

Example 1 Concrete Structure Analysis and Design







CONTENTS

Ι.	OVERVIEW	ERROR! BOOKMARK NOT DEFINED.
II.	INTRODUCTION	ERROR! BOOKMARK NOT DEFINED.
III.	THE NEW ENVIRONMENT	ERROR! BOOKMARK NOT DEFINED.
1.	GENERAL DESCRIPTION	6
1.	1 GEOMETRY	6
1.	2 MATERIALS	6
1.	3 REGULATIONS	6
1.	4 LOAD AND ANALYSIS ASSUMPTIONS	7
1.	5 Notes	7
2.	DATA INPUT – MODELING	8
2.		
2.		
2.	3 IMPORT OF A NEW FLOOR PLAN (NEW DWG FILE) AT THE EXISTING MODEL FOR THE CREATION OF THE RES	ST FLOOR LEVELS:21
2.	4 Typical Floor Modification:	24
2.		
2.	6 AUTOMATIC IMPORT OF FOOTINGS AND FOOTING CONNECTION BEAMS AT THE FOUNDATION LEVEL:	27
2.	6.1 Footings	29
2.	6.2 FOOTING CONNECTION BEAMS	29
2.	7 How to define a raft:	
2.	8 How to insert Footing Beams at the basement walls:	
2.	9 MATHEMATICAL MODEL CREATION:	
2.	10 3D REPRESENTATION:	
2.	11 BASEMENT WALLS NODES CONNECTION – HIGH RIGIDITY BEAM MEMBER:	
2.	12 How to create a slope:	
3.	SLABS	41
3.	1 How to define solid slabs:	41
3.		
3.	3 DEFINE SLAB STRIPS:	45
3.		
4.	LOADS	47
4.	1 How to define the loads:	
4.	2 How to insert Slab Loads:	47
4.		
4.	4 How to assign loads in members:	50
5.		
5.		
5.		
5.	3 How to check the analysis results and create the combination file	67
6.		
6.	1 HOW TO VIEW THE DIAGRAMS AND THE DEFORMATION RESULTS AND THE MESH AREAS STEEL REINFORCE	MENT DEMAND:72
6.	1.1 Model + "Deformed Shape" :	73
6.	1.2 DIAGRAMS – STRESS CONTOURS:	74
7.	DESIGN	76
7.		
7.	2 How to define the parameters of the design for each member type:	77
7.		
Se	LECT THE "CONTINUITY OF BEAMS – OVERALL CONTINUITY OF BEAMS"	
7.	4 How to apply Capacity Design	86
7.	5 How to design columns and walls	
7.	6 How to perform Slab Design:	

SCADA Prot Structural Analysis & Design

EXAMPLE: «CONCRETE STRUCTURE ANALYSIS AND DESIGN»

7.7	How to perform Footing Design	91
8.	DRAWINGS	92
8.1	HOW TO IMPORT DRAWINGS AND BEAM'S DETAILING IN DRAWING ENVIRONMENT:	92
8.2	HOW TO IMPORT ANALYTICAL COLUMNS DETAILS WITH THE ABILITY TO PERFORM MODIFICATIONS DIRECTLY INSIDE THE EDITOR:	96
9.	PRINT	97
9.1	How to create the Project Report:	97



• OVERVIEW

SCADA Pro new version is a result of more than 40 years of research and development while containing all the innovative capabilities and top-notch tools for the construction business.

SCADA Pro utilizes a compact and fully adequate platform for constructing new buildings (analysis and design) or existing ones (check, assessment, and retrofitting).

The software employs the Finite Element Method, combining line and plane finite elements in a smooth way. For design purposes, the user is offered all the Eurocodes as well as all the relevant Greek regulations (N.E.A.K, N.K.O.S., E.K.O.S. 2000, E.A.K. 2000, E.A.K. 2003, Old Antiseismic, Method of permissible stresses, KAN.EPE).

There are numerous possibilities offered for the modeling of various kind of structures. Structures made of reinforced concrete, steel, timber, masonry, or composite structures are now fully feasible.

Several smart operations add on to the practicality and usability of the software. The user can produce the model of a structure no matter how complicated it is, work at ease with the 3D model, process through the steps of analysis and design in a convenient way, up to the conclusion of what initially may seem the most demanding project.

SCADA Pro is presented to you as a powerful tool to meet the highest needs of modern civil engineering!

• INTRODUCTION

The current manual comes as an aid for a new user of SCADA Pro, making the interface of the software as familiar as possible. It consists of several chapters, where one after the other, describes the consecutive steps of a simple example of a loadbearing masonry project. The most useful information is presented, in regards to the best possible understanding of the software commands and logic, as well as the process that has to be followed.

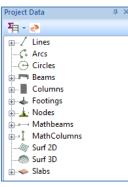
• THE NEW INTERFACE

The new interface of the SCADA Pro software is based on the RIBBON structure, thus, the several commands and tools are reached neatly. The main idea of the RIBBON structure is the grouping of commands that have small differences and work in the same context, in a prominent position different to each group. This converts the use of a command, from a tedious searching procedure through menus and toolbars, into an easy to remember the chain of two or three clicks of the mouse button.

The user can collect his/her most popular commands into a new group, for an even faster access. This group remains as it is for future analyses after the program ends. Different commands can be added to it or removed from it, and its placing in the workspace may be altered through the "Customize Quick Access Toolbar" utility.

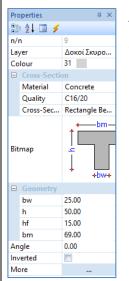






Apart from the RIBBON structure, all the entities that a structure consists of are presented in a tree structure, at the left side of the SCADA Pro main window, either for the whole structure or at each level of the structure. This categorization enhances the use of each entity. When the tree structure is choosing an entity, it is highlighted at the graphical interface and the level of the structure that contains this entity is isolated. At the same time, at the right side of the window, the entity's properties appear. The user can check or modify any of these properties at once. Conversely, the entity can also be chosen at the graphical interface, and automatically it is presented, at the left side in the tree

structure and at the right side with its properties. The right-click mouse button can be very helpful here, since several commands and features, distinct for each entity, can be activated with it.



The "Properties" list that shows up at the right side of the window, not only shows all the properties of the entity shown but can be used for any quick and easy changes, the user wants to make, too.



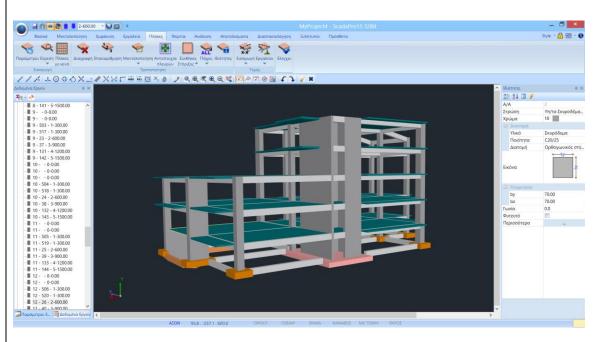
1. GENERAL DESCRIPTION

1.1 Geometry

For educational reasons, in this example, two different ways for modeling the structure will be presented.

The 1st way uses two different plan floors for the model creation along height.
 The 2nd way uses only one plan floor, following the process of modifying the standar floor and the other elements to create the final model that will reflect the structure under consideration.

The considered structure consists of a basement and four upper structure floors. A part of the basement consists of basement walls, while at the 4th floor an inclined section is met. The mixed foundation consists of single footings, footing beams, connection footing beams and raft foundation as well.



1.2 Materials

The quality of the concrete and the reinforcing steel that was used is the same for all members **Concrete:** C20/25, **Reinforcing Steel:** B500C.

1.3 Regulations

Eurocode 8 (EC8, EN1998) for seismic loads. Eurocode 2 (EC2, EN1992) for the design of the concrete elements.



1.4 Load and Analysis assumptions

Dynamic Spectrum Analisys with pairs of torsional moment along the same direction. The loads by the method above are:

(1) G (dead)

(2) Q (live)

(3) EX (node loads, seismic forces along XI axes, derived from dynamic analysis).

(4) EZ (node loads, seismic forces along ZII axes, derived from dynamic analysis).

(5) Erx \pm (node torsional moments, derived from node seismic forces along XI axes, offset by the accidental eccentricity $\pm 2ezi$).

(6)Erz±(node torsional moments, derived from node seismic forces along ZII XI axes, offset by the accidental eccentricity ±2etxi.

(7)EY (seismic vertical component –seismic force along y direction- derived from dynamic analysis).

1.5 Notes

All the commands that were used in this example, as well as the rest of the commands, are explained in detail in the manual that accompanies the program.



2. DATA INPUT – MODELING

2.1 How to start a new project:

SCADA Pro offers several ways to start a new project. Some criteria related to the acceptance of the starting method are: materials, architectural files, floor plan shape, type of elements usage (beam/shell elements) etc.

In this example, the way of using the help dwg file for the modeling of a concrete structure, will be explained in detail.

Right after opening the program, the starting dialog form with a group of commands, related to the initialization of a project, is displayed:



By left clicking on the related icons, one of the following ways, for the project initialization, can be performed:

No matter which way you choose to start a new project, the same form always opens to set the project name and the path of the file, a necessary procedure so that the program commands can work.

New Projec	ct		
Project	-		
Name	example 1		
Details	Concrete Structure		< >
Location Folders:	c:\meletes		_
Drives:	e:	~ Netw	ork
02 04 04 05 05	ELETES 108171 11704A 116086 51711 51711Q 71722Q	Can	

The name of the file can contain up to 8 characters of the Latin alphabet without any symbols (/, -, _) nor spaces. (eg FILE1). The program automatically creates a folder in which all the details of your project are recorded. The location of the folder, that is, the point where the folder will be created, should be on the hard drive. We suggest that you create a folder in C (eg MELETES) where all the SCADA projects are included. (eg C:\MELETES\EXAMPLE1).

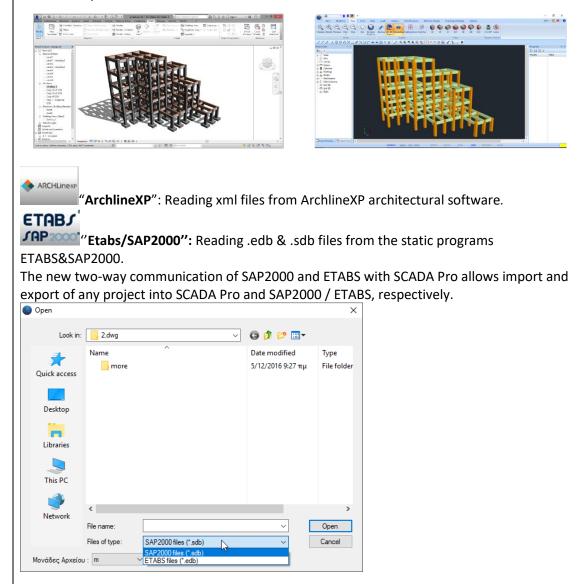
You can add a description or add some information related to the structure, in the "Info" field.



"new": It is used when there is no help file in electronic format. The startup is performed in an empty work sheet. The engineer starts with the definition of the height levels and the sections, and moves on to modeling, using the modeling commands and the snap tools of the program.

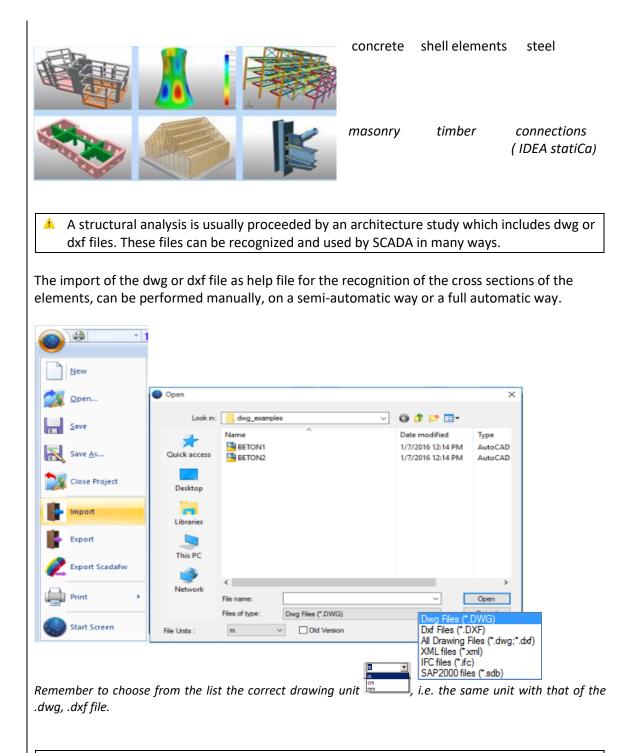
REVIT "**REVIT**": Reading ifc files created by the the Autodesk Revit.

By using appropriate libraries, SCADA Pro automatically recognizes all the structural elements (columns, beams, slabs, etc.) with their respective properties, generating in this way the ready for the analysis model.



"**Templates**": SCADA Pro carries a rich library of structure templates for every type of material. The command can be activated either by clicking on one of the startup icons or by accessing the Modeling>Add-ons>Templates. Detailed explanation of this command can be found at respective chapter of the manual (Chapter 2. Modeling)





- Also, besides the cad files, you may import Revit, SAP2000 etc. files inside SCADA Pro environment.
- ▲ The connection between SCADA Pro and Revit is even more powerful since it regards the import of the whole model and not just the help files.
- The connection between SCADA Pro and SAP2000 enables the import and analysis of any concrete, steel, masonry and timber model inside the SCADA Pro environment by the Eurocodes and the National Annexes.



DWG

"dwg-dxf": Another way to import a dwg or dxf file which in SCADA pro is not just a background that draws snaps on the drawing lines, neither a semiautomatic way to import items by manual selection. It is a completely automated tool that allows you to reproduce a floor plan on the selected floors and automatically create the model.

This command is used for this example and it is extensively described next.

2.2 Automatic recognition of the cross sections derived from a dwg file:

Select the icon

and in the dialog box: dxf

Define the name and the path of the file. If you wish, you can add some information related to your project, inside the "Info" field and click OK.

New Proje	ct		
Project Name	example 1		
Details	Concrete Structure		^
			<u>_</u>
Location			
Folders:	c:\meletes		
Drives:	C:	~	Network
🗁 ME	ELETES 108171		
04	21704A 416086		OK
05	51711 51711Q 71722Q		
	1717	*	Cancel

On the next window, select the dwg file and click Open.

		Open			2
Look in	i 👢 1.dwg		v G 🌶	⊳ 🖽	
Pa	Name	^		Date modified	
	🚬 1.plan1			26/5/2015 4:08 µ	ιμ
Recent places	🚞 1.plan2			26/5/2015 4:09 µ	ιμ
Desktop					
Libraries					
This PC					
Network	<				>
The work	File name:			~ C	pen
	Files of type:	Dwg Files (*.DWG)		✓ C;	ancel
Μονάδες Αρχείου	: m	V Old Version			



▲ In the case of operators without a typical floor, or more typical floors, or with structures with different floor plan per level, there is a need for introducing more help files. SCADA enables the engineerr to import as many dwg / dxf files as he wishes. These are stored in the study file and can be used to create the static model, combining the fully automatic way with semi-automatic and manual way.

In each new file you create, the General Parameters window appears in the interface, where you can define from the start the Materials and the Regulation that you will use, as well as General Project Data and other parameters.

eneral Paramet	ers					×
Other Param Project G	eters Sc ieneral Information	reen	Drawing Mater) ial - Regu	Display Ilation	
Regulation National	EC General				~	
Standard Steel		Euro	~	Metric	~	
	C20/25 ~ C20/25 ~		nbers - Elemen el Plate		Fe430; ~ Fe430; ~	
Steel Main	S400s ~	Weld	3		Fe430; ~	
Stimups	S400s ~	Timb	er	C14	~	
Safety Factors Ultimate S γc 1.5 γs 1.15	Serviceabilit 1	үМ0 1 үМ4 1	γM5	γM2 1.25 γM7 1.1	үМ3 1.25	
E	OK	Cancel	Арр	ly	Help	

ATTENTION: The materials must be defined according to the selected regulation and the data entry, as well as all cross sections mast have the right types (C for new regulations, B for the old ones).

* Predefined scenarios are created according to the Regulation and Attachment option you make at the beginning, within the General Configuration window that opens automatically immediately after the file name is defined.

Click OK and then the project opens automatically in SCADA environment, with all its designed elements.



Basic Modeling Basic Modeling Line Circle Arc Polygon Dratt Project Data 0 X X Project Data 0 X Circles Beams Circles Beams Circles Beams Circles Suff 30 Suff 30 Slabs	View Tools Slabs Loads	Edit	Lines circles	Add multiple levels Number 5 Add Select	set C LiC. Height n 3D	
교 Project Par 전 Project Data		ИСБ Х, Ү.Z	ORTHO OSNAP	SINAP GRID	CROSSING INSIDE	×

The drawing opens inside the Scada environment, with all of its elements.

At the same time, the "Level Management XZ" window, in which you define all the levels of the model, opens.

First modify the level 0 (default). Select "**Edit**" and select the level 0 from the list (blue color indicates the selected level). Now you can change the name and the height value. Select "**New Level**" and then type a name and a height. Complete the range "–" and "+", in case of uneven height, slopes, or vertical mesh elements, to make them belong to the current level (for mass distribution) and visualize them on the current level, too.

Also you can automatically create multiple levels with "Add multiple levels" command.

Level Management XZ		×
Name Edit Delete Move Execute	300 - 0 + 0	Add multiple levels Number 5 Add
n/n Name 0 0 1 2 3 4 5 Connection Method of Columns' Ne	Height R.L.C. Even H 3D 0.00 ♥ ● Ø 300.00 ♥ ● Ø 600.00 ♥ ● Ø 900.00 ♥ ● Ø 1200.00 ♥ ● Ø 1500.00 ♥ ● Ø odes with Mesh Surface Ø Ø	Select All Deselect R.L.C Non R.L.C. Even Height Non Even Height Display in 3D Hide in 3D

Set the number of the levels that will be created and click "Add":

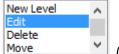


New Edit Delet Move		∧ Nar ∨ Heig	ne 0 ht (cm)	0	-+	0		Add multip Numbe	
n/n	Name			Height	R.L.C.	Even H	3D		Select All
)	0			0.00	Ŷ	e	Ø		Deselect
1				300.00	* ★	e ₽	Ø Ø		R.L.C
2 3				900.00	÷.	₽	a		Non R.L.C.
4 5					i≫ i≫		α α		Even Height
					•				Non Even Height
									Display in 3D
									Hide in 3D
Conn	ection Metho	od of Col	umns' No	des with Me	sh Surfa	ce			

IMPORTANT NOTE:

▲ Make sure that you define **'Kinematic pair to the nearest node of the surface'** at the level 0, so the nodes of the members of the columns can automatically depend on the nodes of the plate that will be created on the foundation.

The list is updated with the levels (with a height difference of 3m (300cm)), which are editable



(for further information view the respective

through the "Edit" command chapter of the Manual)

Close the window and the "Section Identification from Dxf – Dwg file" form automatically opens.

Section Identification from Dxf - Dwg File				×
Selections Section Identification, Global(Beams - Columns) Select Layer for identification		Apply From	(Levels) 0-0.00 5-1500.00	~
Columns	\sim		0 1000100	
Beams	\sim			
Cantilever	\sim	Sec	tion Identification, Autom	natic
Automatic Insertion and Predesign of Footings Automatic insertion of footing connecting beams Automatic creation of matematical model - 3D			tion Identification, Selec	

It regards an automation that recognizes the beams and columns of any shape (T, Π , Γ), slabs and cantilevers, footings and connection footings beams, while it automatically creates the mathematical model of the whole model as well.

The lists next to the element types "Section Identification" contain the layers of the .dwg help file.

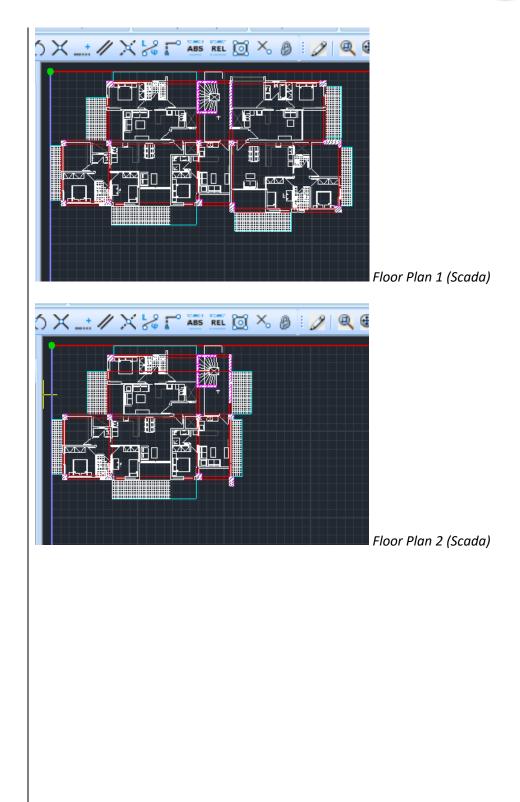
NOTE:

1 To make sure that the automatic procedure of the sections recognition will be performed, some preconditions must be taken into accound.



