

Example 9 Steel Structure using 3D dwg Analysis and Design







CONTENTS

•		OVER\	/IEW	3
•	ļ	INTRO	DUCTION	3
•		THE N		3
	1	CEN		сс
	т.	1 1	Geometry	5 5
		1.1	Materials	5
		13	Regulations	5
		1.5	Sections	5
		1.5	Load – Analysis assumptions	6
		1.6	Notes	6
	2.	DAT	A INPUT - MODELING	7
		2.1	How to start a new project	7
		2.2	New Project	9
		2.3	Project Modelling from a 3D dwg file	10
		2.4	Preparing a 3D dwg file	11
		2.5	Import of the drawing file and sections recognition	12
	,	2.6	Footings configuration	21
	3.	LOA	DINPUT	24
		3.1	How to insert wind and snow loads automatically in accordance with EC 1:	24
	4.	ANA	ALYSIS	32
		4.1	How to create an analysis scenario:	32
		4.2	How to run an analysis scenario:	37
		4.3	How to create load combination:	41
	5.	POS	T-PROCESSOR	44
		5.1	How to view diagrams and the deformed shapes results:	44
	6.	STE	EL MEMBER DESIGN	47
		6.1	How to create design scenarios:	47
		6.3	Steel members design:	53
	1	Merge	Elements	53
		6.3.1	Cross Section Design:	58
	-	6.3.2 Con	Buckling Members Input:	60
	7.	7 1	NECTIONS	05
	0		The to perjoint steel members connection design	ده
	о.	Q 1	How to perform footing design:	08
	٥	D.1 RIII		60
	9. 10)RAWINGS	09
	10.	. D 101	How to import the detailing drawings:	70
	11	-0.1 P	RINTING	,1
	± ± .		How to create the report:	



• 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.





Project Data	ф,	1
제 - 🌏		
C Arcs		
🗄 🖉 Columns		
🖶 📥 Footings		
🗄 📥 Nodes		
🗄 📲 🖁 MathColumns		
- And Surf 2D		
- Surf 3D		
🗄 🛶 Slabs		
		-

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 an entity is being chosen by the tree structure, 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

Current steel structure is a truss created in 3D cad. The upper-structure consists of steel only, while single concrete footings and connecting beams in both directions form the foundation. The final result should look as the following image:



1.2 Materials

For the upper-structure is used steel of quality S275 (Fe430). The modulus of elasticity is E=21000kN/cm2 and the Poisson ration is v=0,30. The specific weight is considered 78,5 kN/m3.

1.3 Regulations

Eurocode 0 (EC0, ENV 1990), for the definition of the load combinations. Eurocode 3 (EC3, ENV 1993), for the design of the steel members. Eurocode 8 (EC8, EN1998), for seismic loads. Euρωκώδικας 1 (EC1, EN1991), for wind and snow loads. Eurocode 2 (EC2, EN1992), for the footing design.

1.4 Sections

Columns: Main Beams: Truss Upper: HEB500 SHS150X8-SHS100X8 IPE300



Truss Lower:IPE300Truss members:CHS193,7X10Griders:IPE200Vetr. Wind bracing:SHS100X5

1.5 Load – Analysis assumptions

Dynamic Spectrum Analisys with pairs of torsional moment of the same direction. The loads in accordance with 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 $\pm 2e\tau zi$).

(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).

For this example we will also include the three following loads:

(8) S (snow)

(9) W0 (wind along x direction)

(10) W90 (wind along z direction)

In seismic analysis involved only dead and live loads. Snow and wind loads are considered in separate "simple" static analysis scenario (see Analysis).

The values of snow and wind loads in this example will be taken arbitrarily without accurate calculation according to the Eurocode 1, for simplicity.

 ψ 0, ψ 1, ψ 2 action factors, will be according to ECO.

1.6 Notes

All the commands that will be used in this example (in fact the whole group of the software commands), are analytically described and explained in the User's Manual of the software.



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 will be explained in detail the way of using a **3D dwg file** for the modeling of a steel structure.

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



By left clicking on the related icons, one of the following ways, to initialize a project, 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 Project			
Project Name MyProject Details		^ ~	
Location Folders: C:\ Drives: C:		Network	
C:\ ACE ERP Arxeia Athens_7_9_1995 Autodesk BULGJORD DUB	^	ОК	
	~	Cancel	

NOTE:

The name of the file can contain up to 8 characters of the Latin alphabet and numbers, without any symbols (/, -, _) nor spaces.

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 worksheet. 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 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.



ARCHLinexp

DWG

: Reading .xml files Read an .xml file from ARCHLine.XP architectural software.

EXEMPTE: Import a cad file and use it as an auxiliary file into the interface or base for <u>Automatic Level Creation</u> and <u>Automatic Section Identification</u>.

A detailed description of the automatic procedure based on the .acad files is given in the concrete structure example.



"**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. A detailed explanation of this command can be found at the respective chapter of the manual (Chapter 2. Modeling).



1. The most common steel structures

contain continuous frames in one or both directions with duo pitch roof. Stringers, purlins, windbreakers, and front columns may be included. In case of using a template structure, you can perform the entire modeling with one single command! However, in case of more models that are complicated as well, the template command can set the bases to complete the entire modeling faster, just by modifying some of the automatically generated characteristics.

2.2 New Project

New Project		
Project Name 3DSTEEL I Details Project using 3D DWG		
Location Folders: C:\meletsteel Drives: C:	Vetwork	
C:\ MELETSTEEL SIDE_Files Matalest SAPSTEEL SAPSTEEL steelDGW	ок	

Select the related icon and in the dialog window

Set the "**Project**" name. If you wish, write in the "**Info**" field, some information related to the project and define the path that your project will be stored to, inside the local disk.

Automatically opens the General Parameters window, to set the parameters of the project, such as Material and Regulation, and other general parameters. Set the parameters and press OK.



eneral Parame	eters		2				>		
Other Para	meters	Scree	en	Drawir	Display				
Project	General Informa	tion		Mate	erial - R	egulation			
Regulation	EC					~			
National	General					~			
Standard Stee	l Shapes	E	uro	~	Metric	•			
Concrete			Stee	el	_				
Foundation	C20/25	\sim	Mem	bers - Eleme	ents S2	75(Fe430) ~			
Upper	C20/25	\sim	Stee	l Plate	S2	S275(Fe430) ~			
			~						
Steel	S400e	\sim	Weld S275(Fe430)						
Stimups	S400s	~	Timb	er	C14	4 ~			
Safety Factor Ultimate γc 1.5 γs 1.15	s Serviceabilit 1 1	Y 1 1	/M0 /M4	үМ1 1 үМ5 1	γM2 1.25 γM7 1.1	үМ3 1.25			
	ОК	С	ancel	Ap	ply	Help			

2.3 Project Modelling from a 3D dwg file

This example is intended to educate the user in modeling a steel structure from a 3D dwg file. With the new version of Scada Pro, it is possible to automatically identify the steel sections from a three-dimensional design.

More specifically, for the automatic identification of the steel sections, the next steps must be followed:



2.4 Preparing a 3D dwg file



For this example, the design steel truss structure of the above image is used. It is a one opening frame structure with six trusses. In the 1st and 5th frame there are vertical wind bracings on both sides, and purlins on the roof.

MAIN DESIGN CONDITIONS:

1. Different layers for each cross section were defined during the design

S	Name	0	Fre	L	Color	Linetype	Lineweig	Trans
_	wind brac.	•	×	•	🔤 yel	Continu	— Defa	0
-	upper	?	×.	6	<mark> 1</mark> 60	Continu	— Defa	0
-	truss	1	×.	•	📃 cyan	Continu	— Defa	0
_	TEXTN	1	×.	•	yel	Continu	—— Defa	0
_	TEXTM	1	×.	•	yel	Continu	— Defa	0
_	griders	?	×.	•	ma	Continu	— Defa	0
-	main beam	?	×.	6	200	Continu	— Defa	0
-	lower	?	×.	6	1 50	Continu	— Defa	0
-	col2	?	×.	6	<mark></mark> gr	Continu	— Defa	0
\checkmark	col1	?	×.	•	red	Continu	— Defa	0
_	0	9	- 🔆 -	б	🗌 wh	Continu	Defa	0



2. Also, at the intersection of the lines, where during the cross-section identification, the model requires the existence of a node, the design was made with segments of lines.

More specifically, the figure below shows that at the point where the lower truss element encounters the column, it must be a node, so the line is not continuous but consists of two successive segments.



2.5 Import of the drawing file and sections recognition

First, give the name of the project, and before the file import, define the levels.

In Basic unit, in Layers – Levels



deactivating also the Rigid Link Constrain:

EXAMPLE 3: "Steel Structure"



New Edit Delet Move	Level te	∧ Na ∨ Hei	me 0 ght (cm)	0	-	0		Number	0 Add
	Execute		Readjustn	nent	+	0			, au
n/n	Name			Height	R.L.C.	Even H	3D		Select All
0	0			0.00	?	∎,	a		Deselect
1 2				487.00 723.00	Ŷ	₽°	o O		R.L.C
					·	_			Non R.L.C.
									Even Height
									Non Even Heigh
									Display in 3D
									Hide in 3D
Conn	ection Meth	od of Co	lumns' Nod	les with Me	esh Surfa	e			
Kiner	natic pair to	the near	est node o	of the surfa	ace			\sim	Ok

A BASIC CONDITION FOR THE 3D DESIGN IMPORTING:

Open the 3D visualization of the empty interface of Scada Pro.



Use Import command to import the draw file:





The 3D drawing appears on the 3D interface.

From "Basic" and the command group DXF-DWG starts the <u>automatic process of inserting the</u> <u>steel sections:</u>

Press Layers to open the Import File Layers including all draw layers and two new commands, Assign Columns Cross-Section and Assign Beam Cross-Section.



	Import File Lavers		Ν	×
Layers Layers Move	Number 0 col1 col2 lower main beam	Visible Q Q Q Q Q	•	Select All Cancel Option Visible Non Visible
Delete	purlins	ā		Convert Lines - Arcs
DWG Layers	TEXTM TEXTN	a		ОК
DXF Freeze Laye	truss	<u>¤</u>	~	Cancel
	Assign Column Cross-	-Section Assign Beam Cross	s-Section	
Rotate	Assign F	Foundation Beams Cross-Section		Purge

Select col1 and col2 and press Assign Column Cross-Section



Set the section IPE300, angle 0, quality S275 and layer Steel Columns:



EXAMPLE 3: "Steel Structure"



Automatically import the column cross sections into the position specified by the col1 and col2 layers of the draw file.



Repeat the same procedure for the Vertical Wind Bracings, selecting wind brac. and Assign Beams Cross-Section





Set SHS100X5 and match the layer Ver. Wind bracings. Ok, and wind bracing cross sections are automatically inserted.



Respectively for the Griders:





For the truss elements, set three New Layers in the "Edit Layer" window: -Upper -Lower -Truss

To insert all the respective cross sections of the draw layers.

Edit Layers						×
Current	Lines circles					Levels XZ - Storeys
New	Truss					Update
Number		Visible	Editable	Colour	^	Select All
 Flat Slab		a a	 	8 8		Deselect All
Drop Panel		Q M		8		Visible
Support Line zz		ã	= ∎ -0	8		Non Visible
Lower		ă	∎ .	1		Editable
Truss		Ø	∎°	1	~	Non Editable
Delete Data						
All Model	By Level XZ	Ву	/ Layer	Model	Only	OK Cancel

And come to the following model:



EXAMPLE 3: "Steel Structure"



And activating the virtual view:



NOTE:

Truss does not transfer moment, So we have to free the members from the moment.

Using the command "Multiselect Edit" and the Select Group by Filter and Layer "Lower" and Add by Filter.

