given the option to have the reinforcement of the pedestals come with the number of rods indicated (or if not checked the reinforcement comes with the number of rods per distance indicated).
- "**Striping of columns**" gives the option to have the columns stripped.

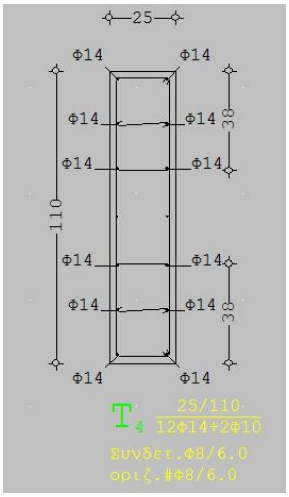
In the fields next to "**Angle**" and "**Distance**", the slope of the lines and the distance between them to be used for the delineation of the column are specified.
In the "**Fasteners**" field you select the type of fastener to be inserted in the formwork.
Activating with "v" the indication

⚠ "**Indication of reinforcement details**", the diameter of each bar will be indicated in the details of the columns.
If you wish to further edit the details, you can to enter the cross section in Editor Armament

(see. Wood types>Edit>Edit>Editor Armament)

⚠ "**Create Beam Connector Table**" inserts the table with the beam connectors into your design,

⚠ "**Create longitudinal beams table**" inserts a table with the reinforcement bars of the beams.



Point to the insertion point and insert the drawing of the selected level, repeating the process for all levels.

From a drawing file (*.scj, *.spj) :

Insert an existing xylem in your plan that has been previously entered as a plan file of the program (*.scj or *.spj). You can also use this option in cases where you want to display xylems of two or more stations on the same xylem. The *.spj files are ScadaPro files, while *.scj is the older format of Scada for Windows

From Block file (*.scb, *.spb) :

Insert into our design a group of entities that you have entered as a block. Files *.spb format are ScadaPro files, while *.scb is the older format of Scada for Windows

From File Ascii File Win(*.*)/Dos(*.*) :

Insert Ascii text file which has come from Windows text editor
/ Dos respectively.

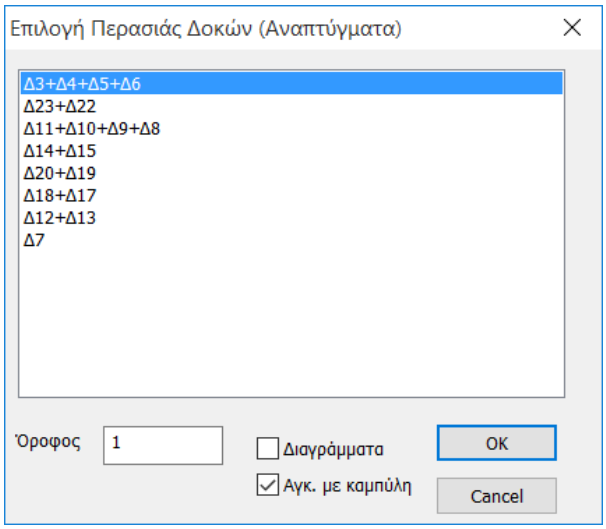
From Autocad Dxf file Win(*.dxf)/Dos(*.dxf) :

Import design file format (*.dxf) derived from any cad program from Windows/Dos respectively.

From file Beam Expansions (*.per) :

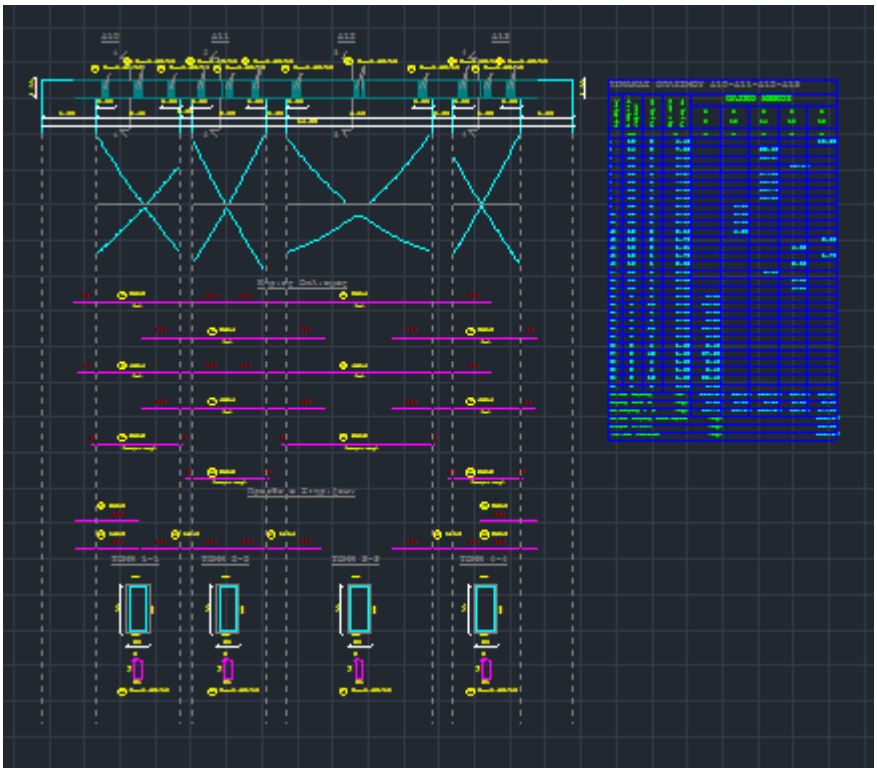
Enter in your design the reinforcement expansions for the beam span that you will choose from those available in our study.
This option is for expansions created with the existing beam editor, while the "New Beam Expansions" option with the same format (*.per) is for expansions created with the new "Armor Details" editor.

Selecting the Beam Expansions (old and new) the path in Find takes you to a new window to select the passes one by one.

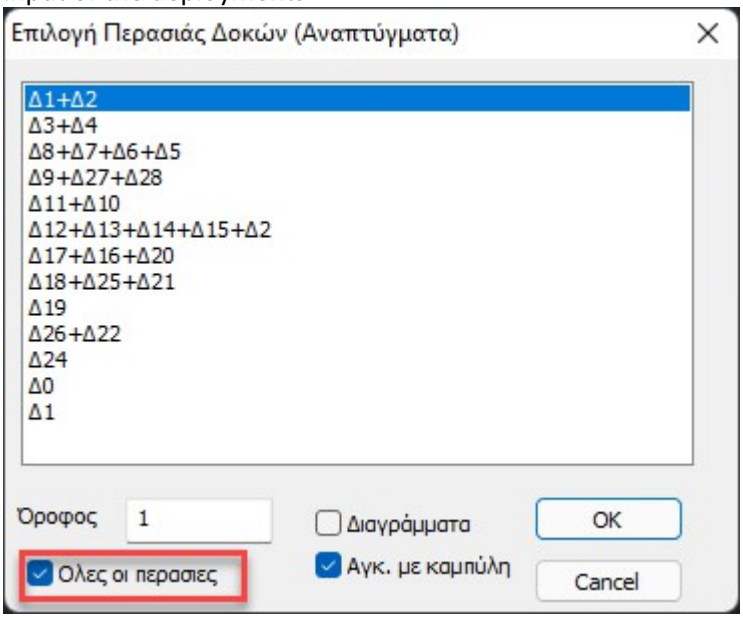


In the "Floor" position, enter the number of the level where the terrace is located, of which you want to draw the developments.
Activating with "v" the indicator:
⚠ "Diagrams": the expansion you enter will be accompanied by the corresponding torque diagram.
⚠ "Curved anchorage": the anchorages will be closed with a curve.

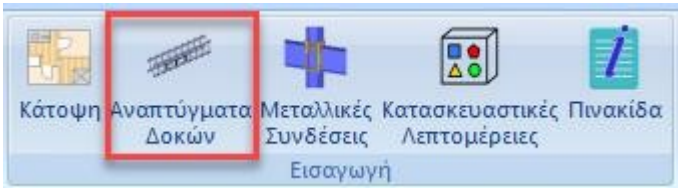
You select one of the available passes that open and by pressing the "OK" button you are invited to place the design of the growths on your paper.
Point to the insertion point and insert the drawing of the selected level, repeating the process for all levels and all details.



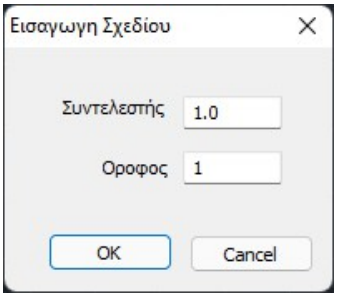
With the new version of SCADA Pro you have the ability to enter with one click the deployments of all the passages for each floor. Just check the corresponding option in the dialog box for the input of the deployments



For the quick import of the beams' developments there is now a separate icon in the formwork section



Also here, in the same command section, you can enter the floor plan of the formwork of any floor

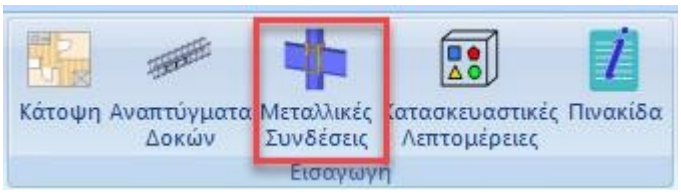


as well as the ready-made construction details that exist in SCADA Pro and their ready-made project plate with the project identity data to be filled in.

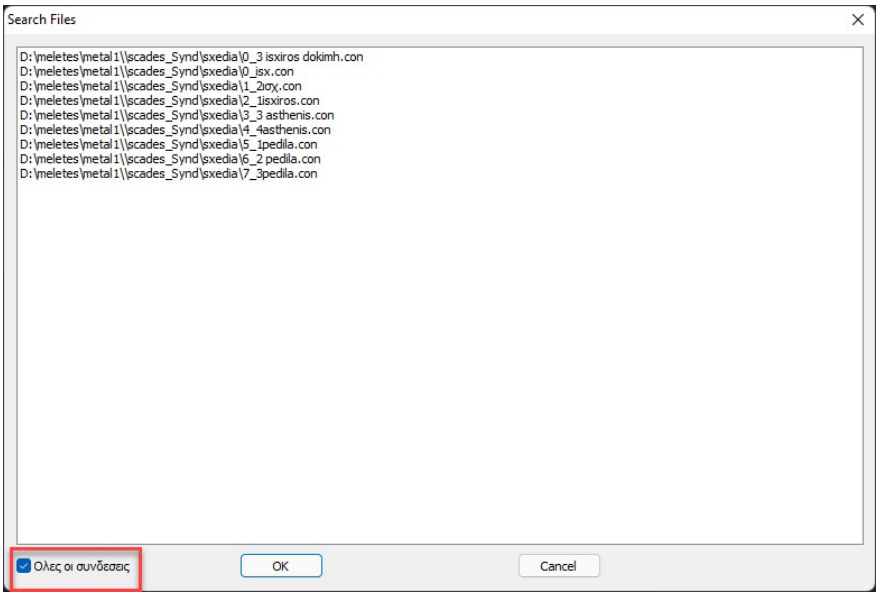
From file Metal Connections (scada connection *.con):

Insert the detail drawings of the metal connections into your design. These files are located inside the design folder in the **Files of Type** "Scada connection (*.con)".

With the corresponding logic of the beam expansions, the program now allows you to import all the metal connections you have created. The import is done by pressing the



and selecting "All connections" in the following dialog box.

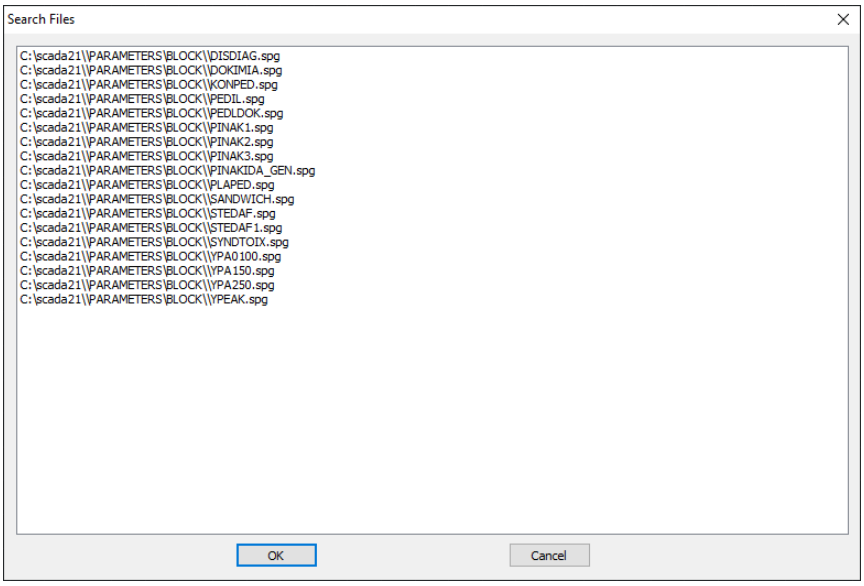


Export Block: to group entities and enter them as one, in block format (*.scb), so that they can be imported into as many projects as you wish.

1. You activate the option "FILE>> Export Block".
2. You select the entities you wish to include in the block that will be created, either selectively one by one by pointing them with the mouse, or collectively by opening a window with the  button.
3. The end of the options is indicated by pressing the right mouse button, at which point the program asks you to indicate with the block insertion point feature. Its placement in your drawings, each time it is called, will be in terms of its feature point, which can be a point of some design entity to be included in the block, or any point on the screen.

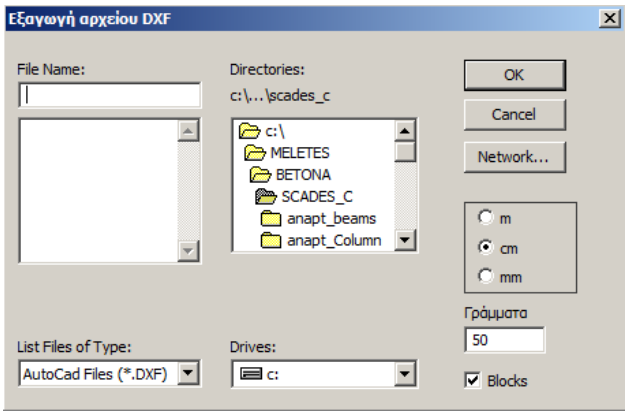
The blocks created by the program are files of the format (*.scb), where (*) is the name you enter for the block.

The program now gives you the possibility to enter all the construction details by pressing the Construction Details button and selecting them from the list that appears:



Export-DXF (Old): with this option the program creates a file of the format (*.dxf) or (*.dwg), where (*) is the name you enter, so that it can be "read" by any other cad program, static or architectural, which is able to read files of this format.

1. Activate the option "FILE>> EXPORT DXF (old)" and the following dialog box opens where you enter :



⚠ File name: The name of the file to be created, without the extension.
⚠ Directories: the directory in which the file to be created will be registered.
⚠ List Files of Type :Here you specify the type of file to be created (dxf or dwg).
⚠ Drives :Specify the drive where the file will be registered

(*.dxf or *.dwg).

⚠ 

: You specify the length units that the exported file will have. The unit to be selected depends on the length units used by the cad program in which the file will be imported (*.dxf).

⚠ Letters : Type the size of the letters of the texts you have entered with the "DESIGN >> Text", in centimeters to scale.

⚠ 

: Enabling this option will export as blocks all entities that are blocks in the drawing you are editing. Otherwise, the entities

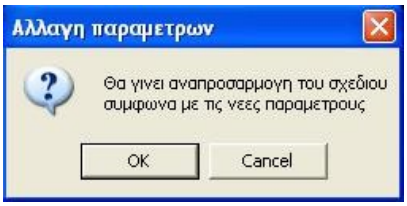
will explode into the synthesized data and will be "translated" into dxf or dwg afterwards.

Export DWG,DXF: With this option the program creates a file in the format (*.dxf) or (*.dwg), where (*) is the name you enter, so that it can be "read" by any other cad program, static or architectural, which is able to read files of this format. The difference between this command and the previous command is that this format is much more modern and sophisticated than the previous one, and is the one you should use. The previous format has been retained for compatibility reasons for older studies. The way to use the command is exactly the same as the previous command.

1.3 Standard:

Existing: this option calls a file with default parameters, so that the new project you call will be imported with these parameters. A prerequisite is the existence of a default parameters file in the format (*.dsc).