	PRECONDITIONS:
1.	Each plan that will be used as a help dwg file must be in a separate file that includes only this plan (without any other drawing entities).
2.	The lines (or/and polylines) that define the columns, the beams and the cantilevers must belong on their one layer without any other type of element or/and drawing entity inside this layer.
3.	The help file is imported to the SCADA environment at the active (current) level by placing the
	upper-left corner of the drawing to the (0,0) point of the SCADA coordinate system During the import of the help files be aware of the floor plans inserting position (by their reference point) so that the correct placement for all the plans is achieved.
	Floor plan 1 (dwg)
	Floor plan 2 (dwg)







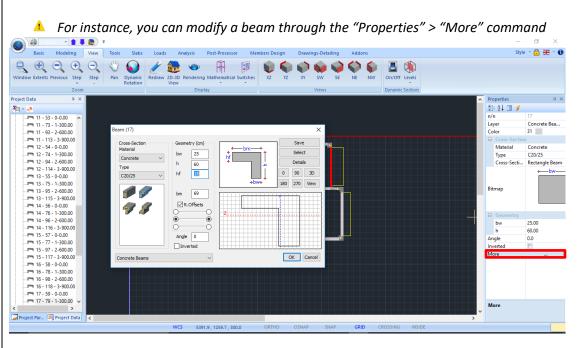
	NOTES:
•	Through the command that appears on the initialization window, you car import a help file and perform the automatic modeling with one single command.
•	For each help file of the same project, use the " Import " command and with the considered floor plan XZ activated import the drawing.
•	To perform the automatic modeling move to the "Modeling" unit and select the "Elements: Creation" command. Next, select Columns – Beams – Foundation Beams, for the <u>automatic</u> or <u>selective</u> elements: creation on the selected levels.
	Section Identification from Dxf - Dwg File Selections Select Layer for identification Columns Column

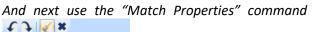
the t beam depth which is rectangular defaulted by the program



ection Identification from Dxf - Dwg Fil	e	×
Selections		
Minimun line distance a (mm)	5	4b
Divergence Parallel Lines b (mm)	5	ay
Minimum beam Depth (cm)	20	
Maximum beam Depth (cm)	100	
Default beam Depth (cm)	60	
ОК	Cancel	

The modification of the beam cross section after the import can be performed in several ways. •

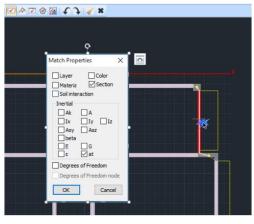






Match Properties to modify the rest of the beams that share the same cross section by selecting the preferred elements with one of the 🗸 🔊 🗹 🙀

available methods





Automatic creation of matematical model - 3D

By activating the "Automatic creation of the

mathematical model – 3D", the program not only does it recognize and insert the cross sections of the physical model, but it calculates the inertial properties and creates the mathematical model too.

Section Identification from Dxf - Dwg File Selections Apply (Levels)	~
Selections Apply (Levels)	\times
Select Layer for identification	
Texts Columns	
Defpoints	
Beams Cantilever	

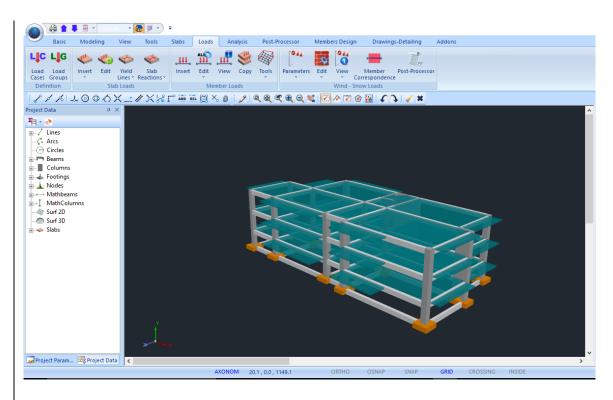
Δ Precondition for the automatic creation of the slabs and the cantilevers is the selection of the columns and beams creation as well as the automatic creation of the mathematical model so that the members that surround the slabs exist.

The "From - To" commands define the levels that the drawing will be applied.

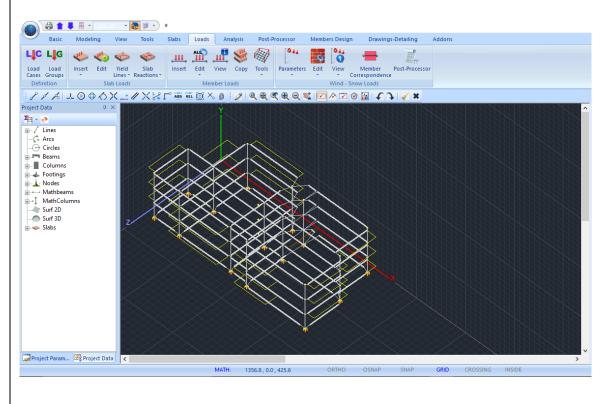
Section Identification, Automatic

Select the "Section identification, Automatic" to view the Rendered representation of the model.





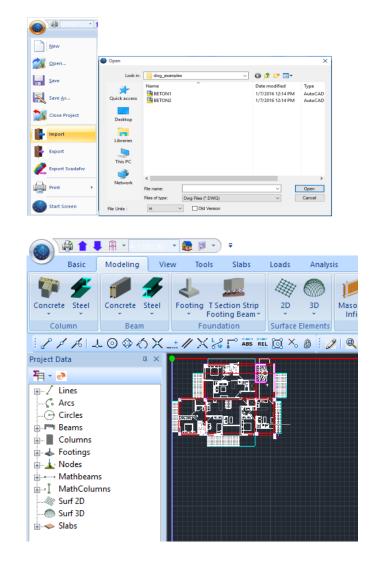
Deactivate the rendering to view the mathematical model





2.3 Import of a new floor plan (new dwg file) at the existing model for the creation of the rest floor levels:

After creating the first floor plan (*plan1.dwg*) for levels 0 to 3, the levels 4 and 5 of the model do not include any elements. For the creation of the elements that belong to the second floor plan (*plan2.dwg* - levels 4 & 5) follow the automatic process described below:



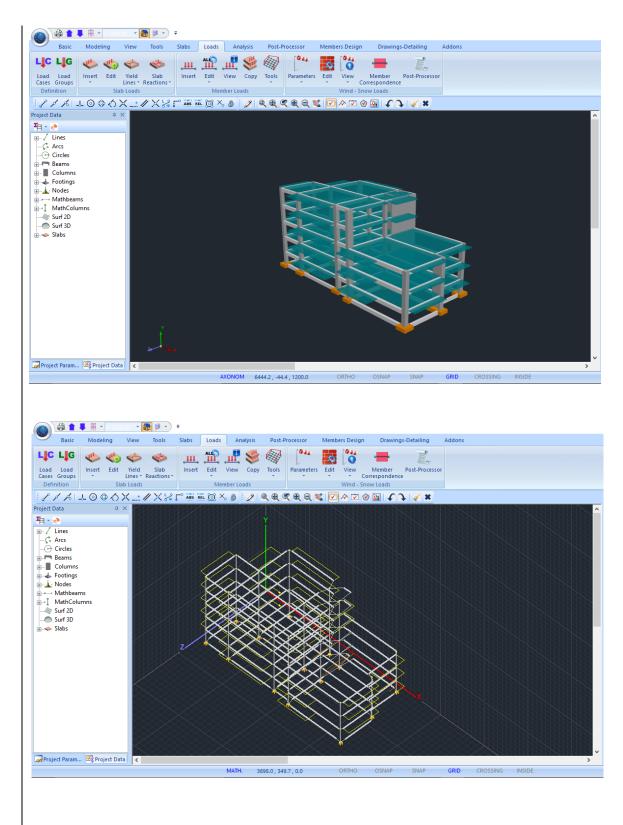
"Import" the plan2.dwg at the current empty XZ level of the SCADA (level 4)



mer	the automatic Elements Creatio nu. nns or Beams and in the dialog boy		ı th	e " Mod e	eling"	Διατομών Κα Υποστυ	 1 EX 10
	tification from Dxf - Dwg File		pply	(Levels)		×	
	n Identification, Global(Beams - Columns) yer for identification	Fr	on	4-1200.00		~	
Columns	Columns	~ To	0	5-1500.00		~	
Beams Cantilever	Beams Cantilever	~ ~	Sec	tion Identifi	cation, A	utomat	
Autor	matic Insertion and Predesign of Footings matic insertion of footing connecting beams matic creation of matematical model - 3D			tion Identif	fication, S	Selection Ok	

- The "Section identification, Global (Beams Columns)" that activates the corresponding • fields for the selection of the respective layers for the creation of Beams, Columns and Cantilevers by selecting their corresponding layer.
- The "Automatic Creation of Mathematical Model 3D" •
- The "Application" at levels 4&5 •
- The "Section Identification, Global"

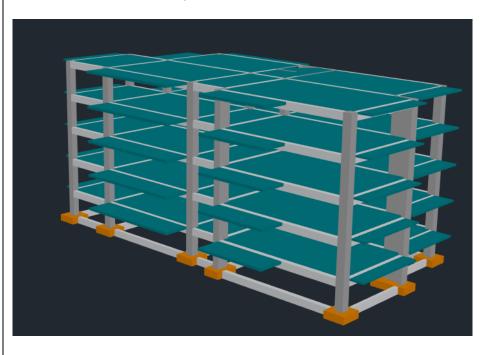






2.4 Typical Floor Modification:

Alternatively, we can follow the 2nd way, i.e. import only one floor plan (*plan1.dwg*) and copy it to the rest of the levels, and perform the desired modifications, to the rest of the levels.



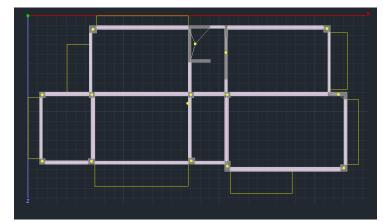
IMPORTANT NOTE:

▲ To modify the physical model, the mathematical model must not exist, but since the ✓ Automatic creation of matematical model - 3D command created the mathematical model you

Mutamatic creation of matematical model - 3D command created the mathematical model, you must delete the mathematical model of the desired levels that you wish to modify. In case that you know that you will perform modifications you may deactivate the "Automatic creation of mathematical model" option, so that you won't have to delete it later.

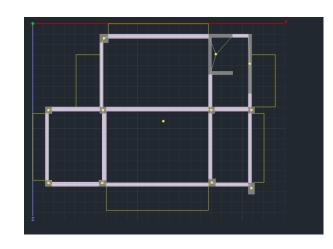
Level 4

Initial floor plan

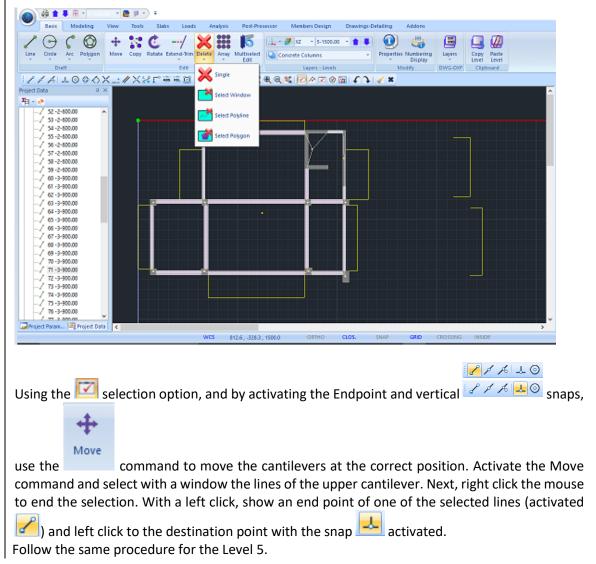




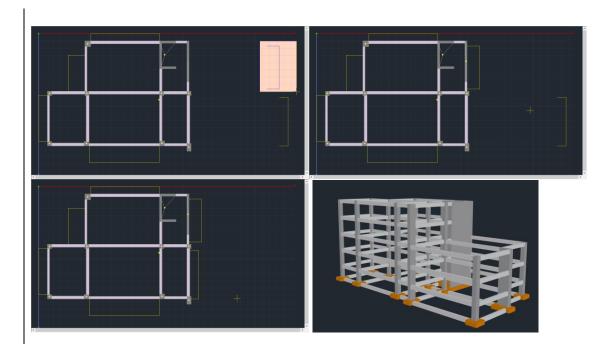
Floor plan after the modifications



Delete the Mathematical Model of the current level and then delete the elements that do not exist at this level (level 4).





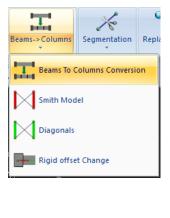


2.5 How to simulate the basement walls:

×

100

Cancel



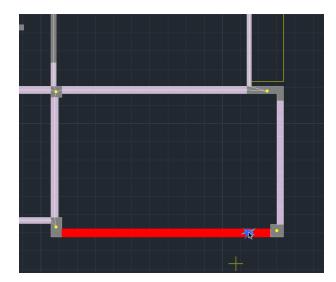
Beam to Columns Conversion

Max Column Length (cm) Auto

O No. of Columns

OK

There are several ways to simulate the basement walls. In this example the "Beams to Column Modification" method was used. For the first level (ground floor level), through the "Tools" >> "Model" unit, select the "Beams to Columns Conversion" command. At the dialog box that appears, select one of the two choices and insert the appropriate number.



Left click to select the beams of the level 1 that will be automatically converted into consecutive columns (after you delete their mathematical representative).

Level 1:



A You can repeat the same process for Level 0 or copy the new columns that were created at

0....

level 1, to level 0 using the "Copy"	Сору	command.

First, delete the Footing Connection Beams from level 0. Next, call the command and select the elements that you want to copy. The selection can be performed either individually or by window, polygon etc. Next, right click to end the selection and define the characteristic point (end point of a line, column, beam etc). Move to level 0 and define the respective point for the selected elements copy.

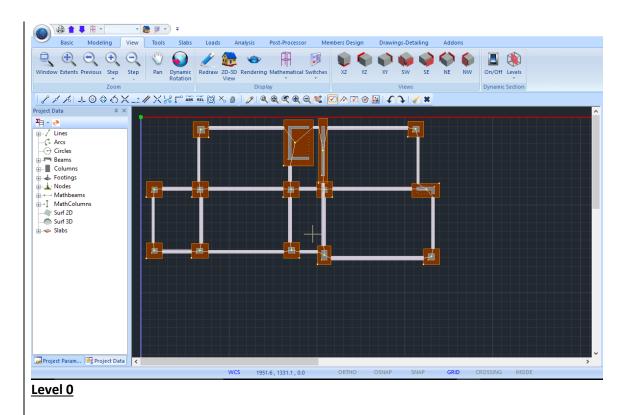
1 The modeling is completed with the creation of the mathematical model and the connection between the nodes of the columns above with high rigidity beam members (see §2.10).

2.6 Automatic import of Footings and Footing Connection Beams at the foundation level:

The new version of SCADA Pro, enables the user to Automatically Insert and Predesign the Footings, and insert Footing Connection Beams as well, during the Section Identification procedure.

Section Iden	tification from Dxf - Dwg File				×	
Select La	Section Identification, Global(Beams - Columns) Select Layer for identification Columns		Apply From To			
Cantilever	Cantilever	~	Sec	tion Identifica	tion, Automatic	
🖂 Auto	matic Insertion and Predesign of Footings matic insertion of footing connecting beams matic creation of matematical model - 3D]		ction Identifica Info	ation, Selection Ok	



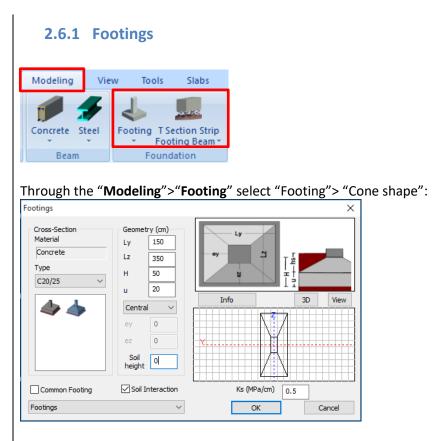


▲ The dimensions of the footings, derive from a predesign, and are imported as fixed (with a Ks=0). The user is to select all the footings through the "Multi Select Edit" command and set a value for the Ks variable by the soil.

	Member Propert Material Cr	ties Elemer	t Releases E Cross-Sectio		Offsets I Nodes	Draw History Member Type	
	Beams (cm)		Beams OEF (cm - MPa/cm	i) Footing	s (cm-MPa/cm)	
	bw	0	bw	0	Пн	0	
	h	0	□ h	0	u	0	8
	🗌 hf	0	🗌 bm	0	hs	0	
	hfo hfo	0	hf	0	Soil Inte		
	hfu	0	🗌 Ks	0	Soil 🗸	Yes 🗸	
	🗌 bm	0	R.Offsets	Yes \sim	🗹 Ks	0.5	l i
			Foundation B				8
	R.Offsets	Yes \vee	bw	0			
	Inverted	Yes \sim	h	0			
					[Apply	
. .					Exit	Help	
				_			-

• For completeness, in this example, the manual way of inserting footings and footing connection beams is presented as well.





In the dialog box define the geometry and material properties of the footing.

Click "**OK**" and place the footing to the model by left clicking on one of the upper structure columns edge.

Repeat the same process to place the rest of the footings as well.

2.6.2 Footing Connection Beams

Select "Footing Beam"> "Footing Connection Beam":

Beam OEF - Footing conne	ection beam (0)			×
Cross-Section Material Concrete Type C20/25 V	Geometry (cm) bw 25 h 50	2	bw	† ⊂
1		Details		3D View
-				
	R.Offsets	Z		
	00 00			
Footing Connection Beams	~		ОК	Cancel



In the dialog box, define the properties of the material and the geometry as well as the edge reference of the $beam^{(*)}$.

Click "**OK**" and place the beam by left clicking at the endpoints. Repeat the same process to place the rest of the footing connection beams.

^(*)During the placement of a beam you can alternate the endpoints reference edges of the beam using the TAB and SHIFT keys.

2.7 How to define a raft:

For completeness, in this example, a part of the foundation will be replaced with raft, so that every foundation type can be presented.

To model a raft, use the 2D Plate Elements.



2D +	3D •	Masonry N Infill
	Mesh	
	External I	Boundary
	Holes	
	Line	
	Point	
H	Edit	
	Calculate	

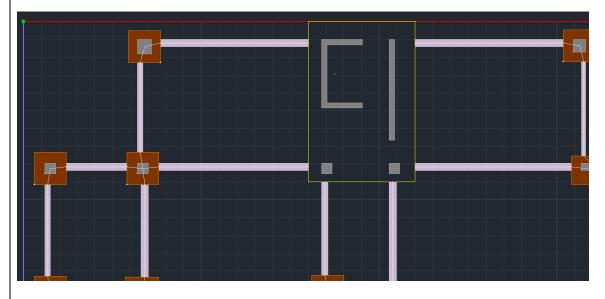
First select "**Mesh**" to define the mesh properties.

Plate Element	s Creation					×
Description	RAFT	Ka (Maa (cm)	Material Concre		Туре	C20/25 ~
Eleme		Ks (Mpa/cm)	 Isotropic 	○ Orthot	ropic	Angle 0
Plate O.E.F.	\sim	0.5				
Density	Width (cm)	Thickness	Exx (GPa)	30	Gxy (GPa)	12.5
0.20 ~	50	50	Eyy (GPa)	30	ε (kN/m3)	25
Descripti	ons Me	sh	Ezz (GPa)	30	atx*10-5	1
Mesh Groups		t Surface	vxy(0.1-0.3)	0.2	aty*10-5	1
I KAP			vxz(0.1-0.3)	0.2	atxy*10-5	1
			vyz(0.1-0.3)	0.2	Exx * vx	z = Eyy * vxy
			Redefinition	n Steel Re	einforcement	OK
			Del From Lis			
			New	S500	~	Exit



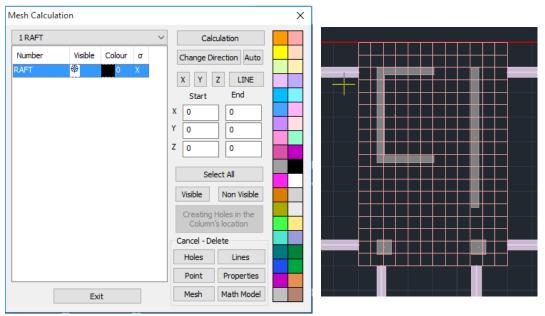
In the dialog box, set a description, the material, the quality, the type of the element, the Ks value, the density and the dimensions of the mesh as well as the quality of the steel reinforcement. Click "**New**" and then "**OK**". Next select "**External Boundary**".

Define the perimeter of the raft by left-clicking on the corners of the boundary. Right click to complete the selection.



Finally perform "Calculation".

In the dialog box that appears select the mesh, so that it turns blue and then click "Calculation". The mesh is automatically created. Click "**Exit**" and the mesh is created.

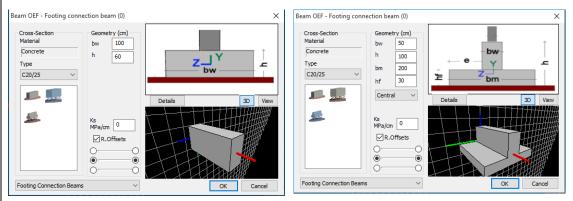


We move on to the rest of the foundation elements and we'll get back to mesh elements later. After completing the import of the physical elements, the mathematical model of the respective physical elements are to be created.