Load Group		×
Material	Concrete 🗸 🗸	Beams - B3D \checkmark Add to List Clear List
🗌 Туре	C8/10 \vee	~ ~ ~
Element	B-3d \vee	B-3d - 1325(1407,1406) - (A)IPE 300 - L:Lower
Member	Beam \vee	B-3d - 1327(1409,1408) - (A)IPE 300 - L:Lower
🗹 Layer	Lower \checkmark	B-3d - 1328(1410,1409) - (A)IPE 300 - L:Lower B-3d - 1329(1411,1410) - (A)IPE 300 - L:Lower
Preference	Cross Section \sim	B-3d - 1331(1413,1412) - (A)IPE 300 - L:Lower
Color	0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	B-3d - 1332(1414, 1413) - (A)IPE 300 - L:Lower B-3d - 1333(1415, 1414) - (A)IPE 300 - L:Lower B-3d - 1334(1416, 1415) - (A)IPE 300 - L:Lower B-3d - 1347(1404, 1240) - (A)IPE 300 - L:Lower
Select	From To Step	B-3d - 1349(1259,1416) - (A)IPE 300 - L:Lower
Objects	0 0 0	B-3d - 1350(1333,1339) - (A)CHS 193,7X10 - L:Truss B-3d - 1351(1333,1339) - (A)CHS 193,7X10 - L:Truss B-3d - 1352(1327,1340) - (A)CHS 193,7X10 - L:Truss
Add By Filter	Remove By Pick + Clear List	B-3d - 1353(1321,1341) - (A)CHS 193,7X10 - L:Truss B-3d - 1354(1342,1315) - (A)CHS 193,7X10 - L:Truss B-3d - 1355(1343,1309) - (A)CHS 193,7X10 - L:Truss
Load Group	Save Group OK	B-3d - 1356(1303,1344) - (A)CHS 193,7X10 - L:Truss



Ok and right clic to open the Multiselect Edit window. In Element Releases check Mz start and end for all Truss and Lower Layer elements. Apply and Exit to complete.

material		Cross-S	ection	(Cross-Se	cti	on Deta	ails	Nodes		Memb	per Typ
Member	Prop	erties	Eleme	ent Re	eleases	ł	Element	Rigi	d Offsets	Dr	aw	Histo
Beam				`	~				B-3d			~
Layer	[Truss										~
Start i	i —	F			т.		End j		F			т.
		From			10		_		From			10
	Ν			N				N			N	
	Vy			Vy				Vy			Vy	
	Vz			Vz				Vz			Vz	
	Mx			Мх				Мх			Mx	
	Му			Му				Му			Му	
	Mz		\checkmark	Mz	\checkmark			Mz		\checkmark	Mz	\checkmark
									Exit			Help
ft click	on	a trus	s eler	nen	t to ch	ec	ck that	t Pro	opertie	s ar	e up	dated
ft click	on	a truss	s eler	nen ¤ ×	t to ch	ec	ck that	t Pro	opertie	s ar	e up	dated
ft click perties ≹↓ ा ∮	on	a truss	s eler	nen # ×	t to ch	ec	ck that	t Pro	opertie	s ar	e up	dated
ft click perties ⊉↓ Ⅲ ∮ er	on	a truss	s eler	nen # ×	t to ch	ec	k that	t Pro	opertie	s ar	e up	dated
ft click perties ≹↓ ा ∮ er pr	on	a truss	s eler	nen # ×	t to ch	ec	ck that	t Pro	opertie	s ar	e up	dated
ft click perties 21 I S	on	a truss	s eler	nen # ×	t to ch	ec	ck that	t Pro	opertie	s are	e up	dated
ft click perties 2 I I I I I er or e Cross-Secti	on a	a truss	s eler	nen # ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	dated
ft click perties 2↓ □ ∮ er cross-Secti nap	on i	a truss	s eler	nen * ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties 2↓ □ ∮ er or e Cross-Secti nap	ON i	a truss	s eler	men # ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties ≵↓ ■ ∮ er or e Cross-Secti nap	ON i	a truss	s eler	men # ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	dated
ft click perties 2↓ □ ∮ er or e Cross-Secti nap Geometry Nodes 'Cross Sect	ON i	a truss 1350 Truss 1 B-3d	s eler $z \rightarrow d$	men # ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	dated
ft click perties 2↓ □ ∮ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse	ON i	a truss	s eler $z \rightarrow d$	men * ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	dated
ft click perties 2↓ ■ ∮ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member R	ON :	a truss	s eler $z \rightarrow d$	men * ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties 2↓ ■ ∮ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member Ri N (Start)	On i ion tion* ets (cm	a truss	s eler $z \rightarrow d$	men # ×	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties 2↓ ■ ∮ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member R N (Start) N (Start) N (Start)	ON i ion	a truss	s eler $z \rightarrow d$	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties 2↓ □ ∮ er or e Cross-Secti nap Geometry Nodes 'Cross Sect Rigid Offse Member R: N (Start) N (Start) N (Start)	ON : ion	a truss	s eler $z \rightarrow d$	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties 2↓ □ ∮ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member R: N (Start) N (Start) N (Start) Vy (Start) Vy (Start)	ON i ion	a truss	s eler $z \rightarrow d$	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties ¢↓ ■ ¢ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member Ri N (Start) N (Start) N (Start) N (Start) Vy (Start) Vy (Start) Vy (Start) Vy (Start) Vy (Start) Vy (Start)	On i ion	a truss	s eler $z \rightarrow d$	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties ¢↓ ■ ¢ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member Ri N (Start) N (Start) Vy (Start) Vy (Start) Vy (Start) Vy (Start)	On i ion	a truss 1350 Truss 1 B-3d B-3d C C C C C C C C C C C C C	s eler $z = \frac{y}{z}$	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties ¢↓ ■ ¢ r or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member Ri N (Start) N (Start) N (Start) Vy (Start) Vz (Start)	ON i ion	a truss 1350 Truss 1 B-3d B-3d C C C C C C C C C C C C C	s eler	men	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties ¢↓ ■ ¢ er or e Cross-Secti nap Geometry Nodes *Cross Sect Rigid Offse Member R N (Start) N (End) Vy (Start) Vz (End) Mx (Start) Mx (Start)	ON : ion	a truss 1350 Truss 1 B-3d B-3d C C C C C C C C C	s eler	men A × C	t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated
ft click perties at a set of the set of t	On i ion	a truss 1350 Truss 1 B-3d B-3d a a a a a a a a a a a a a	s eler		t to ch	ec	k tha	t Pro	opertie	s ar	e up	odated



2.6 Footings configuration



Select Footing in Modeling and define the parameters

Use the 3D visualizzation to insert footings on the bottom of the columns elements.



Correspondingly, for connecting beams, select the cross section and pass them from one node to the other, either in the Mathematical display or Physical. Automatically calculates the beams mathematical model:











NOTE:

One of the new tools that SCADA Pro offers (after the creation of the mathematical model), is the Footing Predesign:



Which predesigns the footings and may change the dimensions of the footings by the soil interaction stress $\sigma(KN/m2)$, the height of the footings H and the height of the supernatant earth hs.



3. LOAD INPUT

3.1 How to insert wind and snow loads automatically by EC 1:

For steel structures the influence of the wind and snow loads is extremely important and must be considered.

		0-0.00	•	1		Ŧ										
	Ba	asic	Modeling		View Tools		Slabs Loads A		An	alysis Post-Pro		rocessor	ocessor Members Design		gn Drawings	-Detailing
L	: Ц	G	4		0	*	<u>,Ш</u> ,		ш.	Sec.		011	\$	011	· ·	
Load Case	l Lo s Gro	ad ups	Insert	Edit	Yield Lines *	Slab Reactions *	Insert	Edit	View	Сору	Tools	Parameters *	Edit	View	Member Correspondence	Post-Processor
De	finitio	n		Sla	b Loads			Me	mber Lo	ads				Wind -	Snow Loads	

Through the "Loads" unit and the "Wind – Snow Loads" command group, the appropriate tools for the calculation and the distribution of the loads to the walls and the roofs, are located. The first step is to define the wind and snow parameters according to the location of the structure.

Parameters:

In the wind parameters dialog window:

EC1 WIND PARAMETERS	×
-	
Zone Rest of Greece	\sim
Altitude from sea level (m) ? A 500	
Mean Wind Velocity (m/sec) Vb,0 27	
Snow Density (Kg/m3) ? ρ 1.25	
Directional Factor Cdir 1	
L, Season Factor Cseason 1	
Soil Type	
IV Area in which the 15% is covered with buildings with height >15 m	\sim
Distance from More than 40 Km	\sim
Z0(m) 1 Zmin(m) 10	
Kr 0.17	
Orthography Factor	
Cliffs and escarpments \checkmark Upwind \checkmark Lu(m) -500	
Crest	
H site x downwind slope < 0.05	
wind > X(m) -150	
Z(m) 150	
X X+ CO(z) 1	
Roughness Factor	_
Automatic Calculation Cr(z) 0.53956204	
Cancel	

Select EC1 and define the Zone, the Soil Type, the Orthography and the necessary wind values. The Roughness factor is automatically calculated if the respective checkbox is activated, otherwise you can fill in a value manually.

In the snow parameters dialog window:



EC1 SNOW PARAMETERS		×
Regulation		~
Topography Normal		~
********	Exposure Factor Ce	1
	Thermal factor Ct	1
	Snow Density γ kN/m3	3
Zone II (Magnesia,Fthiotida	a,Karditsa,Trikala,Larissa,Sp	orades,Ei 🗸
Snow Load (at sea level) Sk	,0 kN/m2	1.7
Altitude (from sea level) A m	?	500
Snow Load (at Altitude A) Si	k kN/m2	2.2054174
Accidental Snow Load		
Design State Case A (No	exceptional falls/No except	ional drift $ \smallsetminus $
Exceptional Loads Facto	or Cesl 1	
ОК	Cancel	

You set the topography which defines the values of the Ce and Ct coefficients, the Zone and the design state.

Edit Walls:

Next, through the "Edit" > "Walls", we define the walls for each direction for the calculation of the Equivalent Wall.

lit Wall	×	
	Wall left (perpendicular wind dir. 0) V	
	Partial Walls	
	a/a b(m) h(m) %	
	1 20.00 3.00 0.00	
	b(m) 20 Pick Openings h(m) 3 Pick 0 % New Del From List Equivalent Wall b(m) = 20 Automatic h(m) 3 Calculation 0 % 0 % Openings 0 % Cancel 0 %	

Starting from the wall on the left perpendicular on the wind direction "0".

You define the length (b) and the height (h) for each

wall (Left, Front, Right, Back), by clicking the button and selecting every time with the mouse the two ending points of the wall in the corresponding direction, (model should be viewed in 3D).

▲ Define "h" from the foundation level.

The goal here is to define all the parts of the wall that are perpedincular to the 0 direction of the

wind, with a graphical way, by using the Pick button and pointing to the corners of the wall for the definition of the length (b) and the height (h) of each section, per level.



Next, set the percentage of the openings and click The program calculates automatically the "Equivalent Wall." Press "OK" command to save the parameters. Repeat for all four directions of the walls.



NOTE:

- ▲ The height of the lower wall always defined starting from level 0 even if the steel structure begins at a higher level.
- ▲ If the front view consists of several walls at one or more levels, press the button "New" and repeat the above procedure to set the whole face.





Edit Roof: Similarly, from "Edit" > "Roof",



Define the type, the orientation and the dimensions Lo, L1, L2, L3, of the roof by clicking/button and showing with the mouse the four corners of the roof.

View Wind: With the command "**View**" > "**Wind**", you can view for each wind direction the distribution of the wind loads along the height of the structure with the respective Cpe+, Cpe-, Cpi coefficients, for each wall and roof.

View Snow: Similarly, you can use the next command "**View**" > "**Snow**", to view the snow load distribution upon the roof by EC1.

Member Correspondence: to assign the calculated loads to the members, through the influence zones.

Select the command and in the dialog box: select a wall, or a roof and define the dimension of the influence zones.

In the new version of SCADA Pro, completed and integrated the automatic calculation of influence zones for linear members to make the distribution of wind and snow loads.

Remind that until now the automatic distribution was only for the structures derived from Templates. Now enable this distribution on any surface.

By selecting the command now opens the following dialog box

EXAMPLE 3: "Steel Structure"



Member Corr	espondenc	e			×		
Wall left (pe	rpendicular v	wind a	dir. 0)		~		
Add Membe Influence Z	ers ones (m)		Vertices	Coordinat	e (cm)		
left	0	1.	Pick	0.0,500.0),0.0		
Right			0		Pick	0.0,0.0	, 0.0
ragine		3.	Pick	0.0,0.0,	300.0		
Pid	k			Distribution	View		
	Initialisat	tion o	f all member	rs (Walls-Roofs)			
Member	s Initialisatio	n		[Cancel		

The part on the old definition of the influence zones did not change but added to the right a new part to define the area with three points.

The definition always concerns the active area
Wall left (perpendicular wind dir. 0)

- Better to start either the manual or semi-automatic procedure by pressing the "Members Initialization" button.
 - Semi-automatic Procedure

Indicate the point graphically with the following particularity:

- The first two points define the direction by which the automatic calculation of influence surfaces made for items which are parallel to this direction.
 Note also that the distribution will be for all linear members belonging to this level and are parallel to the first direction.
- After you define the three points, press the "Distribution" button and the program automatically makes the distribution and displays it.
 Similarly for the other walls.







- Finally it is worth noting that if the walls are properly defined there is NO need of more definition. Just select each wall and press «Distribution». The distribution becomes and simultaneously displays on the linear members belonging to this wall.
- Same for the flat roofs only.

Post-Processor: This is the last command.