⚠ If there is an already loaded drawing on the screen and you select the above command, the program displays a dialog box informing you that the drawing will be updated according to the new parameters.



Listing as: You create a default configuration file in the format (*.dsc) so that loading it will adjust any new or existing design to them.

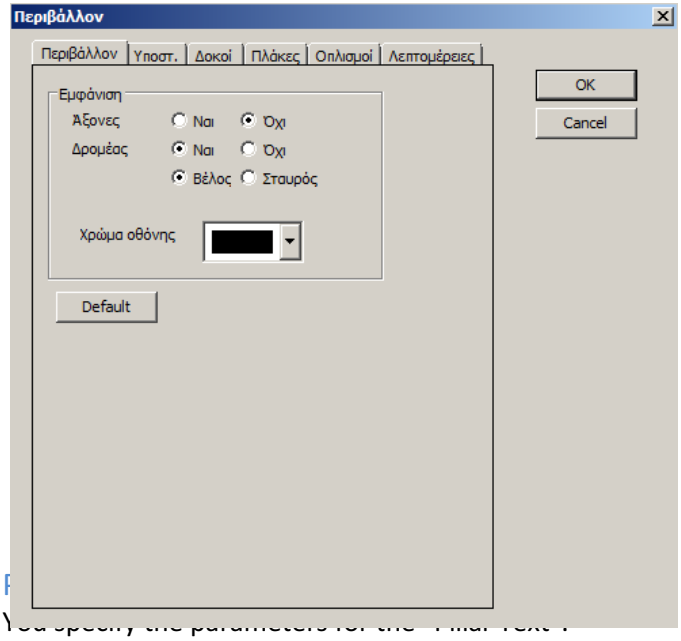
⚠ The default configuration file can contain the parameters described in the "Parameters" options list from "Environment" to "Text".

1.4 Parameters:

Through the dialog box that opens we define parameters concerning the "Design Environment" as well as the parameters of the design entities.

Environment:

In the first "Environment" section that appears, you can set the following parameters:

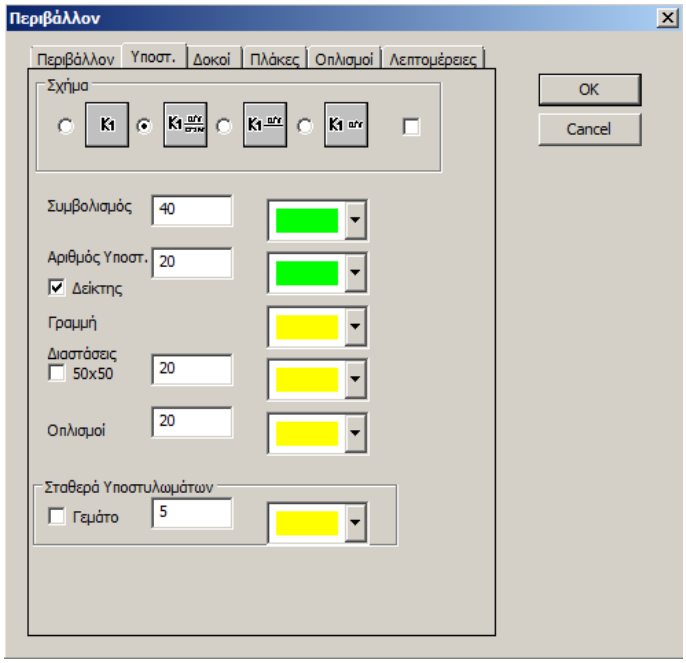


⚠ The appearance or not of the axes (ruler).

⚠ The appearance or not of the cursor and its shape.

⚠ Screen Color : From the field  select the desired colour for the working screen (backround).

: By pressing the button, you can restore all the default parameters of the program in the fields of the dialog box, cancelling any previous modification you have made.



In particular :

Figure :

:In the text of columns only their Symbolism and Numbering will be displayed.

:The column text will be displayed complete with all its elements (Symbol, Numbering, Dimensions, Reinforcements).

:Only the Symbol, the Numbering, the Dimensions and the Dimension-Arms Separation Line are displayed.

:The Symbolism and Dimensions of the poles are displayed.

: If we check the box then the shape of all existing columns of the formwork will be adapted to the "Shape" you select.

In the parameters that follow in the same dialog box, you specify the size (in centimeters to scale) and color of the Symbol, Numbering, Dimension - Reinforcement Separation Line, Dimensions and Reinforcements of the column text that will be displayed on your screen, as well as the Column Stands. Changing the size is done by typing a new value into the corresponding field, while changing the colour is done by selecting a new one from the colours displayed,
by pressing to the right of the color of the parameter we want to change.

Pressing it next to the "Marker" indicator will result in the name of the columns appearing as K6. Otherwise the name will appear as K6.

Pressing next to "50x50" will result in the dimensions of the columns being displayed as 50x50. Otherwise the designation will appear as 50/50. The choice of the format in which the column dimensions will be displayed is determined before the dimensioning is entered into the formwork.

Columns stable :

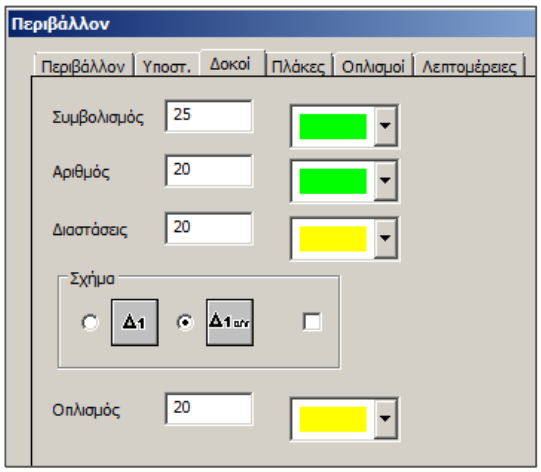
Γεμάτο : By activating the option the circles of the Fixed Pillars will be filled with the same colour.
If you wish to design "filled" Column Stacks you should activate the option from "Parameters" before you enter your design on paper.

The Pillar Stations that exist in your project can be "closed" if you make their Layer "Not Visible".

The entities of the beam deployments belong to separate slides located in the Layers list at positions 50 to 59.
You can accept your choices by clicking on OK or cancel by selecting.

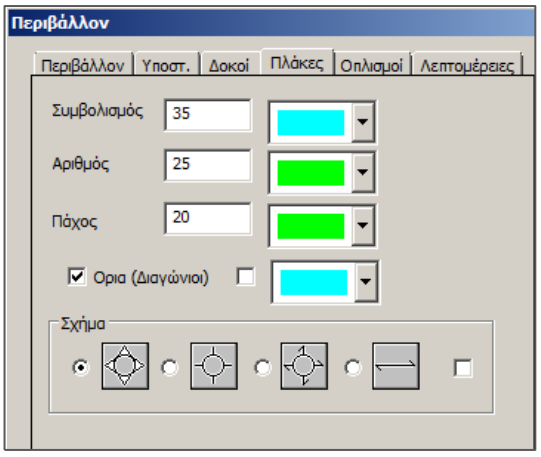
Beams:

Corresponding to the parameters of the poles, you set parameters for the Beam Text.



Plates:

You specify the size and color for the Symbol, Numbering and Thickness of the plates.



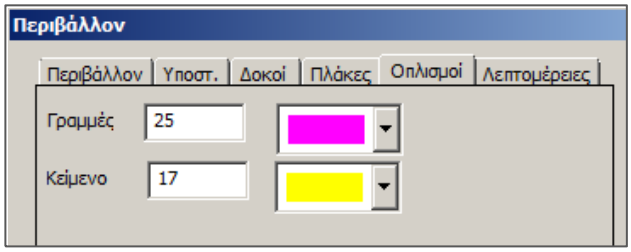
Limits : with the checkbox checked to the left, it indicates that the slabs will be imported from now on with the command "ARCHIVES>> Import" or the command "Lumber Tools >> Nomenclature>>Plakes", will be defined and drawn with their boundaries.

⚠ Deactivating the option, in combination with activating to its right, allows you to remove the boundaries of all existing plates. Conversely, activating it simultaneously with to its right will restore the boundaries of the plates that have been drawn and then removed.

Figure : Here you specify the type of Symbolism of the slabs (, or), while if you activate to the right of the types of symbolism all the existing slabs of our formwork will be updated with the type you selected. Accepting the selections is done with the button OK or cancel by selecting Cancel.

Armaments:

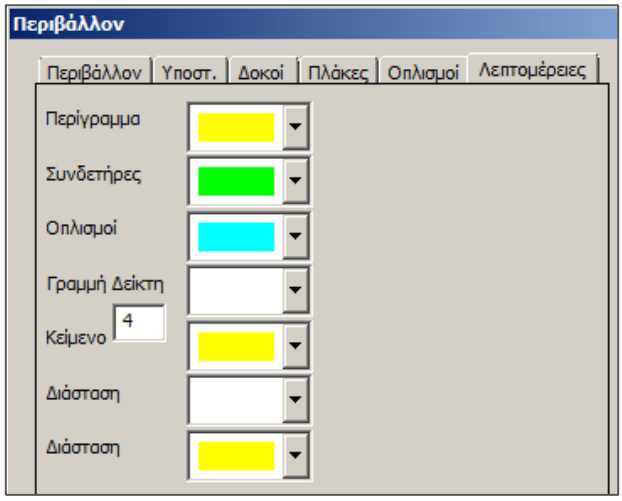
Here you specify the size and colour of the text of the reinforcement of the plates, pedestals and additional beam supports.



In the field next to
"Lines": the distance between the bars (evenly-spaced) of the reinforcement of the plates is specified.
"Text": the relative distance in the text of the reinforcement of the plates is specified.

⚠ It refers to and updates already designed armatures as well as new ones that will be introduced in the future with the corresponding commands.

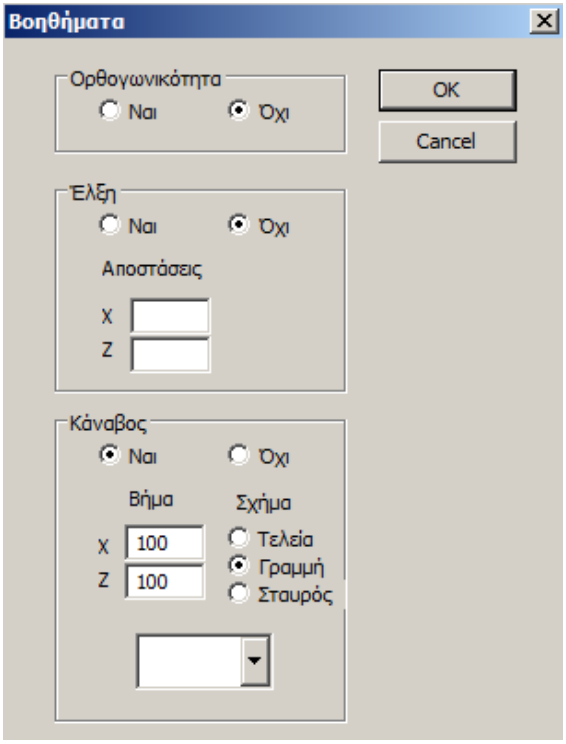
Details:



You specify the colour for the Outline, Connectors and Main Reinforcements of the column details.

⚠ It concerns and updates details already planned and new details to be introduced in the future.

Aids:

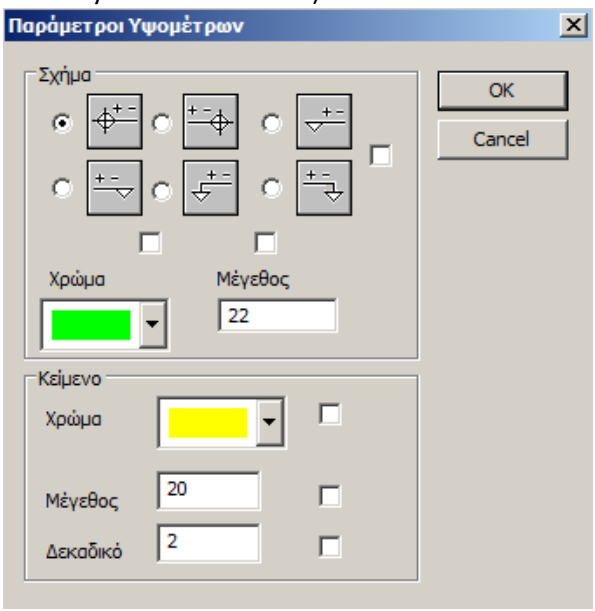


α. Orthogonality : By selecting Ναι you bind the mouse to move horizontally or vertically.
β. Attraction : Whether or not you activate the snap" at the points of the canvas. In the "Distances" by x or z positions, you specify the maximum distances in centimeters where the snap will occur when the mouse pointer approaches the canvas points.

c. Canavus : You choose whether you want the points of the canavus to be displayed on your screen or not, by selecting Ναι or Οχι respectively. If you select Ναι , you define the "Step" of the canavus in centimeters, its "Shape" (Dot, Line or Cross) as well as its "Color".
The acceptance of your choices is done with the key OK or cancel by selecting Cancel.

Altitudes:

Here you specify the shape of altitude symbols, the colour and the size of the symbols in centimetres to scale. (Size refers to the length of the vertical straight line from the horizontal line of the symbol downwards).

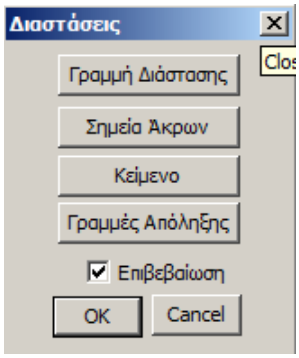


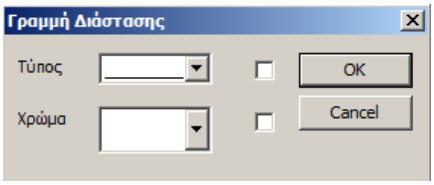
In "Text" you set parameters for the altimeter text (colour, size and number of decimal places).

ATTENTION!

If you activate located to the right of the parameters listed in the dialog box, the corresponding parameter will be updated in all altitude symbols already entered in your xylotype.
The acceptance of our choices is done with the key OK or cancel by selecting Cancel.

Dimensions:

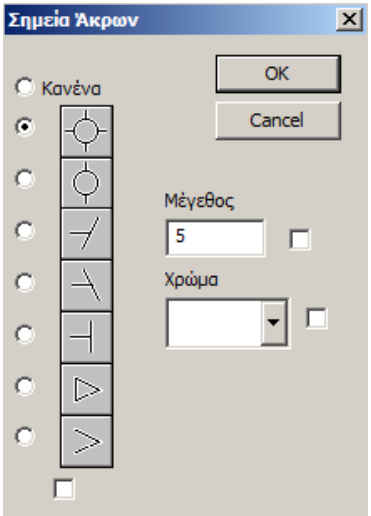




α. Dimension line : In the dialog box that opens, select the "Type" for the dimension lines from available ones displayed, by pressing with the mouse on the corresponding , as well as their "Color".

ATTENTION!

⚠ If you activate located to the right of the parameters listed in the dialog box, the corresponding parameter will be updated in all the dimensions already entered in your formwork.



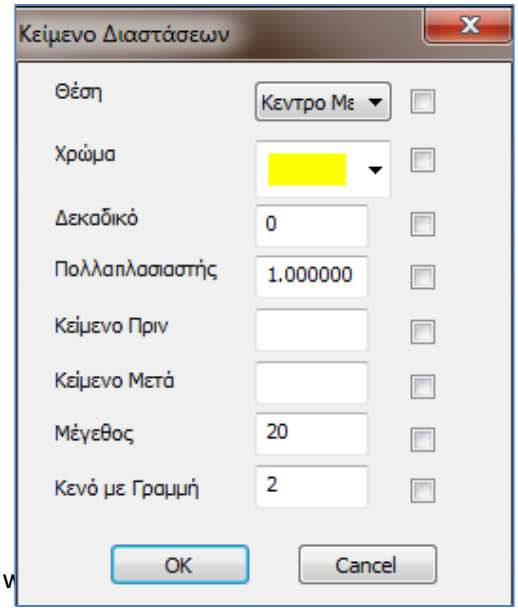
β. Edge points : Here you can select the type of the "edge points" of the dimensions from the available symbols of the program as well as their size and colour.