2.8 How to insert Footing Beams at the basement walls:

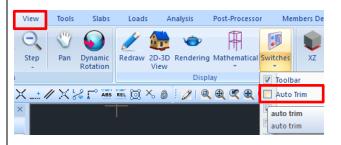
At the "**Modeling**" > "Foundation" unit select "Footing Beam" > "Rectangular" or "T" section.



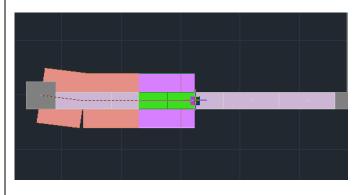
In the dialog box, define the parameters, the material and the geometry of the foundation beam.

To insert foundation beams at the basement walls, first of all, deactivate:

- "R.Offsets" R.Offsets (in the dialog box)
- "Autotrim" Auto Trim (in the View>Switches command group)

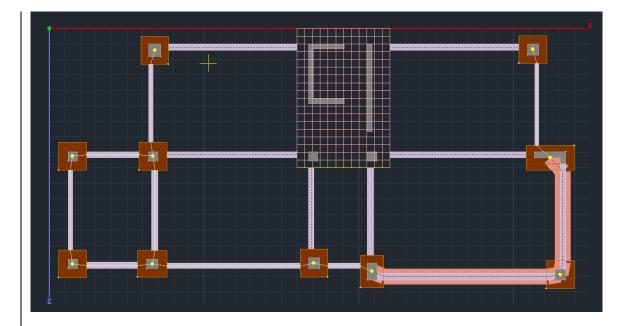


Next, insert footing beams at the basement walls, using the snaps, from center to center.



Complete the insertion of all the foundation above elements, so that level 0 looks like the following image:





2.9 Mathematical Model Creation:

As soon as you complete the modifications of the physical model (copy-delete elements) and the elements insertion, you move on to the creation of the mathematical model.

With the "Calculation" command, the program calculates and produces the mathematical model of the project (nodes and beams). This means that all the physical elements obtain their corresponding mathematical representative.

Select the command and the dialog box opens:

Mathematical Model X						
Select Regulation (inertial)						
EC2 V						
Change Regulation						
Calculation Inertia Redefinition						
Calculation of Inertia − Surfaces with the Boundary Element Method						
OK Cancel						

Select a regulation to calculate the modulus of elasticity accordingly.

▲ In case that you want to change the regulation after the creation of the mathematical model, select the new regulation and click "Change Regulation" to update the modulus of elasticity.

SCADA Pro enables the connection between beam and plate elements in the same modeling environment. The connection between beam elements and the corresponding plate element node is performed automatically with the above mentioned command.



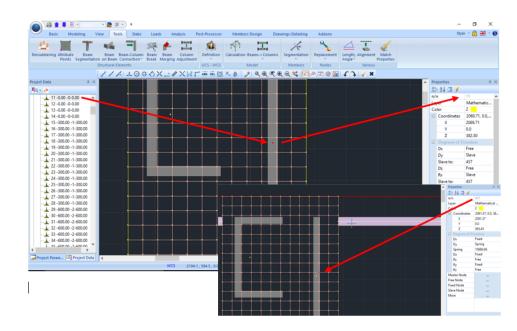
Connection between column nodes with mesh surface

SCADA Pro enables the connection between beam and plate elements in the same modeling environment. Therefore, there is an need for their enchain.

Level Manage	ement XZ					×	<
New Level Edit Delete Move Exec	Height (cm)	300 ent	- +	0		nultiple levels Number 0 Add	
n/n Nar 0 0 1 2 3 4 5	ne	Height 0.00 400.00 700.00 1000.00 1300.00 1600.00	÷ ≻ ≻	-	3D Q Q Q Q Q Q	Select All Deselect R.L.C Non R.L.C. Even Height Non Even Height Display in 3D Hide in 3D	
Kinematic p Kinematic p Rigid links i Rigid links i	Method of Columns' Node pair to the nearest node of pair to the nearest node of pair to the nearest node of n nodes of the surface n nodes of the surface/sid with binding bars with the	f the surfa f the surfa	ce ce tross-sec	tion	nt that belor]

In the bottom side of the window there is an option of the way to connect the column nodes with the mesh surface elements for the selected level, by choosing one of the following ways, that is, connecting the nodes either with a kinematic pair or with binding bars.

▲ Select the column base node inside the raft area (11 node) and notice the automatic slave relation from the nearest surface node (457 node).

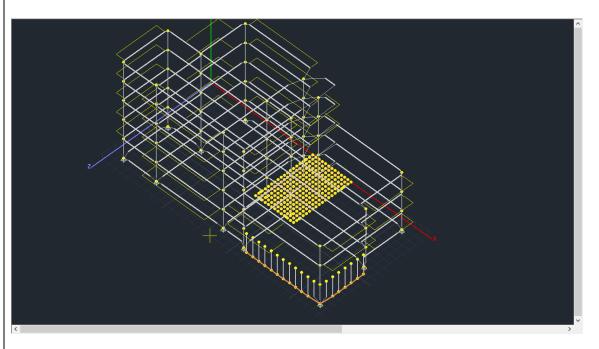




2.10 3D Representation:

After creating the mathematical model, you are allowed to view the 3D representation of the model as well as use the rendering options.

In this way you can modify in real time the mathematical elements, the nodes etc. For instance, you can create a slope and insert mathematical members to connect the unconnected nodes of the basement walls.



2.11 Basement walls nodes connection – High rigidity beam member:

The basement walls simulation through the "Beams to Columns" command is completed with the connection of the column nodes in level 1 (in level 0 the connection is already performed through the insertion of the footing beams).



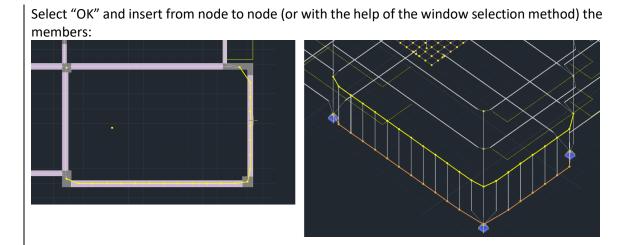


Select the "Member" command and in the dialog box click High rigidity beam member. The parameters' fields are automaticaly completed with the characteristics of a high rigidity cross-section; zero specific weight and without an assignment of a physical cross-section. This element

Linear Men	nber							×
A/A	0	Туре	B-3d	\sim	A(m^2)	0.75	Asz(m^2)	0.625
Nodes i	0	j	0		Ak(m^2)	0.75	beta	0
Material	Concrete			~	Ix(dm^4)	148.04534	E(GPa)	30
Type	C20/25			~	Iy(dm^4)	39.0625	G(GPa)	12.5
	ross-Section				Iz(dm^4)	5625	ε(kN/m^3)	0
Beam	\sim	Cros	s-Sectio	n	Asy(m^2)	0.625	at*10^-5	1
0	O 25/300 Columns 🗸			\sim	Soil Consta	nt Ks (MPa/cr	n)	0
Hig	gh rigidity bea	m mem	ber					
Rigid Off	sets (cm)				Member F	Releases		
S	tart i	End	j		Starti	N Vy	Vz Mx M	1y Mz
dx 0		0			Endj			
dy 0		0			Mathemat	tical Model		~
dz 0		0			ОК	Can	cel	Info
					UK	Can		1110

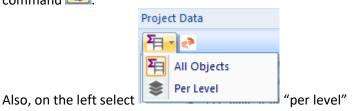
is necessary for the connection of the basement wall column elements.





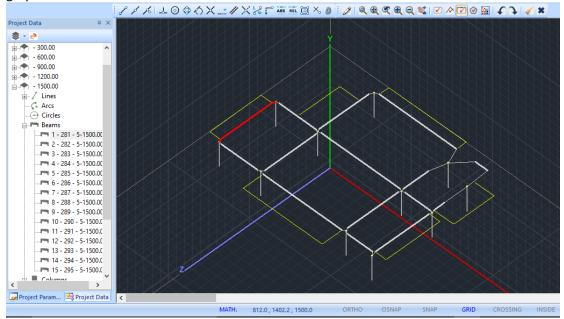
2.12 How to create a slope:

To create a slope simply, take advantage of the 3D representation of the model by selecting the command .



Open the group Level 3 and the subgroup Beam Members. Select the member whose slope will be modified:

The member is colored red while level 5 is isolated helping in this way the localization and the graphical modification.





Through the "View" unit activate the "XY" view:

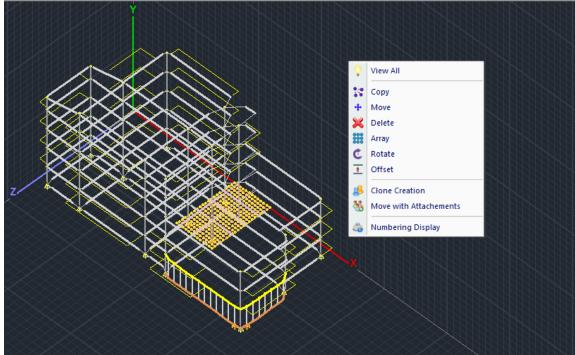
A A				
Basic Modeling View Tools Slabs Window Extents Previous Step Step Pan Dynamic Zoom	Loads Analysis Post-Processor Redraw 2D-3D Rendering Mathematical Switch Display		vings-Detailing Addon SW SE NE Views	s NW NW Dn/Off Levels Dynamic Section
roject Data $\Psi imes \dot{Y}$	> ⟨\ X _ : / X ¦2 [" == ≡ ∅ ×	8 2 9 9 9 9	< < < ∠ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕ ∕	🗑 🕈 🕈 🗶 🕷
Greet Data → - 300.00 → - 900.00 → - 1200.00 → - 1500.00 ⊕ - / Lines → Circles				
Image: Second	totate Extend-Trim Delete Edit		select the attachments activate the window" op	e "Basic" unit "Move with " command select "with ption enclose
· A A .			the node and	d right click.
Project Data	Move v	Creation with Attachements Properties	nodes that	select all the are located elected node ane.
Left click on the node and selec	t the command 🔟.			
Relative Coordinates X	In the "Relative Coor	dinates" winc	low fill in cm	n the relative
X (cm) 0 OK Y (cm) -100 OK Z (cm) 0 Cancel	displacement and click and the slope is create	« " OK ". The no		
Relative to point				

38



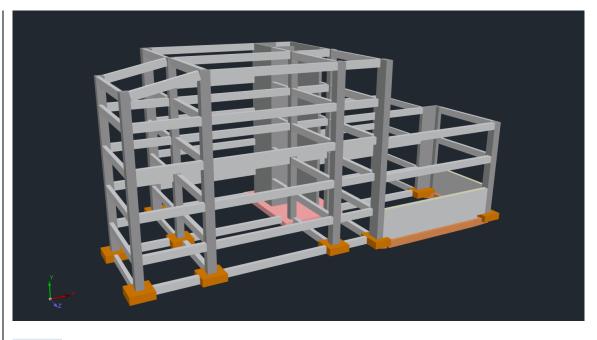


Right click on the desktop and "View All" to view the whole model.



Open the rendering to view your model in final state:







Select the "Model Check Report" to locate any errors created during the modeling process:

roject Data roject Data rojec	Concrete Steel	Image: Stable Code Analysis Post-Processor Members Design Drawings-Detailing Addons Image: Stable Code Image: Stable Code </th <th>Sty</th> <th>0 X</th>	Sty	0 X
2-91-0-0.00 2-91-0-0.00 Color 31 2-203-0-4.00 2-203-0-4.00 CHECKS REFORT Truto 17: The beam 13 (1) is not connected at the end physically but is connected mathematically Erriol 17: The beam 13 (2) is not connected at the end physically but is connected mathematically Erriol 17: The beam 13 (2) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (3) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (3) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (3) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (3) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically Erriol 17: The beam 16 (4) is not connected at the end physically but is connected mathematically by 50.00 Material For Help, press F1 NUM	阳 - 2	🗑 check_er - WordPad — 🗆 X	20:24 💷 🗲	2
A - 107 - 1300.00 Efficitive fill fill fill fill fill fill fill fil		CHECKS REPORT Err1017: The beam 13 (1) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (1) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (2) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (2) is not connected at the end physically but is connected mathematically Err1017: The beam 13 (3) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (3) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (3) is not connected at the end physically but is connected mathematically Err1017: The beam 16 (3) is not connected at the end physically but is connected mathematically Err1017: The beam 15 (4) is not connected at the end physically but is connected mathematically	Color Cross-Section Material Type Cross-Section	Concrete C20/25
- F 5 - 145 - 3-900.00			bw h Angle Inverted	100.00

The window that opens, displays for each error, a sort description, the number-ids of the members that concern the error and possible warnings.

▲ The Error1017 of the current example refers to the "Beam on Beam" support and it is not an error. In case that the model has errors, correct them using the program tools before you move to the next step.



3. SLABS



command, models the slabs as well but in case that you want to modify the auto-generated model you must first delete the mathematical model, thus

the slabs will be deleted as well. To redefine the deleted slabs (or even to define new ones) you must use the commands of the "Slabs" unit.

🔊 🛱 🕇 🖡 🕈 1-300.00	- 🍓 🗊 - 🔻	_				
Basic Modeling	View Tools Slabs	Loads Analysis	Post-Processor Memb	ers Design D	rawings-Detailing	Addons
Parameters Identification Zoellner Slab	Delete Renumbering Mo	deling Model correspondence Edit	Supports Thickness Properties	Insert Tools Strips	Checks	

3.1 How to define solid slabs:

To define the slabs set in 2D view each level and:

In the "Insert" field, select "Parameters" and fill in the values of the minimum width and concrete cover in mm.

Slab Parameters	×
Min Width (mm)	140
Zoellner-Sandwitch	
Upper Slab Thickness (mm)	80
Lower Slab Thickness (mm)	50
Rib Width (mm)	200
Dome (mm)	500
Concrete Cover (mm)	20
Composite slabs	
Auto composite slab chara	acterization
Construction Stage	~
Profile Sheet	
ОК	Cancel





The command "**Identification**">"**Overall**" identifies all the closed contours that exist in the current level and automatically inserts all the slabs.

After inserting a slab, the circular symbol with the corresponding information is displayed; the number and the thickness in cm (the greater value between the minimum you set and the one resulted from the bending resistance check), in a circle. Around the circle, lines are displayed representing the slab's support conditions:

- Thick Line: slab continuity \rightarrow fixed.
- Thin Line: slab discontinuity \rightarrow joint.
- No Line: free end (case of balconies).
- The sign "?" in the symbol of the slab, indicates that the slab has not rendered correctly and needs "Modeling". For this purpose you need to define a new slab, the shape of which can be rectangular, rectangulal with a slope, triangular or equivalent to the original.



Inside the "**Modeling**" command group select one of the three commands.

With a left click inside the area of an arbitrary shape slab, you select it and define the equivalent slab:

- For rectangle shapes: Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.
- For inclined rectangle shapes: Left click on a slab side to define the direction of the equivalent inclined rectangular slab. Left click on the first top, move the mouse diagonally up to the second top drawing a rectangle and left click again.
- For triangle shapes: Left click on the three sides of the equivalent triangular slab.

Last, define the correspondence between the sides and the tops of the equivalent slab and the ones of the real slab. The members and the lengths of the sides of the physical model match the ones of the mathematical model with this process.



Select the "**Model Correspondence**" command and click on the slab. The rectangle or triangle that appears is the mathematical model of the equivalent slab.



Yodeling

Select one side of the mathematical model of the slab. An X appears on it. Left click on the corresponding physical model member (in the middle of the member a circle that takes the color of the corresponding mathematical member is displayed). Right click to complete and left click again to continue with the rest sides of the mathematical model of the slab. Finally assign to each vertex of the equivalent slab (symbolized with a triangle) the corresponding physical point to make the reduction of the length of the sides of the physical to the mathematical model. Consequently, the loads of the equivalent slab will be distributed to the real lengths of the physical members. For the assignment first select the top of the mathematical model and then left click on the new location. Repeat the process for the remaining three vertices of the mathematical model without using the right mouse button.



With the command a TXT file that contains the results of the slab's design checks for all slabs of the current level opens. In case of errors you must correct them before moving on. Repeat the command after the load assignment as well, to be sure that no errors were located.

3.2 How to create a Zoellner Slab:

From the "Insert" field, select "Zoellner Slab" and click on the slab.



In the drop-down list "Type", select if the slab is connected in one or two directions and define solid zones' widths (cm). Click the "Pick" button and left click on the side of the beam considered as an outline of the slab. Then, the boundary of the solid zone will be placed in parallel with the beam at a distance equal to the width that was defined previously. The line is drawn (boundary of the solid zone) and with left click the direction is indicated. Repeat the same procedure for every solid zone.



Zoellner		×
Type Edge Or Tv List	ne-Way vo-Way 50	Pick
Thickne	ss (cm)	
hs	ho	hu
16.2	8	5
Widths	(cm)	
Direction	Ribs	Dome
1 [20	50
2	20	50
Whole	Domes	
OK		Cancel

To define a solid zone of different width, first right click to reopen the dialog box. Modify the width and continue as previously described to place the very last of the solid zones.

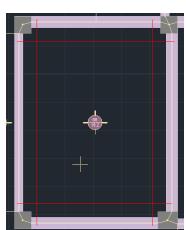
Right click to open the window and complete it by typing thickness, ribs and domes widths, and click "OK". ^(*6)

(*5) hs: type the slab's total thickness (cm).

ho: type the up side solid slab's thickness (cm).

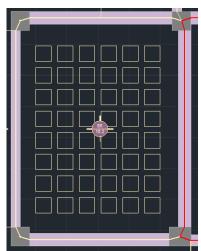
hu: type the down side solid slab's thickness (cm) for Sandwich slabs. Otherwise type 0.

 $^{(*6)}$ Select the checkbox next to "Whole Domes" to receive only whole domes.



Click "OK" to display the mathematical model of the selected slab. Then the program asks you to define Direction 1 (the side of the slab, which will be parallel to the beam of the first direction). Select the side of the slab's model and the gap with the defined geometry is placed automatically in the center.

To define where to start putting the domes, first click on a dome's vertex and then on a slab's vertex. The solid slab is automatically converted to a Zoellner slab.



The image on the left, shows the result of the procedure above.



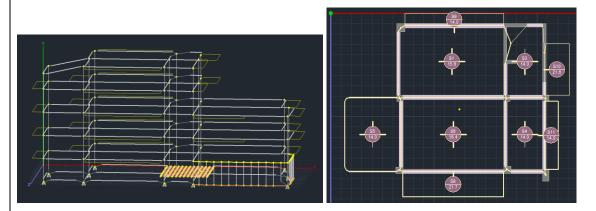
3.3 Define Slab Strips:



Strips Strips' input is essential to the analysis and design slabs' steel reinforcement. They are the "guides" for the design of the steel reinforcement and the diagrams display. From the "**Slabs**" unit, "**Strips**" command group select "**Insert**" along **X** or/and **Z** and define the strips with a left click. The direction of the strip identifies the steel reinforcement main bars direction.

3.4 In case of inclined slabs:

As in the current example, in case of inclined slabs, to achieve correct modeling, specific steps must be followed:



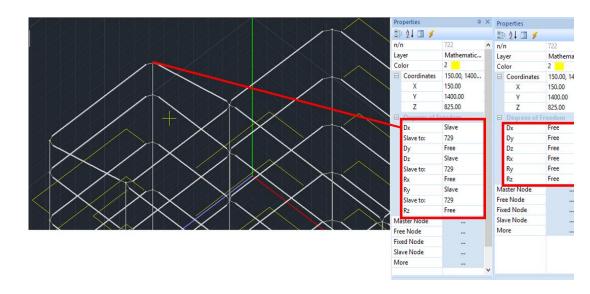
1. To insert slabs strips, all the elements that are connected to the slab, must belong to the same level. Thus, in case of inclined slabs you must define the uneven height value of the considered level:

evel Management XZ New Level Name Edit Name Delete Height (cm) Execute Execute	Add multi Numbe 1500100 + 0	ple levels er 0 Add	89 (140) (140) (153)	
n/n Name 0 0 1 2 3	Height R.L.C. Even H 3D 0.00 ♥ ■ Q 300.00 X ← Q 600.00 X ← Q 900.00 X ← Q	Select All Deselect R.L.C Non R.L.C.) – – – – – – – – – – – – – – – – – – –
4 5	1200.00 🔅 🖬 🚨 1500.00 <table-cell></table-cell>	Even Height Non Even Height Display in 30 Hide in 30		- <u>54</u> - <u>511</u> 14.0
Connection Method of Columns' No	odes with Mesh Surface		S8 217	

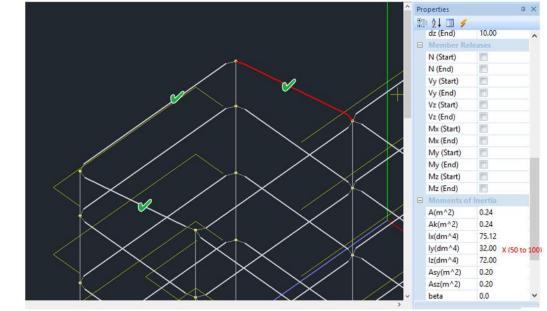
Declare the height value (for example - 100) > Execute > Exit.