On the dialog box, in the "Load attribution" field, there are two units;

Load Attribution 90 180 270 Vind 90 180 270 Cpe_p+Cpi 3 7 11 15 Cpe_p-Cpi 4 8 12 16 Cpe_p-Cpi 4 8 12 16 Cpe_p-Cpi 6 10 14 18 Case ii 21 24 Cpe_n-Cpi 6 10 14 18 Case ii 21 24 Case ii 21 24 Case ii 21 24 Case X Case ii 21 Case X Case X Case X Vind Load Deletion (Snow-Wind Loads) Case X Case X Wind 0 New Scenario New Scenario Wind 180 New Scenario Y Wind 270 New Scenario Y Academia Snow New Scenario Y Xacidental Snow	ases ad cases for a typica ases for an accidenta ear in the fields are the es.	with a total of 16 load cases -snow loads, three load cases snowfall, three load cases for snowfall.	Acci- Typical dental	Snow					Load Attributi
Wind 90 180 270 Cpe_p+Cpi 3 7 11 15 Cpe_p-Cpi 4 8 12 16 Cpe_p+Cpi 5 9 13 17 Case ii 20 23 Case ii 21 Cpe_n+Cpi 6 10 14 18 Case ii 21 Case ii 21 24 Case ii 21 24 Total Load Deletion (Snow-Wind Loads) Case iii 21 24 Load Attribution in Members(from Wind and Snow) Results The numbers that appear in the fields numbers of the load cases. Scenarios Wind 0 New Scenario Results Wind 180 New Scenario Wind 270 New Scenario Wind 270 New Scenario Vertical Snow New Scenario Accidental Snow New Scenario Vertical Snow New Scenario	ad cases for a typica ases for an accidenta ear in the fields are the es.	-snow loads, three load cases snowfall, three load cases for snowfall.	Acci- Typical dental	Snow				on	Louid Attailout
0 90 180 270 Typical dental Typical dental Typical dental Cpe_p-Cpi 4 8 12 16 Cpe_p-Cpi 4 8 12 16 Case ii 19 22 Cpe_p-Cpi 6 10 14 18 Case ii 21 24 Cpe_n-Cpi 6 10 14 18 Case ii 21 24 Total Load Deletion (Snow-Wind Loads) Case iii 21 24 The numbers that appear in the fields numbers of the load cases. Scenarios NWind 0 New Scenario Results Results New Scenario Wind 180 New Scenario New Scenario New Scenario New Scenario Y Ypical Snow New Scenario New Scenario New Scenario New Scenario Accidental Snow New Scenario New Scenario New Scenario New Scenario	ar in the fields are the	snowfall, three load cases for snowfall.	Acci- Typical dental						Wind
Cpe_p+Cpi 3 7 11 15 snowfall, three load cases for an act snowfall. Cpe_p-Cpi 4 8 12 16 Case ii 19 22 Cpe_n+Cpi 5 9 13 17 Case ii 20 23 Cpe_n+Cpi 6 10 14 18 Case ii 21 24 Total Load Deletion (Snow-Wind Loads) Load Attribution in Members(from Wind and Snow) The numbers that appear in the fields numbers of the load cases. Scenarios Results Results New Scenario Wind 180 New Scenario Results Wind 270 New Scenario New Scenario Yypical Snow New Scenario New Scenario Accidental Snow New Scenario New Scenario	ases for an accidenta ear in the fields are the es.	snowfall, three load cases for snowfall.	dental		270	180	90	0	
Cpe_p-Cpi 4 8 12 16 Case ii 19 22 Case ii 20 23 Case ii 21 24 Snowfall. The numbers that appear in the fields numbers of the load cases. Total Load Deletion (Snow-Wind Loads) Load Attribution in Members(from Wind and Snow) Scenarios Wind 0 New Scenario Wind 180 New Scenario Wind 180 New Scenario Y Wind 270 New Scenario Y Wind 270 New Scenario Y Accidental Snow New Scenario Acadivisis Scenario Acadivisis Scenario Case iii C	ear in the fields are the es.	snowfall.			15	11	7	3	Cpe_p+Cpi
Cpe_n-Cpi 5 9 13 17 Case ii 20 23 Cpe_n-Cpi 6 10 14 18 Case ii 21 Case ii 21 24 Total Load Deletion (Snow-Wind Loads) Load Attribution in Members(from Wind and Snow) Scenarios Scenarios Wind 0 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario New Scenario Accidental Snow New Scenario Must 270 New Scenario Accidental Snow New Scenario Must 270 New Scenario Accidental Snow New Scenario Must 270 </td <td>ear in the fields are the es.</td> <td></td> <td>19 22</td> <td>Case i</td> <td>16</td> <td>12</td> <td>8</td> <td>4</td> <td>Cpe_p-Cpi</td>	ear in the fields are the es.		19 22	Case i	16	12	8	4	Cpe_p-Cpi
Cpe_n-Cpi 6 10 14 18 Case iii 21 24 Inumbers of the load cases. Ital Load Deletion (Snow-Wind Loads) Load Attribution in Members(from Wind and Snow) Scenarios Wind 0 New Scenario Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Typical Snow New Scenario Accidental Snow New Scenario	es.	The numbers that appear in the	20 23	Case ii	17	13	9	5	Cpe_n+Cpi
Total Load Deletion (Snow-Wind Loads) Load Attribution in Members(from Wind and Snow) Scenarios Wind 0 New Scenario Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario		numbers of the load cases.	21 24	Case iii	18	14	10	6	Cpe_n-Cpi
Load Attribution in Members(from Wind and Snow) Scenarios Wind 0 New Scenario Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario				ind Loads)	n (Snow-Wi	d Deletior	otal Load	To	
Scenarios Wind 0 New Scenario v Wind 90 New Scenario v Wind 180 New Scenario v Wind 270 New Scenario v Typical Snow New Scenario v Accidental Snow New Scenario v Analysis Scenario v New Scenario v			w)	Vind and Sno	pers(from V	n in Memb	ttribution	Load Att	
Wind 0 New Scenario Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario									Scoperios
Wind 0 New Scenario Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario		1	Results						scenarios
Wind 90 New Scenario Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario					• ~	v Scenari	New		Wind 0
Wind 180 New Scenario Wind 270 New Scenario Typical Snow New Scenario Accidental Snow New Scenario					• •	v Scenari	New		Wind 90
✓ Wind 270 New Scenario ∨ ✓ Typical Snow New Scenario ∨ ✓ Accidental Snow New Scenario ∨ Analysis Scenario ∨					• •	v Scenari	New		Wind 180
✓ Typical Snow New Scenario ✓ Accidental Snow New Scenario ✓ Analysis Scenario ✓					• ~	/ Scenari	New		Wind 270
Accidental Snow New Scenario V					• •	v Scenari	New	iow	Typical Sr
Analysis Scenario Creation					• ~	/ Scenari	New	Snow	Accidenta
Cancel		1	Cancel			Creation	enario C	alysis Sce	An

And now 16 more cases for wind (from 3 to 18) and 3 for snow (19, 20 and 21). In this example we will not consider the cases of the accidental snowfall.

Sele	ect the co	omman	d	Loa	d Attribution in Members(from Wind and Snow)							to apply	
the	wind	and Total	snow Load Dele	loads tion (Snow	to -Wind	the Loads)	members	of to	the delet	struc e them	ture, all.	or	click
The aut	"Scenari omaticall	os" fiel y create	ld, inclue ed by clie	des a list king the	: with	n all th An	e possible so alysis Scenario (cenai Creati	rios of on	the ar	nalysis, utton!	whic	ch are
1	Thus SCA snow loa	DA Pro ds it au	besides tomatica	of the a ally creat	utom :es all	atic ca I the ai	lculation of t nalysis scena	he di: rios a	istribu as well	tion of	the wii	าd an	d

EXAMPLE 3: "Steel Structure"



	-weight	L DEGU LOGO		V Incor	+	Renumbe	ering					51
				Inser		Nodes	Cuthill-Mck	(ee(II)	~	Advanc	ed preaded S	olve
LC	S.W.	Description		Pel From	n Lis'					Huru-11	in couleu 5	onve
1	Yes	Dead Load				Disable			Mana			
2	No	Live Load		Delet	e				Name			
3	No	Wind 0 Cpe_p+Cpi		Load	s	EC8_Gener	al Static (0)	0	Analysis	EC8_0	General	
4	No	Wind 0 Cpe_p-Cpi				EC8_General	al Dynamic (Inc 0 (2)	1)	_			
5	No	Wind 0 Cpe_n+Cpi		Delete	All	Static Aven	ioc 90 (3)		Type	Static		
6	No	Wind 0 Cpe_n-Cpi		Load	s	Static Aveu	ος 180 (4)		Propertie	2S		
7	No	Wind 90 Cpe p+Cpi		×		Static Aveu	ος 270 (5)		Fleme	onte	Nod	les
<				> OK		Static Xiovi	Топіко (6)		Lienie			
									Load C	ases	Mas	ses
									New	I	Upd	date
										Run all	Analisies	
										E	xit	
ne om o	F each (command of t	commar he "Wir	nd, opens nd – Snow	the tx / Loads	t results " group	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
ne om (d001 - e Edit	F each (WordPad View In	command of t	commar he "Wir	nd, opens nd – Snow	the tx / Loads	t results group	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit	F each (WordPad View In I I I I I I I I I I I I I I I I I I	command of t	commar he "Wir	nd, opens nd – Snow	the tx Loads	t results " group –	s file, wl are ana	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit	F each d ^{WordPad} View In	command of t	commar he "Wir	nd, opens nd – Snow	the tx / Loads	t results " group _	s file, wl are ana	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit	F each (WordPad View In B B B B ATA	Results command of t sert Format Help ACCORDING TO EN	commar he "Wir SNOW LOA	nd, opens nd – Snow	the tx v Loads	t results " group _	s file, wl are ana	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit	F each o View In B B C	Results command of t sert Format Help MIND ACCORDING TO EN	commar he "Wir SNOW LOA 1991-1-3/4	nd, opens nd – Snow	the tx v Loads	t results " group _	s file, wl are ana	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit Edit	WordPad View In Control Control Contro	Results	commar he "Wir SNOW LOA	nd, opens nd – Snow	the tx v Loads	t results " group 	s file, wl are ana	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - Edit Coroce Edit Coroce HERMA	WordPad View In Control Control WATA	Results	commar he "Wir SNOW LOA	nd, opens nd – Snow	the tx / Loads 	t results " group 	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - e Edit wow D woopogra XxPosu HERMA XxPosu	WordPad View In B B C VATA VATA VATA VATA VATA VATA VATA VAT	Results	commar he "Wir SNOW LOA 1991-1-3/4	nd, opens nd – Snow	the tx / Loads NN ECE : Normal : 1.00 : Case A : 1.00 : 3.00	t results " group 	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - e Edit e Edit NOW D OPOGR XPOSU HERMA VESIGN EEIGHT EEIGHT	WordPad View In WATA WATA UCOEFL STATE STONAL S: TONAL S: TONAL S: TONAL S: TONAL S: TONAL S:	Results	Commar he "Wir SNOW LOA 1991-1-3/4	nd, opens nd – Snow	the tx / Loads N ECE : Normal : 1.00 : Case A : 1.00 : 3.00 : 0.00	t results " group 	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - e Edit e Edit i Edit i Edit i Elit i Elit	WordPad View In I I I I I I I I I I I I I I I I I I	Results	Commar he "Wir SNOW LOA 1991-1-3/4 NT Cesl Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 d001 d001 d001 e Edit e Edi	WordPad View In View In Marka View In View In	Results	NT Cesl Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wl	nere a lytical	ll the c ly prese	alcula	ations I.	de
d001 - e Edit e Edit	WordPad View In View In C View Ino C View In C View In C In C View In C View	Results command of t sert Format Help MIND ACCORDING TO EN WIND ACCORDING TO EN WIND COM LOADS COEFFICIE (OF SNOW Y(Kn/m^3) Rest Greece) r SEE LEVEL) Sk,0 (LEVEL 500.00m) Sk (NT Ces1	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wl	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- d001- d001- e Edit COPOGR C	WordPad View In View In C View C View In C View In C C View In C View In C View In C View In C In C View In C View In C View In C View In C View In C View In C View In C View In C View In C In C In C In C View In C In C View In C In C In C In C In C In C In C In C	Results command of t sert Format Help MIND ACCORDING TO EN WIND ACCORDING TO EN FICIENT CE ICIENT CE ICIENT CE ICIENT CE COM LOADS COEFFICIE X SOW V(Kn/m3) Rest Greece) r SEE LEVEL) Sk,0 (LEVEL S00.00m) Sk (DRY	NT Ces1	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- d001- e Edit COPOGR	WordPad View In View In Control Control View In Control View In Control View In View In View In View In View In View In Control View In View I	Results	NT Cesl Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001 - d001 - Edit d001 - Edit More L d001 - Concerned Concern	WordPad View In View In Marka	Results	Commar he "Wir / SNOW LOA 1991-1-3/4 / SNOW LOA 1991-1-3/4 / SNOW LOA (M/m^2) Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- d001- Edit Generation Edit Copege	WordPad View In View In I I I I I I I I I I I I I I I I I I	Command of t command of t sert Format Help ACCORDING TO EN ACCORDING TO EN WIND ACCORDING TO EN WIND ACCOR	Commar he "Wir / SNOW LOA 1991-1-3/4 NT Cesl Kn/m^2) Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- d001- time Edit edita edit edit edit edit edit edit edit edit	F each (View In I ▲ Correct View In I ▲ Correct View In I ▲ Correct View In I & Correct I & State I & State I & Correct I & State I &	Results command of t set Format Help MIND ACCORDING TO EN ACCORDING TO EN FICIENT Ce COENT Ct NOW LOADS COEFFICIE (COF SNOW Y(KN/m ³)) R SEE LEVEL) Sk,0 (LEVEL 500.00m) Sk (DRY OF THE BASIC WIND (Kg/m ³) TOR Cdir N CSEason	NT Cesl Kn/m^2)	nd, opens nd – Snow	the tx / Loads // Loa	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
A CONTRACTOR CONTRACTO	ATA WordPad View In Wiew In	Results command of t command of t tert Format Help MIND ACCORDING TO EN WIND ACCORDING TO EN WIND ACCORDING TO EN WIND ACCORDING TO EN WIND COLLENT Ct NOW LOADS COEFFICIE (OF SNOW ecce) T SEE LEVEL) Sk, 0 (LEVEL 500.00m) Sk (LEVEL 500.00m) Sk (CRY OF THE BASIC WIND (Kg/m^3) TOR Cdir NG Cseason APHY	Commar he "Wir SNOW LOA 1991-1-3/4 NT Cesl Kn/m^2) VELOCITY (nd, opens nd – Snow	the tx / Loads / Loads // Load	t results " group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
A COURT AND A COUR	WordPad View In Control Control View In Control View In Control View In Contro	Results command of t tert Format Help ACCORDING TO EN ACCORDING TO EN ACCORDING TO EN FICIENT Ce CIENT Ct NOW LOADS COEFFICIE (OF SNOW V(Kn/r^3) REST ELEVEL) Sk,0 (LEVEL 500.00m) Sk (COEFFICIENT CE CE SOURCE) TOR Cett CE SOURCE) CE SOURCE CE SOURCE) CE SOURCE CE SOURCE) CE SOURCE CE	Commar he "Wir SNOW LOA 1991-1-3/4 NT Cesl Kn/m^2) VELOCITY (E Lu (m)	nd, opens nd – Snow	the tx / Loads / Loads // Load	t results "group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- e Edit cond (cond (F Construction	Results	Commar he "Wir SNOW LOA 1991-1-3/4 NT Cesl Kn/m^2) VELOCITY (E Lu (m) E H (m)	nd, opens nd – Snow	the tx / Loads // Loads // Loads // Loads // Loo //	t results "group 	s file, wi are ana 	here a lytical	ll the c ly prese	alcula	ations I.	de
d001- e Edit e Edit	F Constraints Co	Command of t command of t sert Format Help ACCORDING TO EN ACCORDING TO EN ACCORDING TO EN FICIENT CE CELENT CE C	Commar he "Wir Solution / SNOW LOA 1991-1-3/4 / NT Cesl Kn/m^2) VELOCITY (E Lu (m) E H (m) E TOP OF T	nd, opens nd – Snow	the tx / Loads // Loa	t results "group 	s file, wi are ana 	nere a lytical	ll the c ly prese	alcula	ations I.	de



4. ANALYSIS

After the modeling and the distribution of the loads to the members of the structure, the analysis of the structure, by the selected regulation, the creation of the load combination and the results of the checks are next.