By selecting None no edge points will be drawn on the dimensions to be entered.

ATTENTION!

⚠ If you activate  located to the right of the parameters listed in the dialog box, the corresponding parameter of all the dimension edges already entered in your formwork will be updated.

Dimensional Text:



Position : Specifies the position of the text in relation to the dimension line. Specifically :

i. Center-Middle : The text will be written in the middle of the dimension line and in the center of the dimension line, interrupting it.
ii. Center-Up or Down : The text will be written in the center of the line up or down respectively, without interrupting it.
iii. Left-Right : The text will be written to the left or right of the dimension line. This option is appropriate to use in cases where the dimension is smaller than the size of the text.

Color : You select the color of the dimension text and thus the thickness with the

w

Decimals : Give the number of decimal digits of the dimension texts.

Multiplier : Specify a factor by which each dimension to be inserted will be multiplied.

EXAMPLE:

Let's say you have chosen centimetres as unit of length measurement, so all dimensions will be written in centimetres. By setting the "Multiplier" option to 0.01, all dimensions will automatically be given in metres (e.g. dimension 580 will be given in 5.80).

Text before/after : Enter some text that you want to be displayed before or after the dimension.

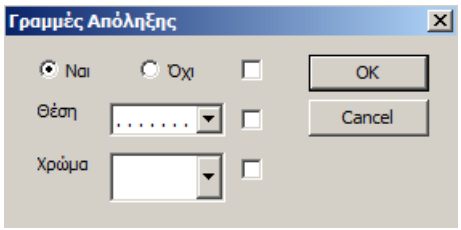
Size : Specify the size of the text in centimeters to scale.

Text Offset : Specify the distance of the text from the dimension line.

ATTENTION!

⚠ If you activate located to the right of the parameters listed in the dialog box, the corresponding parameter of the dimension text already entered in your xylotype will be updated.

Outlet Lines:



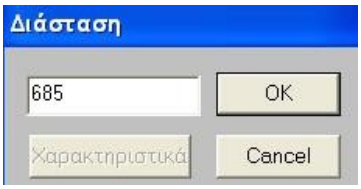
You specify the color, style of the end lines of the imported dimensions and whether they will be drawn (Ναι) or ( ). Οχι

ATTENTION!

⚠ If you activate located to the right of the parameters listed in the dialog box, the corresponding parameter of the end lines of the dimensions already entered in your formwork will be updated.

End lines are straight segments that start from the points in the drawing, the distance between which you want to calculate, and end at the ends of the dimension.

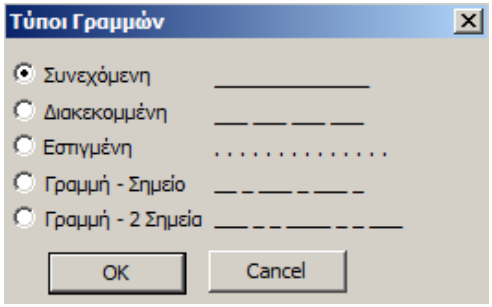
e. Επιβέβαιωση : If the option is enabled before a dimension is entered, a dialog box will open with the value of the calculated dimension which can be modified.



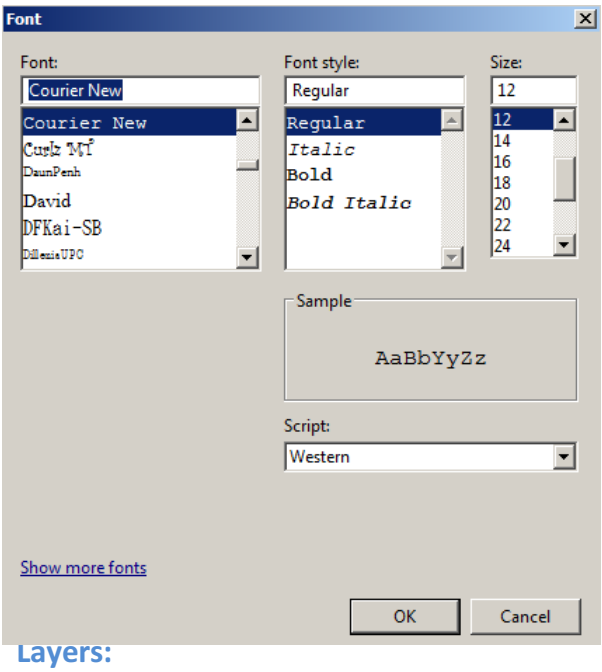
This option does NOT apply to dimensions entered by intersection.

Lines :

Select the type of lines to be inserted in the styluses with the command "Lines-Arcs-Circles >> Lines ".



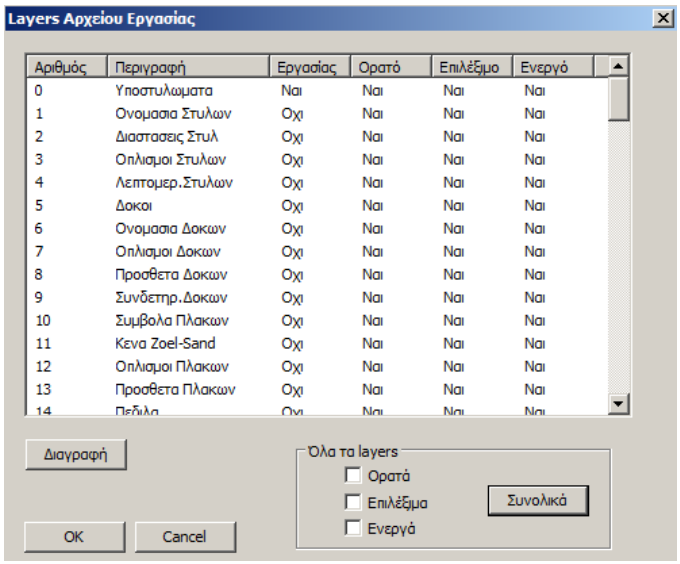
Text:



You choose the type of font to which each text that will be inserted in the xylotype will obey, either automatically by importing a design, or by the user with the option "OTHER FACTS >> Text".

Layers:

With this option you specify the parameters of the slides (layers) available in the program or those you create with the same option.



1. Status

α. Work (Work) : Every entity that is inserted in your formwork is registered in the work layer. Working layer can be only one at a time. Changing the state of a layer to make it work (from "No"¨ to switch to "Yes" state) is done by double clicking twice with the mouse on the "No" of the "Work" column. next to the layer name. The reverse is NOT true. The change from status "No" to "Yes"

is done automatically by doing some other work.

ATTENTION!

⚠ Making a layer "Working" automatically makes it "Visible (Visible)", "Eligible (Eligible)" and "Active (Active)".

The "Work" layer appears on your screen in the "Layers" group.

β. Visible : "Yes" : The contents of the layer are displayed on the screen.

"No": the contents of the layer are not displayed on the screen, but they still exist and can be displayed when you change the layer to visible.

ATTENTION!

⚠ Changing the state of a layer to make it visible (from "No" state to "Yes" state and vice versa) is done by double clicking twice the mouse on the column "Or." next to the name of the layer.

Layers in positions 1 to 32 relate to design entities of the formwork drawings, while those in positions 50 and 59 relate to entities of the beam reinforcement deployments.

c. Eligible (Selectable) : "Yes" : Entities included in an Eligible layer can be selected for processing.
"No": entities included in an Ineligible layer CANNOT be selected for processing.
Changing the state of a layer to make it selectable (from "No" state to "Yes" state and vice versa) is done by double clicking twice with the mouse on the "Select" column next to the layer name.

By making a layer "Selectable" it automatically becomes "Visible (Or.)" while the "Visible" layer is not necessarily "Selectable".

δ. Active : Making a layer "Active" automatically makes it "Selectable" and "Visible". It is a quick way to make a layer "Selectable" and "Visible" together.
Individual modifications of the state of a layer can be made by pressing twice with the mouse on the corresponding field. If you want to change the state of all layers as a whole, you will select the state from the ¨All Layers¨ option group and press on the 

button.

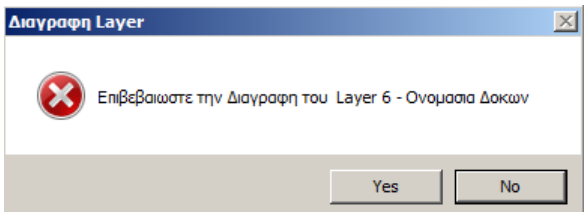
: With this option you can delete the content of a layer without deleting the layer itself. You press once with the mouse on the layer whose contents you want to delete to darken the entire layer.

line and press the button 

. A dialog box appears where confirm the deletion by selecting 

or cancel by selecting 

.

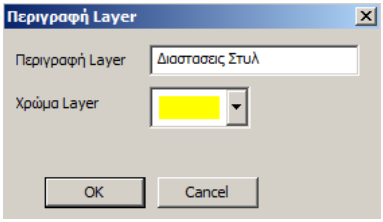


ATTENTION!

⚠ Deleting the "Work" layer NOT be done. You must first make any other layer active.

2. Modify title/colour layer

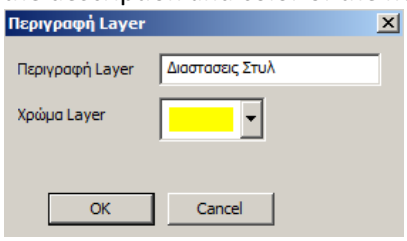
If double click on the name of an existing layer, a dialog box appears where you can modify the description or color of the layer.



Whenever you insert an entity into the xylotype, it is drawn in the color of the active layer and inserted into it.

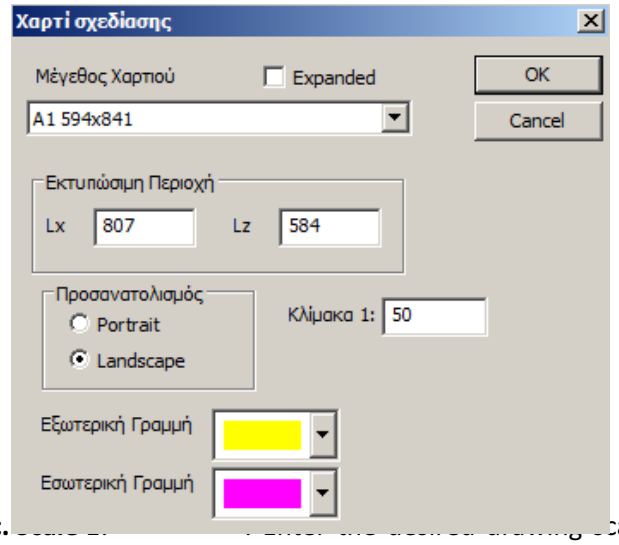
3. Insert new layer

To insert a new layer, go to the first empty position under the available layers of the program and double click twice, which opens a dialog box similar to the "Modify" dialog box where you enter the description and color of the new layer.



Drawing paper:

Here you select the drawing paper and set its parameters. In the dialog box that opens, you can see :



α. Paper Size : Pressing to the right of the paper dimensions field opens the list of available standard paper sizes. If select "User....." you can set your own paper dimensions in the Lx and Lz positions in the "Printable Area" field.
For any other standard size the printable areas are automatically displayed.

β. Expanded **Expanded** : Activating this field increases the limits of the printable area, as long as the same state is selected on the printing medium (e.g. plotter).

c. cale in the box (e.g. for a drawing scale of 1:100, enter the number 100 in the box). The default scale from the program is 1:50.

δ. Orientation : You choose Portrait in case you want the paper to be drawn with its small dimension (Lz) parallel to the x-x axis.

The default selection from the program is Landscape where the Lx dimension of the paper is horizontal.

ε. Inner / Outer Line : Select from the available palette the color of the paper boundary lines to be printed with the corresponding thickness.

You accept the above options by pressing on the OK button and a frame appears on your screen (with orientation and size according to your choices) which is "dragged" by the mouse pointer and positioned (by pressing the left button) appropriately so that your formwork is drawn in the desired position.

Introduction

The "Import" command group contains commands that allow you to quickly import all the drawings of the study (see command 1.2 Import)

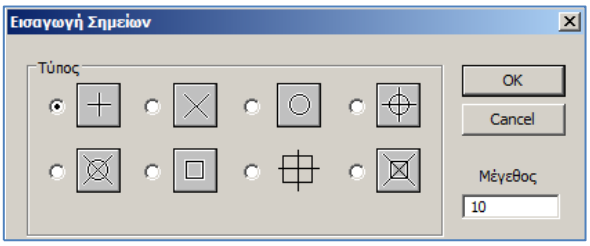
Selecting each command opens the list of corresponding drawings and details from which you can select one (or all) of the beam extensions and metal connections for import into the formwork environment.

Design

The "Drawing" command group contains commands that allow you to draw geometric shapes.

3.1 Lines - Arcs - Circles:

Point: to insert auxiliary points within your drawing. When inserting them you can use the auxiliary tools to determine their exact positioning.



In the dialog box that appears, select the format of the point and its size in centimeters.

⚠ The size of the point is purely visual and determines how distinct the point is in your drawing.

Line - Segment: to draw individual line segments by defining with the mouse the two points of their edges.

Line - Segments: to insert continuous line segments, composing a broken line which will consist of individual line segments in continuity. These segments can be edited (move, delete, etc.) independently. When you insert them, the end of each drawn segment is also the beginning of the next one. The points are indicated with the left mouse button, and after inserting the last one, you press the right button to complete the insertion process.

Line - Continuous Line: to draw continuous lines using the same process as described above, except that the resulting polygonal line is a single entity (pline) that you can edit as a whole as a single entity.

⚠ To "break" the Continuous Line into its constituent parts, use the "Explode" command from the "Interventions" command list.

Remarks :

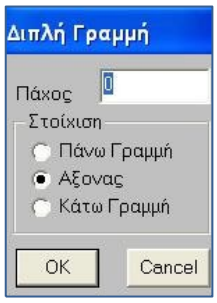
1. The type of line to be drawn with the line drawing commands is specified with the "Parameters - Lines" command (continuous, focused, etc.).
2. When drawing "Line >> Section" and "Line >> Sections", you can differentiate the Line type from the parameters, so that you get, for example, a broken line consisting of sections of different types.

⚠ This does NOT apply to the "plines" drawing which will obey the original default line type.

Double Line - Department/Sections/Continuous Line: to draw double lines in exactly the same way as described in the command "PLANNING>> Lines" (Department, Sections or Continuous Lines).

⚠ THE "DESIGN >> Double Line" can be used to insert into your design, lines representing beams.

After activating the command, a dialog box opens in which you will specify the parameters of the double line which are :



Thickness : Enter the distance between the two lines, in centimetres.
Alignment : You specify which of the two lines, or their axis, the double line will be drawn.

The type of lines is also specified here with the command "Parameters - Lines" (see remark in the previous paragraph).

Remark :

⚠ During the drawing of the double line and until you press the right mouse button to complete the insertion, a single line is drawn on our screen, the one you have specified as an alignment in the dialog box that opens when the command is activated.

Arc: to draw arcs in 4 alternative ways, depending on the data you have available for entering them.

Center, Ray Graphics

1. After activating the command, you use the left mouse button to set a point as the centre of the arc.
2. You define a second point on the screen (which can also be defined with the help of the design tools), which will determine, together with the centre, the radius of the arc. Immediately the circle is drawn from which you will graphically cut off (with the help of the left mouse button by defining 2 points as edges) the arc are interested in.

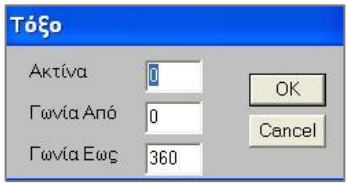
⚠ The positive direction for the definition of the ends of the arc is counterclockwise. This means that each time the part of the circle that starts from the first end you define and ends, if you move counterclockwise, at the second end, remains on the screen.

Center, Radius Data

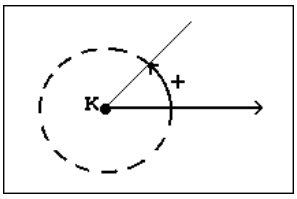
The import procedure is the same as described above, except that you specify the radius of the arc by typing its value in centimeters in the dialog box that opens immediately after selecting the arc center.

Centre, Data, Data

You define the point of the arc's center and a dialog box opens immediately in which you enter, in order, the Radius of the arc, the Angle from which it will start to be drawn and the Angle that defines the point of the end of the arc.