2. The nodes of the inclined slab, must be excluded from the diaphragm. Select the nodes one by one and release all of their constraints related to the diaphragm node.

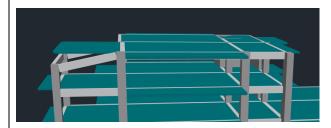


3. Next, you must multiply the ly of the edge beams with a factor 50 to 100.

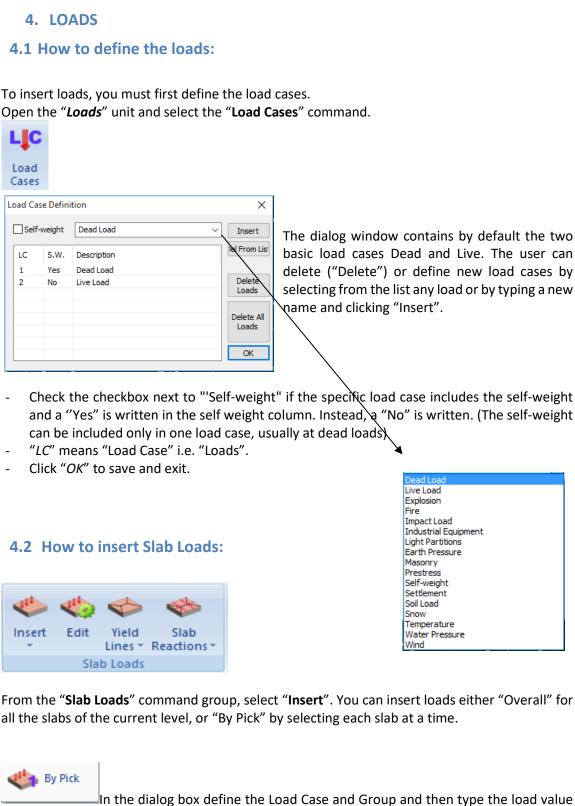


💧 NOTE

The rendering displays the slab as if it wasn't inclined. In any case the slabs are considered as if they were horizontal. Only the beams are inclined.







for each type of slab (KN/m²). "Select" and click on the slab.



Insert Slab Loads	;	×	Predefined Load		×
Load Case	Dead Load	\sim	Import from	MOSAIC	~
Load Group	Group 1	\sim	Description	MOSAIC	
Load Type	Uniform V Predefined		Load (kN/m2)	1.8	Add to Library
Load(kN/m2)	2		Height (m)	0	ОК
Selec	t		Final Load (kN/m)	0.00	Cancel

The "Predefined Load" button, contains a library with coating materials that automatically updates the assigned load value. The user can update or/and enrich the library with new materials by defining the corresponding load values.

Uniformly Dist	ributed l	Loads		×
Load Case	Dead L	oad		\sim
Group	Group	1		\sim
Loads (kN/m2)			
1	G	eneral	Predefined	
Slab Type		Solid	Zoellner	
Cantilev	/er	1	1	
Two-Way In	nclined	1	1	
Two-Way	Slab	1	1	
Three-Way	/ Slab	1	1	
Four-W	ау	1	1	
Triangu	lar	1	1	
Lc=1/Lg=1	~	Insert	Add to List	
Apply		Delete		
Replaceme	ent		Exit	

Overall

In the dialog window that appears, fill in a load value (KN/m^2) and click the "General" button to assign this value to every slab type.

The "Insert" button, creates the loads but it won't be applied until you click "Apply". Select another Load Case and repeat the same process.

To apply the loads that you just defined click "Apply". The loads are automatically distributed uniformly on the slabs area of the current level.

The first time that you insert a load (for example dead load) after the "Insert" command you select "Apply". Next, if you want to add live load you define it and then click "Add to List".

In this example the load values that were considered are 1KN/m2 for dead loads and 2KN/m2 for live loads for all slabs.



4.3 How to assign the slab loads to the members:

After inserting slab loads, select:



Lines "Yield Lines": Load areas' calculation resulting from geometric partitioning of the slab, and then used to calculate the design forces for beams (slab loads which will be imposed on beams),



Calculation is automatically made by the program according to the support conditions, either Overall or By Pick.

and



Reactions "Slab Reaction": To assign slab loads on beam members as reaction - Load distribution from slabs on beams and columns, based on the geometric partitioning done previously (Yield lines).

*	Overall
	By Pick
Equiva	lent
جه	Overall - Equivalent
	By Pick - Equivalent

Overall: select the command (Load distribution from all the current level slabs).

By Pick: select the command and then left click inside one or more slabs (Load distribution from the selected slabs).

Equivalent: With this command, you can assign (Overall **ver** or By

Pick respectively) the slab loads on the connected members, without considering the yield lines evaluation (rectangular and triangular areas). Instead the assignment is implemented by the

convertion of the entire area corresponding to the member, in an equivalent rectangle.



4.4 How to assign loads in members:

From the "Member Load" field, select "Insert" and define the elements (member, node and surface) to assign the loads to. For elements selection use $\square \land \square \land \square$. Complete selection by pressing the right mouse button and then the following dialog box appears

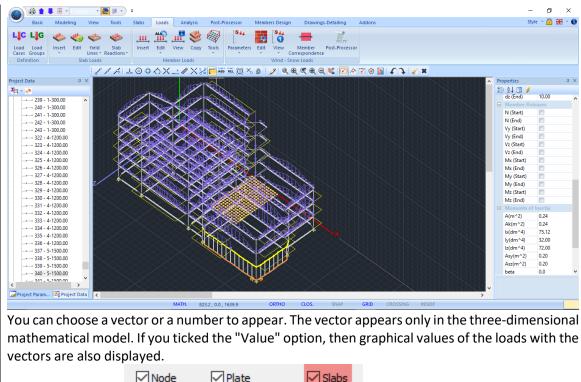
1 200	d Case	Dead Load \checkmark Group	Group 1
Load Load	Property Type	Load Kind	777
Eleme	ent 🗸	Uniformly Distributed Loads $\qquad \lor$	i 🗖 😽 😽 🚽
Descr	iption	COATING	ি⊷dis.i⊷ ⊸dis.j⊶ Y1\
Value	i (kN/m)	8.4 Value į (kN/m) 8.4	
Dist.i	(cm)	0 Dist.j (cm) 0	$\gamma \rightarrow \gamma$
Angle Apply		0 Predefined Local xy ~ Load	
LC	LG	Description	Insert
1	1	U.D.F. COATING 8.40/8.40/0.00/0.00/0.00	Clear Lis
			Clear by Selec

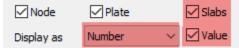
Click the "Insert" button to insert the defined load to the table and "OK" to apply the loads to the selected members.



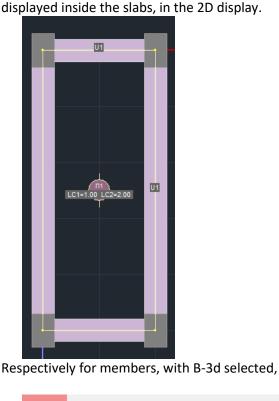
Click the command to display the loads for all elements, in 3D view as vectors, with or without values, or in 2D view as number.







the values of the slab loads are

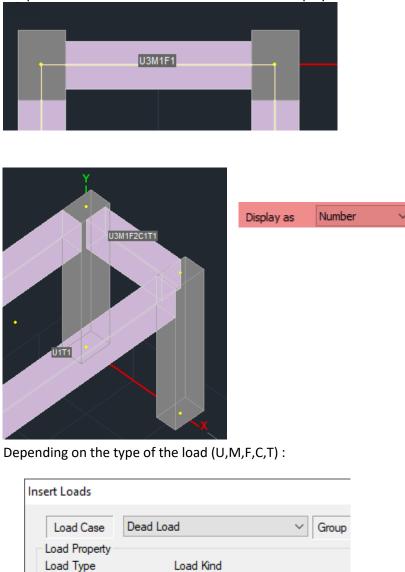


Also, by choosing

B-3d	Truss	✓ B-30	loef	✓ Node	🗹 Plate		🗹 Slabs
Scale (1 L	.oad Unit) =	10	cm	Display as	Number	~	🗹 Value



the presence of loads in letters and numbers is displayed on the member,



And the number indicating how many loads of that type exist

Uniformly Distributed Loads

Uniformly Distributed Loads

Torsional Moment

Trapezoid Forces

Slab Reactions

0

Local xy

Concentrated Forces

Member Temperature

Transverse temperature

U

М

F

С

т

Element ~

Value i (kN/m)

Description

Dist.i (cm)

Angle

Apply To

 \sim

Predefined Load



200			Display a	as Vecto	or v
	000				
THUR HER HER HER HER HER HER HER HER HER HE					
Finally, in the option Filter	Filter :	From 0	То	0	you can d
for the loads you wish to ap	pear.				

In this example a value of 8,4KN/m for dead loads was defined for every beam of the perimeter in 1,2,3 and four levels.

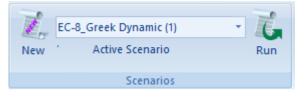


5. ANALYSIS

As soon as modeling and loads' assignment have been completed, the "Analysis" of the structural member, for the design of the structure follows, based on the provisions of the current design codes, to get the results, the loads combination and the final checks.

5.1 How to create a new analysis scenario:

The command group "Scenario" allows sceneries creation (choosing regulation and type of analysis) and implementation.



Press "New" and in the dialog box you can create analysis scenarios by choosing different design regulations and methods of analysis. By default there are two scenarios based on the selected "language" codes (including local Annex if there are any, or "EC-General" if there are not)

*Predefined scenarios are created according to the Rule and Attachment option you made at the beginning, within the General Parameters window that opens automatically immediately after the file name is defined.

				Analysis	EC-8_Greek	~
				Type Properties		
Scenario			×	Elemen	EC-8_Greek NTC_2008	
Renumbering Nodes Cuthill-McKee(II)	~	Advanced Multi-Threaded Solve	er	Load Ca	EC8_Italia EC8_Cyprus EC8_Austrian EC8_General	
				New	SBC 301	-
Disable	Name			Туре	Dynamic	~
EC-8_Greek Static (0)	Analysis	EC-8_Greek	\sim	Properties		
EC-8_Greek Dynamic (1) Static Wind 0 (2) Static Wind 90 (3)	Туре	Dynamic	~		Dynamic Nonlinear Linear	
Static Wind 180 (4) Static Wind 270 (5) Static Typical snow (6)	Propertie			Load Ca	Elastic Dynamic Preliminary Static Preliminary Dynamic	
Static Typical show (b)	Load C	Cases Masses		New	Time History Linear	
				Analysis	EC8_General	~
	Nev	v Update	•	Туре	Static	~
		Exit		Properties		
		Ext		Elemen	Dynamic Nonlinear	
				Load Ca	Linear Elastic Dynamic Preliminary Static Preliminary Dynamic	
				New	Time History Linear	



New	sis" list and the analysis method from the "Type" list and ysis scenario. Optionally, type a name.
Select among the possible scenarios pro	ovided in SCADA Pro:
<u>For Greece:</u> <u>LINEAR</u> -	– NON LINEAR METHODS
- EAK Static	Simplified spectral analysis according to EAK
- EAK Dynamic-eti	Dynamic spectral analysis according to EAK
- EAK Dynamic	Dynamic spectral analysis (masses displacement) according to EAK
- Old 1959-84	Seismic analysis according to 1959 Regulation
- Old 1984-93	Seismic analysis according to 1984 Regulation
- Static	Static Analysis without seismic actions
- EC 8 Greek static	Static analysis according to Eurocode 8 and the Greek Appendix
- EC8 Greek dynamic	Dynamic analysis according to Eurocode 8 and the Greek Appendix
- EC 8 Greek Preliminary Static	Static Preliminary analysis according to KANEPE
- EC8 Greek Preliminary Dynamic	Dynamic Preliminary analysis according to KANEPE
- EC 8 Greek Time History Linear	Static analysis according to Eurocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis according to Eurocode 8
- EC 8 Greek NonLinear	Nonlinear analysis according to Eurocode 8 & KANEPE.

For other countries:

LINEAR - NON LINEAR METHODS

- NTC 2008	Seismic analysis according to the Italian Regulation
	2008
- EC8 Italia	Seismic analysis according to Eurocode 8 and the
	Italian Appendix
- EC8 Cyprus	Seismic analysis according to Eurocode 8 and the
	Cyprus Appendix
- EC8 Austrian	Seismic analysis according to Eurocode 8 and the
	Austrian Appendix
- EC8 General	Seismic analysis according to Eurocode 8 with no
	Appendix (enabled typing values and coefficients)
- EC 8 General Non Linear	Nonlinear analysis according to Eurocode 8
- SBC 301	Seismic analysis according to Saudi Arabia code (SBC
	301)

For this example a Dynamic Scenario by the Eurocode 8 will be used.



Scenario		×		
Renumbering				
Nodes Cuthill-McKee(II)	✓ Advance Multi-Thr	readed Solver		
Disable EC8_General Static (0) EC8_General Dynamic (1)	Name Analysis EC8_Ge Type Static Properties Elements Load Cases New Run all A	Vodes Masses Update		
The "Renumbering" field ind options: The choice of each option a ✓ Default choice: "Cuthil ✓ "Cuthill-Mckee" and "A complete the analys recommended. Select Exit	offects the compo I-Mckee(II)". Ascending Order is, while choo	utational time. " take more time to		Cuthill-McKee(II) ~ No Ascending Order Cuthill-McKee Cuthill-McKee(II) Scada
Click "Elements" to open properties of the linear elen	-		nultipliers	of the characteristic



EC8_General Dynamic								E	ic 📐 🗸
Linear Element Proper	ty Value	Multiplier							
Concrete \lor	E	G	Ak	Asy	Asz	3	Ix	-/	.0 .A.K.
BEAMS - B3D	1	1	1	1	1	1	0.1	0.5	
BEAMS - TRUSS	1	1	1	1	1	1	0.1	0.5	0.5
BEAMS - B3Def	1	1	1	1	1	1	0.1	0.5	0.5
COLUMNS - B3D	1	1	1	1	1	1	0.1	0.5	0.5
COLUMNS - TRUSS	1	1	1	1	1	1	0.1	0.5	0.5
WALLS - B3D	1	1	1	1	1	1	0.1	0.5	0.5
WALLS - TRUSS	1	1	1	1	1	1	0.1	0.5	0.5
	_		-						_

By default, the values of the multipliers are defined according to the design code, while any modification is acceptable.

If for example, you select "EC" the values of the multipliers will automatically be updated by the Eurocode provisions.

Click the "Nodes" button to open the following dialog box:

Select whether to consider slab's Master Node (FSR) by selecting "Yes" (default) or not by selecting "No"

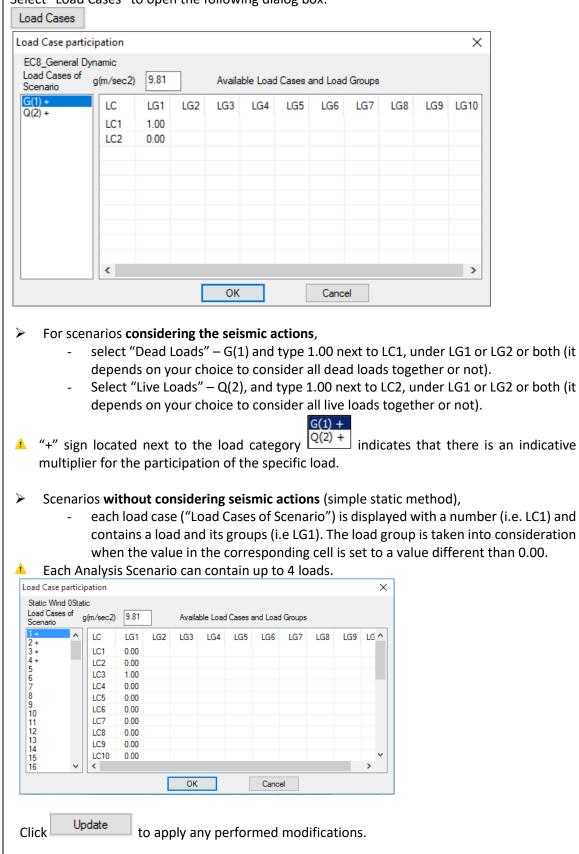
Nodes		×
EC8_General Dy	namic	
Master Nodes	Yes	\sim
Springs		
Dx	Dy	Dz
Yes 🗸	Yes 🗸 🗸	Yes 🗸
Rx	Ry	Rz
Yes 🗸	Yes 🗸 🗸	Yes 🗸 🗸
ОК	(Cancel
In cases that	t a Dynam	nic Analysis is r

Moreover, you can choose whether to allow the corresponding displacement or rotation of the foundation's springs or not (fixed support conditions).

In cases that a <u>Dynamic Analysis</u> is required, if you select "Nodes" and you activate the springs ("Yes"), then you will be able to use the combinations of the dynamic analysis for the footing design as well.

Press Update to update the scenario with the performed changes.





Select "Load Cases" to open the following dialog box:



5.2 How to run an analysis:

New	EC8_General Static (0) EC8_General Static (0) EC8_General Dynamic (1)	Run
_		<u> </u>

select from the scenario list the considered scenario, i.e. the scenario that will be used for the analysis.

In the scenarios list, apart from the two predetermined, all the previously created scenarios are created. Choose one scenario at a time and continue with the definition of the parameters of the corresponding analysis



Click the "Run" button to open the parameters of the current analysis window which differs for:

- ✓ EAK Scenarios
- ✓ Eurocodes Scenarios
- ✓ Non-Linear Analysis Scenarios

First of all, press Initialize Data to update the parameters of the current scenario.

Then press Parameters to define the parameters of the project.

Based on the selected scenario, the parameters dialog box differs accordingly. In this example, having selected the Eurocode 8 scenario, the dialog box will have the following format:



EC8 Parameters				X
Seismic Area	Characteristic Periods			
			Apply seismic actions on Levels XZ Down 0 - 0.00 V Up 0 - 0	
Seismic Areas		0.9	Down 0 - 0.00 V Up 0 - 0	.00 V
Zone I v a 0.16 *g	Type I Systy		Dynamic Analysis	
	Soil TB(S) 0.15	0.15	Eigenvalı 10 Accuracy 0.001 C	QC V
Importance	B C CC		Spectrum Participation factors	
Zone II V Vi 1	TD(S) 2	1 P	PFx 0 PFy 0 PFz	z 🗌 🕛
Spectrum			Acc.Eccentricities Sd (T)	
Response Spectrum Design	 Ductility Class 	DCM \checkmark	е тих □ 0.05 *Lx Sd (TX)	1
ζ(%) 5 Horiz	ontal b0 2.5 Vertical b	b0 3	Sd (TY)	1
Response Spectrum U	pdate Spectrum Sd(T) >=	0.2 a*g	е ті 0.05 *Lz Sd (ТZ) (1
Structural Type	50(i) / 2		Davis Datharda	
Concrete v qx	□ 3.5 qy □ 3.5 qz [3.5	Bays Setbacks	
Structural Type			X One X All the other c	ases
X Frame System	Z Frame System	m	Z One Z All the other c	ases
Fundamental Periods				
r undumentari enous	Calculation Method	EC8-1 par. 4	1.3.3.2.2 (3)	\sim
X Concrete Moment Resisting	Frames	V Z Concrete Mo	ment Resisting Frames	\sim
Interstorey Drift Limit	0.005		Walls KANEPE Default O	K Cancel
				Cancer
Seismic Triangul	ar 🗸		ΚΡΙΤΗΡΙΑ ΑΠΑΛΛΑΓΗΣ ΣΤΑΤΙΚΗΣ ΕΠ/	ΑΡΚΕΙΑΣ
the area, type of seismic Areas Seismic Areas Seismic Area Seismic Areas Zone I v a 0.16 *g	soil, importance of the a file that contains a list eismicity zone, pops up.	ne structure	in this dialog box (level of sector). By clicking " Seis ne national annex, with the ent "a" will be filled ir	smic areas"
automatically.				
Characteristic Periods	C 11	•	trum Type (for Greece Typ	-
Spectrum Type			t all the coefficients for bot	in norizontal
Type 1 v S,avg	1.2 0.9 and	vertical spec	ctrums are filled in	
Soil TB(S)	0.15 0.05			
TC(S)	0.5 0.15			
5				
TD(S)	2 1			

Choose the type of "Response spectrum" and "Ductility class" to suit your analysis



EXAMPLE: «CONCRETE STRUCTURE ANALYSIS AND DESIGN»