4.1 How to create an analysis scenario:

Through the "Analysis" unit, the commands of the "Scenarios" group allow the creation of the analysis scenarios (regulation and analysis type selection) and the execution.



According the selections made in the initial General Parameters window, comes the predefined analysis and members design scenarios.

eneral Parame	eters					>
Other Para	meters	Scr	reen	Drawi	ng	Display
Project	General Info	mation		Mat	erial - Re	egulation
Regulation	EC					~
National	General					~
Standard Stee	l Shapes		Euro	~	Metric	· · ·
Concrete			Stee			
Foundation	C20/25	\sim	Mem	bers - Eleme	ents S27	75(Fe430) 🗸
Upper	C20/25	\sim	Steel	Plate	S27	75(Fe430) 🗸
oppe.			Bolts		4.8	~
Steel			Weld		S27	75(Fe430) 🗸
Main	S400s	\sim				
Stimups	S400s	\sim	Timbe	er	C14	4 ~
– Safety Factor Ultimate	s Serviceabilit		yM0	yM1	γM2	YM3
vo 15	1		1	1	1.25	1.25
γc 1.5		г	γM4	γM5	γM7	_
γs 1.15	1		1	1	1.1	
[ОК		Cancel	Ap	ply	Help

To create more analysis scenarios, select "New". In the dialog box that opens, besides the predefined ones, you can create as many scenarios as you want.



Scenario			×	Aridiysis	EU-ŏ_Greek ∨ Statio
- L ·		N	~	Type	Dynamic
Renumbering		Advanced		Properties	Seismic EC-8_Greek
Nodes Cuthill-McKee(II)	~	Multi-Threaded Solver		Elemer	NTC_2008 EC8 Italia
				Load Ca	EC8_Cyprus
Disable	Name				EC8_General
EC9. Conoral Static (0)				New	SBC 301
EC8_General Static (0) EC8_General Dynamic (1)	Analysis	EC8_General	\sim	Туре	Dynamic
Static Aveµoç 0 (2)	Type	Static	\sim	Properties	Static Dynamic
Static Aveµoc 90 (3)	Propertie	2		Elemen	Nonlinear
Static AVELIOG 180 (4) Static Avelioc 270 (5)	Toperac	-	_	Load Ca	Elastic Dynamic
Static Xióvi Tunikó (6)	Elemer	nts Nodes			Preliminary Dynamic
				New	Time History Linear Time History Non Linea
	Load Ca	Ases Masses			
				Analysis	EC8_General
	New	Update		Туре	Static
				Properties	Static Dynamic
		Run all Analisies		Eleme	Nonlinear Linear
		Evit		Load Ca	Elastic Dynamic
		Exit			Preliminary Static Preliminary Dynamic
				New	Time History Linear Time History Non Linea
Nake your selections from ew scenario. If you want y elect among the possible s	the "Analysi ou can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr	lists and c	lick	to crea
Nake your selections from ew scenario. If you want y elect among the possible s or Greece:	the "Analysi you can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr	lists and c	lick	to crea
Nake your selections from ew scenario. If you want y elect among the possible s or Greece:	the "Analysi you can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr – NON LINEAR ME	ro: THODS	lick	to crea
Nake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static	the "Analysi you can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr <u>– NON LINEAR ME</u> Simplified spect	lists and c ro: <u>THODS</u> ral analys	lick	to crea
Nake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti	the "Analysi you can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra	lists and c ro: <u>THODS</u> ral analysis	lick is according according	ng to EAK g to EAK
Nake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic	the "Analysi you can type scenarios pro	s" and the "Type" a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra	ro: THODS ral analysis al analysis	is according according (masses of	to created by the second secon
Make your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic	the "Analysi you can type scenarios pro <u>LINEAR</u>	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI	lists and c ro: <u>THODS</u> ral analysis al analysis K	is according according (masses of	ng to EAK g to EAK displacement)
Aake your selections from ew scenario. If you want y elect among the possible s <u>or Greece:</u> - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84	the "Analysi you can type scenarios pro	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis	lists and c ro: <u>THODS</u> ral analysis al analysis Al analysis K according	is according according (masses of g to 1959	ng to EAK g to EAK displacement) Regulation
Aake your selections from ew scenario. If you want y elect among the possible s <u>or Greece:</u> - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93	the "Analysi you can type scenarios pro	s" and the "Type" I a name as well. ovided in SCADA Pr <u>– NON LINEAR ME</u> Simplified spect Dynamic spectra according to EAI Seismic analysis Seismic analysis	lists and c ro: <u>THODS</u> al analysis al analysis al analysis <u>K</u> according according	is according according (masses g to 1959 g to 1984	to created by the second secon
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static	the "Analysi you can type scenarios pro <u>LINEAR</u>	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis w	lists and c ro: <u>THODS</u> al analysis al analysis K according according vithout se	is according according according (masses of g to 1959 g to 1984 ismic actio	to created by the cre
Aake your selections from ew scenario. If you want y elect among the possible s <u>or Greece:</u> - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static	the "Analysi you can type scenarios pro	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Seismic analysis Static Analysis a	lists and c ro: <u>THODS</u> al analysis al analysis al analysis <u>k</u> according vithout se ccording t	is according according according (masses of g to 1959 g to 1984 ismic action to Eurocoo	to created to created to created to created to EAK g to EAK displacement) Regulation Regulation Dns de 8 and the
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static	the "Analysi you can type scenarios pro	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis w Static analysis a Greek Appendix	lists and c ro: THODS ral analysis al analysis al analysis al analysis according vithout se ccording t	is according according according (masses g to 1959 g to 1984 ismic action to Eurocoo	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek dynamic	the "Analysi you can type scenarios pro <u>LINEAR</u>	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis a Greek Appendix Dynamic analysi	lists and c ro: <u>THODS</u> al analysis al analysis Al analysis Al analysis Cording tithout se ccording t	is according according according (masses) g to 1959 g to 1984 ismic action to Eurocoop ng to Eurocoop	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the pcode 8 and the
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC8 Greek dynamic	the "Analysi you can type scenarios pro	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Seismic analysis Static Analysis a Greek Appendix Greek Appendix	lists and c ro: <u>THODS</u> al analysis al analysis al analysis al analysis cording vithout se ccording t	is according according according (masses of g to 1959 g to 1959 g to 1984 ismic action to Eurocoor ng to Euro	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the code 8 and the
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek dynamic	the "Analysi you can type scenarios pro LINEAR	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis w Static Analysis a Greek Appendix Dynamic analysi Greek Appendix	lists and c ro: <u>THODS</u> ral analysis al analysis al analysis al analysis al analysis coording vithout se ccording t is according ry analysis	is according according according according according to 1959 g to 1959 g to 1984 ismic action to Eurocoo ng to Euro s accordin	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the pcode 8 and the g to KANEPE
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek dynamic - EC 8 Greek Preliminary D	the "Analysi you can type scenarios pro <u>LINEAR</u>	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Seismic analysis Static Analysis an Greek Appendix Dynamic analysi Greek Appendix Static Prelimina Dynamic Prelimi	lists and c ro: <u>THODS</u> ral analysis al analysis al analysis al analysis (according vithout se ccording t s according ry analysis inary anal	is according according according according (masses) g to 1959 g to 1959 g to 1984 ismic action to Eurocool ing to Euro s accordin ysis accordin	to creation regulation Regulation Regulation Regulation Data de 8 and the prode 8 and the g to KANEPE ding to KANEPE
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek static - EC 8 Greek Preliminary D - EC 8 Greek Time History	the "Analysi you can type scenarios pro LINEAR	s" and the "Type" I a name as well. Divided in SCADA Pr NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Seismic analysis Static Analysis a Greek Appendix Dynamic analysi Greek Appendix Static Prelimina Dynamic Prelimi	lists and c ro: THODS ral analysis al analysis al analysis al analysis al analysis al analysis according vithout se ccording t is according ry analysis inary anal ccording t	is according according according (masses of g to 1959 g to 1959 g to 1984 ismic action to Eurocoo s according ysis according ysis according to Eurocoo	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the code 8 and the g to KANEPE ding to KANEPE ding to KANEPE ding to KANEPE
Aake your selections from ew scenario. If you want y elect among the possible s <u>or Greece:</u> - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek static - EC 8 Greek Preliminary S - EC 8 Greek Preliminary D - EC 8 Greek Time History - EC 8 Greek Time History	the "Analysi you can type scenarios pro LINEAR	s" and the "Type" I a name as well. ovided in SCADA Pr - NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis w Static Analysis w Static analysis a Greek Appendix Dynamic analysi Greek Appendix Static Preliminal Dynamic Prelimi	lists and c ro: THODS ral analysis al analysis al analysis al analysis al analysis al analysis al analysis al analysis according timout se ccording timout in ary anal ccording timout is according timout is	is according according according according according (masses (masses) according to 1959 g to 1984 ismic action to Eurocoo ng to Eurocoo ng to Eurocoo ng to Eurocoo	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the prode 8 and the g to KANEPE ding to KANEPI de 8 brode 8
Aake your selections from ew scenario. If you want y elect among the possible s or Greece: - EAK Static - EAK Dynamic-eti - EAK Dynamic - Old 1959-84 - Old 1959-84 - Old 1984-93 - Static - EC 8 Greek static - EC 8 Greek static - EC 8 Greek tatic - EC 8 Greek Preliminary D - EC 8 Greek Preliminary D - EC 8 Greek Time History - EC 8 Greek Time History - EC 8 Greek NonLinear	the "Analysi you can type scenarios pro LINEAR LINEAR	s" and the "Type" I a name as well. ovided in SCADA Pr – NON LINEAR ME Simplified spect Dynamic spectra according to EAI Seismic analysis Static Analysis at Greek Appendix Dynamic analysis Greek Appendix Static Preliminat Dynamic Prelimi Static analysis at Greek Appendix Static analysis at Monlinear analysis	lists and c ro: <u>THODS</u> ral analysis al analysis al analysis al analysis al analysis al analysis according tis according t inary analysis inary analysis inary analysis is according t is according t	is according according according according according (masses) g to 1959 g to 1984 ismic action as according lysis according lysis according lysis according to Eurocording to Eu	to creation ng to EAK g to EAK displacement) Regulation Regulation ons de 8 and the scode 8 and the ding to KANEPE ding to 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)

▲ In this example you'll only choose the scenarios EC8 dynamic for the earthquake, as well as the scenarios Snow Typical, Wind 0 and Wind 90, which were automatically created as previously explained.

Select the EC8 Dynamic. The command **Elements**, includes the properties modifiers for the beam members.

1 The program automatically chooses the appropriate inertial modifiers, by the selected regulation while you can modify at any time these modifiers.

Multipliers of Property	Multipliers of Property Values (Concrete)									
EC-8_Greek DynamicE	C-8_Gre	ek						E	c v	1
Linear Element Proper	rty Value	Multiplier	s						-	
Steel 🗸	Е	G	Ak	Asy	Asz	З	Ix	Iy	Iz	
Concrete Steel	1	1	1	1	1	1	1	1	1	
BEAMS - TRUSS	1	1	1	1	1	1	1	1	1	
BEAMS - B3Def	1	1	1	1	1	1	1	1	1	
COLUMNS - B3D	1	1	1	1	1	1	1	1	1	
COLUMNS - TRUSS	1	1	1	1	1	1	1	1	1	
WALLS - B3D	1	1	1	1	1	1	1	1	1	
WALLS - TRUSS	1	1	1	1	1	1	1	1	1	
Walls Filter (Lmax/Lmir	1) > 4	•			C	К	(Cancel		1



Select the EC8 Dynamic. The command Nodes, opens the following window:

Nodes		×					
EC-8_Greek DynamicEC-8_Greek							
Master Nodes	Yes	\sim					
Springs							
Dx	Dy	Dz					
Yes 🗸 🗸	Yes 🗸 🗸 🗸	Yes 🗸 🗸					
Rx	Ry	Rz					
Yes 🗸 🗸	Yes 🗸 🗸	Yes 🗸 🗸					
ОК		Cancel					

Here you can choose to perform the analysis without considering Rigid Link Constrain at any level even if master nodes exist and consider a fixed base for the whole model even if an elastic foundation is defined.