⚠ The angles are measured counterclockwise. The starting point of measurement is the semi-major line having its origin at the defined centre of the arc and its direction in the positive direction of the x-x' axis.



⚠ The values of : "Angle From" and "Angle To" can also be negative.

By 

closes the dialog box and draws the arc on the screen.

Three Points

To draw the arc, you define 3 points with the left mouse button (the beginning, an intermediate point and the end of the arc).

Circle - Center, Beam Graphics/ Center, Data/ Three Points:

Draw circles by giving their centre and radius (graphically or by typing its value) or three points. The drawing process is similar to that of the arcs described in the previous paragraph.

3.2 Other

Schemes: Ring:

When inserting a Ring, a circle is drawn (with the geometric elements we define) filled with active color.
The method of use is similar to that of drawing circles.

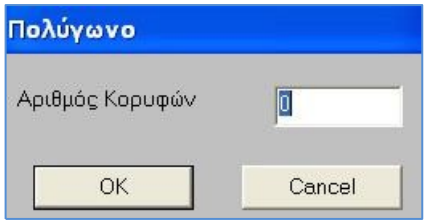
Polygon:

With this command you can draw polygons inscribed or outlined on an existing circle or arc. The drawing procedure for both cases is the same.

1. After activating the command, you select with the mouse the circle on which to draw an inscribed or circumscribed polygon.
2. With the help of the semi-line with the centre of the circle as a vertex which follows the mouse pointer, you define the position of the vertex of the polygon which is determined as follows:

i. For an inscribed polygon it will be the point of intersection of the circle and the semilinear line to be defined, or its extension.
ii. For a circumscribed polygon the point will be on the half-line and outside the existing circle.

3. Immediately after defining the first vertex of the polygon, a dialog box opens where you type in the number of vertices of the polygon.



With  the dialogue box is closed and the polygon is drawn.

Rectangular:

Horizontal: Draw a rectangular parallelogram with sides parallel to the two principal axes, using the mouse to define the vertices of one of its two diagonals.

Oblique : Draw a rectangular parallelogram with sides oriented with respect to the principal axes. The program, before suggesting the two points of its diagonal, asks you to indicate an existing line in the drawing, the direction of which will be the direction of the rectangle to be drawn.

Striping:

Object: With this option you draw delineation within plines, circles and general design entities that "come" or are drawn with continuous lines.

How to use :
1. You activate the command " OTHER FIGURES>> Object Deletion".
2. Use the left mouse button to select the objects to be deleted.
3. You indicate the end of the options with the right mouse button.
4. In the box that opens, enter the angle of inclination of the lines and the distance between them, in centimetres to scale.
5. Selecting "O.K." the diagram is designed which is included in the active layer and as a design entity, it obeys the entity editing commands (drag, delete, etc.).

⚠ The "open continuous line" delineation results in the drawing of a delineation of the area defined by the open line and imaginary straight line segment connecting its beginning and end.

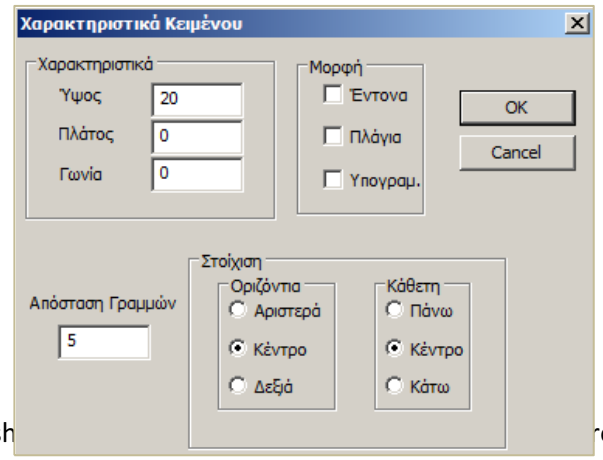
Area : The program allows you to draw an area that you indicate with the mouse, defining its vertices, without a drawn continuous line.

How to use :
1. You activate the command "OTHER FIGURES>> Area Delineation".
2. You define the delineation area by pointing the mouse at its vertices.
3. Selecting the last vertex of the area, you press the right mouse button and the chart parameters panel appears.

Text:

With this option, you can insert text into your design in the color that is currently active. The text will be inserted into the working layer.
Activating the command opens a dialog box in which you set its parameters. Specifically :

1. Height/Width : Enter the Height/Width you want the text to have in centimeters.



⚠ If you give a value to only one of the two parameters and leave the other one equal zero (0), the program will draw text with a size, for this parameter, of the value you gave and for the other one the corresponding value for the given font.
If you provide values in both fields then the text will be drawn obeying the values you provide.
2. Angle : Type the angle of the text tilted to the horizontal in degrees.
Format : You can choose whether the text sh ressing the mouse on the corresponding box.

Gap between lines : Enter the distance in centimeters between lines, in case we enter text on more than one line.

Alignment :
Horizontal : Specify the alignment of the text to the beginning, middle or end by selecting Left, Center, Right respectively.
Vertical : The vertical alignment of the text is specified accordingly.

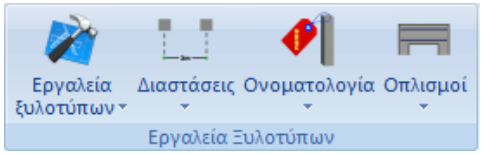
Accepting your choices is done by pressing the 

button, closing the dialog box and indicating with the mouse the point of insertion of the text. You type the text and press the "Enter" key each time you want to change the line. Each line obeys the attributes you previously defined and is an independent entity.

The end of the text input process is done by pressing the right mouse button.

⚠ Modifying the content of the text and/or its attributes is done with the command "MODIFY >> Correction".
The color change is done with the command "MODIFY >>Color".

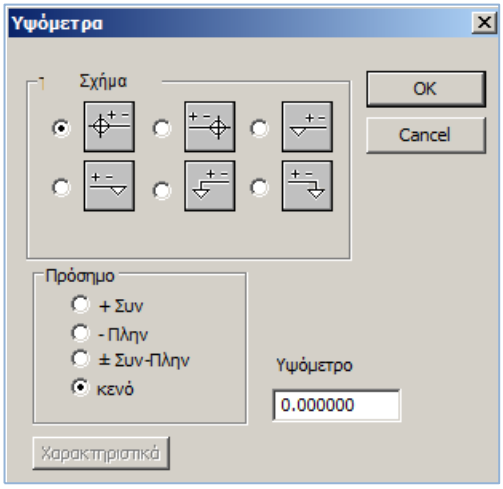
Wood type tools



The command group "Wood type tools" contains useful commands for displaying and completing the wood type drawings.

4.1 Woodworking tools:

Altitudes: to insert altitude symbols into your project.
After indicating the point where the altimeter will be placed, in the options panel that opens you enter :



1. Figure : You select the symbol type by pressing the mouse on the corresponding circle.
2. Sign : Set the sign that the altitude value will have.
3. Altitude : Enter the altitude without the sign (previously set)

Pressing the 

button closes the dialog box and the altitude is plotted at the position originally specified.

⚠ The 

field becomes active only when "Correct" altimeters from the "MODIFY" options list where you can modify the parameters of the altimeters (color, size, etc.) through the corresponding dialog box.

Slab gaps: to introduce "symbolism" indicating that there is NO slab (it is a gap) at the specific location of the formwork so that it is obvious during the construction phase of the project. The program at the corner of the panel blank you indicate will insert two "delineated triangles" with delineation parameters that you define in the dialog box that opens :

How to use :

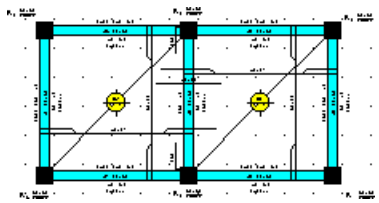
1. You activate the command "Lumber Tools>> Slab Gaps".
2. Point the mouse at three inner vertices of the void. At the intermediate vertex you indicate (second in order) the bases of the two drawn triangular areas will be drawn.
3. After indicating the third vertex, the above dialog box will open where you will enter the parameters of the diagram (angle and distance between the diagrams, in centimeters).

Pressing on the 

button closes the dialog box and draws the blank of the plate in the position you specified.

⚠ Removing the symbols of the empty plates is done with the command "INSERT >> Delete" and pointing with the mouse to the deleted area.

Slab diagonals: to insert into an existing slab two diagonal auxiliary lines that indicate its boundaries.



How to use :

After activating the command, select with the mouse the symbol of the plate where you want to insert the auxiliary boundary lines.
Use the mouse to indicate two points where the boundary lines start and end.

With the indication of the second point, the boundary lines are drawn and at the same time the position of the symbol of the plate is changed so that it is in the middle of the boundary line.

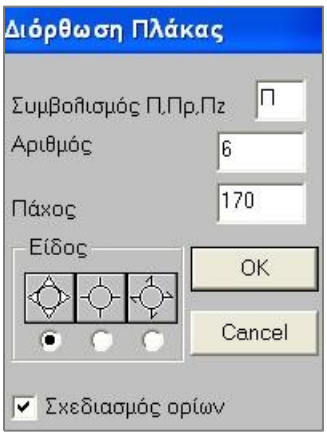
To remove designed boundary lines from slabs : a. Individual or

selective removal.

1. From the "EDIT" command list, activate the "Correct" command and select the slab from which you want to remove the boundaries.
2. In the dialog box that opens, disable the parameter 

and press 

, whereupon the drawn boundaries are removed.



If you ask for "Correction" again and activate the "Correction" button Σχεδιασμός ορίων , then the limits will appear again. This is only true in those cases where there were drawn boundaries and they were removed by the procedure described above. NO ALTERNATIVE WAY OF INTRODUCING BOUNDARIES.

β. Total removal of the boundaries from all the plates designed

1. From the "PARAMETERS" command list, activate the "Plate faces" command
.
2. In the dialog box that opens, click with the mouse on Ορια to remove the inside indication from the field and on ☐ under the number fields (to the right of "Limits") to display the inside indication.
This way you instruct the program not to draw boundaries and to update all the already drawn plates with this option.

Pressing on the OK button closes the dialog box and leaves the drawn boundaries from all the existing plates.

If you re-enter the dialog box and check Σχεδιασμός ορίων and the box to the right of it, the boundaries will be redrawn. This only applies to those cases where there were designed boundaries and they were left using the process described above. NO ALTERNATIVE WAY TO IMPORT BOUNDARIES.

Fixed Columns: to enter in your design "Fixed Columns", (circles), "Filled" or not, depending on whether the corresponding option in "Parameters >>" is enabled. Pillars".

The Pillar Stands introduced in this way are design entities. If, for example, a substructure already has a fixed point, it will not be deleted if you design a new one with the above command. Deleting Fixed Subpillars is done with the corresponding command from the "OPTIONS" option list.

4.2 Dimensions

to enter dimensions (distances between points) in any direction in your drawing, well as dimensions for arcs, circles and angles.

α. Standard:

i. Horizontal : Dimensions are drawn parallel to the horizontal axis (x-x'). The values shown are for horizontal projections regardless of the position of the points in your drawing.

ii. Vertical : The dimensions are drawn parallel to the vertical axis (z-z'). The values shown are for vertical projections regardless of the position of the point in your drawing.

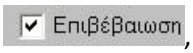
iii. Lucky you: The dimensions are drawn parallel to the imaginary line defined by the two points chosen to dimension. The values shown are the "real" distances between the defined points.

How to use :

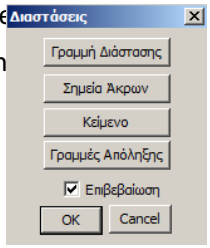
1. You activate the command "DIMENSIONS>> Standard" (Horizontal, Vertical or Random).
2. You select the location where the dimensions will be displayed by pointing the mouse at any point in your drawing.
3. You select the points between which the dimension is requested, which is calculated and written at the position you previously set.

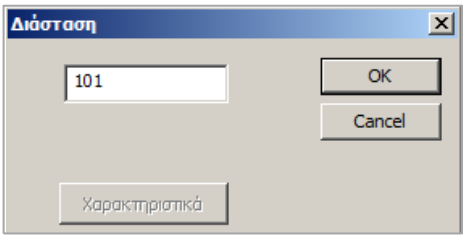
Remarks :

1. The program allows you to enter consecutive dimensions if you specify more than two points. In this case, each new point you specify is the second end of the dimension which has as its origin the immediately preceding point specified. (The end of one dimension is the beginning of the next dimension).

2. If the "Parameters - Dimensions" option is activated in the "Parameters - Dime 

, then each dimension will be prompted for confirmation through dialog box that opens before it is entered.



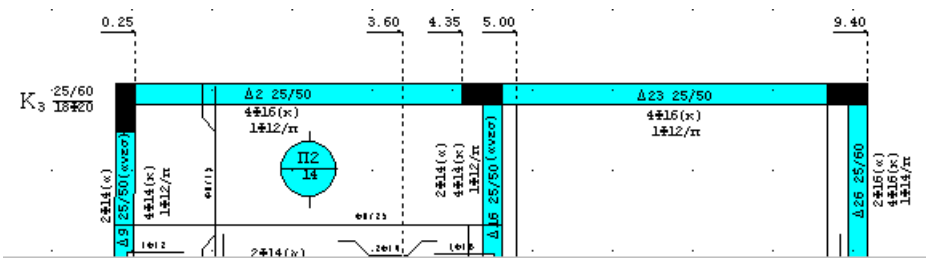


β. Engraving - Horizontal/Vertical/Random: to enter consecutive, continuous dimensions with the measurement start point indicated by you, in horizontal, vertical or random direction respectively.

How to use :

1. You activate the command "DIMENSIONS>> Engraving - Horizontal".
2. Select with the mouse the position where the dimensions will be drawn.
3. You select the starting point of the measurements.
4. You indicate successive points to calculate the distances from your starting point.

The format of the written dimensions will have the following format :



c. By Cross-section View X/Y View/ Line View: to automatically enter multiple, contiguous dimensions in any direction.

How to use :

1. From the command list "LINES - AXES - CIRCLES", draw a line that intersects your formwork at the points between which you want to calculate dimensions.
2. Activate the command "DIMENSIONS >> With intersection". Select "View x-x', z-z'" or "View line" if you want the dimensions to be written as horizontal or vertical projections, or projected on the intersection line respectively.
3. Use the mouse to indicate the point where the dimensions will be written.
4. You select the intersection line with the mouse and the dimensions are automatically drawn.
5. Delete the auxiliary intersection line.

Remarks :

1. The intersection line can be a Section, Sections or a Continuous Line (pline).

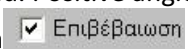
In the case of Line - Segments you must indicate each time with the mouse the position where the dimension will be indicated before selecting each line segment.

2. In the case where the line of intersection is a continuous line and intersects points on the formwork which will give zero dimensions, these are NOT indicated.
3. The format of the imported dimensions, where possible, obeys the "Dimensional Parameters".
4. Importing dimensions "By intersection" does not obey the "Confirm" parameter

δ. Angle degrees : Calculate and plot the size of an angle in degrees. The indicated dimension is in the form of an arc between the sides of the indicated angle. **Mode of Use :**

1. You activate the command "DIMENSIONS>> Angle Fates".
2. Select with the mouse the position where the dimension will be drawn. The program will draw an arc with a radius defined by the point you indicate and the vertex of the corner.
3. You select the two half-lines (sides of the corner) that you want to dimension.

The program will dimension the angle defined between the first and second semi-line in the order defined. Positive angle direction is counterclockwise.

If the option 

is activated in the "Parameters - Dimensions", then before entering the dimension of the angle you will be asked to confirm it in a dialog box that opens.

ε. Arc:

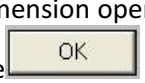
Length: The length of the arc is calculated and displayed. The dimension line is an arc concentric with the drawn one at the position indicated with the mouse.

Radius: Enter the length of the radius of the circle of which the drawn arc is a part. Draw the radius from the center of the arc to a point in the direction indicated with the mouse.

l= : Similar to Length option. Here the calculated dimension is entered in form l=a, where a is the length of the arc, and an arrow is drawn indicating the arc being dimensioned.

R=: Similar to the previous option except that the length of the radius is indicated here.