Spectrum Response Spectrum Design V Du ζ(%) 5 Horizontal b0 2.5		
	ctility Class DCM 🗸	
S(70) S Horizontal DU 2-5	Vertical b0 3	
Response Spectrum Update Spectru	m Sd(T) >= 0.2 a*g	
Choose the "Structural Type		
Structural Type		
Concrete V		
Concrete		
S Steel		
Composite Unreinforced masonr		
Confined masonry S Reinforced masonry		
Low seismity masonr		
The " Rehavior factor a " o	f the structure is a result of	a computation procedure
Additionally, the "Structure ty		u computation procedure.
Additionally, the Structure ty		
Q		
qx 3.5 qy 3.5	qz 3.5	
Structural Type		
X Frame System	Z Frame System	
		4 1
Canada Dua at a tha		
described in the next chapter: "In the field <i>Structure periods:</i>	he opportunity to get rid of the How to calculate the behavior fact	or q
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section:		or q
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S	How to calculate the behavior fact	or q te the fundamental period. Now
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section:	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3)	or q te the fundamental period. Now
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods	How to calculate the behavior fact	or q te the fundamental period. Now
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method Calculation Method Z Concrete Moment Resist	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3)	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3)	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method Calculation Method Z Concrete Moment Resist	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5)	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method Calculation Method Z Concrete Moment Resist	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3)	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method Calculation Method Z Concrete Moment Resist	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5)	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Z Concrete Moment Resist calculate the period in three ways.	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to (EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Z Concrete Moment Resist calculate the period in three ways.	or q te the fundamental period. Now ng Frames
described in the next chapter: " In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to the EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis The first two are the approximation	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Z Concrete Moment Resist calculate the period in three ways. te methods of EC8-1. 4.3.3.2.2 (3) it is necessary:	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis The first two are the approximation 1. In the first one EC8-1 par. 4 To choose, per direction, the	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Z Concrete Moment Resist calculate the period in three ways. te methods of EC8-1. 4.3.3.2.2 (3) it is necessary:	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis The first two are the approximate 1. In the first one EC8-1 par. 4 To choose, per direction, the X Concrete Moment Resisting Frames	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Z Concrete Moment Resist calculate the period in three ways. te methods of EC8-1. 4.3.3.2.2 (3) it is necessary: he structure type Z Concrete Moment Resisting Tames	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis The first two are the approximate 1. In the first one EC8-1 par. 4 To choose, per direction, the X Concrete Moment Resisting Frames	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Concrete Moment Resist calculate the period in three ways. Calculate the period in three ways. te methods of EC8-1. 4.3.3.2.2 (3) it is necessary: he structure type V Z Concrete Moment Resistin Tames The Frames	or q te the fundamental period. Now ng Frames
described in the next chapter: "I In the field <i>Structure periods:</i> In previous versions there was the S it is replaced by the section: Fundamental Periods X Concrete Moment Resisting Frames There is now an opportunity to a EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis The first two are the approximate 1. In the first one EC8-1 par. 4 To choose, per direction, the X Concrete Moment Resisting Frames	How to calculate the behavior fact Structure Type X and Z field to calcula Calculation Method EC8-1 par. 4.3.3.2.2 (3) Concrete Moment Resist calculate the period in three ways. te methods of EC8-1. 4.3.3.2.2 (3) it is necessary: he structure type Z Concrete Moment Resisting Tames To Frames ic bracings	or q te the fundamental period. Now ng Frames





		X or/and Z c field "Bays"	lirection, the Bays X One Z One)	structure on	Ily consis	sts of one frame you activate the
Afterw	ards choos	e the comman	d "Walls"	Valls to assi	an a value	e for the minimum length that a
				d as a wall inste	•	
Walls I	Definition EC	8 - SBC301			×	
m	nin Columns Le	ngth (cm) >=	200			
Col	lumn	Element	Vy	Vz	hw ^	
1		265			0.0	
2		266			0.0	
3		267			0.0	
4		268			0.0	
5		269			0.0	
6		270			0.0	
7		271			0.0	
8		272			0.0	
9		273			0.0	
10		274			0.0	
<					>	
	Add All	Clear All	ОК	Car	ncel	
		all length (cm checked in ea	-	าย	ength (cm)	button, and automatically, according to paragraph 4.3.3.2.2
				EC8-1 par	43322	2 (5)

- 2. For the second approximate method [EC8-1 par. 4.3.3.2.2 (5)], there is no need to do any further action as long as it is selected.
- 3. The third method includes a Modal Analysis to calculate the periods.

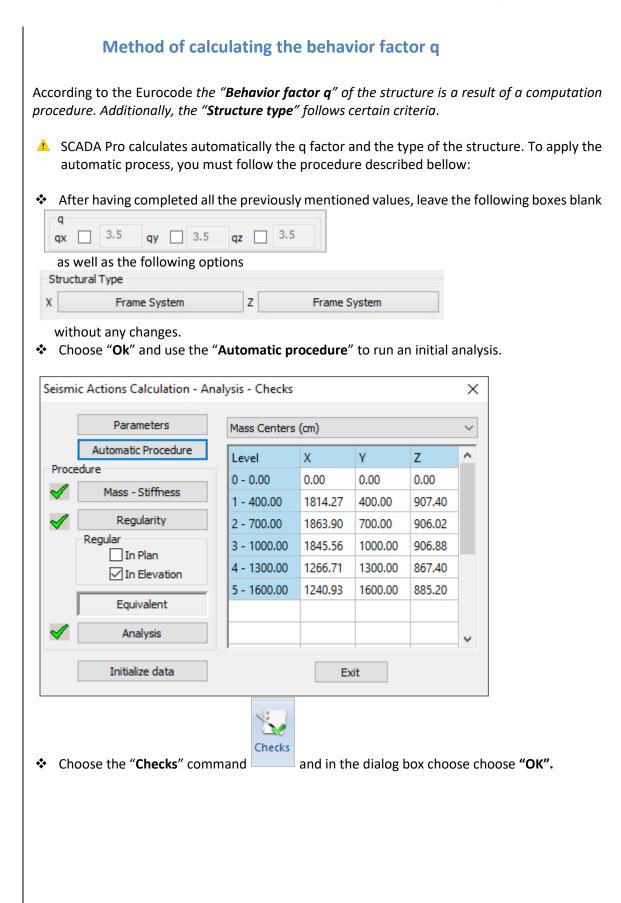
The program takes into consideration the period which corresponds to the dominant modal in each direction. (the modal which has the biggest percentage of the activated mass)

The user can increase or decrease the number of eigenvalues in case of dynamic or static analysis, as long as the calculation of the eigenvalues with Modal Analysis and the percentage of accuracy are chosen.



Dynamic Analysis
Eigenval 10 Accuracy 0.001 CQC ~
Spectrum Participation factors CQC (10%)
Also, there is also the opportunity to choose the method of combining the modal responses
according to Complete Quadratic Combination CQC and CQC (10%)(3.6 EAK), or the square root
of the sum of squared (SRSS) method.
Moreover, the results of the modal analysis for the static scenarios are included in the results of
seismic action. To modify the coefficients of the eccentricities, select the respective checkbox and type the new
value on the right.
Acc.Eccentricities
е тіх □ 0.05 *Lх
е ті 2 0.05 *Lz
In the same way, the engineer can modify the X, Y, and Z spectrums by typing his values in the
respective fields,
sd (T)
Sd (TX)
Sd (TY)
Sd (TZ) 1
as well as the spectrum participation factors.
Standard David a factor
Spectrum Participation factors DIRECTION X
PFx U PFy V PFz V All the other cases
In the Indents field, select for each direction the case that is
appropriate for the particular study and is defined by the Eurocode.
Setbacks
X All the other cases
Z All the other cases
OK Cancel
Seismic Triangular V
The engineer can also choose the Type of
Distributionç of the seismic force between two options. Triangular







In the dialog box "Seismic analysis control coefficients" you are asked Seismic Analysis Control Coefficie... to assign a value for the minimum length that a vertical member must Angular Distortion y li <= 0.005 have to be regarded as a wall instead of a column. Click the min Wall Length (cm) >= 200 Column Element Vv Vz ^ min Wall Length (cm) >= button, and automatically, all the walls are checked in each direction. ~ ✓ Wall Adequacy ratio (nv) Additionally, by checking the boxes File with internal forces from load combinations (combin.txt) next to the two last options, two .txt files will be created and saved to the 7 10 10 folder of the project, ready to be viewed or printed afterwards. Add All Clear All Mass - Stiffness Limits As far as the "Wall adequacy" is concerned, the relevant .txt file Reduction 0.5 Reduction 0.5 contains the computation of the shear acting to each wall, at each level Increase 0.35 Increase 0.35 of the structure and for all the load combinations considered. Wall Adequacy ratio (nv) File with internal forces from load combinations (combin.txt) OK Cancel Mass - Stiffness Limits The "Mass - Stiffness limits" area, since no specific Stiffness Mass limitation is prescribed by EC8 (in contrast with EAK – Greek Reduction 0.5 antiseismic regulation), modifications may be incorporated Reductior 0.5 to those limits. Consequently, the building's regularity state 0.35 0.35 Increase Increase in elevation will be altered, too.

In the "Checks" file, the program "defines" the structural type by the base shear undertaken by the walls.

	• V:						
e Edit	t view	Insert Form	пат негр	1			
<u>ן ש </u> נ		🛕 🏘 🛓	6 🖻 🛍 🖻 💁				
						*	
							(m)
wails	Shear	Force Par	r. 5.1.2.	F	eference Level	1:0 0.000	(m)
					*		
n/n	Wa:	lls Shear,	/Total Shear =		* Walls Shear,	/Total Shear	
n/n		lls Shear,			*	/Total Shear	
n/n Level	Wai	lls Shear, (Kn)	/Total Shear =	nvx	* Walls Shear,	/Total Shear (Kn)	= nvz
n/n Level	Wa ** 25-	lls Shear, (Kn) 621.141	/Total Shear = (Kn)	nvx 0.31 NO	* Walls Shear/ (Kn) *	/Total Shear (Kn) 1737.703	= nvz
n/n Level 1 ** 2	Wa: ** 25- 18-	lls Shear, (Kn) 621.141 500.708	/Total Shear = (Kn) 2025.586	nvx 0.31 NO 0.26 NO	* Walls Shear/ (Kn) * 40-1169.332	/Total Shear (Kn) 1737.703 1170.361	= nvz 0.67 0 0.49 N
n/n Level 1 *1 2 3	Wa ** 25- 18- 25-	(Kn) 621.141 500.708 707.106	/Total Shear = (Kn) 2025.586 1940.827	nvx 0.31 NO 0.26 NO 0.27 NO	*	/Total Shear (Kn) 1737.703 1170.361 1085.765	= nvz 0.67 OI 0.49 N(0.41 N(
n/n Level 1 *1 2 3	Wa ** 25- 18- 25- 5-	(Kn) 621.141 500.708 707.106 148.031	/Total Shear = (Kn) 2025.586 1940.827 2576.280	nvx 0.31 NO 0.26 NO 0.27 NO 0.18 NO	* Walls Shear, (Kn) *	/Total Shear (Kn) 1737.703 1170.361 1085.765 558.370	= nvz 0.67 OI 0.49 N(0.41 N(

Since the **"Building system definition"** has been determined, it should be included in the **"Parameters"** dialog box. With these changes, analyze for a second time. Now, the proposed values for the **"Behavior coefficient q"** can be found in the "Parameters" dialog box. For the example considered, in the **"q"** area, one can read.



q		2.76	qy 🗌	1.38 q	z 🗌 2.7	76	
from	the	beginn es, at all 2.76	ing of the I).	• •	ure, how	vever,	he latter one is an option that could be utilized in this occasion the software would not propose
Click				to	update th	ne spe	ctrum by the new values of the q factor and click
		Spectrum Spectru	to see i	it.			×
A	/A	T(s	RdTx	RdTy	RdTz	^	Rds = 1.8822
1		0.000	1.884	1.413	1.884	- 1	Σmi/M = 0.00%
2		0.050	2.109	2.895	2.109	_	
3		0.100	2.334	2.895	2.334	_	
4		0.150	2.559	2.895	2.559	_	Ty = 0.2027
5		0.200	2.559	2.171	2.559	_	Rds = 2.1425
6		0.250	2.559	1.737	2.559	_	Σmi/M = 0.00%
7		0.300	2.559	1.447	2.559	_	
8		0.350	2.559	1.241	2.559	- 1	
9		0.400	2.559	1.086 0.965	2.559	- 1	
	•	0.430	2.339	0.905	2.339	~	Tz = 0.6798 Rds = 1.8822
	Defa	ult	Write	ТХТ	OK		Σmi/M = 0.00%
	Read	TXT			Cance	el	
		amaged	Structures	check			
B		s' catego			nstruction r	period h	pefore 1985 EAK ???
	S	eismic ma	agnification	coefficien	t 0	a*/g	0 Spectrum Calculation

Click "Ok" and conduct the analysis one more time, for the **q values to be accounted**.



5.3 How to check the analysis results and create the combination file

Right after running the selected scenario analysis, use the "Results" command group to create the combinations (to apply the EC8 checks and design) and view the results of the analysis:



Click "Combinations" to open the "Load Group Combinations" where you can define your combinations or use the results derived from the "Default Combination" button, which completes the table with the combinations of the active scenario analysis.

ad Groups Con	nbinations																
G 1.35 Q 1.5	γE 1 γE0.3 0.3		γGE	1		ψ2	0.3 Wind - Sno	w	✓ ΣG-	3+γ +ψ1	/Q+Σγψ0Q 1Q+Σψ2Q +Σγψ2Q	2	erviceability 2 ΣG +Q +ΣψΟΟ 2 ΣG +ψ 1Q +Σψ 2 ΣG +Σψ2Q	-	Calco		
	Туре	-	Direction		LC1		LC2	Τ	LC3		LC4		LC5		LC6	Т	LC /
Scenario					EC8_Ge	ner 💌	EC8_Gener	-	EC8_Gener	•	EC8_Gener	¥	EC8_Gener	•	EC8_Gener	-	EC
Load Case					1		2		3		4		5		6		5
Load Type					G	-	Q .	-	ExD	•	EzD	•	Erx	•	Erz	•	Eyĺ
Actions						-	Category A	-		•		•		•		•	
Description																	
		_														4	
Comb.:1		-	No	_	1.35		1.50	_								_	
Comb.:2	Ultimate	-	No	_	1.00		0.50	_								_	
Comb.:3	Ultimate	-	Dir. +X		1.00		0.30	_	1.00		0.30		1.00		0.30	_	0.3
Comb.:4	Ultimate	-	Dir. +X	_	1.00		0.30	_	1.00		0.30		1.00		0.30	_	-0.
Comb.:5	Ultimate	_	Dir. +X	_	1.00		0.30		1.00		0.30		1.00		-0.30		0.3
Comb.:6	Ultimate	•	Dir. +X	_	1.00		0.30		1.00		0.30		1.00		-0.30		-0.
Comb.:7	Ultimate	•	Dir. +X	•	1.00		0.30		1.00		0.30		-1.00		0.30		0.3
Comb.:8	Ultimate	•	Dir. +X	•	1.00		0.30		1.00		0.30		-1.00		0.30		-0.
Comb.:9	Ultimate	-	Dir. +X	-	1.00		0.30		1.00		0.30		-1.00		-0.30		0.3
Comb.:10	Ultimate	•	Dir. +X	-	1.00		0.30		1.00		0.30		-1.00		-0.30		-0.
Comb.:11	Ultimate	-	Dir. +X	-	1.00		0.30		1.00		-0.30		1.00		-0.30		0.3
Comb.:12	Ultimate	-	Dir. +X	-	1.00		0.30		1.00		-0.30		1.00		-0.30		-0.

After running a scenario analysis, combinations are automatically generated by the program. "Combinations" opens the table with the combinations of the active scenarios.

- The same results are derived from the "Default Combination" button, which completes the table with the combinations of the active scenario analysis.
- The default combinations of the executed analysis, are automatically saved by the program.



- You can create your combinations without using the "Default", or add more loads of other scenarios and calculate the new combinations either by modifying the defaults, or deleting all "Delete All" and typing other coefficients. Furthermore you can type the factors and select the combinations and then press 'Calculation" to complete the table. The tool "Laod Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.
- Predefined combinations concerning seismic scenarios. To create combinations of scenarios without seismic loads you can use both **automatic** and **manual** mode.

5.4 Checks:

Press "Checks" and in the dialog box:

- ✓ Type in the minimum length for defining the walls and click the corresponding button,
- ✓ set limits on the mass and the stiffness considering the regularity conditions of the building,
- ✓ Activate the creation of the two .txt files
- ✓ "OK"

A TXT file that contains design check's results according to the "active scenarios", opens automatically:

- Regularity
- Second Order effects
- Interstory Drift Limitation
- Interstory Drift sensitivity coefficient θ
- Walls Shear Force ratio nv,z
- Seismic joint's calculation
- Torsional sensitivity

Se	eismic Ana	lysis Contr	ol Coeff	fic X						
	Angular	Distortion	y li <=	0.005						
	min Wall	Length (cm)>=	200						
	Column	Element	Vy	Vz ^						
	1	265								
	2	266								
	3	267		✓						
	4	268								
	5	269								
	6	270								
	7	271								
	8	272								
	9	273		 Image: A start of the start of						
	10	274		└ ∨						
[Add A	I	Clea	ar All						
	Mass - Stif	fness Limits								
	Mass		Stiffne	ss						
	Reduction 0.5 Reduction 0.5									
	Increase 0.35 Increase 0.35									
I	Wall Adequacy ratio (nv)									
I	File with combinat	internal for ions (comb	ces from txt)	load						
	OK		Ca	ancel						



Check		Format Help	1				
Check	<u>sq</u>	X 🖻 🛍					
			<u>a</u>				
		CHE	KS REPORT				
	DYNAMIC I	RESPONSE SPI	CTRUM ANALYSI	S WITH PAIR MOM	ENTS (EC8)		
	for mass	and stiff.	differences pe	r build.level (par.4.2.3.3	3.)	
		*		**	-		-
			tal Stifness	Diffe) (Mi+1-M		s - Stifness	
				-Z)*(ΔMi)			_
				6.261		(1112 2)	
				1.855 inc. 0.02			
				1.855 red 0.01			
				9.631 red 0.38			
5	10.000	JU4.206 S	9420.903 6943	9.631 red 0.09	inc. 0.00	Inc. 0.00	_
sses	: The in	ncrease must	; be <=0.35 - 1	The reduction m		.50	
				tion must be <=			
eck sa	tisfy the	regular.in	elevation cri	teria			
enter	Weight - (Center of St	iff				
	-			*	*	·	-
n/n	Total	CENTER	R WEIGHT	CENTER OF	STIFF	Distance	
				X Coor.(m) Z			
		* 18.1427		* 18.9978	7.5303		-
				17.8566			
		18.4556			7.6826		
4	13.000	12.6671	8.6740	16.4116	7.8626	3.8314	
5	16.000	12.4093	8.8520	16.6140	7.9362	4.3032	
					*		-
lls Sh	ear Force	Par. 5.1.2		Reference Leve	1:0 0.00	00 (m)	
				*			-
-				Walls Shear		ar = nvz	
vel	(Kn)	(Kn)		(Kn)	(Kn)		
1 ***1	25- 621.1	41 2025 5	36 0.31 M	0 40-1169.332	1737.703	0,67,08	-
2	18- 500.7	08 1940.82	27 0.26 N	0 44- 575.583	1170.361	0.49 NO	
3	25- 707.1	06 2576.28	30 0.27 N	0 44- 443.958	1085.765	0.41 NO	
				0 51- 187.460			
5	5- 302.0	49 1256.8	58 0.24 N	0 51- 184.078	581.493	0.32 NO	
							-
ilding	system de	erinition	X : Frame Sp	ystem wall system (c	oupled or "	incounled)	
		nv from reg		warr system (C	oupred of t	mcoupred)	
10	oncok		,				
gulari	ty in plan	n check -	Par. 4.2			ection X	
							-
				> Coef. 1s			
ver	nerg(M) .	Lmax/Lmin	SQTU(2KU/2K_)	sqrt(IO/mass)	ε_0 (m)	TREGULAR	



5.5 Seismic force:

Select the command and a TXT file which contains the parameters considered in the calculation of the seismic actions as well as the calculated results, automatically opens. Enilic Force Enilic Etilic Eti

- ✓ Fundamental Periods
- ✓ Accidental Eccentricities
- ✓ Distribution of the equivalent static force along height (Shear-Moment) Response Spectrum values
- ✓ Response Spectrum values

🗒 Histor.txt - WordPad	_		×
File Edit View Insert Format Help			
SCENARIO : 4 - DATA AND RESULTS OF SEISMIC FORCE			^
DATA FILE LOAD CASES			
Load Case 1 (Dead-G)			
Load Case 2 (Live-Q)			
MASSES CALCULAT. FROM : G+\2*Q			
RESULTS FILE - INTERNAL FORCES			
Load Case 1 (Dead-SG)			
Load Case 2 (Live-SQ) Load C. 3 (Horizontal Seismic Force x)			
Load C. 4 (Horizontal Seismic Force z)			
Load C. 5 (Eccentricity of seism. force x from maxez)			
Load C. 6 (Eccentricity of seism. force x from minez)			
Load C. 7 (Eccentricity of seism. force z from maxex)			
Load C. 8 (Eccentricity of seism. force z from minex) L. Case 9 (Vertical Seismic Force y)			
1. ouse 5 (Activital Science Torde 3)			
SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF BUILDING			
Calculation Parameters			
Ductility Class : DCM			
Response Spectrum Type : Type 1			
Sesmic Zone : II			
Acceleration of Gravity g (m/sec2) : 9.810 Design ground acceleration αgR : 0.24*9.810=2.3544			
Build. system along X : Frame System			
Build. system along Z : Frame-equivalent dual system			
Ground Type : B			
Characteristic Periods : TB=0.15 TC=0.50 TD=2.00(sec) Factor-Importance Category : γι=1.000 - Σ2			
Behavior Factor : qx=2.760 - qz=2.760 - qy=1.830			
Lower bound factor : ßo=2.50			
Viscous damping ratio : ξ=5.000%			
n/n Level Plan Dimensions Coef.ψ2 Acc. Eccenticities			
Height (m) LIX (m) LIIZ (m) L.C.2 etix(m) etiz(m)			
0 0.000 31.900 16.500 0.300 1.595 0.825 1 4.000 31.900 14.900 0.300 1.595 0.745 2 7.000 31.900 14.900 0.300 1.595 0.745 3 10.000 31.900 14.900 0.300 1.595 0.745			
2 7.000 31.900 14.900 0.300 1.595 0.745			
3 10.000 31.900 14.900 0.300 1.595 0.745			
4 13.000 19.500 14.600 0.300 0.975 0.730			
5 16.000 19.500 14.600 0.300 0.975 0.730			
 ετix = 0.05 *LIx , ετiz = 0.05 *LIIz			
			~
For Help, press F1		N	IUM /
			11