In cases of <u>Dynamic Analysis</u>, if you select "Nodes" and you "open" the springs "Yes", then you will be able to use the combinations of the dynamic analysis for the design of the footing as well. Select the EC8 Dynamic. The command **Load Cases**, opens the following window:

Load Case parti	cipation											\times
EC-8_Greek Dy Load Cases of	/namicEC-8_(g(m/sec2)	Greek 9.81	7	Availa	ble Load	Cases a	and Load	Groups				
Scenario G(1) + Q(2) +	LC LC1 LC2 LC3 LC4 LC5 LC6 LC7 LC8	LG1 1.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG	^
	LC9 LC10	0.00										•
	<										>	
				ОК			Cance	el				

Where, for each scenario load case (for the current scenario only) on the left column, you match one or more Load Cases (LC) of those that you created.



- Select the value 1.00 for LC1 (after having selected the category "Dead Loads" G(1), that are colored blue) and 1.00 for LC2 (after having selected the category "Live Loads" Q(2), that are colored blue).
- The "+" sign next to the load category Q(2) + shows that for the specific category (scenario Load Case) there is a load participation. The maximum "+" signs for each scenario is 4.

G(1) +

Click Update to update the scenario by the performed modifications.

The program fills automatically a unit factor to the corresponding Load Case. Any modification is acceptable here.

For Static wind and snow scenarios the respective loads participate to the corresponding categories without including the dead and live loads derived from cases LC1 and LC2, since these are already included in the seismic analyzes.

Static Typical sno	owStatic										
Load Cases of Scenario	g(m/sec2)	9.81		Availa	ble Load	Cases a	and Load	Groups			
1+ ^	LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG ^
3+	LC17	0.00									
4 5 6 7 8 9 10 11 12	LC18	0.00									
	LC19	1.00									
	LC20	0.00									
	LC21	0.00									
	LC22	0.00									
	LC23	0.00									
	LC24	0.00									
4	LC25	0.00									
15											~
16 🗸 🗸	<										>

itatic Win oad Case	d 0 Stat s of	tic g(m/sec2)	9.81		Availa	ble Load	Cases a	and Load	Groups			
+	^	LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG /
+		LC1	0.00									
+		LC2	0.00									
		LC3	1.00									
		LC4	0.00									
		LC5	0.00									
0		LC6	0.00									
ĭ		LC7	0.00									
2		LC8	0.00									
3		LC9	0.00									
5		LC10	0.00									×
6	\sim	<										>


tatic Wind	1 90Sta	atic										
oad Cases cenario	s of	g(m/sec2)	9.81		Availa	ble Load	l Cases a	and Load	Groups			
+	^	LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG 🖉
+		LC1	0.00									
+		LC2	0.00									
		LC3	0.00									
		LC4	0.00									
		LC5	0.00									
0		LC6	0.00									
1		LC7	1.00									
2		LC8	0.00									
3		LC9	0.00									
5		LC10	0.00									•
6	\sim	<										>

When a category is activated the + symbol appears next to it.

NOTE

For each scenario, you can activate up to 4 scenario load cases.

4.2 How to run an analysis scenario:



Inside the scenarios list, besides the two predefined scenarios, the scenarios related to wind and snow now exist. Select each scenario and define the corresponding parameters of the selected analysis.

By clicking the "Run" button, depending on the selected scenario, the following dialog box opens:

- ✓ Eurocode scenarios
- ✓ Static scenarios

First of all, select Update to update the parameters of the current scenario and delete the data of the previously executed analysis.

Then, select Parameters to defi

to define the parameters of the current scenario.

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



C8 Parameters				
Seismic Area	Characteristic Periods	Apply	seismic actions on Levels XZ	
Seismic Areas	Spectrum Type Horiz	ontal Vertical Down	0 - 0.00 🗸 Up	2 - 723.00 🗸
	Type 1 V S,avg 1.2	0.9	mic Applycia	L
Zone I v a 0.16 *g	Soil TB(S) 0.1	5 0.05 Siere	nic Analysis	
Imperiment	TD(3) 0.5	Eigen		τųτ
		-Spect	rum Participation factors	
Zone II V Y	TD(S) 2	PFx	0 PFy 0	PFz 0
Spectrum		Acc.E	ccentricities Sd (т)
Response Spectrum Design	 Ductility Class 	DCM V	Sd ((TX) 1
ζ(%) 5 Hori	zontal b0 2.5 Verti	cal b0 3	Sd (TY) 🗌 1
Response Spectrum	Jpdate Spectrum	е пи	0.05 *Lz Sd (TZ) 1
Structural Type	Su(1) /			
Concrete V qx	3.5 qy 3.5 q	z 3.5 Bays	Setbacks	
Structural Type		X [One X All the c	other cases
X Frame System	Z Frame Sv	z Z	One z All the c	other cases
	2			
Structural Type) (E)			
X Concrete Moment Resisting	1 Frames	× Z Concrete Memori	Desisting Frames	
Condicte HomenerCostang	, runes	- Concrete Moment	Residing Frances	•
Interstorey Drift Limit	0.005	Wall	KANEPE Default	OK Cancel
Seismic Triangu	ılar 🗸			
Seismic Triangu this dialog box, yo portance factor, the	u enter all the neces	sary data related nd the levels of th	to the seismic re e seismic loads ap	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area	u enter all the neces e safety coefficients an	sary data related nd the levels of th	to the seismic re e seismic loads ap	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas	u enter all the neces e safety coefficients ar	sary data related nd the levels of th	to the seismic re e seismic loads ap	gion, the soi plication.
Seismic Trianguthis dialog box, yo bortance factor, the eismic Area Seismic Areas ne II \checkmark a 0.24	u enter all the necess e safety coefficients an	sary data related nd the levels of th	to the seismic re e seismic loads ap	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24	u enter all the neces e safety coefficients an	sary data related nd the levels of th	to the seismic re le seismic loads ap	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24	u enter all the neces e safety coefficients ar	sary data related nd the levels of th	to the seismic re e seismic loads ap	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 tk the "Seismic Area	u enter all the necess e safety coefficients an g	sary data related nd the levels of th ectly the value of	to the seismic re e seismic loads ap coefficient "a".	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 ck the "Seismic Area	u enter all the necess e safety coefficients an g	sary data related nd the levels of th ectly the value of	to the seismic re e seismic loads ap coefficient "a".	gion, the soi plication.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 Ck the "Seismic Area Importance	u enter all the neces e safety coefficients ar	sary data related nd the levels of th ectly the value of	to the seismic re le seismic loads ap	gion, the soi plication.
Seismic Triangue this dialog box, yo portance factor, the cortance factor, the point factor, the eismic Area Seismic Areas ne II a 0.24 ck the "Seismic Area Seismic Areas Importance Importance Yi 1	u enter all the neces e safety coefficients an safety coefficients an Select the in	sary data related nd the levels of th ectly the value of mportance facto	to the seismic re te seismic loads ap coefficient "a".	gion, the soi plication. fficient "γι'
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 tk the "Seismic Areas Importance Zone II v Yi 1	u enter all the necess e safety coefficients an so u enter all the necess e safety coefficients an Select the ir automatically f	sary data related nd the levels of th ectly the value of mportance facto filled.	to the seismic re le seismic loads ap coefficient "a".	gion, the so plication. fficient "γι'
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 tk the "Seismic Areas Importance Zone II v i 1	u enter all the necess e safety coefficients an so button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled.	to the seismic re te seismic loads ap coefficient "a".	gion, the soi plication. fficient "γι'
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 tk the "Seismic Areas Importance Zone II v Yi 1	u enter all the necess e safety coefficients an safety coefficients and Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled.	to the seismic re e seismic loads ap coefficient "a". or and the coe	gion, the soi plication. fficient "γι'
Seismic Triangue this dialog box, yo portance factor, the cortance factor, the seismic Areas seismic Area Seismic Areas ne II a 0.24 ck the "Seismic Area Importance Importance Zone II Yi 1 characteristic Periods Importance Importance	u enter all the necess e safety coefficients an so button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled.	to the seismic re te seismic loads ap coefficient "a".	gion, the soi plication. fficient "γι'
Seismic Triangu This dialog box, yo portance factor, the dismic Area Seismic Area Seismic Area Ne II a 0.24 tk the "Seismic Area Importance Zone II Yi 1 haracteristic Periods Spectrum Type	u enter all the necess e safety coefficients an "g as" button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled.	to the seismic re e seismic loads ap coefficient "a". or and the coe	gion, the soi plication. fficient "γι'
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 Ck the "Seismic Area Importance Zone II v i 1 Characteristic Periods Spectrum Type	u enter all the necess e safety coefficients an "g as" button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled. Next define the so that the	to the seismic re te seismic loads ap coefficient "a". or and the coe e Spectrum Type a horizontal and ve	gion, the soi plication. fficient "γι' and the soil t ertical spect
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 Ck the "Seismic Areas Importance Zone II v Yi 1 Characteristic Periods Spectrum Type Type 1 v S,aw	u enter all the necess e safety coefficients an "g as" button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled. Next define the so that the l coefficients are	to the seismic re e seismic loads ap coefficient "a". or and the coe e Spectrum Type a horizontal and ve	gion, the soi plication. fficient "γι' and the soil t ertical spect culated.
Seismic Triangu this dialog box, yo portance factor, the eismic Area Seismic Areas ne II v a 0.24 Ck the "Seismic Area Importance Zone II v vi 1 characteristic Periods Spectrum Type Type 1 v S,av Soil TR(4)	u enter all the necess e safety coefficients an "g as" button or type dire Select the in automatically f	sary data related nd the levels of th ectly the value of mportance facto filled. Next define the so that the l coefficients are	to the seismic re e seismic loads ap coefficient "a". or and the coe e Spectrum Type a horizontal and ve e automatically cale	gion, the soi plication. fficient "γι' and the soil t ertical spect culated.
Seismic Triangu this dialog box, yoo portance factor, the eismic Area Seismic Areas ne II v a 0.24 ck the "Seismic Areas Importance Zone II v Yi 1 haracteristic Periods Spectrum Type Type 1 v S,au Soil TB(S	u enter all the necess e safety coefficients an "g as" button or type dire Select the ir automatically f	sary data related nd the levels of th ectly the value of mportance facto filled. Next define the so that the l coefficients are	to the seismic re e seismic loads ap coefficient "a". or and the coe e Spectrum Type a horizontal and ve e automatically cale	gion, the soi plication. fficient "γι' and the soil t ertical spect culated. <i>Ids and fill in</i>
Seismic Triangu this dialog box, yoo portance factor, the eismic Area Seismic Areas ne II v a 0.24 tk the "Seismic Areas Importance Zone II v Yi 1 characteristic Periods Spectrum Type ype 1 v S,av Soil TB(s A v TC(s)	u enter all the necess e safety coefficients an "g as" button or type dire Select the in automatically f Horizontal Vertical (g 1 0.9 5) 0.15 0.05 5) 0.4 0.15	sary data related and the levels of the ectly the value of mportance factor filled. Next define the so that the l coefficients are A You can mod very own parage	to the seismic re e seismic loads ap coefficient "a". or and the coe e Spectrum Type a horizontal and ve e automatically calc lify any of these fiel meters set	gion, the soi plication. fficient "γι' and the soil t ertical spect culated. lds and fill in



Select the "Spectrum Type" and the "Ductility Class" before you click "Update Spectrum"
Spectrum
Response Spectrum Design V Ductility Class DCM V
ζ(%) 5 Horizontal b0 2.5 Vertical b0 3
Response Spectrum Update Spectrum Sd(T) >= 0.2 a*g
Structural Type
Select the "Structural Type"
A) Select the "Structural Type" along X and Z direction to calculate the basic eigenperiod
Structural Type
T1 according to 4.3.3.2.2.(5)
X Steel Moment Resisting Frames V 2 Steel Moment Resisting Frames V
(in case of structures with a single frame along X or/and Z direction check the respective checkbox
on the "Bays" group)
Otherwise B) activate the checkbox to calculate the T1 by the paragraph 4.3.3.2.2. of the EC8 regardless of the structural type
Structural Type
✓ T1 according to 4.3.3.2.2.(5)
X Concrete Moment Resisting Frames V Z Concrete Moment Resisting Frames V
Structural Type
X Moment resisiting frames type a Z Moment resisiting frames type a
Select the "Structural type" per direction from the list:
Moment resisiting frames type a Moment resisiting frames type b Moment resisiting frames type c Frames with concentric diagonal bracings Frames with concentric V-bracings Frames with eccentric bracings Inverted Pendulum type a Inverted Pendulum type b Moment resisting frames with diagonal bracin
Moment resisting frames with infills (in contac Moment resisting frames with infills (without c



According to Eurocode the **"Behavior Coefficient q"** derives from calculations and the **"Structural type"** must follow specific criteria.

- ▲ SCADA Pro calculates automatically the q factor and the structural type. The process is explained next:
- After having completed all the previously mentioned values, leave the following boxes blank

q			
qx 🗌	3.5	qy 🗌 3.5	qz 🗌 3.5

Choose "**Ok**" and using the "**Automatic procedure**" run an initial analysis.