How to use :

1. Activate the command "DIMENSIONS>> Arc dimensions" (Length, Radius, l or r).
2. Use the mouse to point to a point on the screen where the dimension will be drawn. This can be inside or outside the arc.
3. You select the arc you want to dimension and immediately the dialog box for confirming the dimension opens, if the corresponding option in the dimension parameters is enabled. Pressing the 

button displays the calculated dimension.

f. Circle - Horizontal/Vertical/Night:

The dimension you enter refers to the diameter of the circle you indicate and is drawn parallel to the horizontal or vertical axis or in a random position respectively, at the point you select with the mouse.

Circumference l = : Write the length of the circumference of the circle in the form l=a, and draw an arrow indicating the circle dimensioned.

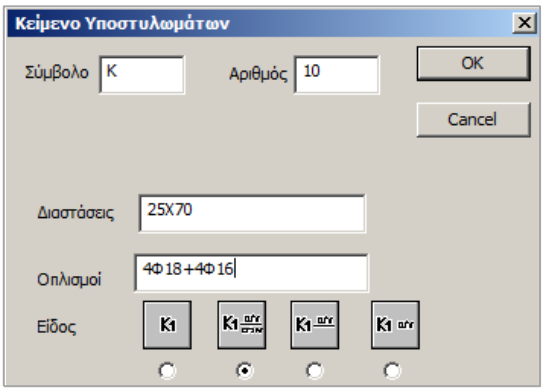
Radius r : Similar to the previous option except that the radius of the circle is indicated here.

4.3 Nomenclature

to enter the labels on Poles, Beams and Slabs .

Pillars:

First you choose a spot on your plan where the "sign" of the pole will be placed. Immediately a dialog box opens where you enter the data:



Symbol K,T,F: Type a letter for the column symbol. .
program suggests : K for "columns", T for "wall", F for "planted", but you can give any character you wish.

Number.
subtree, which will be written as an index of the symbol.

Dimensions: Enter the dimensions of the column e.g. 100/30/30/80 for C-shaped columns with sides of 100 and 80 cm respectively and a leg thickness of 30 cm.

Reinforcements : Give the main reinforcements of the post : e.g. 4Φ18+4Φ16

Type: in this position you select the format in which the label of the column will be drawn, by pressing with the left mouse button on the corresponding circle.

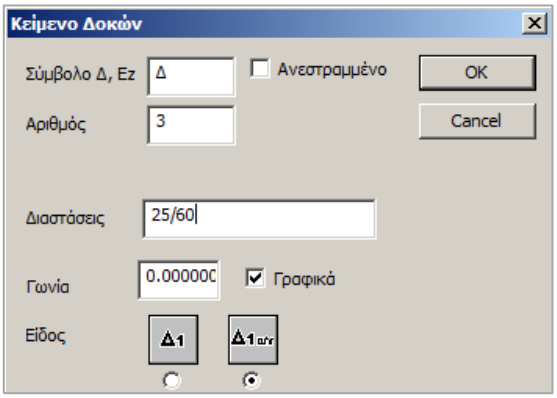
For presentation of formwork in the final design phase where the drawing and indication of the reinforcements is not required, but only the dimensions of the entities, you will choose the format .

By OK accept the above options or cancel by pressing with the mouse on the button Cancel

Remark :

In case you enter values in the "Dimensions" field and select "Type , the dimensions will NOT be indicated but not lost. With the "Correct" option within the "Modify" command list you can change the "Type" for the Pillar text and the dimensions you previously typed will also appear.

Beams:



Symbol D, EZ: For the beam symbol the program suggests : D for "Beam", EZ for "Reinforced Zone", but you can give any other character you wish.

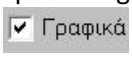
Number: enter the number of the beam which will be written as an index of the symbol.

Dimensions: Enter beam dimensions e.g. 25/60 for a beam with bw=25 and h=60 cm.

: Press with mouse on  if the beam that you want to place is inverted. In this case the text on the formwork will be in the form D5 25/60 (inverted).

Corner: Specify the angle of inclination at which the beam text will be written. Angles are measured counterclockwise starting from the horizontal direction to the positive of the x-x' axis.

The specification of the address at which the text of the beams will be written can also be specified graphically if the option



After accepting your choices by pressing the 

button, the program asks you to indicate the line of the drawing with which the text will be parallel. (The message appears in the bottom status bar of the screen).

Type: choose whether to display only the Symbol and Number of the beam 

or the full text 

(Symbol, Number, Dimensions).

In case you enter values in the "Dimensions" field and select "Type 

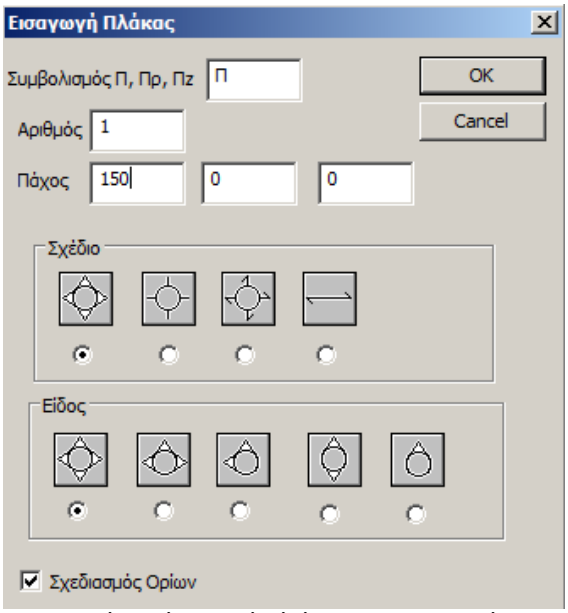
, the dimensions will NOT be indicated but not lost. With the "Correct" option within the "Modify" command list you can change the "Type" for the Beam text and the dimensions previously typed will also appear.

The acceptance of our choices is done by pressing the 

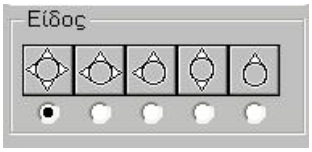
button or their cancellation by pressing the 

field.

Plates:



Symbol Π,Πρ,Πz : The symbol of the plate to be inserted :Π for solid plates in general, Πρ for slabs with projections, Πz for plates with ribs. Alternatively, you can type any character you wish.
Number: enter the number of the plate which will be written as an index of the symbol.
Thickness: enter the thickness of the plate in millimeters.
e.g. for a 15 cm thick plate we will give 150. On the formwork the thickness will be given in centimetres.
Plan: You choose the type of symbol of the plate (, or)



Type : Select the symbol that represents the support conditions of the plate (in order of four-row, three-row, two-row, two-row, two-row prop)

Σχεδιασμός Ορίων : Activate this option when you want to draw, together with the symbol of the slab, the two "diagonal" lines indicating its boundaries.

α. The Σχεδιασμός Ορίων option is **disabled**. **How to use :**

1. You activate the command " TIMBER>> Nomenclature>> Slabs " and the dialog box entitled "Insert Slab" opens.
2. Enter values in the Symbol, Number, Thickness fields and select Design and Plate Type.
3. Pressing the OK button closes the dialog box.
4. Point the mouse at the point in the drawing where the entity will be placed.
5. Select a straight line (side of the slab) in relation to which the slab symbol will be placed (see also note below).

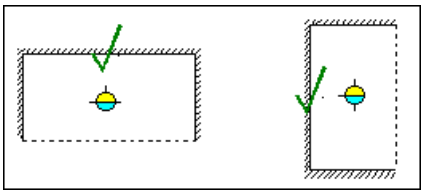
β. Σχεδιασμός Ορίων is **active**.

Follow steps 1 to 3 of the course described above.
4. Use the mouse to indicate the 2 points where the boundaries of the plate start and end.

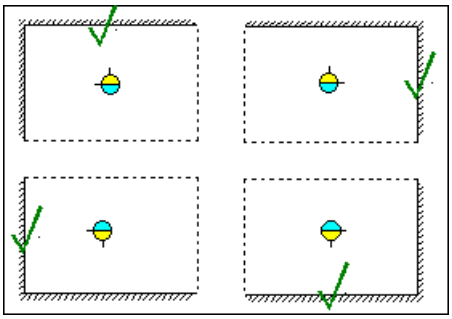
5. You select the side towards which the symbol will be placed.

Remarks :

1. For the insertion of the three-way plate symbol, always indicate the intermediate support of the plate (√).



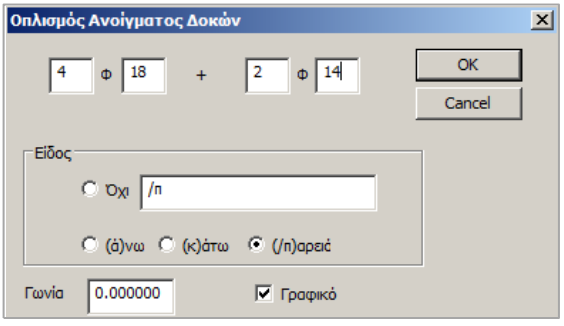
2. To insert the symbol of the dividing plate, you indicate the side that finds the next bearing if you move counterclockwise (√).



4.4 Armament

s: Opening beams:

This command allows you to place reinforcements for beam spans in tension or compression. After you have indicated the point where the reinforcement is to be placed, in the dialog box that opens enter :

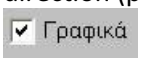


: The number and diameter of irons. It is possible to combine two different steel diameters.

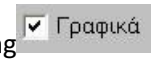
Type : Select the type of reinforcement to be inserted. (top down, /back). The corresponding comment will be written right of the armaments. If you select 

, will be written, and you can type any comment you wish up to 10 characters in the corresponding field.

: Enter the angle of inclination of the text with respect to the horizontal direction (positive or anti-weather).



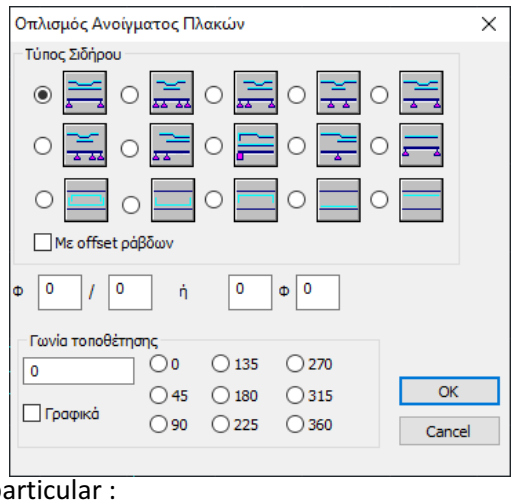
: The Beam Armour text is placed parallel to an existing direction of the design, which you indicate immediately after closing the "Beam Opening Armour" dialog box.

By selecting 

the "Angle" option is automatically disabled, so any value entered in the field is NOT taken into account.

Plate Opening:

In the dialog box that opens, you specify the data of the plate reinforcement you want to place:



Iron type : Here you can select the type of iron according to the type of plate and its geometry, by pressing with the mouse on the corresponding circle.

In particular :


Reinforcement of an amphibious plate (in the direction in which you will place the reinforcement) without continuity on either side of its supports. For its placement you will indicate the outer faces of the bearing beams
(2 lines).


Amphitheatre slab with continuity on both sides of the action with amphitheatre slabs. You will indicate in order :

- the opposite boundary of the first adjacent plate

- the outer face of the first bearing beam of the plate on which the reinforcement is to be placed.
- the outer face of the second bearing beam of the plate on which the reinforcement is to be placed.
- the opposite boundary of the second adjacent plate (4 straight lines)

 Amphitheatre slab with continuity on one side of its seat with another amphitheatre slab. You point with the mouse:

- the opposite boundary of the adjacent plate
- the outer side of the common bearing beam of the plates
- the outer face of the second bearing beam of the plate (3 straight lines)

 Uncontested slab with continuation on both sides of its seat with projections. You define in order:

- the free end of the first sheepskin
- the outer face of the first bearing beam of the plate
- the outer face of the second bearing beam of the plate
- the free end of the second projection (4 lines)

 Amphitheatrical slab with a continuation on one side with a projection. Define:

- the free end of the sheepskin
- the outer face of the first bearing beam of the plate
- the outer face of the second bearing beam of the plate (3 straight lines)

 Amphibious slab with a continuation on one side with a projection and on the other side with an amphibious slab. You define :

- the free end of the sheepskin
- the outer face of the first bearing beam of the plate
- the outer face of the second bearing beam of the plate
- the beam of the adjacent amphibious slab (4 straight lines)

 Triple plate with continuity to the bearing side with amphibious plate. Indicate:

- the boundary of the adjacent plate opposite their joint bearing.
- the outer face of the bearing beam of the truss
- the free end of the tricuspid (3 straight lines)

 Three-way slab without continuity with others in the direction of reinforcement placement. Indicate

- the outer side of the bearing beam of the plate
- the free end of the tricuspid (2

straight lines)

 Three-tiered slab with a continuation on the side of the seat with a projection. Set :

- the free end of the sheepskin
- the outer side of the bearing bracket
- the free end of the tricuspid (3 straight lines)

 Distribution armament. Indicate the two reinforcement limits for each plate.

 Upper+ Lower bars (with hook). Indicate the two reinforcement limits for each plate.

 Individual top or bottom irons (with or without hook). Indicate the two reinforcement limits for each plate. On these reinforcements you can use the tool of **etching**.

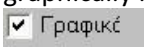
By pressing the  button next to the indication "With offset bars", it is possible to separate the straight bars from the broken bars of the plates.

: Enter the diameter of the reinforcement in the first field and the distance between the bars in the second field. When the reinforcement to be installed is a ribbed plate spacer, enter the values of the number of bars and their diameter respectively in the third and fourth boxes.

Remark :

At any one time you should only give values to one of two types of iron. If you give both, then the second one will prevail (ribbed plate intersection reinforcement).

Mounting angle : Select one of the default mounting angles for the direction of the reinforcement or type any angle in the corresponding field. The reinforcement direction can also be specified graphically if you activate the



in which case the program will ask you to indicate an existing straight line to determine the address.

The acceptance of all the above options is done by pressing the 

button or cancelled by pressing the 

button.

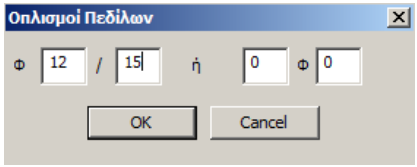
How to use :

1. You activate the command and set the parameters of the reinforcement to be placed as described above.
2. Pressing the OK button closes the dialog box and the program prompts you to define the lines for the placement of the reinforcement, depending on the type of reinforcement.
3. After you have set the last line, the armature appears on the screen and the program asks you to specify the point where it will be placed. The movement of the pointer on the screen is followed by the drawn armature until you indicate the positioning point with the left mouse button, at which point the armature is fixed there.

If you have enabled the parameter Γραφικά, before defining the lines for placing the rebars, you must indicate a straight line to define the direction in which the rebar will be placed. The reinforcements are designed as single entities (blocks).

Fields:

In the dialog box that opens, type :



: In the first field you give the diameter of the bars and in the second the distance between them,

ή

: In the first field you give the number of bars and in the second field the diameter their.

The dialogue box is closed by pressing on the OK field or cancelled by pressing on the Cancel.

How to use :

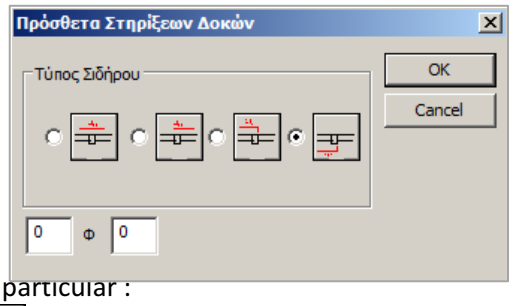
1. You are activating the command "Lumber Tools>> Armaments>> Field Armaments ".
2. The "Field Armaments" dialog box opens, where you type the details of the armament you want to place.
3. Press the OK field and the dialog box closes.
4. Indicate with the mouse the line from the sides of the pedestal to which the reinforcement will be placed parallel, which will be drawn with the same length as the length of the side of the pedestal you indicated. Moving the pointer moves the reinforcement visibly

on the screen and is finally placed in the position you select by pressing the left mouse button.

The reinforcements are designed as single entities (blocks).

Beam Support Additives:

With this command you place additional reinforcement for supporting beams on a tension or compression footing.
In the dialogue box that opens, enter :



Type of iron : Here you can select the type of iron according to the type of support, by pressing with the mouse on the corresponding icon.