When the scenario regards to a Dynamic Analysis, the following units are included to the exported data as well:

- ✓ Fundamental Periods derived from dynamic analysis
- ✓ Eigenvalues Participation Factors
- ✓ Masses participation factors / Direction
- ✓ Active Modal Masses

```
Fun.Periods (Modal Resp.Spect. analysis)
   _____
  n/n Cyclic Frequency Frequency Period
Eigenvalue w (Rad/sec) v (Cycles/sec) T (sec)
   _____

      7.2512E+000
      1.1541E+000

      7.9224E+000
      1.2609E+000

      1.4538E+001
      2.3138E+000

                                        8.6651E-001
      1
      2
                                         7.9309E-001
      3
                                         4.3218E-001
      4
           2.6808E+001
                          4.2667E+000
                                         2.3438E-001
           2.9671E+001
3.1355E+001
3.4995E+001
3.7508E+001
                          4.7222E+000
                                         2.1176E-001
      5
                                         2.0039E-001
      6
                          4.9903E+000
                          5.5696E+000
                                         1.7955E-001
      7
                                         1.6752E-001
                          5.9695E+000
      8
         3.8419E+0016.1146E+0004.2227E+0016.7206E+000
                                         1.6354E-001
      9
                                      1.4880E-001
     10
     _____
  Eigenvalues Participation Factors
   _____
     n/n Dir in reference to the global coord.system
   Eigenvalue Dir X Dir Y Dir Z
    1 2.1433E+001 -6.9810E-001 2.7847E+001
2 -3.7677E+001 -2.0058E-001 2.1469E+001
                                         2.5324E+001
      3
           8.4483E+000
                          -1.6991E+000
          8.8804E+000
-1.4702E-001
                          2.9948E+000
      4
                                         2.1994E+000
                           2.9863E+001
      5
                                         4.0690E-001
            1.8737E-001
                          -1.5404E+001
                                        -3.3619E+000
      6
      7
           1.6543E+000
                           2.1082E+000
                                         -2.0242E+000
           9.7196E+000
      8
                          -6.0046E+000
                                         -1.8085E+000
      9
           -8.7209E+000
                          -1.2616E+000
                                         2.2461E+000
                           1.5838E+001
     10
            3.3796E+000
                                         -2.0670E+000
     _____
  Masses participation factors / Direction
      _____
   Dir X = 1 Dir Y = 1 Dir Z = 1
       _____
For more details you can see the User Manual § 7. ANALYSIS
```





6. RESULTS

6.1 How to view the diagrams and the deformation results and the mesh areas steel reinforcement demand:

Move to "Results" Unit, to get a detailed observation of the internal forces, the diagrams (M, V, N) and the deformed shape of the model as a result of an individual load or load combination.

	Basic	Modeling	View	Tools	Slabs	Loads	Analysi	s Post-	Processor	Members Desi	gn Drawir
-	1	Model		Ŧ			1	•		<u>ш</u>	
Combin	nations				Deformed	Animation	2D Diagrams	Numbering Display	Edit Properties *	Load Properties (By Pick)	Stress Failure Criterion
		Deform	ation Diagr	ams					Vario	JS	

Select "Combinations" and load a combination's file, depending on the results you want to see. In the dialog box:

Load Combinations X										
C:\IO\essor	C:\IO\lesson2\scaanal\EC8_Genera									
Load	Load 7									
Load	Load 101									
EC8_Genera	EC8_General Dynamic (4).cmb									
	Combinations Select									
	Calculation									
End Calc										
ОК		Cancel								

- Choose a combination from the list that includes the combinations of all the analyses that have been performed, and wait to complete the calculation automatically, or

- press "<u>Combinations Select</u>", select the combinations file from the correspondent folder and press "Calculation".

1 To see the deformed shape of the corresponding eigenvalues, choose a dynamic scenario .cmb file.

Model	-
Model	
Diagrams-Stress Contours	

From the list on the right, based on the desired results select:

- Model or
- Diagrams Stress Contours

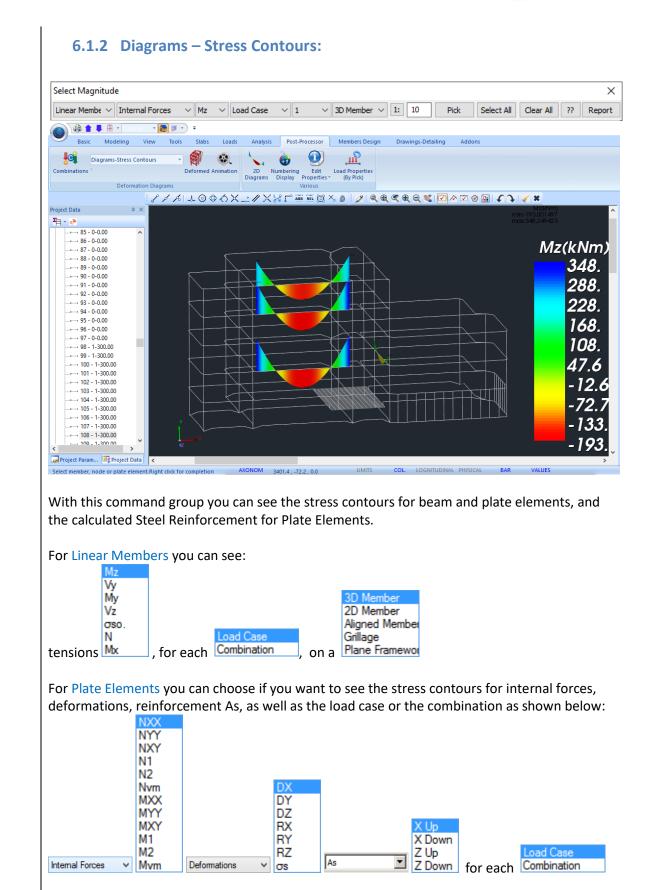
 \checkmark

✓



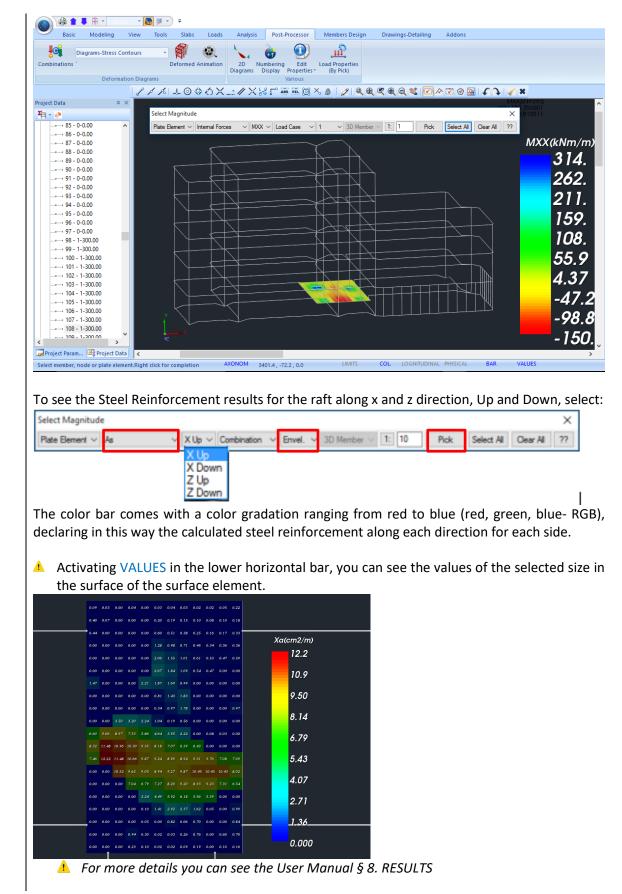
Model Combinations	View Tools Slabs Loads Analysis Post-Processor Members Design Drawings-Detailing Addons View Tools Slabs Loads Analysis Post-Processor Members Design Drawings-Detailing Addons Deformed Animation 2D Numbering Edit Load Properties By Pick)
Deformation	- アイイー クス アメンデ 華 亜 回 ふ & ア
Proj Deformed Model Elast Load Case	
Load Case No:3 As:2 Lc=3	
Load Case No:1 As:2 Lc=1 Load Case No:2 As:2 Lc=2 Load Case No:3 As:2 Lc=3	
Load Case No:4 As:2 Lc=4 Load Case No:5 As:2 Lc=5	
	pe of Dynamic
EC-8_Greek Static V Dyn Eigenvalues 3	
Color Gradient	
Magnification 10 Direction Animation Step	
+- 10	
AVI	
↓.5/7_0.00_0_0.00 ¥	
Project Param 🖄 Project Data	AXONOM 2674.2, 0.0, 3634.5 MATH DEFOR PHYS-MATH PHYS-DFRM TRAN-MATH TRAN-DFRM
L L L	Load Case Combination Eigenvalues Pushover the general deformation cause and the next list, a general Load Case No:1 Sen:7 Lc=1 Load Case No:2 Sen:7 Lc=2 Load Case No:3 Sen:7 Lc=3 Load Case No:5 Sen:7 Lc=5 Load Case No:5 Sen:7 Lc=5
Activate Color G	^{Gradient} , modify "Magnification" and type in the value of the "Animation Step" t visualization.
	Bar" check (double click, blue=active, grey=inactive) the type of the visualization
On the "Statue P	
of the deformed	S-MATH PHYS-DFRM TRAN-MATH TRAN-DFRM
of the deformed	S-MATH PHYS-DERM TRAN-MATH TRAN-DERM





74







7. DESIGN

Since model analysis has been completed, the design checks of the structural elements are applied according to the design code provisions.

7.1 How to create design scenarios :



Move to "Design" unit and click "New" to create the desired scenario by selecting the considered regulation.

* Predefined scenarios are created according to the Regulation and Attachment option you make at the beginning, within the General Configuration window that opens automatically immediately after the file name is defined.

1 Name 1 Type EC2-EC3 ✓ EKOS 2000-EAK EC2-EC3 ✓ New EC2-EC3 NTC_2008 Design Del EC2_Italia EC2_Cyprus Concre Greek old 1959-84 Greek old 1984-93 Steel Austria SBC304 EC5 EC5 EC2	Scenario			Х
EC6-EC8(3)		Type New Design De Concre	EC2-EC3 EKOS 2000-EAK EC2-EC3 NTC_2008 EC2_Italia EC2_Cyprus Greek old 1959-84 Greek old 1984-93 Austria SBC304 EC5	~

Type a name, select the type and click New, to add the new scenario to the list.

In this example we used a scenario by the Eurocode.

In the field "Design Delete" activate the corresponding checkbox and then press "Apply", to delete the results of previous design checks (concrete elements, steel elements or connections). Repeat using other combinations or parameters or scenarios, etc.

Design Delete	
Concrete	Connections
Steel	Apply



S	ve Scenar	io Par met	a- of ers re	or the c the inforce	reation of a paramete	a new ers o ery typ	scena f the pe of s	rio as wo e desig tructura	s the comman ell as the editi n checks a l elements.
elect the cons			EC2-EC3 ' Active	1 (0) e Scenar	io and o	open t	he par	ameters	
tructural Compo		meters						×	
Steel Reinfor Combination		Capa Slabs	city Design		Steel		er struct		
				ams	Columns		Foot	_	
Combinations of	r Load Sets	; (101	l) Ult.	Serv.	+XX	+Z	Z	No	
Combinations						ULS/SI	.S Dir.	^	
1(5) +1.35Lc1						ULS			
2(1) +1.00Lc1						ULS		_	
					30Lc6+0.30Lc7		+X		
					30Lc60.30Lc7 30Lc6+0.30Lc7		+X +X	_	
					30Lc60.30Lc7		+X		
					30Lc6+0.30Lc7		+X		
8(2) +1.00Lc1	+0.30Lc2+	+1.00Lc3+0.	30Lc41.0	0Lc5+0.3	30Lc60.30Lc7	ULS	+X		
9(2) +1.00Lc1	+0.30Lc2+	+1.00Lc3+0.	30Lc41.0	0Lc50.3	30Lc6+0.30Lc7	ULS	+X		
10(2) +1.00Lc <	:1+0.30Lc2	+1.00Lc3+0).30Lc41.	00Lc50	.30Lc60.30	ULS	+X		
Level Multipliers		1	/ (1-θ)	EC8	_General Dynar	nic (4).cr	nb	\sim	
Level	Х	Y	Z	<u> </u>	Insert Co	ombinatio	ns		
0 - 0.00	1.000		1.000		Combinatio	ns Calcu	lation		
1 - 400.00	1.000		1.000		En	d Calc			
2 - 700.00	1.000	1.000	1.000		Combination G	+ψ2Q	101		
3 - 1000.00	1.000	1.000	1.000		Automa	atic Desig	IN		
4 - 1300.00	1.000		1.000	~	Recalculate K	AN.EPE.	values		
1 1000100	4 000	4 000	4 000						
Save		Load				ОК		ancel	

After you define the design parameters you have now the opportunity to save them in a file and use them in your next project.





By clicking the button «**Save**» the storage window opens.

🕇 🔜 > T	his PC > BOOTCAM	P (C:) > meletes > 1dol	ios > scaanal >		~ 0	Search scaanal	P
ganize • New fold	ler						 0
OneDrive	Name Scen000	0	Date modified	Type File folder	Size		
This PC	Scen002		10/5/2017 1:43 µµ	File folder			
Desktop	🛓 sbc.sdp		10/5/2017 3:53 µµ	VLC media file (.s	9	6 KB	
Documents							
- Downloads							
Music							
Notures							
Videos							
BOOTCAMP (C:)							
New Volume (D: INTENSO (F:)							
INTENSO (F:)							
File name: test.	sdp						 _
Saus ar huner Deriv	gn Parameter(*.sdp)						

and you type a name (it is good that the name is relevant to the design scenario).

The extension of the files is sdp scenery design parameters.

Respectively, by clicking "Reading" you can load into one of your projects a parameter file you have already saved.

	his PC > BOOTCAMP (C:) > mel	ietes > Idokos > scaanal >		νÖ	Search s	caanal		٩
rganize 👻 New fold						888 •	. 💷	0
🦲 iCloud Drive 🖈 ^	Name	Date modified	Type	Size				
Dropbox	Scen000	10/5/2017 1:18 μμ	File folder					
	Scen002	10/5/2017 1:43 µµ	File folder					
ConeDrive	🛓 sbc.sdp	10/5/2017 3:53 μμ	VLC media file (.s	96	KB			
This PC								
Desktop								
Documents								
Downloads								
Downloads Music								
👌 Music								
Music Pictures								
Music Pictures Videos								
Music Pictures Videos								
Music Pictures Videos BOOTCAMP (C:)								
Music Pictures Videos BOOTCAMP (C:) New Volume (D:								

ATTENTION

A necessary condition to load a parameter file is that the active design scenario should be the same as the parameters scenario you are loading. Othewise, you will see an error message.

Recalculate KAN.EPE. values

A new command that allows the recalculation of all values provided by KANEPE (Greek norm) for all members of the study and is used in cases where the strength of the materials is changed while the reinforcement has been placed by the existing situation

Combinations

A *Regardless of the material, the calculation of combinations is a condition for designing.



The selection of the existing .cmb combinations file is made:

	~	
default.cmb		
EC-8_Greek Dynamic (2).	mb	
- , , ,		
= ()		
	· · ·	
EC-8_Greek Προέλεγχος	Dynamic XΩPIΣ (4).	with automatic calculation or
Insert Combi	nations	
		that opens the folder with the
ect the file and press	Combinati	ons Calculation
	EC-8_Greek Dynamic (2). EC-8_Greek Dynamic (3). EC-8_Greek Static (2).cmb EC-8_Greek Ανελαστική Ν EC-8_Greek Ανελαστική Ν EC-8_Greek Προέλεγχος Insert Combi	default cmb EC-8_Greek Dynamic (2).cmb EC-8_Greek Dynamic (3).cmb EC-8_Greek Static (2).cmb EC-8_Greek Ανελαστική ΜΕ (1).cmb EC-8_Greek Ανελαστική ΧΩΡΙΣ (0).cmb EC-8_Greek Προέλεγχος Dynamic ΧΩΡΙΣ (4).c

Depending on the situation and the conditions being satisfied, you can use either the static or dynamic combinations so as the superstructure is designed (as long as you have the springs free, not fixed). You may also have performed analyzes taking into consideration different regulations (eg EAK and EC8) and by designing according to the corresponding combinations you will be able to see the differences that arise

In "Combinations" tab the combinations list is displayed.

In "Level Multipliers" tab :

Level Multipliers				1 / (1-	θ)	
Level	Х	γ		Z		^
0 - 0.00	1.000	1.0	00	1.00	0	
1 - 300.00	1.000	1.0	00	1.00	0	
2 - 600.00	1.000	1.0	00	1.00	0	
3 - 900.00	1.000	1.0	00	1.00	0	-
4 - 1200.00	1.000	1.0	00	1.00	0	¥

Combination G+w2Q

You can increase or decrease the seismic actions in any direction and level, by typing different factors You can modify the default coefficients of the seismic loads per direction and level, by typing values different than the unit.

 $1/(1-\theta)$ to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, where

0.1 < θ <0.2.

ATTENTION:

Combinations Calculation

For modification purposes, press the following button

refers to the scenarios of the Greek Regulation.

A NOTE:

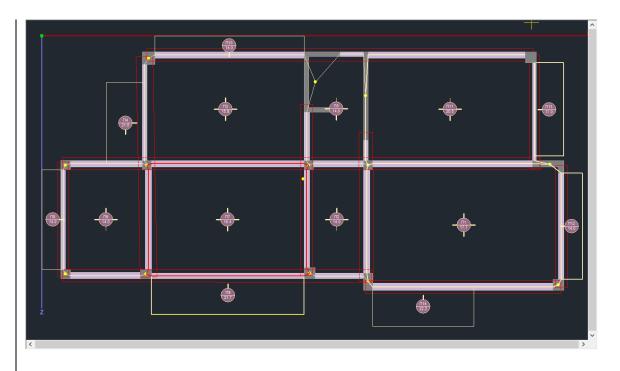
The field

The Automatic Design command offers the possibility for an automatic application of the appropriate design checks and the automatic designing of all structural elements for <u>concrete</u> structures, just by pressing the corresponding button. Set the parameters in the following tabs :



Steel Reinforcemen Combinations	t Capaci Slabs	ty Design Beams	Steel Columns		tructures Footings		
press the button "Au						and th	
elements concernin	-			, procedul			Tur
7.3 How to p	perform Bear	n Design:					
a II		The "Beams	s" command	group, coi	ntains the "	Continuity	of
Continuity Check		Beams", "(Check – Rei				
of Beams * Reinforcer Beams	ment * *	commands.					
Merge Beams							
Merge Beams	Delete						
-							
Single Continu	uity of Beams						
Overall Contin	uity of Beams						
Single Deletio	n of Continuity o	f Beams					
Overall Deletio	on of Continuity (of Beams					
Preferences of	Beam Reinforcer	nent					
Select the "Continui	ty of Beams – (Overall Contin	nuity of Beam	s″			
Design of Beams Aligi	nment X						
O Level	Total		ne the beams building auton	-	nt of the cu	rrent level	or
			m creates au	itomatical	ly all the co	ontinuities	of
ОК	Cancel	beams.					





Use the "Preferences of Beams Reinforcement" command to, insert one common bar or two different bars on the support of the continuous beams, to take into account both of them, to change the anchorage length and if you wish to modify the support widths.

Select the "Check Reinforcement > Overall" to perform the design of every beam of the structure

ignment 🗙
Total
0
Cancel

The program makes the design checks and displays the results using colors and symbols indicating in this way the type of the failure.

The <u>colored</u> indicators of the beam's failure:

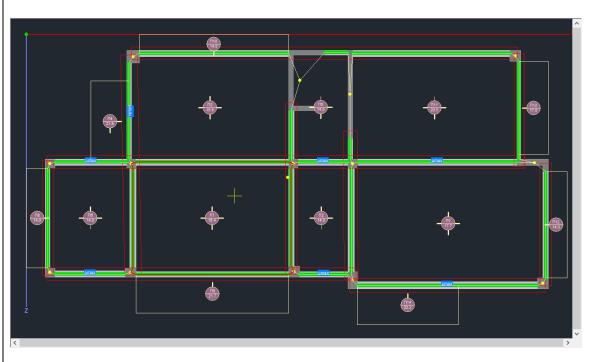
- Red. Failure in Bending. It has exceeded the maximum steel reinforcement ratio ρ_{max}.
- Pink. Failure in Shear / Torsion.
- Cyan. Passed the design checks.