Seismi	ic Actions Calculation - An	alysis - Checks				\times				
	Parameters	Mass Centers	(cm)			\sim				
	Automatic Procedure	Level	х	Y	Z	^				
Proce	edure	0 - 0.00	0.00	0.00	0.00					
~	Mass - Stiffness	1 - 380.00	534.78	380.00	541.32					
V	Regularity	2 - 670.00	555.05	670.00	461.75					
	In Plan	3 - 950.00	446.99	950.00	554.88					
	In Elevation									
	Equivalent									
\checkmark	Analysis	ļ				¥				
	Initialize data Exit									

- Now, the proposed values for the "Behavior coefficient q" can be found in the "Parameters" dialog box.
- The proposed values may be kept or altered (the latter one is an option that could be utilized from the beginning of the procedure, however, in this occasion the software would not propose any values by EC8).

q			_				
qx	✓	2	qy	✓	1	qz	2



4.3 How to create load combination:

Right after the analysis execution, use the command group "Results", to create the load combinations (for the EC8 checks and the design) and display the results of the analysis:



The "Combinations" command, opens the "Load Groups Combinations" dialog box where you can create your very own combinations or call the predefined combinations that SCADA Pro has.

G 1.35 Q 1.5	γE 1 γE0.3 0.3	γGE 1	ψ2	0.3 Wind - Snow		γQ+Σγψ0Q 1Q+Σψ2Q +Σγψ2Q	Serviceability $\checkmark \Sigma G + Q + \Sigma \Psi 0 Q$ $\checkmark \Sigma G + \Psi 1 Q + \Sigma \Psi 2 Q$ $\checkmark \Sigma G + \Sigma \Psi 2 Q$	Calculation Delete All
	Туре	Direction	LC1	LC2	LC3	LC4	LC5	LC6 L
Scenario			EC-8_Gree 💌	EC-8_Gree 💌	EC-8_Gree 💌	EC-8_Gree	EC-8_Gree	EC-8_Gree 🗾 E
Load Case			1	2	3	4	5	6 5
Load Type			G 💌	Q 💌	ExD 💌	EzD .	Erx 💌	Erz 🗾 E
Actions			-	Category A 💌	-		· •	-
Description								
Comb.:1	Ultimate 👤	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
Comb.:4	Ultimate 🔄	Dir. +X 💌	1.00	0.30	1.00	0.30	1.00	0.30 -
Comb.:5	Ultimate 🔄	Dir. +X 💌	1.00	0.30	1.00	0.30	1.00	-0.30 0
Comb.:6	Ultimate 🔄	Dir. +X 💌	1.00	0.30	1.00	0.30	1.00	-0.30 -
Comb.:7	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	0.30	-1.00	0.30 0
Comb.:8	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	0.30	-1.00	0.30 -
Comb.:9	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	0.30	-1.00	-0.30 0
Comb.:10	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	0.30	-1.00	-0.30 -
Comb.:11	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	-0.30	1.00	-0.30 0
Comb.:12	Ultimate 💌	Dir. +X 💌	1.00	0.30	1.00	-0.30	1.00	-0.30 -
<								:

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.

EC8_General Dynamic (1) · Active Scenario

The default combinations of the executed analysis, are automatically saved by the program.

÷



G 1.35 Q 1.5	γE 1 γE0.3 0.3	3	γGE	1	Ψ	2	0.3 Wind - Snow		Ultimate ΣγG+γ ΣG+ψ ΣG+E-	γQ+Σγψ0Q 1Q+Σψ2Q +Σγψ2Q		erviceability 2 ΣG +Q +Σψ0Q 2 ΣG +ψ 1Q +Σψ 2 ΣG +Σψ2Q	! 12Q	Calcu Dele	lati	on All
	Туре		Direction		LC1		LC2	LC3		LC4		LC5		LC6	Т	LC
Scenario					EC-8_Gree	•	EC-8_Gree 💌	EC-8	Gree 💌	EC-8_Gree	•	EC-8_Gree	•	EC-8_Gree	-	EC
Load Case					1		2	3		4		5		6		5
load Type					G	•	Q 💌	ExD	-	EzD	•	Erx	•	Erz	-	Ey
Actions						•	Category A 💌		-		•		•		-	
Description																
Comb.:1	Ultimate	-	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

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. 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.

The **automatic** mode requires that the automatic procedure for the calculation and distribution of the loads of wind and snow as well as the automatic creation of the loads and combinations (as in current example) is already done.



Arameters Edit View Member Post-Processor Correspondence Wind - Snow Loads Ad Case Definition X Self-weight Dead Load V Insert LC S.W. Description I Yes Dead Load V Insert bel From Lis Pelete Loads Delete All Loads Delete All Loads	Load Attrik Wind Cpe_p-Cpi Cpe_n-Cpi Cpe_n-Cpi	bution 0 90 3 7 4 8 3 5 9 6 10 Total Los Load Attributio	180 270 11 15 12 16 13 17 14 18 d Deletion (Snow-Winc	Snow Typi Case i 19 Case ii 20 Case ii 21 Loads)	cal Acci- dental	
arameters Edit View Member Correspondence Wind - Snow Loads ad Case Definition × Self-weight Dead Load v Insert LC S.W. Description kell From Lis 1 Yes Dead Load v Insert 1 Yes Dead Load v Delete Loads No Wind 0 Cpe_p-Cpi 5 No Wind 0 Cpe_n-Cpi 6 No Wind 0 Cpe_n-Cpi	Wind Cpe_p+Cp Cpe_p-Cpi Cpe_n+Cp Cpe_n-Cp	0 90 pi 3 7 i 4 8 pi 5 9 pi 6 10 Total Load	180 270 11 15 12 16 13 17 14 18 d Deletion (Snow-Winc	Snow Typi Case i 19 Case ii 20 Case ii 21 ILoads)	Accidental	
Add Case Definition Self-weight Dead Load LC S.W. Description 1 Yes Dead Load 2 No Live Load 3 No Wind 0 Cpe_p-Cpi 4 No Wind 0 Cpe_p-Cpi 5 No Wind 0 Cpe_n-Cpi 5 No Wind 0 Cpe_n-Cpi 6 No Wind 0 Cpe_n-Cpi 5 No Wind 0 Cpe_n-Cpi 6 No Wind 0 Cpe_n-Cpi 6 No Wind 0 Cpe_n-Cpi 7 No Wind 0 Cpe_n-Cpi	Cpe_p+Cp Cpe_p-Cpi Cpe_n+Cp Cpe_n-Cp	pi 3 7 i 4 8 pi 5 9 pi 6 10 Total Load Attribution	11 15 12 16 13 17 14 18 d Deletion (Snow-Wind	Typi Case i 19 Case ii 20 Case ii 21	Acci- dental 22 23 24	
Wind - Snow Loads Ad Case Definition Self-weight Dead Load ✓	Cpe_p-Cpi	i 4 8 oi 5 9 oi 6 10 Total Loa Load Attributio	12 16 13 17 14 18	Case i 19 Case ii 20 Case iii 21 Loads)	22 23 24	
ad Case Definition X Self-weight Dead Load V Insert LC S.W. Description A 1 Yes Dead Load Delete Loads 2 No Live Load Delete Loads 4 No Wind 0 Cpe_p+Cpi 5 No Wind 0 Cpe_n+Cpi 6 No Wind 0 Cpe_n-Cpi	Cpe_n+Cp Cpe_n-Cp Scenarios	pi 5 9 ji 6 10 Total Loa Load Attributio	d Deletion (Snow-Wind	Case ii 20 Case iii 21 I Loads)	23	
Ad Case Definition × Self-weight Dead Load v Insert LC S.W. Description hel From Lis 1 Yes Dead Load Decemper Cpi 2 No Live Load Decemper Cpi 4 No Wind 0 Cpe_p-Cpi 5 No Wind 0 Cpe_n-Cpi	Cpe_n-Cp	pi 6 10 Total Loa Load Attributio	d Deletion (Snow-Wind	Case iii 21	. 24	
ad Case Definition X Self-weight Dead Load Insert LC S.W. Description Nel From List 1 Yes Dead Load Dead Load 2 No Live Load Delete Loads 3 No Wind 0 Cpe_p-Cpi Delete Loads 4 No Wind 0 Cpe_n-Cpi Delete All Loads 5 No Wind 0 Cpe_n-Cpi Delete All Loads	Scenarios	Total Loa	d Deletion (Snow-Wind	l Loads)		
Self-weight Dead Load Insert LC S.W. Description A 1 Yes Dead Load A 2 No Live Load B 3 No Wind 0 Cpe_p+Cpi B 4 No Wind 0 Cpe_n-Cpi B 5 No Wind 0 Cpe_n-Cpi Delete All Loads	Scenarios	Total Loa Load Attributio	d Deletion (Snow-Wind	l Loads)		
LC S.W. Description Intert 1 Yes Dead Load Intert 2 No Live Load Intert 3 No Wind 0 Cpe_p+Cpi Delete 4 No Wind 0 Cpe_n-Cpi Delete All 5 No Wind 0 Cpe_n-Cpi Delete All	Scenarios	Load Attributio	n in Members(from Wir			
LC S.W. Description Pel From Lis 1 Yes Dead Load Delete 2 No Live Load Delete 3 No Wind 0 Cpe_p+Cpi Delete Loads 4 No Wind 0 Cpe_p-Cpi Delete All Loads 5 No Wind 0 Cpe_n-Cpi Delete All Loads	Scenarios		nin Henbers(nom wi	nd and Snow)		
1 Yes Dead Load Delete 2 No Live Load Delete 3 No Wind 0 Cpe_p+Cpi Loads 4 No Wind 0 Cpe_p-Cpi Delete Loads 5 No Wind 0 Cpe_n+Cpi Delete All Loads 6 No Wind 0 Cpe_n-Cpi Delete All Loads	Scenarios					
2 No Live Load Delete 3 No Wind 0 Cpe_p+Cpi Loads 4 No Wind 0 Cpe_p-Cpi Delete Loads 5 No Wind 0 Cpe_n+Cpi Delete Loads 6 No Wind 0 Cpe_n-Cpi Delete All Loads	Wind 0					
3 No Wind 0 Cpe_p+Cpi Loads 4 No Wind 0 Cpe_p-Cpi Delete All 5 No Wind 0 Cpe_n+Cpi Loads 6 No Wind 0 Cpe_n-Cpi Loads	Vind 9) Sta	tic Wind 0 (2) 🗸 🗸	f	Results	
4 No Wind 0 Cpc_o-cpi Delete All 5 No Wind 0 Cpc_n+Cpi Loads 6 No Wind 0 Cpc_n-Cpi Loads	e wind s	90 Sta	tic Wind 90 (3) $ \sim $			
6 No Wind 0 Cpe_n-Cpi	Wind 1	180 Sta	tic Wind 180 (+ \sim			
	Wind 2	270 Sta	tic Wind 270 (! 🗸			
7 No Wind 90 Cpe p+Cpi	Typica 🗹	Snow Sta	tic Typical sno v			
C > OK	Accide	ntal Snow Ne	v Scenario 🗸 🗸 🗸 🗸 🗸 🗸 🗸			
		Analysis Scenario	Creation			
				L	Cancel	
mbinations by using the command Wind - ter running the seismic scenario and all the	Snow .	cenarios	of wind	and sn	ow, act	ivat
smic scenario and choose the command "	'Comhin	ations"	The com	ninatio	ns of t	he a
smic scenarios are completed automatically	y. To crea	ate auto	matically	the co	mbinati	ons
Wind - Snow					п •	
nd and show loads, press	Automati	ically the	coefficier	its of a	li wind a	and
enarios will be filled, offering a complete loa	ds comb	inations	file.			
Groups Combinations					×	
	Ultimate	-	Serviceability			
1.35 γE 1 Ψ2 0.3	V ΣγG-	+γQ+Σγψ0Q				
15 500 02		0.00	ZG+Q+Σψ0Q		Calculation	
Vind - Snow	∕ ΣG+	ψ1Q+Σψ2Q Ε+Σγψ2Q	✓ ΣG+Q+Σψ0Q ✓ ΣG+ψ1Q+Σψ2 ✓ ΣG+Σψ2Q	Q	Calculation Delete All	
Vind - Snow		ψ1Q+Σψ2Q E+Σγψ2Q	∑G+Q+Σψ0Q ∑G+ψ1Q+Σψ2 ∑G+Σψ2Q		Calculation Delete All	
L.5 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 LC8 LC9 LC10 LC11	LC12	ψ1Q+Σψ2Q E+Σγψ2Q LC13		LC15	Calculation Delete All	,
L.5 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 :enario Static Win \$ Static Win <	LC12	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win		Q LC15 Static Win	Calculation Delete All LC	
LC8 LC9 LC10 LC11 senario Static Win Static Win Static Win Static Win ad Case 1 2 3 4	LC12 Static Win 2	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2	∑ ΣG+Q+Σψ0Q ∑ ΣG+ψ1Q+Σψ2 ∑ SG+Σψ2Q LC14 ✓ Static Win 3	Q LC15 Static Wir 4	Calculation Delete All LC ^ n Sta 1	· ·
L.5 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 senario Static Win Y Static Win Y sad Case 1 2 3 4 ad Type NULL Y NULL Y NULL Y	LC12 Static Win	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2 ✓ NULL	ΣG+Q+Σψ0Q ΣG+Q+Σψ0Q ΣG+ψ1Q+Σψ2 LC14 Static Win 3 NULL NULL	Q LC15 Static Wir 4	Calculation Delete All LC ^ 1 V Sta 1 V NL	
LC8 LC9 LC10 LC11 icenario Static Win \$ Static Wi	Z 2 44 Z 2 4	ψ1Q+Σψ2Q E+Σγψ2Q Cl3 Static Win 2 NULL Wind (0.60	ΣG+Q+Σψ0Q ΣG+Q+Σψ0Q ΣG+ψ1Q+Σψ2 LC14 Static Win 3 NULL Wind (0.60	Q LC15 Static Win 4 NULL Wind (0.6	Calculation Delete All LC ^ 1 V NL 0 V Wi	· ·
LC8 LC9 LC10 LC11 tenario Static Win \$ Static Win	LC12 Static Win 1 NULL	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2 ✓ NULL ✓ Wind (0.60	 ∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+∑ψ2Q LC14 Static Win 3 NULL Wind (0.60 	LC15 LC15 Static Wir 4 NULL Wind (0.6	Calculation Delete All	
LC8 LC9 LC10 LC11 ienario Static Win \$ Static Win	LC12 Static Win 2 NULL Wind (0.60	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2 ✓ NULL ✓ Wind (0.60	ΣG+Q+Σψ0Q ΣG+Q+Σψ0Q ΣG+ψ1Q+Σψ2 LC14 Static Win 3 NULL Wind (0.60	LC15 Static Win 4 NULL Wind (0.6	Calculation Delete All LC ^ n Sta 1	
LC8 LC9 LC10 LC11 ienario Static Win \$ Static Win	✓ ZG +4 ✓ ZG +4 ✓ ZG +4 ✓ ZG +4 ✓ XG +4 ✓ XULL ✓ ✓ Mind (0.60 2	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Z Static Win 2 NULL Wind (0.60	ΣG+Q+Σψ0Q ΣG+Q+Σψ0Q ΣG+ψ1Q+Σψ2 LC14 Static Win 3 NULL Wind (0.60	Q LC15 Static Win 4 NULL Wind (0.6	Calculation Delete All LC ^ 1 V NL 0 V Wi Vi	
L.5 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 senario Static Win Static Win.	LC12 Static Win 1 NULL Wind (0.60	ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+ΣųQ+Σψ2 ∑ SG+Q+Q2Q LC14 ▼ Static Win 3 ▼ NULL ▼ Wind (0.60	Q LC15 Static Win 4 NULL Wind (0.6	Calculation Delete All LC ^ Sta NU Wi Wi	
L-3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 senario Static Win	LC12 Static Win 1 NULL _ Wind (0.60	ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60 Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ2 ∑ SG+Q+Q+Σψ2 LC14 ▼ Static Win 3 ▼ NULL ▼ Wind (0.60	Q LC15 Static Win 4 NULL Wind (0.6	Calculation Delete All LC ^ Sta NU Wi NU U U U U U U U U U U U U U	
L.5 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Static Win.	LC12 Static Win 1 NULL _ Wind (0.60	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ2 ∑ SG+Q+Q+Σψ2 LC14 ▼ Static Win 3 ▼ NULL ↓ Wind (0.60	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC ^ Sta NU Wi Wi Sta	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 static Win Static Win Static Win Static Win Static Win bad Case 1 2 3 4 bad Type NULL NULL NULL NULL Vind (0.60 ctions Wind (0.60 Wind (0.60 Wind (0.60 Wind (0.60 Wind (0.60 omb.1	V V	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ2Q ∑ SG+Q+1Q+Σψ2 LC14 ▼ Static Win 3 ▼ NULL ▼ Wind (0.60	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC ^ Sta NU Wi NU NU NU NU NU	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Wind (0.60 Wind (0.60 Wind (0.60 Static Win Static Win<	V 20 + 4 Z 55 + 4 Z 55 + 4 Static Win _ 1 NULL Wind (0.60 _	ψ1Q+Σψ2Q E+Σγψ2Q LC13 ▼ Static Win 2 ▼ NULL ▼ Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ0Q ∑ SG+Q+Σψ2Q LC14 Static Win 3 NULL Wind (0.60	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC ^ Sta NU VI VI VI VI VI VI VI	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Wind (0.60 Wind (0.60 Static Win Static Win </td <td>Image: Wind Wind Wind Wind Wind Wind Wind Wind</td> <td>Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60</td> <td>∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ0Q ∑ SG+Q+Σψ2Q LC14 Static Win 3 NULL Wind (0.60</td> <td>Q LC15 Static Wind NULL Wind (0.6</td> <td>Calculation Delete All LC ^ Sta NU V NU NU V</td> <td></td>	Image: Wind Wind Wind Wind Wind Wind Wind Wind	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ0Q ∑ SG+Q+Σψ2Q LC14 Static Win 3 NULL Wind (0.60	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC ^ Sta NU V NU NU V	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win	Image: Wind (0.60 0.90	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60 V N00 0.90	∑ ΣG+Q+Σψ0Q ∑ SG+Q+Σψ0Q ∑ SG+Q+Σψ2Q LC14 Static Win 3 NULL Wind (0.60	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC ^ n V Sta 1 V NL 0 V Wi 0 V i	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Wind (0.60 Wind (0.60 Static Win Static Win<	Image: Constraint of the second sec	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60 Wind (0.60 0.90	∑G+Q+Σψ0Q ∑G+Q+Σψ0Q ∑G+Q+QQ ∑G+Zψ2Q LC14 Static Win 3 Wind (0.60 Wind (0.60 0 0 0.90	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win	Comparison of the second	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Static Win 2 NULL Wind (0.60 Wind (0.60 0 0 0 0 0.90	∑G+Q+Σψ0Q ∑G+Q+ΣψQ ∑G+Q+QQ LC14 Static Win 3 NULL Wind (0.60 U 0.90	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Static Win </td <td>Image: Constraint of the second sec</td> <td>ψ 1Q+Σψ2Q E+Σγψ2Q LC13 × Static Win 2 × NULL × Wind (0.60 0 0 0 0 0 0 0 0.90</td> <td>∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+Zψ2Q LC14 3 NULL Wind (0.60 0 0.90</td> <td>Q LC15 Static Wind NULL Wind (0.6</td> <td>Calculation Delete All LC</td> <td></td>	Image: Constraint of the second sec	ψ 1Q+Σψ2Q E+Σγψ2Q LC13 × Static Win 2 × NULL × Wind (0.60 0 0 0 0 0 0 0 0.90	∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+Zψ2Q LC14 3 NULL Wind (0.60 0 0.90	Q LC15 Static Wind NULL Wind (0.6	Calculation Delete All LC	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win	Image: Constraint of the second sec	ψ 1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2 ✓ NULL ✓ Wind (0.60 Ø	∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+Zψ2Q LC14 3 NULL Wind (0.60 0.90	Q LC15 Static Wind VILL Wind (0.6 Wind (0.6 U U U U U U U U U U U U U U U U U U U	Calculation Delete All LC I I I I I I I I I I I I I I I I I I	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win Static Win <td>Image: Constraint of the second sec</td> <td>Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Z Static Win 2 NULL Wind (0.60 0 0.90 0.90</td> <td>∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+Zψ2Q LC14 Static Win 3 NULL Wind (0.60 0.90</td> <td>Q LC15 Static Wind NULL Wind (0.6 Wind (0.6 0.90</td> <td>Calculation Delete All LC</td> <td></td>	Image: Constraint of the second sec	Ψ1Q+Σψ2Q E+Σγψ2Q LC13 Z Static Win 2 NULL Wind (0.60 0 0.90 0.90	∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 ∑G+Zψ2Q LC14 Static Win 3 NULL Wind (0.60 0.90	Q LC15 Static Wind NULL Wind (0.6 Wind (0.6 0.90	Calculation Delete All LC	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win	Image: Constraint of the second sec	ψ 1Q+Σψ2Q E+Σγψ2Q LC13 ✓ Static Win 2 ✓ NULL ✓ Wind (0.60 0 0 0 0.90	∑G+Q+∑ψ0Q ∑G+Q+∑ψ0Q ∑G+Q+∑ψ2 LC14 Static Win 3 NULL Wind (0.60 0.90	Q LC15 Static Wind VILL Wind (0.6 Wind (0.6 U U U U U U U U U U U U U U U U U U U	Calculation Delete All LC I LC Vi Vi Vi O O O O O O O O O O O O O	
L.3 VED.3 0.3 Wind - Snow LC8 LC9 LC10 LC11 cenario Static Win	Default Co	ψ 1Q+Σψ2Q E+Σγψ2Q LC13 Y Static Win 2 NULL V Wind (0.60 0 0 0 0 0 0.90	∑G+Q+∑ψ0Q ∑G+Q+∑ψ2Q ∑G+Zψ2Q LC14 Static Win 3 NULL Wind (0.60 0 0.90	Q LC15 Static Wind VILL Wind (0.6 Wind (0.6 U U U U U U U U U U U U U U U U U U U	Calculation Delete All LC Sta I VINU U U U U U U U U U U U U U U U U U U	