In particular :

 Common additional reinforcement in intermediate support of beams in tension or compression. For its placement, you must indicate the two beams involved in the common support and the final position to be drawn on the formwork.

 Straight additional reinforcement, intermediate or end-supported, for a beam, tension or compression.
For positioning you first indicate the beam, a point the reinforcement will start (e.g. a top of the column where the beam rests) and the final position to be drawn on the formwork.

 Additional C-shaped reinforcement, intermediate or end support, for a beam, on its upper leg.
For positioning you first indicate the beam, a point the reinforcement will start (e.g. a top of the column where the beam rests) and the final position to be drawn on the formwork.

 Additional C-shaped reinforcement, intermediate or end support, for a beam at its lower leg.
For positioning you first indicate the beam, a point the reinforcement will start (e.g. a top of the column where the beam rests) and the final position to be drawn on the formwork.

How to use

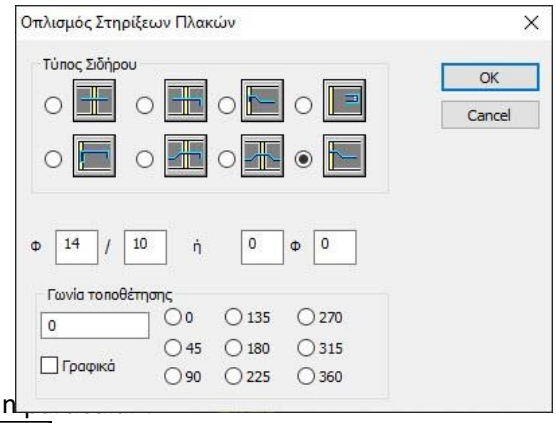
1. You activate the command "Lumber>> Reinforcements>> Beam Support Additions".

2. In the dialogue box that opens, select the type of reinforcement by pressing with the mouse on the corresponding circle.
3. Enter the number of irons in the first field and their diameter in the second.
4. Depending on the type of , indicate the beams and/or the necessary points for its installation.

The reinforcement is initially displayed on the screen, can moved parallel to the selected beams and is finally placed in the position you indicate with the mouse.
The reinforcements are designed as single entities (blocks).

Additions to Plate Supports:

In the dialogue box that opens, you specify the data of the additional reinforcements for the plate support that you will install.



Iron type : Here you can select the type of iron according to the topology of the plates.

In

 Straight-line reinforcement for intermediate support of continuous slabs (four-row, two-row or a combination thereof).
For its placement you must indicate the two sides of the common support beam of the plates.

 Rectilinear reinforcement for supporting a cantilever with continuity with another plate on the side of its engagement.
To position it, you should first point to the free end of the cantilever and then to one of the two sides of its supporting beam.

 Additional end plate support reinforcement or spalling reinforcement in amphibious slabs.
To install it, point to the two sides of the bearing beam starting from the outer one.

 Additional reinforcement of the free end of the free end of headlamps or tripods.

To place it, you point to the free end of the plate, select the semi-level to be placed by moving the mouse and set the point of its final placement.

 Armouring of a canopy support without continuity with another plate to the side of the support.
To draw it, indicate in order the outer side of the beam, the inner side and finally the free end of the cantilever.

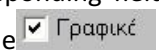
 Bendable reinforcement for supporting the cantilever with continuity with another plate on the side of its engagement. To place it, point first at the beam face, then at the free end of the cantilever.

 Flexible reinforcement for intermediate support of continuous slabs (four-strand, two-strand or a combination of both). For its placement you must indicate the two sides of the common support beam of the plates.

 New! Additional separation reinforcement. To install it, you indicate the two sides of the bearing beam starting from the outer one.

: Enter the diameter of the reinforcement in the first field and the distance between the bars in the second field.
ή


: In the first field you give the number of bars and in the second field the diameter their.

Mounting Angle : Select one of the default mounting angles for the reinforcement direction or type any angle in the corresponding field. The reinforcement address can also be specified graphically if you activate the 

option, in which case the program will ask you to indicate an existing straight line for specifying the address.

Accepting all the above options is done by pressing the 

button or cancel by pressing the 

button.

 **How to use :**

1. You activate the command and set the parameters of the installed reinforcement.

2. Pressing the 

button closes the dialogue box and the program prompts you to define the lines for the placement of the reinforcement, depending on the type of reinforcement.

3. After you have set the last line, the armature is displayed on the screen and the program asks you to specify the point where it will be placed. The movement of the pointer on the screen is followed by the drawn armature until you indicate the positioning point with the left mouse button, at which point the armature is fixed there.

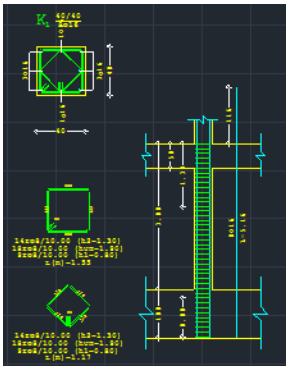
Remark :

If you have activated the parameter 

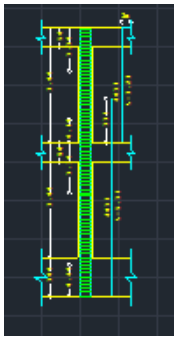
, before defining the lines for placing the rebars, you must indicate a straight line to define the direction in which the rebar will be placed.
The reinforcements are designed as single entities (blocks).

Vertical post cut:

Prerequisite: you must have edited the pole via the "Arming Details" command (it is sufficient to simply open the window).

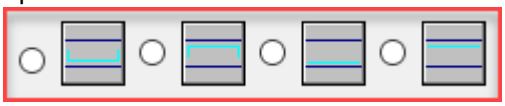


Then, on the xylotype, along with the detail of the pole, the vertical section of the pole will appear for the specific level.
To display the total vertical intersection, select the command "Vertical intersection of pole", the pole and a point on the barrel (a small square will appear, which will be the insertion point).

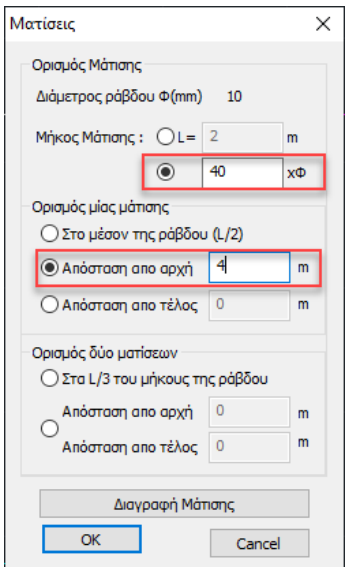


Weaponization:

Prerequisite: you must have designed the reinforcement by "opening plates" between these options.



Select the armature on which you want to draw the pattern and the following mask opens:



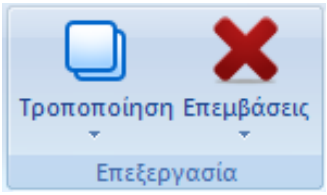
Here, you define the length of the etching, either in meters, or as a function of F (multiple)

You then choose whether to set one or two matches.

In one etch, you have among the options of etching in the middle, either spacing from the beginning or end of the bar.

In the two mates respectively, the options are in the thirds of the rod or at some of the distances from the ends.

Edit



The "Edit" command group contains useful commands for editing the woodblock designs.

5.1 Amendment

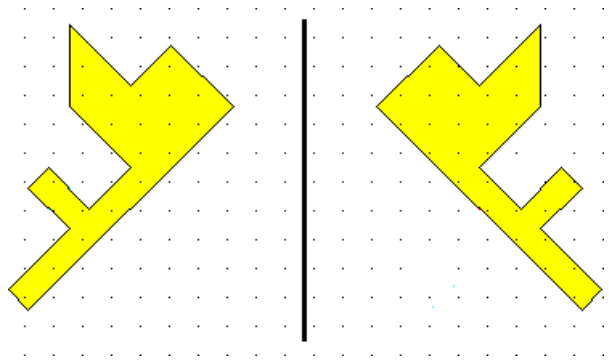
Copy : to create copies of one or more entities at the same time.

How to use :

1. You activate the command from the "MODIFY" option list.
2. Indicate the entity(ies) to be copied.
3. Cancel by pressing the right mouse button.
4. You select a feature point as the starting point for determining the new location of the copy and finish with the final point where the selected entities.

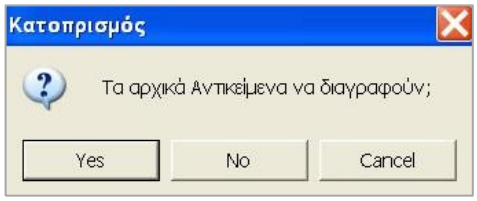
Mirroring :

With this option you can create a "mirror copy" of one or more entities with respect to a line or two points. In addition there is the possibility to keep or delete the original objects.



How to use :

1. After activating the command, you select the objects you want create a mirror copy of with the left mouse button.
2. Cancel with the right mouse button.
3. You choose an existing line of the drawing as the axis for the mirroring or two points instead. In this case the mirroring will be done in relation to the line defined by these two points.
4. After indicating the line or two points, a dialogue box opens where you can choose whether to delete the objects to be mirrored or not.



The process is completed by selecting Yes or No respectively with the left mouse button or cancel the "Mirroring" command by selecting . Cancel

⚠ The text is "mirrored" only as a position, i.e. it keeps its original orientation so that it can be read.

Offset :

The "offset" command allows you to create copies of outlines at a specified distance from the original object. The command is particularly useful in cases where you want to draw lines or plines parallel to existing lines or plines at a specific distance from them.

How to use :

1. You activate the "Offset" command from the "MODIFY" command list. At dialog box that opens, enter in centimeters (cm) the distance you want the offset to be.

2. Select the line or pline and press the left mouse button on the side you want to copy.

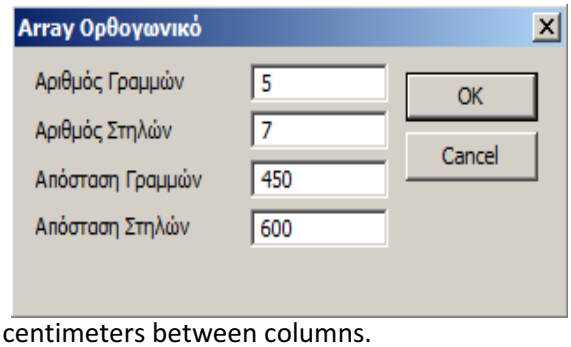
⚠ The "Offset" command on circles and arcs is used to create copies of the circle or arc with the same center and a larger or smaller radius.

Array:

With the "Array" command, you can produce an entity in specified distances. There is the possibility forRectangular¨ and ¨Circular¨.

α. Array Rectangular

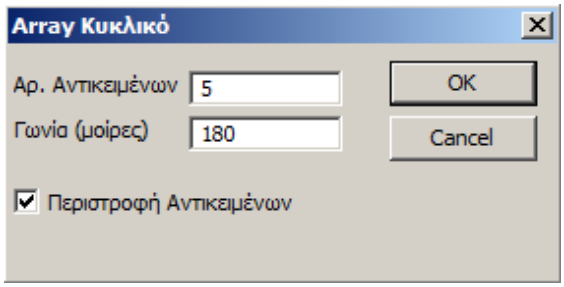
If you select this command, a dialog box appears in which you give:



1. Number of lines : You define how many horizontal lines the entity you are interested in will be repeated.
2. Number of columns : You define how many columns (vertical rows) the entity of interest will be repeated.
3. Line spacing : Set the distance in centimeters between the lines.
4. Column spacing : Set the distance in centimeters between columns.

β. Array Circular

Selecting the command displays a dialog box in which you give:



1. Number of Objects : Set the final number of objects to be produced.
2. Angle :You set the angle (the opening of the arc) at which to deploy the objects you want.

Activating 

option results in the rotation of the reproduced objects towards the center of the circle in which the "Circular Array" will be made.

Transportation:

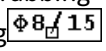
This command allows you to move one or more objects (beams, columns, lines, circles, etc.) in parallel.

How to use :

1. Select the command "MODIFY>> Transfer"
2. You specify the entity (or entities) to be selectively transferred one by one or in groups using a window.
3. The completion of the entity selection is done by pressing the right mouse button.
4. You select a characteristic point of the entity or entities (e.g. a top of a pole or a point inside the entity or any point in general) to which the selected entities will be moved.
5. You specify a new location where the entities will be placed.

Remark :

In case after the transfer there are "remnants" of the entities that were transferred on your screen, select the "Redraw" command or press the  button and they are cleaned up.

Plate and pedestal reinforcements behave as single entities (blocks) and are thus transferred by selecting the reinforcement design. The program also allows you to transfer the text, independently of the reinforcement design, without "breaking" the symbol, by "grabbing" it with the mouse at the base of the vertical, between diameter and bar spacing 

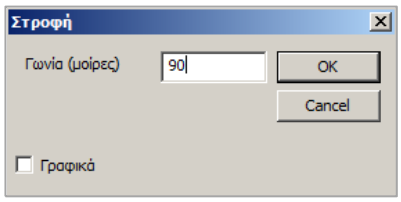
. The reinforcements are still blocks.

Perassia:

The ways of using the command and examples are described in detail in the chapter of the Section "TOOLS>Miscellaneous>Passwords".

Turn:

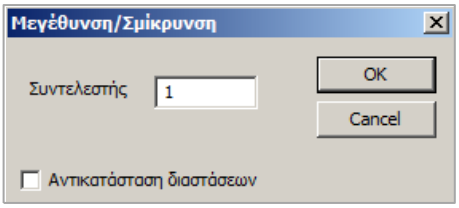
With this command you can rotate an object so that it comes to the position you want.



The ways of using the command and examples are described in detail in the chapter of the section "BASIC>Edit>Rotate".

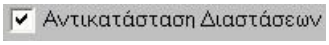
Zoom in/out:

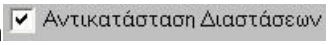
This option allows you to zoom in or out one or more entities according to a scale factor that you type in the corresponding dialog box.



How to use :

1. Activate the command and select the entities you want to resize.
2. Cancel with the right mouse button.
3. Point with the mouse to a point to which to zoom in or out, or reduction. This point can be one of the points of the selected entities or a random point in your drawing.
4. A dialog box opens immediately in which you enter the zoom factor (value > 1) or reduction factor (value < 1).



: The dimensions are also "obeyed" by the command "Zoom in/out". If the option 

, in the dialog box where you enter the coefficient, is active, the value of the selected dimension will be multiplied by the coefficient you specified. Otherwise the original numerical value will remain.

With 

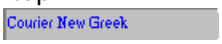
you complete the task or cancel by clicking . 

Text/Font Style:

With the "Style" command you can modify the font type for texts you have already entered in your design, or the type of drawn lines, plines, circles, etc.

Suppose you have imported text with an Arial Greek font and you want modify the Style of the text to Courier New Greek.

How to use :

1. From the "PARAMETERS" command list, select text and choose the Courier New Greek font. The list of available fonts can also be opened from the field where the current font type is shown, at the top status bar of the screen 

.
2. You activate the "Style" command and indicate with the mouse the text of which you want to modify the type.

Remarks :

In general, to change the text font type, you must first have selected the new font (it has become the current font). Then any text you select to change gets the current font style regardless of what it had previously.

With a corresponding procedure you modify the type of drawn lines, continuous lines, arcs, circles.

From the command list "PARAMETERS >> Line", select the line type. Activate the "Style" command and indicate the entity you want to design with the above line type.

Color (Modify):

With the command "MODIFY >> Color" you can change the color of designed entities. Each entity you indicate with the left mouse button gets the current color. The selected, current color is shown in the icon  .

The color is selected by pressing with the mouse on one of the colors in the palette shown in the lower part of the screen, to the right of the bottom status bar, either in advance (before activating the "MODIFY >> Color" option), or while you are already in the process of changing a color.

Transparency (Amendment):

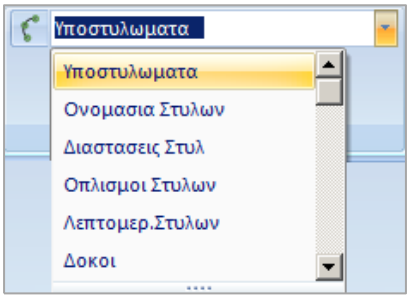
The same procedure is used to modify the Transparency (Layer).