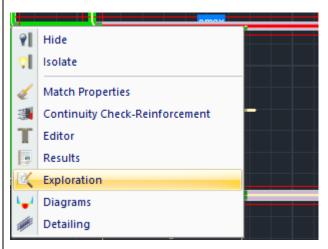
The symbol on the beam indicates:

Failure in Bending	Μ
Failure in Shear	V
Failure in Torsion	Т
Dense positioned Stirrups	Asw
It has exceeded the maximum steel reinforcement ratio	ρ
It has exceeded the maximum anchorage length	ldb
Capacity Design	αcd
It has exceeded the maximum crack width	Wk
Deflection failure	L/d



In this example the beam design process indicated some failures related to the exceedance of the maximum steel reinforcement on the supports "pmax".





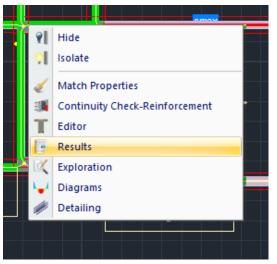
Right click on the beam member that fails the checks to open a list of commands related to the design of the beam continuity.

Click "**Exploration**" to see the results for the selected beam continuity on the file that opens:



=		B00021 - WordPad	_ 🗆 🗙
File Ed	it View Insert Form	at Help	
99		.000 0.13998	· · · · · · · · · · · · · · · · · · ·
99		.000 0.11157	
100		.000 0.13030	
100		.000 0.10369	
101		.000 0.12643	
101		.000 0.10054	
TEA03 99	-	.000 0.14921	
100		.000 0.13909	
101		.000 0.13505	
101	420.052 -0	.000 0.13505	
FAFTS	COΣ pmax		
		.600 c=0.025 d=0.575 hf=0.000	
		45 (7/fyd)=0.01610	
-	Αριστερά (EC8)=		
-	$\Delta \epsilon \xi_1 \dot{\alpha} = 0.01108$		
	Δεξιά (EC8) = 0.0		
-		$\rho_1 \xi_{\eta \zeta} = 23.405 > Asmax=21.576)$	
	Μέσον = 0.01610		
-	Μέσον (EC8) = 0.0		
BEAM	2 37 b=0.400 h=0	.600 c=0.025 d=0.575 hf=0.000	
pmax	Αριστερά = 0.016	67 (7/fyd)=0.01610	
pmax	Αριστερά (EC8)=	0.01173	
pmax	Δεξιά = 0.01677	(7/fyd)=0.01610	
pmax	Δεξιά (EC8) = 0.0	1128	
pmax	Μέσον = 0.01610	(7/fyd)=0.01610	
pmax	Μέσον (EC8) = 0.0	4000	
BEAM	3 35 b=0.400 h=0	.600 c=0.025 d=0.575 hf=0.000	
pmax	Αριστερά = 0.010	78 (7/fyd)=0.01610	
-	Αριστερά (EC8)=		
	- · ·	ριξης = 21.865 > Asmax=20.508)	
	Δεξιά = 0.01172		
	Δεξιά (EC8)= 0.0		
		ριξης = 31.604 > Asmax=26.540)	
	Μέσον = 0.01610		
-	Μέσον (EC8) = 0.0		
		.600 c=0.025 d=0.575 hf=0.000	
		52 (7/fyd)=0.01610	
	Αριστερά (EC8)=		
-	$\Delta \varepsilon \xi_1 \dot{\alpha} = 0.01102$		
-	Δεξιά (EC8) = 0.0	1136 ριξης = 35.469 > Asmax=27.262)	
	Μέσον = 0.01610		
-		(//134)-0.01010	
<			>
or Help,	press F1		NUM

NOTE: You can retrieve information related to most of the failure types through the right click menu "**Results**" command



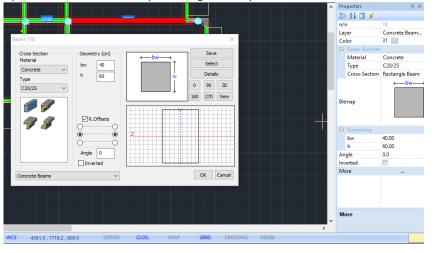
For example, for a failure described as " $\Delta\Sigma$ ": Select "**Results**" to check the failure data at the file summary results that opens:



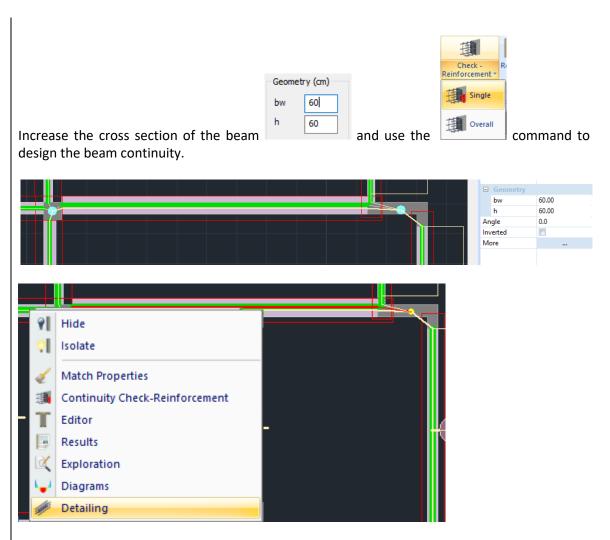
B00021 - WordPad						- 0	
ile Edit View Insert Format Help							
D FR 60 M 30000	₽6						
per Face/Critical Combin(cm2)					I	I	L
SHEAR VERIFICATION							4
Seis.Shear force (KN) Start			HEAR CAP			7- 0.27	1
			/ max				
			3.4				1
Seismic participation+	No	Yes	No		No	Yes	4
Applied Shear force VEd (KN)							
Applied Tors. Moment TEd (KNM)		8.1		8.1		8.1	
[Resist.without reinf.VRd.c(KN)]		56.1		45.1		51.0	
Resist.comp.struts VRdmax(KN)		476.11		328.3		476.1	i -
Resist.tors.moment TRdmax(KNM)	i	53.8	i	37.1	i	53.8	i
TEd/TRdmax + VEd/VRdmax <= 1.0		1.6		0.5		1.6	î.
Critical Combinations		52 (A)		39(A)		52 (A)	î.
Required Stirrups+							÷.
Reinforc. Asw/s, Doub. (cm2/m)	p4.96		p2.86		p3.08	1	1
Additional incl.reinf. (cm2)	1					1	1
+			+		+		1
FINAL REINFORC.							1
+							
Req. reinforcement As (cm2)	3.30	1.65	1.32	3.30	3.30	1.65	1
Final reinforcement As (cm2)							1
I			req.=(
CRACK CONTROL Wk (mm) <0.30	0.28		0.16	0.07		0.02	
With addit. reinfor. Wk (mm)						1	
Critical Load Combinations		(min)	99(3)	99(8)	(min)	99(S)	
REQUIRED REINFORC. As (cm2)							5
Reinf. Bars (Longitudinal)			5014	4414			1
Incrut: pars (roudicudinar)	1410		0414	4414			1
Common support bars						I I 4014	
I sommon subbore pars						1 1414	
Face reinforcement bars		1412	Left-Rid	The second		' . · · ·	1
Additional crack reinf.bars		1412	Dere-Ri			. '	
	-Ver	-Doub	+-Ver	Doub	-Ver	Doub	1
	\$12/7 I		48 /10		φ12/7 I		
SUITTUDS W/DISC.(Cm)							

After locating the elements that fail the checks, you must perform the necessary modifications to pass all the checks for every element

Select the beam with left click on 2D view of the floor plan. On the left, the "Properties" list that opens, click "More" to open the geometry definition window of the beam.







Right click on the beam and select "**Reinforcement Detailing**", to open a window with the respective details of the steel reinforcement of the beam continuity, derived from the design process (the direction of the display result meets the direction of the local axis)

Attention: beams of the same continuity must share the same local axis direction.



Beams Editor	— 🗆 X
	OK Cancel
▲4 7∑\$8/10 33∑\$8/10 7∑\$8/10 7∑\$8/10	Δ5 85Σ#8/10
· 0.60 - 3.40 - 0.60 · 0.60 - 0.60	
- 60 - 60 - 60 - 60 - 60 - 60 - 60 - 60	9.75
0.52 4.60 0.60 0.60	
∽ (1)5¢14 L=6.49 0.60 0.60	9.75
(3) 11 	¢14 L=12.15
Geometry Span Main Reinforcement Support Reinforcement Stirrups Additional Crack control Diagrams Retrofitting method	
General Data	Supports Width (cm)
Number of Spans 4 Number 1 Length 4.60 Critical Length Left (m) 0.6 Critical Length Right (m) 0.6	Left 60
Cover (mm) 25 Name 4 Lsp.(cm) 460 Reinforcement Pattern	Right 60
b(cm) 40 h0(cm) 0	
h0 h1(cm) 60 h1(cm) 0 0.80m B4 (125) 0.80m	

Here, you can modify the main and secondary steel reinforcement.

▲ Detailed description on how to perform the desired modifications can be found in the Use Manual. (Chapter .A "Beam's Detailing")

7.4 How to apply Capacity Design

After defining the limits of the acd (capacity design coefficient factor) at the "**Capacity Design**" tab of the **Parameters** section, along each direction (x, z), that will be used during the capacity design, use the command "**Node Releases**" to define the support condition for each node along each direction.

 		Node Releases	×	
T	T,	Direction y-z	Direction y Direction z	
Node Releases *	Design	Free	Direction y-z	1
Capacity	Design	Free End Middle Fixed No Capacity Design	1	

It is reminded that if you don't define the Node Releases all nodes are considered to be free along both directions (except for the fixed ones)





The capacity design can be performed either selectively or overall.

Select the **Results** command to display the TXT file that contains the results of the main design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results (along each direction).

Node = 33 Col. bottom = 19 Column up = 33 COMB. SMBby SM

COMB. SMRby SMEby acdy acdy SMRbz SMRbz acdz acdz calc 3 1016.600 184.132 7.177 3.500 541.400 113.397 6.207 3.500 4 1016.600 174.484 7.574 3.500 541.400 99.594 7.067 3.500 5 1016.600 182.850 7.149 3.500 541.400 99.693 7.060 3.500 6 1016.600 181.740 7.272 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 99.264 7.090 3.500 9 1016.600 164.927 8.013 3.500 541.400 99.363 7.083 3.500 11 1016.600 164.927 8.013 3.500 270.700 19.907 17.678 3.500 12 1016.600 164.210 8.048 3.500 143.900 20.006	Colum	-	33			-	-		
$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	COMB.	SMRby	SMEby	acdy	acdy	SMRbz	SMEbz	acdz	acdz
4 1016.600 174.484 7.574 3.500 541.400 99.594 7.067 3.500 5 1016.600 184.850 7.149 3.500 541.400 113.496 6.201 3.500 6 1016.600 175.202 7.543 3.500 541.400 99.693 7.060 3.500 7 1016.600 181.740 7.272 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 99.264 7.090 3.500 10 1016.600 182.458 7.243 3.500 541.400 99.363 7.083 3.500 10 1016.600 164.927 8.013 3.500 541.400 13.888 50.678 3.500 12 1016.600 164.210 8.048 3.500 270.700 19.907 17.678 3.500 13 1016.600 164.210 8.048 3.500 143.900 20.006 9.351 3.500 14 1016.600 162.535 8.131 3.500				calc				calc	
4 1016.600 174.484 7.574 3.500 541.400 99.594 7.067 3.500 5 1016.600 184.850 7.149 3.500 541.400 113.496 6.201 3.500 6 1016.600 175.202 7.543 3.500 541.400 99.693 7.060 3.500 7 1016.600 181.740 7.272 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 99.264 7.090 3.500 10 1016.600 182.458 7.243 3.500 541.400 99.363 7.083 3.500 10 1016.600 164.927 8.013 3.500 541.400 13.888 50.678 3.500 12 1016.600 164.210 8.048 3.500 270.700 19.907 17.678 3.500 13 1016.600 164.210 8.048 3.500 143.900 20.006 9.351 3.500 14 1016.600 162.535 8.131 3.500									
5 1016.600 184.850 7.149 3.500 541.400 113.496 6.201 3.500 6 1016.600 175.202 7.543 3.500 541.400 99.693 7.060 3.500 7 1016.600 181.740 7.272 3.500 541.400 113.067 6.225 3.500 8 1016.600 172.092 7.679 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 99.264 7.090 3.500 10 1016.600 172.810 7.648 3.500 541.400 99.363 7.083 3.500 11 1016.600 164.927 8.013 3.500 241.400 13.888 50.678 3.500 12 1016.600 155.280 8.511 3.500 241.400 7.586 71.052 3.500 13 1016.600 164.210 8.044 3.500 143.900 20.066 9.351 3.500 14 1016.600 152.877 8.644 3.500	_								
61016.600175.2027.5433.500541.40099.6937.0603.50071016.600181.7407.2723.500541.400113.0676.2253.50081016.600172.0927.6793.500541.40099.2647.0903.50091016.600182.4587.2433.500541.40099.2647.0903.500101016.600172.8107.6483.500541.40099.3637.0833.500111016.600164.9278.0133.500541.40013.88850.6783.500121016.600155.2808.5113.500270.70019.90717.6783.500131016.600164.2108.0483.500414.6007.11975.7063.500141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.1708.6853.500143.90020.2379.2443.500171016.600152.1708.6853.500143.90020.3369.1993.50020304.000154.5622.5572.557541.40019.90727.0753.50021304.000154.2622.5572.557541.40020.23734.7793.50022304.000164.2102.4072.707									
7 1016.600 181.740 7.272 3.500 541.400 113.067 6.225 3.500 8 1016.600 172.092 7.679 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 99.264 7.090 3.500 10 1016.600 172.810 7.648 3.500 541.400 99.363 7.083 3.500 11 1016.600 164.927 8.013 3.500 541.400 13.888 50.678 3.500 12 1016.600 164.210 8.048 3.500 414.600 7.586 71.052 3.500 13 1016.600 164.210 8.048 3.500 143.900 20.006 9.351 3.500 14 1016.600 162.535 8.131 3.500 414.600 7.119 75.706 3.500 15 1016.600 162.187 8.644 3.500 143.900 20.237 9.244 3.500 17 1016.600 152.170 8.685 3.500									
8 1016.600 172.092 7.679 3.500 541.400 99.264 7.090 3.500 9 1016.600 182.458 7.243 3.500 541.400 113.166 6.219 3.500 10 1016.600 172.810 7.648 3.500 541.400 99.363 7.083 3.500 11 1016.600 164.927 8.013 3.500 270.700 19.907 17.678 3.500 12 1016.600 164.210 8.048 3.500 414.600 7.586 71.052 3.500 13 1016.600 164.210 8.048 3.500 143.900 20.006 9.351 3.500 14 1016.600 162.535 8.131 3.500 143.900 20.237 9.244 3.500 15 1016.600 161.818 8.167 3.500 143.900 20.336 9.199 3.500 17 1016.600 152.170 8.685 3.500 143.900 20.336 9.199 <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td>									
9 1016.600 182.458 7.243 3.500 541.400 113.166 6.219 3.500 10 1016.600 172.810 7.648 3.500 541.400 99.363 7.083 3.500 11 1016.600 164.927 8.013 3.500 541.400 13.888 50.678 3.500 12 1016.600 155.280 8.511 3.500 270.700 19.907 17.678 3.500 13 1016.600 164.210 8.048 3.500 414.600 7.586 71.052 3.500 14 1016.600 154.562 8.550 3.500 143.900 20.006 9.351 3.500 15 1016.600 162.535 8.131 3.500 144.600 7.119 75.706 3.500 16 1016.600 161.818 8.167 3.500 143.900 20.237 9.244 3.500 17 1016.600 152.170 8.685 3.500 143.900 20.336 9.199 3.500 18 1016.600 152.280 2.545 2.545									
101016.600172.8107.6483.500541.40099.3637.0833.500111016.600164.9278.0133.500541.40013.88850.6783.500121016.600155.2808.5113.500270.70019.90717.6783.500131016.600164.2108.0483.500414.6007.58671.0523.500141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50020304.000155.2802.5452.545414.60019.90727.0753.50021304.000164.9272.3962.396143.90013.88813.4703.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597 <t< td=""><td>8</td><td>1016.600</td><td>172.092</td><td>7.679</td><td>3.500</td><td>541.400</td><td>99.264</td><td>7.090</td><td>3.500</td></t<>	8	1016.600	172.092	7.679	3.500	541.400	99.264	7.090	3.500
111016.600164.9278.0133.500541.40013.88850.6783.500121016.600155.2808.5113.500270.70019.90717.6783.500131016.600164.2108.0483.500414.6007.58671.0523.500141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000164.2102.4072.407270.7007.58646.3913.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000162.5352.4312.431270.7007.11949.4303.50024304.000162.5352.4312.4422.70.7007.11949.4303.50025304.000152.1702.5972.597 <td< td=""><td>9</td><td>1016.600</td><td>182.458</td><td>7.243</td><td>3.500</td><td>541.400</td><td>113.166</td><td>6.219</td><td>3.500</td></td<>	9	1016.600	182.458	7.243	3.500	541.400	113.166	6.219	3.500
121016.600155.2808.5113.500270.70019.90717.6783.500131016.600164.2108.0483.500414.6007.58671.0523.500141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000164.2102.4072.407270.7007.58646.3913.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000162.5352.4312.431270.7007.11949.4303.50024304.000162.5352.4312.431270.7007.11949.4303.50024304.000162.5352.4312.4422.70.7007.11949.4303.50025304.000152.1702.5972.5975	10	1016.600	172.810	7.648	3.500	541.400	99.363	7.083	3.500
131016.600164.2108.0483.500414.6007.58671.0523.500141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000164.2102.4072.407270.7007.58646.3913.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000162.5352.4312.431270.7007.11949.4303.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	11	1016.600	164.927	8.013	3.500	541.400	13.888	50.678	3.500
141016.600154.5628.5503.500143.90020.0069.3513.500151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	12	1016.600	155.280	8.511	3.500	270.700	19.907	17.678	3.500
151016.600162.5358.1313.500414.6007.11975.7063.500161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	13	1016.600	164.210	8.048	3.500	414.600	7.586	71.052	3.500
161016.600152.8878.6443.500143.90020.2379.2443.500171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	14	1016.600	154.562	8.550	3.500	143.900	20.006	9.351	3.500
171016.600161.8188.1673.500414.60013.42240.1583.500181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	15	1016.600	162.535	8.131	3.500	414.600	7.119	75.706	3.500
181016.600152.1708.6853.500143.90020.3369.1993.50019304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	16	1016.600	152.887	8.644	3.500	143.900	20.237	9.244	3.500
19304.000155.2802.5452.545414.60019.90727.0753.50020304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	17	1016.600	161.818	8.167	3.500	414.600	13.422	40.158	3.500
20304.000164.9272.3962.396143.90013.88813.4703.50021304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	18	1016.600	152.170	8.685	3.500	143.900	20.336	9.199	3.500
21304.000154.5622.5572.557541.40020.00635.1813.50022304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	19	304.000	155.280	2.545	2.545	414.600	19.907	27.075	3.500
22304.000164.2102.4072.407270.7007.58646.3913.50023304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	20	304.000	164.927	2.396	2.396	143.900	13.888	13.470	3.500
23304.000152.8872.5852.585541.40020.23734.7793.50024304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	21	304.000	154.562	2.557	2.557	541.400	20.006	35.181	3.500
24304.000162.5352.4312.431270.7007.11949.4303.50025304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	22	304.000	164.210	2.407	2.407	270.700	7.586	46.391	3.500
25304.000152.1702.5972.597541.40020.33634.6103.50026304.000161.8182.4422.442270.70013.42226.2203.500	23	304.000	152.887	2.585	2.585	541.400	20.237	34.779	3.500
26 304.000 161.818 2.442 2.442 270.700 13.422 26.220 3.500	24	304.000	162.535	2.431	2.431	270.700	7.119	49.430	3.500
26 304.000 161.818 2.442 2.442 270.700 13.422 26.220 3.500	25	304.000	152.170	2.597	2.597	541.400	20.336	34.610	3.500
	26	304.000		2.442	2.442	270.700	13.422	26.220	3.500
	27	304.000	174.484	2.265	2.265	143.900	99.594	1.878	1.878

7.5 How to design columns and walls



The "Columns" field contains the commands related to the Design, Reinforcement Check, columns and walls Results. (You can see the User Manual Chapter 9 "Design")

Select the command **"Check Reinforcement > Overall"** to perform the design of all the columns and walls of the building (the design will be performed automatically by level for the whole building).



×
◯ Total
✓ Walls
Cancel

Activate the command and the following dialog box opens:

Select whether to design all columns/walls of the current level or the building in total

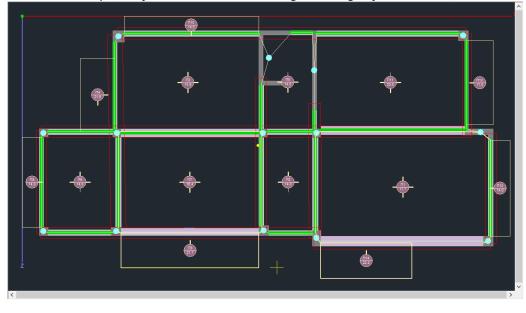
After the design A colored dot is displayed in the center of the element. The <u>color</u> changes according to the type of failure as follows

- Red: Failure caused by biaxial bending. The steel reinforcement exceeded the maximum ratio of 4%. Dense stirrups. No results are displayed.
- Pink: Failure by Shear / Torsion or exceedance of the ductility level. The results show the reason of failure.
- Cyan: All design checks are verified.