5. POST-PROCESSOR

5.1 How to view diagrams and the deformed shapes results:

Activate "Post-Processor" to view the deformed shapes of the model for each load case or/and combination scaled accordingly and see the M, V, N diagrams for each member as well.

6		- 🎁 🕒	0.00	-		Ŧ			
		Basic		Modeling	View	То	ols	Slabs	Loads
Í	-	1	Diag	rams-Stress C	Contours		-		
	Combir	nations					D	eformed /	Animation
l				Deform	ation Dia	grams			

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

Load Combir	.oad Combinations X								
C: MELETST	TEEL \3DSTEEL 1b \	scaana							
Load	26								
Load	772								
ALL.cmb		O [×]							
	Combinations Se	lect							
	Calculation								
	Beam 234 /42	8							
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".

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 required results, select:

- Model or
- Diagrams Stress-Contours









46



6. STEEL MEMBER DESIGN

After having completed the analysis, checked the results and the deformations the section design follows.

6.1 How to create design scenarios:

Go to "Members Design" section and click "New" to create the scenario by choosing desired regulation.

Scenario			\times
1	Name	1	
	Туре	EKOS 2000-EAK	\sim
	New	EKOS 2000-EAK EC2-EC3	
	Design De	NTC_2008 EC2_Italia EC2_Cyprus Greek old 1959-84 Greek old 1984-93	
Exit	Steel	Austria SBC304 EC5	

Type a name, select a type and click New to fill in the Scenarios list.

1 For this example, a Eurocode scenario was used.

Comment: For steel structures the EC3 is applied to every scenario. EC2 regards the analysis method as well as the design method of concrete cross sections.

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

Design Delete	
Concrete	Connections
🗖 Steel	Apply



6.2 How to define the parameters of the steel members design:

-		-	
Z,	EC2-EC3 1 (0) 🔹	6	
New	Active Scenario	Para- meters	Merge Elements *
	Scenarios		

Select from the Scenarios list the scenario that you want to use for the design.

Steel Reinforce	ment	Capaci	ty Design		Steel	Timbe	r struct	ures		
Combinations		Slabs	Beam	s	Columns		Foot	ngs		
Combinations of L	oad Sets	(772)	Ult. Se	erv.	+XX	+Z	Z	No		
Combinations						ULS/SL	S Dir.	^		
1(5) +1.35Lc1+	1.50Lc2					ULS				
2(1) +1.35Lc1+	1.50Lc2+0	.90Lc8				ULS				
3(2) +1.35Lc1+	1.50Lc2+0	.90Lc9				ULS				
4(2) +1.35Lc1+	1.50Lc2+0	.90Lc10				ULS				
5(2) +1.35Lc1+	1.50Lc2+0	.90Lc11				ULS				
6(2) +1.35Lc1+	1.50Lc2+0	.90Lc12				ULS				
7(2) +1.35Lc1+	1.50Lc2+0	.90Lc13				ULS				
8(2) +1.35Lc1+	1.50Lc2+0	.90Lc14				ULS				
9(2) +1.35Lc1+	1.50Lc2+0	.90Lc15				ULS				
10(2) +1.35Lc1+	+1.50Lc2+	0.90Lc16				ULS		×		
<								>		
Level Multipliers		1/	(1-θ)					~		
Level	х	Y	Z		Insert Co	ombinatio	ns			
0 - 0.00	1.000	1.000	1.000		Combinatio	ns Calcula	ation			
1 - 487.00	1.000	1.000	1.000							
2 - 723.00	1.000	1.000	1.000		-		101	7		
					Au	+ψ2Q tomatic D	esign			
Save		Load				ОК	C	ancel		
eauisite for tl	ne desig	n is the o	calculatio	on of t	he load co	ombina	ations			
selection of t	he .cmb	, file that	was stor	ed af	ter the an	alvsis i	s perf	orm	ed eith	her:
		default	.cmb]			
		EC-8_0	Greek Dyna	mic (1)	.cmb					
selecting from	m the l	ist total.cn	nb				whi	ch a	utoma	atically per
alculation										
through the		Insert	Combinatio	ons		ommar	nd. w	hich	is loc	cated insid
ect folder, yo	u select	from the	e existing	ones	the desire	ed com	binat	tion	file acc	cording to
design will be	perfor	med and	next clicl	k the	Co	mbinatio	ns Cal	culatio	on	butto



In this example, the combination file of the dynamic analysis, including wind and snow will be used.

The **Plates, Beams, Columns, Footings, Steel Reinforcement** tabs, include the parameters that affect concrete sections.

For steel structures, to define the design parameters of the steel sections, select the "Steel" field.

The dialog box that opens is divided in two areas: on the left there is a list with all the layers and on the right there is the checks list including all of the respective parameters.

First select a layer. Click one from the list, or more using "ctrl", or all using "Select All".

(By pressing the button "Deselect All" cancel the previous layers' selection.)

Then activate one or more design checks with a tick on the corresponding checkbox and press the corresponding button to specify the parameters.