How to use :

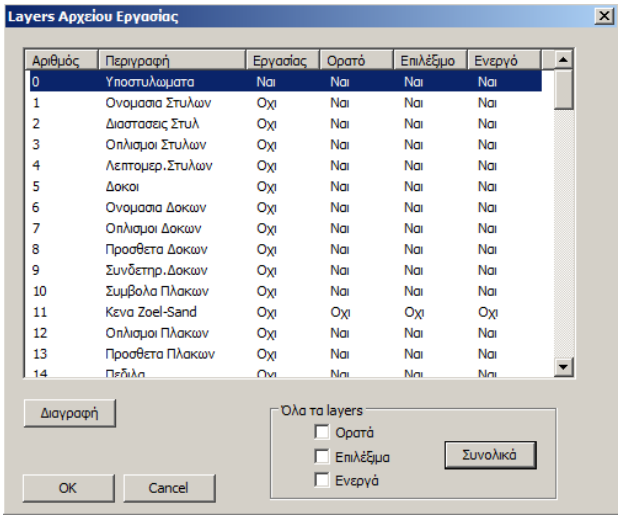
1. You select a slide to make it work.
2. You activate the "MODIFY>> Slide" command and select entities you want to "move" to the current slide, regardless of which one they were on until now (as long as it is active so that the object can be selected).

The choice of transparency can be made in two ways:

α. By opening the list of "Layers", from which you select one to work on.

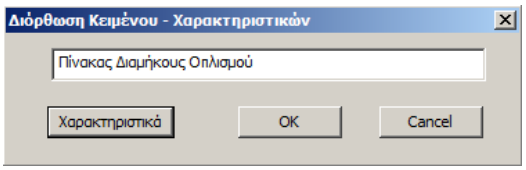


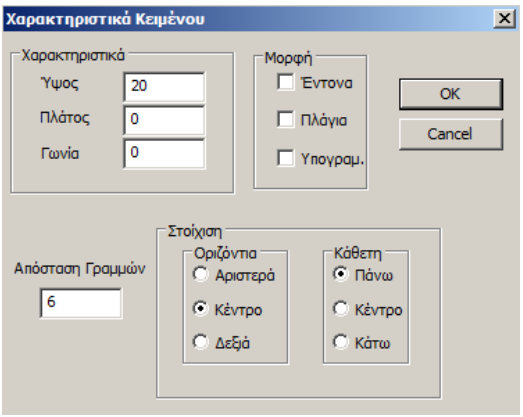
β. From the "Layers" group, select "Layer" and press twice consecutively with the left mouse button (double click) on the "Work" column next to the name of the desired transparency. The value changes from NO to YES. The change will be confirmed by pressing the mouse on the OK button.



Correction:

The "Correct" command is a general, dynamic entity correction tool. Selecting an entity for correction opens the corresponding dialog box where you can modify its content or attributes. If, for example, you select a text for correction, a dialog box will appear where you can modify the content of the text or its attributes.





Remarks :

In detail, with the "Correct" option you can correct
:

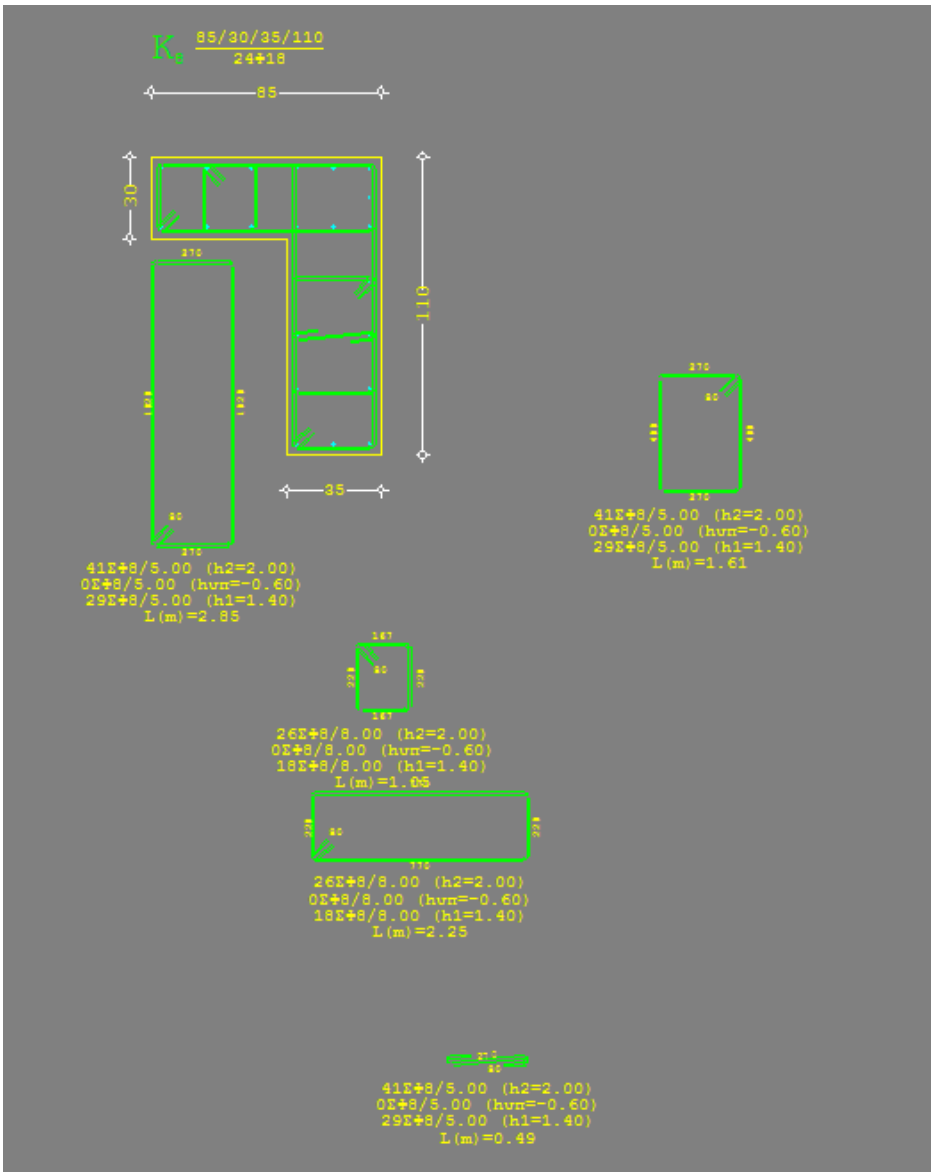
- Details of columns, text of beams, columns
- Plates (symbolism, numbering, symbol type, thickness, boundaries)
- Reinforcements of beams, poles
- Beam support additives
- Symbolisms and text plates
- Plate irons and plate support additives
- Foundation irons
- Dimensions (text, features)
- Text features and content
- Altimeters (type, price, characteristics)

With the above command you can NOT modify :

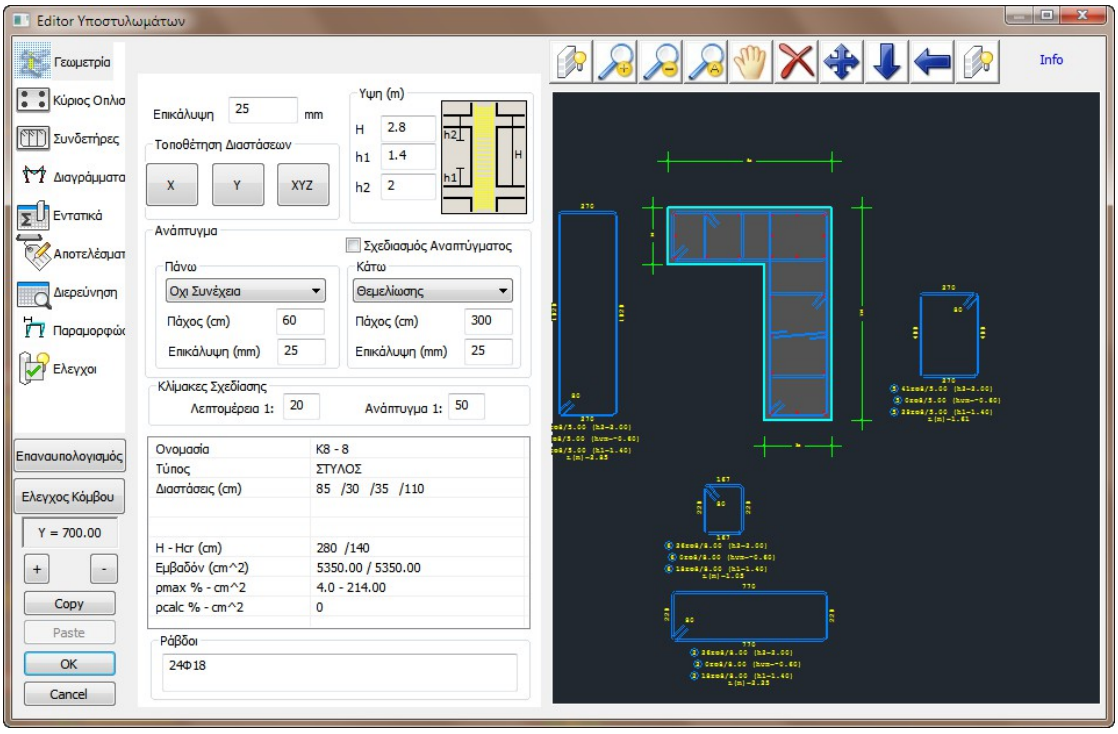
- Text and Line Style
- Colours of objects
- Layer

These corrections are made with the corresponding commands (Style, Color, Transparency).

With the creation of the Subcolumn Detail Armor in Dimensioning, you can edit the subcolumn detail as a whole with the "Edit" command through the same dialog box as in Dimensioning.
More specifically by selecting the command and showing the detail of the column,



the following Armament Details dialog box appears:



Where you can edit the detail of the substrate based on the instructions in the corresponding manual. A prerequisite for importing the detail in its new form is open the corresponding substructure with "Reinforcement details" command in the dimensioning and select "OK".

Armament Editor:

With this command you can create or edit a reinforcement cross-section detail. For detailed instructions, please refer to the corresponding manual "REINFORCEMENT DETAILS.doc".

4.2 Interventions

Undo : Each time you activate the "UNDO" command, you return to a previous state with step 1, cancelling the last command. The 'UNDOcommand is subject to the 'Delete' and 'Transfer' commands.

Redo : Reset to pre-Undo state

Delete : delete entities from your design (lines, points, plines, plines, blocks, texts, etc.).

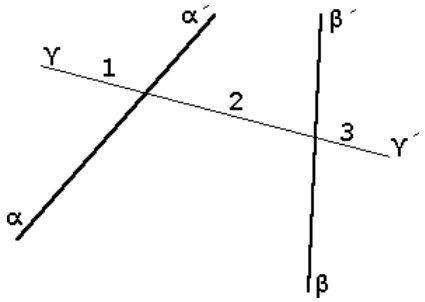
To delete an entity the layer to which it belongs must be SELECTED.
Deletion can be done by selecting individual objects or by selecting a group of objects with the help of a window (see also paragraph II.A.). In this case, the objects which belong entirely within the window will be deleted.
After selecting the items to be deleted, with the left mouse button (either individually or in a window), press the right mouse button and the deletion process is completed,

In case there are "remnants" of the deleted entities left after the deletion, on our screen we select the "Redraw" command or press the  button and they are cleaned.

Restoring deleted entities can be done with the "UNDO" command or with the  .

Trim : to remove parts of entities which are enclosed by some boundaries.

Example :
Suppose you want to delete the part of line c-c' lying between lines a-a' and b-b' (section 2).

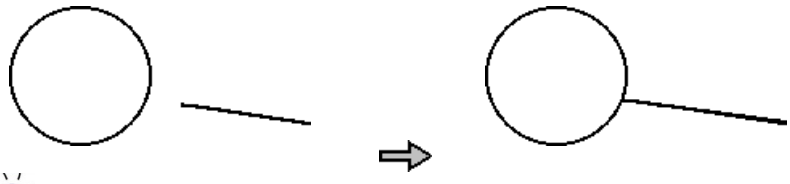


1. You activate the "trim" option.
2. Use the mouse to point to the first cut-off point (line a-a') and then to the second (line b-b').
3. Cancel by pressing the right mouse button once.
4. Pressing the left mouse button on section 2 of line c-c' deletes it.

Remarks :

1. If, after indicating the boundaries a-a' and b-b', you press with the mouse on section 1 of line c-c' (or 3) then it will be deleted.
2. The "trim" command can also work for multiple "trims" of objects that are between the same boundaries. In this case, after setting the boundaries and cancel with the right mouse button, you indicate successively all the entities you want to cut off.

Extend : to extend single lines or extreme segments of plines to a boundary which can be a line, pline segment, arc, circle, ring.



How to use :

1. After activating the command, you use the left mouse button to set the expansion limit.
2. You select the lines that will be extended up to this limit.

Remarks :

1. If the lines to be extended do not intersect the selected boundary the extension will not be performed.
2. To expand internal parts of plines up to a certain limit you must first "Expole" the pline

Extend : to "break" an object (line, pline, circle, arc, etc.) by removing the part of the object between two (2) points.

How to use :

α) Line segmentation

- Activate the "Split" option and first indicate the line to be split by pointing to it with the left mouse button.

- You select the first and second point between which the line segment will be removed.

Remark :
The points can be points on the line or random points in your drawing. In the latter case, the projections of the random points on the line to be segmented must belong to the line.

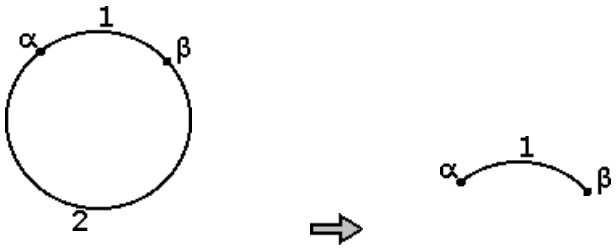
b) Cycle segmentation

Select the circle to be segmented and point to the two points to remove the part of the circle between them.

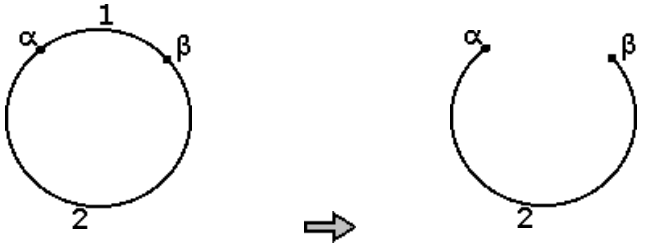
ATTENTION:
⚠ In cases of segmenting circles, the order in which you indicate the two cut-off points is important.

⚠ Positive time is counterclockwise.

Example :
If you want to remove part 1 of the circle, you would indicate point b first and then point a.



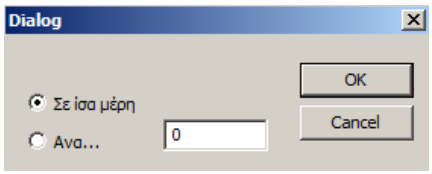
If you want to remove section 2, you will first show a and then b.



c) Arc segmentation

Observation:
The same procedure is followed as for the cycle segmentation. Here the order in which the points are indicated does not affect the result. The part of the arc between the 2 boundary points is always cut off.
During the process of entity segmentation the selection of the 2 cut-off points can also be done with the help of the "Point Definition Tools". (intersection of lines, projection of a point on a line, etc.).

Break : to split a line or a segment of a continuous line (pline), arc or circle into individual segments.



The division can be made into equal parts of an integer number or into parts of a specific length that you specify.

How to use :

You activate the "Split" command and immediately a dialog box opens where you set the parameters of the split.

α) In Equal Parts : Type the number of parts you want the entity to be divided, press on 

with the left mouse button, the configuration box closes and you indicate the line the arc or circle.

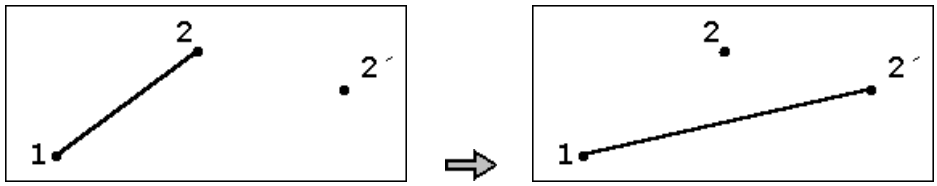
b) Per : Select this separation mode and type :

i. The length of each line segment in centimetres (cm). The division into segments starts from the nearest end of the line at the point you indicate.
ii. The size of each part of the arc or circle in degrees.