The <u>initially</u> indicated type of failure appears above the element as well:

Failure by biaxial bending	M-N
Failure by Shear	V
Confinement failure	ωwd
Buckling failure	λ
Failure by Torsion	Т
Dense Stirrups	Asw
Exceedance of 4% steel reinforcement ratio	ρ
Exceedance of the ductility index	v

In this example, no failure was located during the design of the columns/walls.



Right click on the section to open a list of commands related to the elaboration and the display of the results derived from the design checks of columns/wall design.



Ide Isolate ✓ Match Properties Check-Reinforcement Editor ✓ Exploration ↓↓ Detailing ↓↓	Select the "Results" command to open the .txt file and the results:
C00030 - WordPad	- D X
-	
ile Edit View Insert Format Help	-1
	····· ^
fck (Mpa)=20.00 ycu/ycs =1.50/1 fctm (Mpa)= 2.20 trd (Mpa)=0.25 REINF MAIN : B500C Es (Gpa)=200. STIRRUPS : B500C Es (Gpa)=200. BIAXIAL BENDING WITH P O S I T I O N	RETE : C20/25
CONCRETE DEFOR Apex Comb. Deform. Apex Comb. D Column Bottom 1 62 -0.2785 2 62 -0 3 38 -0.0868 4 38 -0 S H E A R F O R C E Seis.shear force Y (KN) Start	<pre>y= 216.18 z= 175.04 y= -53.51 z= -58.67 MATIONS ENVELOPE (0/00)</pre>
End Seismic direction+ Applied Shear force VEd (KN) Applied Tors. Moment TEd (KNM)	
<pre> Resist.comp.struts VRdmax(KN) Resist.tors.moment TRdmax(KNM) TEd/TRdmax + VEd/VRdmax <= 1.0 Shear critical combinations Req. stirrups Asw/s (CM2/M) Moment resis. Mrd-(KNM)+</pre>	Top
	+y -y +z -z +y -y +z -z 168 -167 457 0 0 -382 0 -529 579 -363 826 0 0 -525 0 -644

Select "**Detailing**" to open a window for editing the reinforcement of the column - wall in an integrated environment of verification and design:



Column Editor	ŝ.		_	
Geometry				Info
Main Reinfor Stirrups M Diagrams	Concrete Cover 25 m Insert Dimensions X Y XYZ	H 3 h2 H		
Results	Detailing Top Continuity Width (cm) 60 Cover (mm) 25	Detailing Design Bottom Continuity Width (cm) 300 Cover (mm) 25		
Recalculation	Design Scales Cross Section 1: 20	Detailing 1: 50		
Recalculation	Name	K9 - 23		
Joint Check	Туре	WALL		
	A Strategy and a stra	201 /40 /35 /80		
Y = 600.00	Boundary zone	by:60 70		
+ M-N -	H - Har (am)	300 /201		
		9440.00 / 9440.00		
Сору		4.0 - 377.60		
Paste	pcalc % - cm^2	0.52 - 49.39		
OK	Rebars			
Cancel	8010+28014			

With this command, you can modify the reinforcement of the column – wall, apply retrofitting methods and calculate the new moment diagrams.

 Detailed instructions on how to use this command refer to the related user manual (chapter B. Column's Detailing.

7.6 How to perform Slab Design:



The "Slabs-Mesh" command group includes commands related to the analysis of slabs with the strip method and the corresponding results, and commands to insert, delete, edit and generate a mesh.

Select the command "Slab – Strip Calculation > Overall" to calculate all the strips of the current level.

The slab strips are analyzed, the stress resultants are calculated and the designing of the slab is performed. The program calculates the <u>tension</u> (Fe) and <u>compression</u> (Fe') and the steel reinforcement in cm². Also the reinforcing bars in span, additional and <u>secondary</u> reinforcement and stirrups, for solid and Zoellner slabs, are calculated.



7.7 How to perform Footing Design



The "Footing" command group contains commands for footing design check, design calculation, editing and the respective results.

Select the command "Check Reinforcement > Overall" to check all footings on the current level (foundation).

The program performs the design checks and the corresponding results are displayed by colors and symbols that indicate the type of failure.

The color of the node indicates that the design checks of the footing:



The footing was satisfied.



The footing failed

The type of failure is mentioned with a symbol as well:

(i) "Z" symbol corresponds to the exceedance of the critical load

(ii) "e" symbol corresponds to the exceedance of the load eccentricity

(iii) " σ " symbol corresponds to the exceedance of the normal stresses.

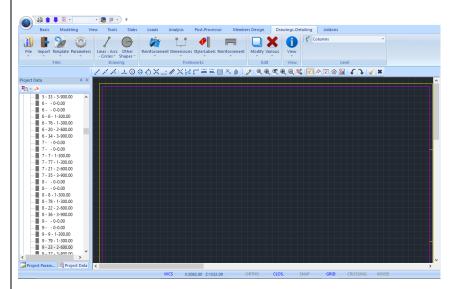
A Necessary precondition for the footing designing, is the columns designing in level 1.



8. DRAWINGS

Since the design and reinforcement of the structural elements of the concrete structures or the design of steel connections of the steel structures have been completed, you can open, modify and finally produce all the drawings in the "Drawing-Detailings" Ribbon.

The "Drawing-Detailings" Ribbon incorporates a drawing application in the interface.



8.1 How to import drawings and beam's detailing in drawing environment:



Selecting "Import" command opens the following dialog box for choosing the project's folder. Then select:

- the type of project from Files of Type
- the number of the floor
- the Scale factor

and press Find.



Open			×
			~
Look in:	Documents	✓ Ø Ø №	
_	Name	Date modified	Type ^
	📙 ace erp	9/12/2015 1:06 µµ	File folder
Quick access	ACE ERP_Updater	11/12/2015 10:48	File folder
	ace-hellas	22/2/2016 10:37 πμ	File folder
Desktop	Archline 2015 ARCHlineXP Draw	7/1/2016 3:40 μμ 25/8/2016 5:02 μμ	File folder File folder
	ARXEIA ARCHLINE	5/9/2016 4:58 μμ	File folder
	Autodesk Application Manager	9/12/2015 3:51 μμ	File folder
Libraries	📜 Avaya	9/12/2015 1:07 µµ	File folder
	Camtasia Studio	24/8/2016 1:41 μμ	File folder
	Custom Office Templates	11/12/2015 2:02 μμ	File folder
This PC	Direct Connect	11/4/2016 1:37 μμ	File folder
S	Downloaded Installations Inventor Server SDK ACAD 2016	7/1/2016 3:44 μμ	File folder
Network	<		>
	File name:	~	Open
	Files of type: Scada Pro(*.inf)	~	Cancel
		Drawing (*.SPD)	
Scale Fac	tor 1.0 Level 1 Find Find	Block (*.SPG) Drawing 32BIT (*.S	
		Block 32BIT (*.SPE Ascii file Win(*.*)	3,*.SCB)
		Ascii file Win(*.*) Autocad Files (*.D) Beams Detailing(*.	(F, *.DWG)
		Scada Pro(*.inf)	
		Scada connection Beams New Detail	
	OK Concel		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		
In the dialog	box:		



Insert Proje	ct Drawing		
Columns		Slal	b Reinforcement
Column D	imensions	Sup	port Inclined
Column R	einforcement		No 🔿 Ye
🔽 Beams		- Det	aile
🔽 Beam Dim	nensions	scal	
🔽 Beam Rei	inforcement		- 2.5
🔽 Slabs Sha	вре		
Slab Rein	forcement 🗌	With Ba	ars offset
Footings			
Footing R	einforcemen 🔲	Numbe	r of Bars
Column D	etails		ОК
Various(Li	nes-Circles)		
Column R	eference Point		Cancel
Column H	atch	Angle	45
		Dist	5
Strups	Μανδύας		•
Display D	etails Reinforcem	ent	
	ole od connection		
Create tab		rocams	

Select the elements that will be imported in the design by activating the corresponding checkboxes.

✤ In "Slab Reinforcement" select "Yes" if you want the inclined reinforcing bars to be designed as well as the additional reinforcement in slab's supports. Otherwise check "No".

In "Scale" type the zoom factor for the columns' detailing.
 Example: In case that you design a drawing in scale 1:50 and the columns' detailing is in scale 1:20, you will have to type the factor 50/20=2.5

From File Beams' Detailing drawing (*.per) :

Import the beams' detailing in the drawing for the beams 'alignment you will choose from the available ones.

This choice is suitable for the beams' detailing which are created with the existing editor of the beams, while the choice "Beams New Detailing (*.per)". refers to the beams' detailings which are created with the new editor "Design Details"

By choosing the Beams' Detailing (old and new) the path in Find guides you to a new window to select the alignments one by one

Select Beam Alignment (Detailing)	×
B3+B4+B5+B6 B23+B22 B11+B10+B9+B8 B14+B15 B20+B19 B18+B17 B12+B13 B7	
Level 1 Diagrams	OK Cancel

Select the "Level" by typing the level's number. Active:

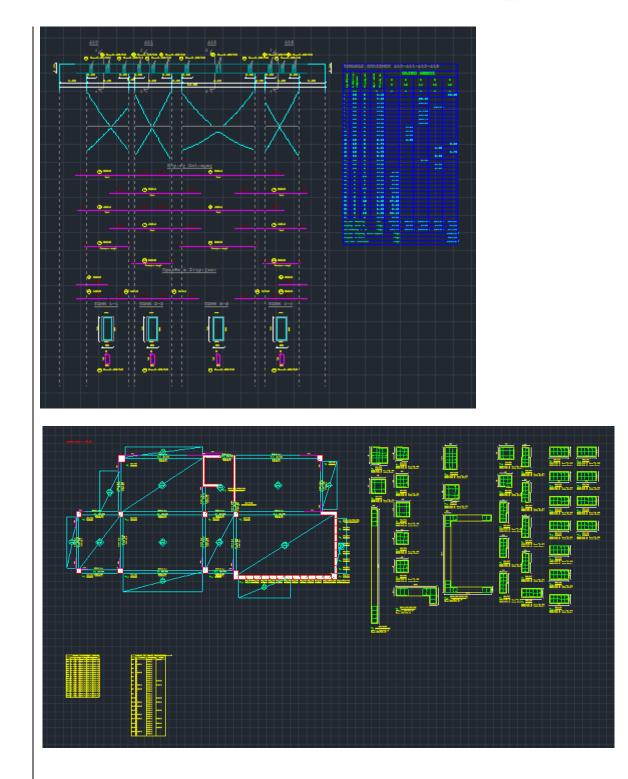
• "Diagrams": Beams' Detailing the corresponding moment diagrams will accompany drawing.

• "Curved Anchorage": The anchorages will be curved at the end.

Select a beam alignment from the list and press

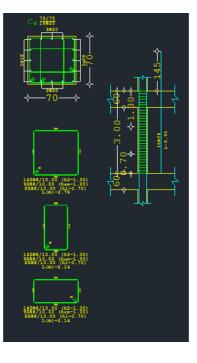
"OK". Then left click on the screen to define the position in the design. You xan repeat this procedure for the rest of the levels and details.







8.2 How to import analytical columns details with the ability to perform modifications directly inside the editor:



Precondition for the import of analytical columns and walls detailing inside the drawing environment is:

1. To have already selected the "Detailing" for the respective columns and walls, and

2. To click "OK" to the respective windows.

Then, and only the, importing the "project. Inf" file, will also import the analytical details of columns and walls.

Using the "Modify > Edit" command you can modify detailing drawings directly inside the editor.

	● ♣ ★ ₩ ₩ + · · · ● ₩ • · · Basic Modeling View Tools Slabs L	Loads Analysis Post-Processor Member Dauge Dri	awings-Detailing Addons	Style - 🔷 💥 -
		Accement Dimensions Style/Labels Reinforcement Modify Various Formworks	View Level	
Сору		OX_/XXII THE CARCO	€ < % < ^ 2 @ ≦ € 	
	ject Data 🛛 🗴 🖹 Column Editor		- • ×	Properties
Mirror	E · 🕐 🖉 Geometry		🔀 岐 岐 🕵 🐱 👘 👘	
9	- 1 - 15 - 2-600.00 ^ . Main Reinfor			n/n 2
Offset	- 1 - 29 - 3-900.00	Height (m)		Layer Concrete Col
	2 0-0.00 CILLI Sorrups	24		Color 10
Rectangular Array	2 0-0.00 M Dispress	ht 13		B Cross-Section
	2 - 2 - 1 - 300.00	Y 1/12 N2 L3 NJ		Material Concrete
Chanadan Annua				Type C20/25
Circular Array	2 - 16 - 2-600.00 Results Detailing	Detailing Design		Cross-Section Rectangle set
Move Alignment	3 0-0.00 3 0-0.00 3 - 3 - 1-300.00 3 - 3 - 1-300.00 3 - 3 - 1-300.00 3 - 7 - 2-600.00 3 - 17 - 2-600.00 3 - 17 - 2-600.00 3 - 17 - 2-600.00	0 00 Weth (on) 60 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		est to by
Rotate	4 0-0.00 Recalculation	ection 1: 20 Detailing 1: 50		by 70.00
Scale Scale	4 0-0.00 Joint Check Type Desensions	COLUMN 20 /70	100/10.0	bz 70.00
	4 - 4 - 1-300.00 4 - 74 - 1-300.00 Y = 300.00			Angle 0.0
Text Style	4 - 18 - 2 - 60200 4 - 32 - 3 - 500.00 4 - 32 - 3 - 500.00			Planted More
Color	- 5 - 0-0.00 Patter Pat			
Layer	\$ - 75 - 1-300.00 Gancel 26020		ate ate	
Edit .	Project Param_ I Project Data			· · · · · · · · · · · · · · · · · · ·

Select "**Edit**" and click on an object. The corresponding dialog box, in which you can change the corresponding parameters, opens. Click OK to save the changes that automatically update the drawing and the report as well.



9. PRINT

9.1 How to create the Project Report:



To create the project report, select the "Addons" unit and click Print.

In the dialog box "Calculation Printout", on the left, the list with the Available Chapters is displayed. Double click on the selected chapter to show it on the right list.

Τεύχος Fill in the Printout list by double clicking on the available chapters and then press "Project Report".

Calculation's Printout			×
Availiable Chapters	Printout	Number of Pages :	
🖃 General	Cover		Building Data
Cover Short Description	Short Description Regulations	n	Move Up
Regulations	Solemn Declara	tion	Move Down
Solemn Declaration Program Assumptions	Program Assum Seismic Analysis	ptions Parameters (EC)	Delete
	Material Descrip		Delete All
Seismic Analysis Parameters Material Descriptions	Load Combinati		Insert File
Combinations	Checks		Error Correction
EC-8_Greek Static			Format Page
EC-8_Greek Dynamic EC-8_Greek Dynamic2 Load Combinations Checks			Paging 0
			Export Printout
Reinforcement			Print
			Project Report
Masonry Assessment			Save
⊕ Bill of Materials			Cancel

Click "Project Report" to open the print preview of your report.



You can save the file in PDF, or DOC, XLS, XML format and edit it. Export to PDF Current page	Prepared report Adobe Acrobat W Adobe Acrobat W Rich Text Re HTM, Rie MHT Rife (neb archive) M Style (Secol) bable M Excel 2007 Be W Microsoft Word 2007 Re OpenOffice Writer Norselfice Writer Norselfice Writer Dep Indice Dep Indice Text File Makis Printer	URBAN & RURAL PLANNING OFFICE		
Protected	W Rich Text fle HTM, fle HTM, fle HTM, fle HTM, fle W Back 2007 fle W Marzos/Wac2007 fle V Marzos/Wac2007 fle OpenOffice Calc OpenOffice Calc OpenOffice Witter Marzos/Wac56 fle Cyl fle Der balle Text Field Main Printer Text Field Main Printer	PLANNING OFFICE	TECHNICAL REPORT	
Image: Start Market Name PLANNING Image: Start Market Name Image: Start Market Name Image: Start Name Image: Start Name <th>H41T file (web-archive) H 304, Except table Except 2007 File W Moraroff-Word 2007 File OpenOffice Valler OpenOffice Valler CopenOffice Valler CopenOffice Valler CopenOffice Valler Defi table Text File/Makin Printer</th> <th>PLANNING OFFICE</th> <th>TECHNICAL REPORT</th> <th></th>	H41T file (web-archive) H 304, Except table Except 2007 File W Moraroff-Word 2007 File OpenOffice Valler OpenOffice Valler CopenOffice Valler CopenOffice Valler CopenOffice Valler Defi table Text File/Makin Printer	PLANNING OFFICE	TECHNICAL REPORT	
Image: Second	W Microsoft Word 2007 Fle PowerPoint 2002 Fle OpenOffice Calc OpenOffice Writer Microsoft XP5 File CSV Fle DB* table Text File/Matrix Printer	MUNICIPALITY	TECHNICAL REPORT	
Specific vite Breach State Breach State <th>OpenOffice Calc OpenOffice Writer Microsoft XPS file CSV file DBF table Text File/Matrix Printer</th> <th></th> <th>TECHNICAL REPORT</th> <th></th>	OpenOffice Calc OpenOffice Writer Microsoft XPS file CSV file DBF table Text File/Matrix Printer		TECHNICAL REPORT	
Notanit 95 Billing TECHNICAL REPORT PROJECT TITLE PROJECT TITLE Nota of 31 Project to Rich Text You can save the file in PDF, or DOC, XLS, XIML format and edit it. Export to Rich Text Sport of 70 Implementation and/or pageranges, separated by commas, For example, 1,2,5-12 Security Page: main/or pageranges, separated by commas, For example, 1,2,5-12 Vener Options Options Pagerange Vener Pagerange Pagerange Pagerange Reported Form Fackground Information Implementation Options Pagerange Pagerange Pagerange Reparated by commas, For example, 1,2,5-12 Options Implementation Implementation Security Background Implementation Pagerange Security Background Implementation Pagerange Implementation Pagerange Reparated by commas, For example, 1,2,5-12 Options Pagerange Pagerange Reparated for the second for the seco	Mcrosoft XPS file CSV file DBF table Text File/Matrix Printer		TECHNICAL REPORT	
Spect of 20	DBF table Text File/Matrix Printer		TECHNICAL REPORT	
TecHNAGer Protection Image Ret. PROJECT TITLE PROJECT TITLE Provide Provide Page range Current page Pages: Page range Pages: Pages: Pages: Page range Pages: Pages: Page range Pages: Pages: Page range Pages: Page range Pages: Pages: Page range Page range Pages: Page range Page range Pages: Page range Page range Page range Pages: Page range	Text File/Matrix Printer		TECHNICAL REPORT	
PROJECT TITLE Prove to fish You can save the file in PDF, or DOC, XLS, XML format and edit it. Export to PDF Information Page range Pages: Enter page numbers and/or pageranges, separated by commas. For example, 1,3,5-12 Options Vewer Options Page breaks Pitures	Image fie			
Prote 16/32 Prove				
Page 1d 31 Image: Constraint of the second of the seco				
Prote 16/32 Prove				
Page 1d 31 Image: Constraint of the second of the seco				
Page 1d 31 Image: Constraint of the second of the seco				
Page 1d 31 Image: Constraint of the second of the seco				
You can save the file in PDF, or DOC, XLS, XML format and edit it. Export to PDF Image Current page Image Current page Pages: Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12		PROJECT TITLE		
You can save the file in PDF, or DOC, XLS, XML format and edit it. Export to PDF Image Current page Image Current page Pages: Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12				
You can save the file in PDF, or DOC, XLS, XML format and edit it. Export to PDF Image Current page Image Current page Pages: Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Options Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12 Image by commas. For example, 1,3,5-12				
Export to PDF X Export Page range • All Current page • Current page Pages: • Current page Pages: • Export • Pages: • Pages: • Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 • Viewer • Options • Compressed • Background • Embedded Forts • Print Optimized		file in DDE or DO	C VIS VMI format and adit it	
Export Page range Export Page range Information C current page Security Pages: Enter page numbers and/or pageranges, separated by commas, For example, 1,3,5-12 Vewer Options Imbedded Forts Print Optimized	fou call save the	The in PDF, of DO	2, ALS, AIVIL IOITIIAL AITU EUIL IL.	
Export Page range Information C current page Security Pages: Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Wewer Options Options Imbedded Forts P mint Optimized	Export to PDF	X		• All
Export © All © regist Information © Current page Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Security Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Viewer Options Øptions Øptions Øc compressed Ø Background Embedded Forts Ø Print Optimized	-			
Internation Control page Security Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Viewer Options Options Impressed Doptions Impressed Embedded Forts Print Optimized	Export			
Security Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Viewer Options Options Image: Page breaks Image: Compressed image: Print Optimized Print Optimized		e		Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12
Viewer separated by commas. For example, 1,3,5-12 Options Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image: Separated by commas. For example, 1,3,5-12 Image:	Security	bers and/or pageranges		
Options Image Decades Image Decades Image D	separated by o	commas. For example, 1,3,5-12		
M Compressed M Background Embedded Fonts M Print Optimized	Options			
		-		
✓ Open after export		Fonts M Print Optimized		
				Open after export
✓ Openafter export OK Cancel OK Cancel	✓ Open after export	OK Cancel		OK Cancel
Conceptus and OK Cancel	✓ Open after export	OK Cancel		