Drag : to modify the size and position of an entity while keeping some points fixed.

1. Pull line segment or segments

After activating the command, you select with the mouse a point on the line near the edge you want to move (the line initially becomes dashed) and indicate its new position. The other end of the line will remain fixed.



EXAMPLE:

You have drawn line 1-2. You want to move end 2 of the line to position 2' while keeping end 1 fixed.

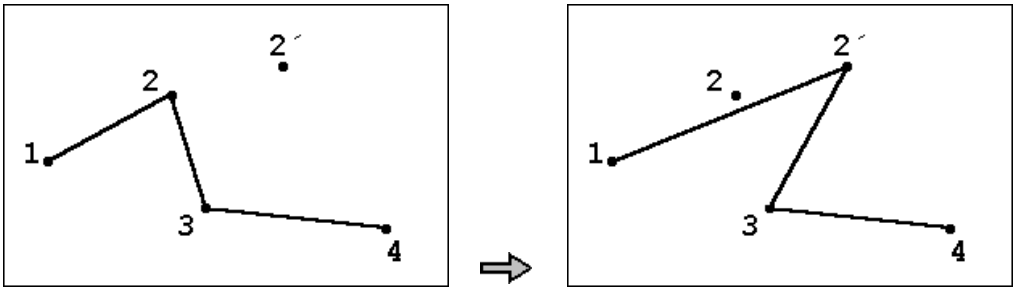
1. You activate the "pull" command.
2. Point the mouse at a point on the line at end 2.
3. Press with the mouse to position 2'.

The new position of the line will be 1-2'.

2. Continuous Line Pull (Pline)

EXAMPLE:

Suppose you have drawn Continuous line 1,2,3,4 and you want to move end 2 to position 2'.



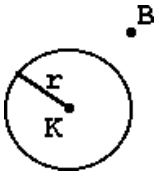
1. Select with the mouse the vertex 2 or a point on line 1-2 near edge 2, or line 3-2 near edge
2.
2. By moving the mouse you can see that the positions of both lines (1-2 and 3-2) are modified while points 1, 3 and 4 remain fixed.
3. You indicate position 2' so the new Continuous Line is now 1, 2', 3, 4.

3. Drawing Arc, Circle, Ring

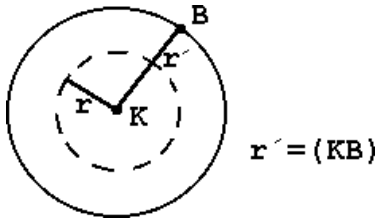
Using the "Drag" command on Arc, Circle, Ring allows to change the radius of these entities while keeping their center fixed.

EXAMPLE:

You want to enlarge the circle with centre K and radius r so that it passes through point B.



1. You activate the "Draw" command and point to any point on the circle.
2. You choose point B so that the new circle has centre K and radius
   r'=(KB).



A similar procedure is used for pulling the bow and ring. Drawing

Polygon - Rectangle

The "Draw" Polygon or Rectangle command works similarly to the "Draw" Continuous Line command.

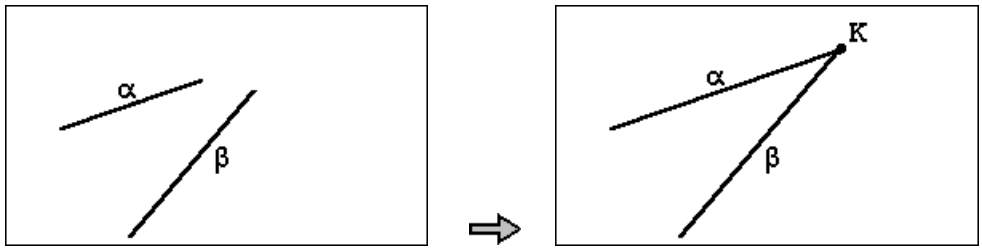
5. Dimensional Pulling

Using the command for dimensions allows you to move a dimension parallel to its original position. Essentially, the "Drag" here refers to the dimension's End Lines which are elongated, keeping their two starting points constant.

Extend - Intersect : to extend two lines that do not intersect to their point of intersection, or to cut off parts of two intersecting lines beyond their point of intersection.

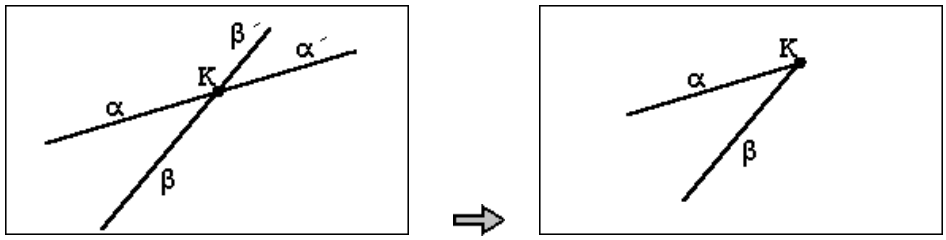
EXAMPLE 1:

- To extend lines (a) and (b) to point K, simply activate the "Intersection" command and press the left mouse button on the two lines in sequence.



EXAMPLE 2:

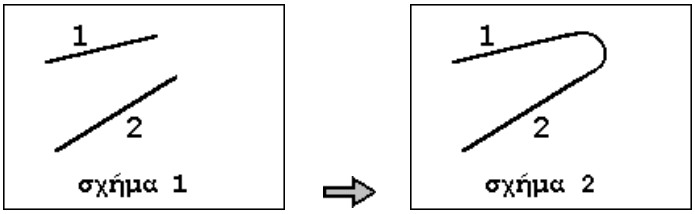
- To cut off parts a' and b' of lines a-a' and b-b', beyond their intersection K, indicate successively the parts of the two lines you wish to cut off.
keep.



Fillet : to connect two intersecting or non-intersecting lines with an arc of a predefined size.

It works just like the "Intersection" command after you have typed the radius of the arc of adjustment in degrees in the dialog box that opens.

EXAMPLE:
Connecting lines 1 and 2 with an arc 45 (figure 1) will give you the result shown in figure 2.



Explode : to "break down" an entity into the individual units that make it up:

1. Dissolving Groups (blocks)

Blocks can be managed as an object (e.g. copy, move, rotate, etc.). To manage the entities that make up a Group individually, you must first "dismantle" it.
This is done by pointing to the group with the mouse, after activating the "Explode" command.

2. Solving Continuous Lines (plines)

The "Explode" command on consecutive lines has the effect of "breaking" the pline into its individual line segments.
The reinforcements of the plates are pline and can divided into individual rectilinear sections.

⚠ The details of the substructures are blocks and often require "Explode" before exporting them to a dwg/dxf file.

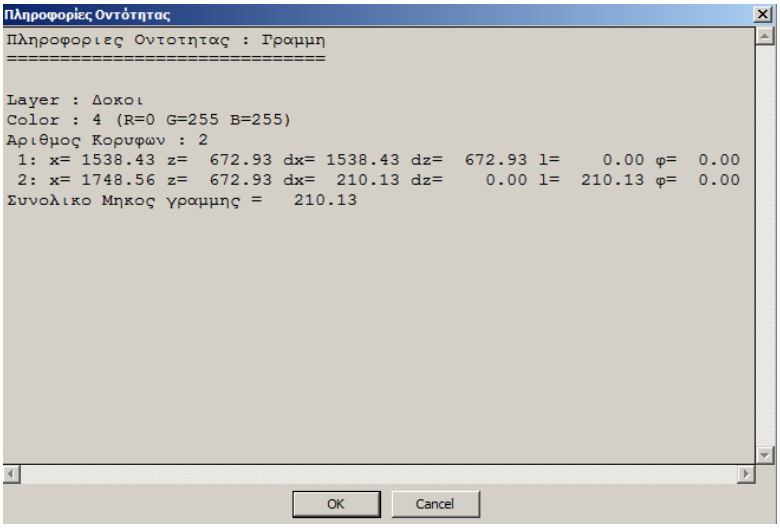
Show

6.1 Show Identity :

The "Show" command group includes:

Ruler - Ruler : When these commands are enabled, you see the length measuring ruler, and the ruler on the working screen.
By starting the stylus program, all these options are enabled. Next to each enabled command a (In case you do not want the ruler and the canvas to be displayed on the screen, after selecting the "DISPLAY" command, press with the mouse on the corresponding indication.
If you want the ruler, and/or the canvas to reappear on the working screen, follow the same procedure as you used when you wanted them not to appear on the screen.

Identity : You activate the option and select a design entity with the mouse. A dialog box opens on your screen in which information about it is displayed.

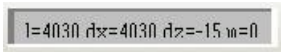


Distance : With this option you can find the distance between any two points in your drawing.

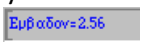
How to use :

1. You activate the "DISPLAY >>Distance" option.
2. Use the left mouse button to indicate the first point which from now on will be considered the starting point for distance calculation.
3. Select the second point whose distance you want to find in relation to the first. If you do not cancel the command with the right mouse button or if you do not activate the same or another command again, you can continue to indicate second, third and so on points to calculate their distance from the initial one.

The distance and the projections of the imaginary line segment connecting the two points on the x and z axes are displayed in the third field in the top status bar, to the right of the x and z coordinate fields.



Area-frame :

The area of an area enclosed by a continuous closed line (pline) is calculated. The value of the area is given in m2 and is displayed ¨E=.....¨ in the upper status bar, in the third field to the right of  of the active color. 

(Point with the left mouse button to the pline after activating the command "INSERT >> Area >> Lines"))

With the command "Area >> Lines" command calculates the area of areas defined by pline, Circles, Rings, Polygons (inscribed or circumscribed) and Arcs (circular segments).

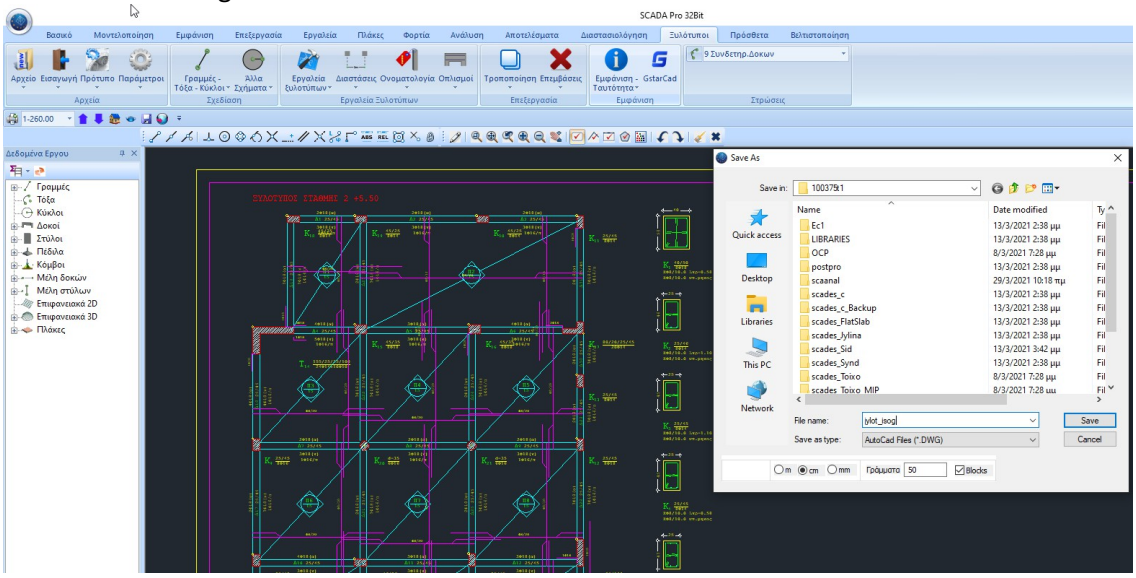
If you request an area calculation for an area enclosed by an open pline, the program will calculate the area assuming that the line is closed by the line segment connecting the beginning and end of the pline.

Area-Area :

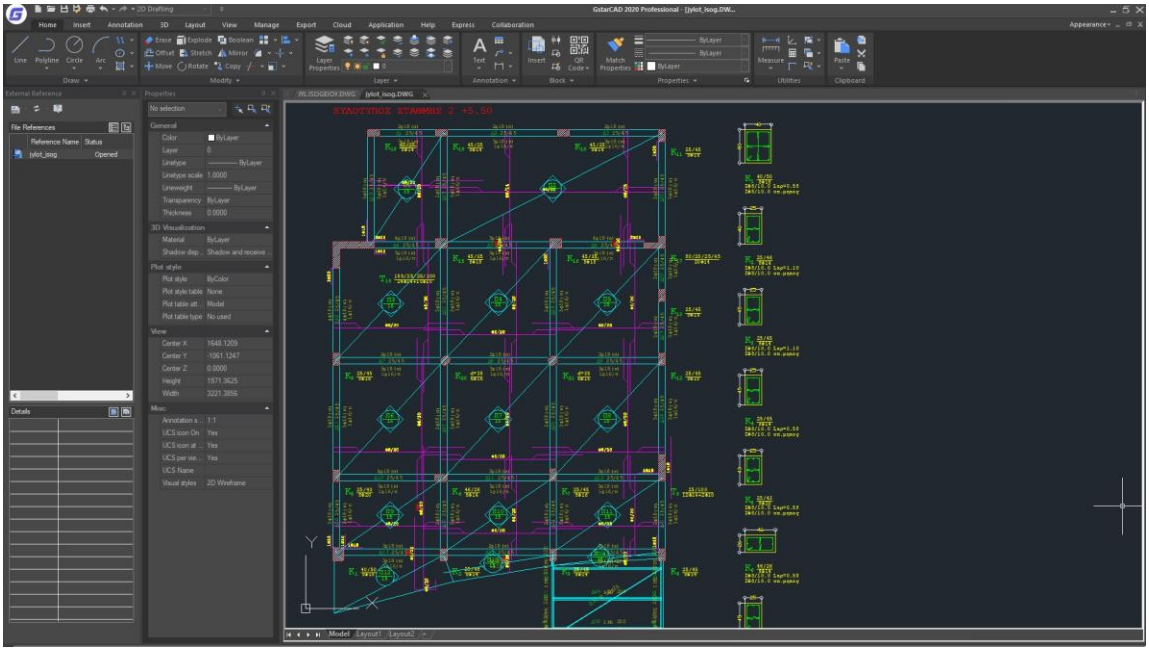
Here you calculate the area of an area, the vertices of which you select with the mouse. By pointing to the last vertex, WITHOUT pressing the mouse button, its area is displayed in the same position as in "Line area".

6.2 GstarCad

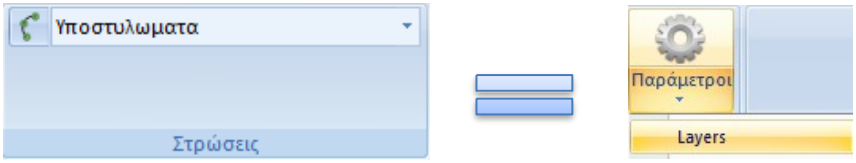
In the new version of SCADA Pro, the function of automatic export wood types to the general design program **Gstarcad** for further processing has been fully integrated. Communication is carried out via dwg files and is done with one click.



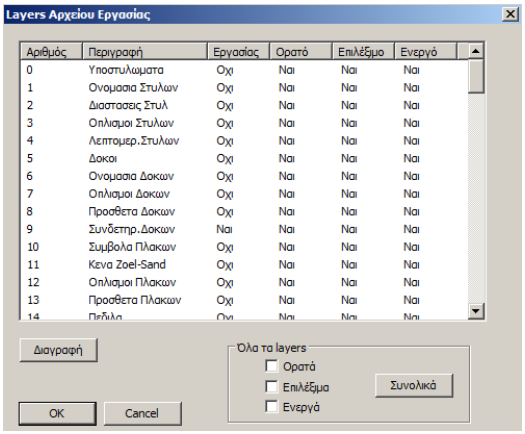
Select the command and give a name to the dwg file that will be created. Select Save and you will automatically be taken to the GstarCad interface.



Layers



Select the button  button to open the Layers window:



Open the list to change the "Active Layer", simply by selecting another one, which will now appear on the surface.

⚠ Detailed instructions can be found in the chapter "FILE>>Parameters>>Layers"