The parameters defined for one layer can be copied to other layers, using the command "Copy". Select a layer \rightarrow define the parameters \rightarrow press "Copy" \rightarrow select another layer \rightarrow press "Paste".

ctural Component Pa	rameters					
Combinations	Slabs	Beams		Columns	F	ootings
Steel Reinforcement	Capaci	ty Design	5	Steel	Timber str	uctures
Name		^	S	elect All	Dese	lect All
Lines circles Concrete Columns				Сору	Pa	iste
Concrete Jackets			Param	eters		
Concrete Beams Concrete Foundation B	leams		\Box	G	ENERAL	
Footing Connection Be	ams			T	ENSION	
Steel_Columns					SHEAR	
Steel_Beams Surface Mesh				Т	ORSION	
Mathematical Model				CON	IPRESSION	
Mesh 3D				F	ENDING	
Mesh 2D						
Slabs-Strips				BENDING	& AXIAL FOR	RCE
Steel Columns			\Box	BENDING	& SHEAR FO	RCE
Main Beams				BENDING SH	IEAR AXIAL	FORCE
Purlins					Dofault	
Girders		~			Derault	
Save	Load				ОК	Cancel



The definition of the design parameters for steel sections is performed per layer. First you select the layer of which the parameters are to be defined, (for example Steel Columns) and for each check category (General, Tension, Shear etc.), you set the respective parameters. As soon as you defined the parameters for one layer, the program gives you the ability to copy these parameters to another layer using the Copy - Paste commands.

Suppose you have set all parameters for the layer Steel Columns and you want to pass these parameters to Steel Beams. Activate the check box next to "Default" and automatically all parameters become selected. Then press "Copy", select layer Steel Beams and press "Paste" (which is now activated). Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams.

An alternative method to set the same parameters to all layer including steel sections, is selecting all layer pressing "Select all" button and set once the parameters for each check category. Note that at least one (or more) layer should be selected to set parameters.

Next, all the parameters for each category is analytically explained.

 By clicking the button "GENERA 	AL" the following dialog box opens:
GENERAL	to set the γ_{Mi} safety factors:
General Parameters × Safety Factors	γ_{M0} : partial factor for resistance of cross-sections whatever the class is γ_{M1} : partial factor for resistance of members to instability assessed by member checks γ_{M2} : partial factor for resistance of cross-sections in tension to fracture In the "Limit of Internal" field define an upper limit. Under this value the program will not consider the corresponding stress resultants. These values are recommended by Eurocode.
OK Cancel	



"TENSION"

 $\mathbf{\nabla}$

TENSION

to define the parameters that correspond to the shear design check as well as the position of the hole check (EC3 §1.8 §3.5):

ension Parameters				×
Holes No In Web only	◯ Web and Flange	OL	Safety Factor	OK Cancel
Bolt Holes Geometry Web Holes Diameter (mm) 0 Number of Bolt Rows (perp. to the force pic. 1) P2 P2 P2 P1 P2 P1 P1 Distant e1 p1 0	$\begin{bmatrix} e_{2} \\ p_{2} \\ e_{1}p_{1} \\ e_{1}p_{1} \\ e_{1}p_{1} \\ e_{2} \\ e_{1}p_{1} \\ e_{1}p_{1} \\ e_{2} \\ e_{1}p_{1} \\ e_{1}$	Flange Holes Dia Number of the force p P2 P2 P2 P2 P2 P2 P2 P2 P2 P2 P2 P2 P2	imeter (mm) 0 Boh Rows (perp. to 0 p1 0 0 P1 0 0 Distance between the holp p1 e2 p2	es (mm)
Section L EZ e101	lles Diameter (mm)	Number of Bol	t Rows (parallel to the force)	0

Specify the spacing of the centers of two consecutive holes, the diameter of the hole and the number of rows of bolt holes.

In case of L section specify the parameters on the bottom of the dialog box in the field "Section L".

Here the user defines whether to consider the reduction of the tensile strength of the section due to the bolt holes of the connections or not. The data in the fields of the dialog box are derived from the design checks of the connections. For that reason the verification of the connections must be preceded.

The safety factor for all design checks is fixed and equal to one, which means that the program calculates the ratio of the stress resultant versus the resistance. A value of the calculated ratio greater than 1.0 indicates failure.



"SHEAR"			
	SHEAR	1	
Shear Para	meters	×	H
Safety Factor	1		w
Stiffeners			sp
Dist between Stiffeners (m) 30		no
Digid			
		_	
	7/111,		
O Non Rigid			
d			
	nt in ,		
ОК	Cance		
"TORSIO	N "		
	TORS	SION	
Torsion Paramete	ers	;	×
Safety Factor		1]
Torsional Mome	nt	L	1
No	◯ Distri ◯ Conc	ibuted centrated	
Distance from St	tart (cm)	0	
Distance from Er	0		
Valu	ie (KNm)	0	
Element's Length	(cm)	300]

Support Conditions

Type

OK

0

Cancel

Here define if the elements of the selected layer contain stiffeners and what type; web stiffeners or intermediate stiffeners. Also define the spacing between the stiffeners and the type of the connection (rigid or not rigid).

Here you define whether the structural elements of the selected layer are loaded by a distributed or concentrated torsional moment, or not. If yes you define the load data. You also define the support conditions based on the corresponding figures.



For all design checks presented in the figure on the left, define the "**Safety Factor**" in the dialog box that appears when you click one of the five buttons. The safety factor is the ratio of the resistance value versus the corresponding design value, which is set 1.0 by default.

	Parameters ×
BENDING	Safety Factor
BENDING & AXIAL FORCE	
BENDING & SHEAR FORCE	OK Cancel
BENDING SHEAR AXIAL FORCE	
6.3 Steel members de	esign:
Steel Timber Design Design Design Diag Cross-Section Design Buckling Members Input Connections	"Steel Design" command group contains commands for the cross-sections design, the buckling resistance and the steel connections design.
Merge Elements	
EC2-EC3 1 (0) New ' Active Scenario Par met	ra- ters Eleme ts *
LScenarios	

In the new version of the program, added a new command group, which concerns merging of steel (concrete and timber) members for the calculation and buckling and deformation checks display according EC3.

IMPORTANT NOTES:

▲ Using this command, is now possible to define correctly, the initial length of the member per

Buckling Members Input

direction to be taken into account in the buckling checks.



1	Until	now,	this	condition	was	considered	defining	the	length	coefficients	(see
	/	Bucklin	ng Men	nbers Input							
D	✓ Later irection ` Member	al Bucklin Y 's Length	ig n		D	irection Z Member's Lengtl	h				
	○ Real ● Coef	ficient	1			○ Real ● Coefficient	1				

- Now, using merging per direction, there is no need for the coefficient process, and merging will be, in most cases, automatically.
- Also, note that with the merging process, the buckling length, is calculated correctly, and in the print outs of the results a merged element is printed once with annotation of the individual members that contains.
- Basic concepts of buckling along major and minor axies you can find in the User's Manual Chapter "9. Members Design"

NOTE:

Generally, making a rule, we could say that, we consider the **merged length Ly** in the direction where the local axis y-y is parallel to the supporting elements. While in the other direction, if no supporting elements, **Lz** is **the length of each member**.

Press Merge Elements command and then Auto:



Merge elements means that, either automatically or manually, the individual parts of a single element, merge in each buckling direction.

Meaning that, the buckling length is considered computationally, not the actual length of the element, but the unified from the beginning to the end of the column or beam, respectively.

Also, in the presentation of the results, for these merged elements, the worst results display only once and not for each part, as it was so far.

Finally, in automatic merging, there is the definition of discontinuity levels, horizontal or vertical, used as merging boundaries of a continuous element.

Discontinuity levels are levels that are boundaries of beams and columns, used to break merging in each direction.



Auto merge

Using this command displays the following dialog box

Automati	c Merge			×
Layer	Steel Colu	mns		\sim
Columns		~	Calculation	
Discont	inuity Level	S		_
N	lew	View	Delete	
Pick	:// XY	Pick // ZY	Pick	
		Cancel		
		Cancer		

First, choose the layer of the elements to merge.

Just below, specify the type of element contained in the selected layer. The program automatically understands the type of the element: Column if vertical, Beam all the others.

Merged elements display colored:

- Yellow color for the merged elements along y-y local axis
- Cyan color for the merged elements along z-z local axis
- Pink color for the merged elements along both local axes

Press "Calculation" and the program merge the elements of the active layer, based on what was mentioned above.





Discontinuity levels are levels that are boundaries of beams and columns, used to break merging in each direction.

- For the columns, the discontinuity levels are horizontal levels defined by the floor levels.
- For the beams, the discontinuity levels are always vertical levels defined by two points.

Predefined limits:

- For the horizontal levels are: the foundation level and the last level.
- For the beams are: the vertical limits of the model.
- **1** The predefined limits never display in the **discontinuity levels** list.

In this example there are three levels 0.00, 487.00 and 725.00 in **discontinuity levels** list of the columns, only the level 487.00 will be specified by default (that is, only the intermediate level without the limits) considering that, the columns merging will be interrupt at 487.00 cm. The column will merge from 0.00 to 487.00 cm.



Automatic Merge				
Layer Steel Colu	umns	~		
Columns	~	Calculation		
Discontinuity Leve	els			
1 487.00				
New	View	Delete		
Pick // XY	Pick // ZY	Pick		
	Cancel			

Users merging

Select the command, and then point to the start point and the endpoint of the members to merge.

Selecting the second point (endpoint), the following dialog box appears:

Merge	Х
Buckling	
Direction Z	
Direction Y	
Columns	
View	
OK	
Cancel	

where you define the direction of merging for Buckling and Deformations.



6.3.1 Cross Section Design:

The command Cross section design is used to check the adequacy of the steel cross-section.

Select this command to open the following dialog box.

Name	Cross Section 1	Cross Section 2	Cross Section 3	Cross Section 4	Cross Secti
Mesh 3D					
Mesh 2D					
Slabs-Strips					
Steel Columns	HEB 500				
Steel Beams					
Main Beams	SHS 150x8,0				
Purlins	IPE 200				
Girders					
Secondary Columns					
Hor.Wind bracings					
Vert.Wind bracings	SHS 100x5,0				
– Timber Columns					
Timber Beams					
Timber top main beams					
Timber Purlins					
<					>

The checks are performed for all the elements of the selected layer.

The program, for each stress, locates the element with the less favorable value for this stress. The first column contains the layers of the current project and the next columns contain the cross sections of those layers. In this example, the "Steel Columns" layer contains the columns of the structure with the cross section HEB500. Similarly the steel beams belong to the layer "Steel Beams" and the rest of the elements on respective layers.

Choose the command "**Calculate all**" to perform the design checks for each cross section of each layer, for all combinations. Layers with full adequacy of every cross section, will be colored green while the layers that contain at least one cross section with inadequacy will be colored red (which does not mean that all the cross sections of this layer are inadequate necessarily).

Select the layer and click "Edit".

On the dialog form you can view in a tabular format the checking results of the cross sections of the selected layer with colored values.



During the automatic process, the program locates the 12 most critical combinations of every member for each type of stress (Max N and the rest of the stress types, Min N etc.) and performs the design-checks (see Members-Design Manual).

Steel Design	- Layer D	ata	•			0			×	
Layer: Stee	el Column	s	VERIFIC	ATION OK					Capacity Design Amplification	
Different Cro	oss	н	EB 500					\sim	CHECK SELECTION	
Description	Member	Comb	N	Vy	Vz	Mx	Му	Mz	NO Auto N M V Mx M-V M-V-N	
MaxN (kN)	1084	406	2538.13	44.62	-0.38	0.03	-0.35	83.56		
MinN (kN)	1084	431	-2465.56	10.45	-0.76	0.02	-0.40	-66.20		
MaxQY (kN)	1096	208	67.59	98.96	-0.37	-0.07	1.63	126.34		
MinQY (kN)	1083	429	2.21	-98.72	-0.82	-0.02	-0.46	-108.21		
MaxQZ (kN)	1086	151	41.59	22.96	107.51	-0.11	102.69	64.35		
MinQZ (kN)	1076	87	53.32	35.06	-110.48	0.11	-111.06	78.16		
MaxMX (kNm)	1098	200	46.13	54.08	3.33	0.73	-5.93	68.51		
MinMX (kNm)	1098	189	-19.44	-4.51	-4.44	-0.60	-6.11	-3.53		
MaxMY (kNm)	1086	151	41.59	22.96	107.51	-0.11	102.69	64.35		
MinMY (kNm)	1076	87	53.32	35.06	-110.48	0.11	-111.06	78.16		
MaxMZ (kNm)	1084	200	152.10	77.76	-1.47	-0.00	-4.04	149.20		
MinMZ (kNm)	1095	208	149.63	-75.51	-1.15	0.02	-2.45	-150.14		
User			0	0	0	0	0	0		
	For a	ll mem	bers that be	long to this	; GROUP					
ОК		Ca	ncel		Layer De	esign		La	ayer Explorer Calculation Printout	

When your mouse hover a cell, the value will show up. Observing that the adequacy ratios are very small, there is margin for reducing the cross-section.

In Basic unit, select the Multiselect Edit command and the Group filter, and choose the elements of the Main Beams Layer.





In Cross-Section choose a smaller section, SHS100X8.



NOTE:

Now you have to run the analyses again to calculate the new intensive forces.

Re-run the analyses and reload the combinations in the "Parameters" window, otherwise the section design will be performed by the new cross section but with the previously calculated stresses.

6.3.2 Buckling Members Input:

The buckling resistance check is one of the main design checks for steel structural members. Select the command "Buckling Members Input", to apply on each member of each layer the following resistance checks:

ULS (Ultimate limit state)	SLS (Serviceability limit state)
Flexural Buckling check	Member Deflection check
Torsional Flexural Buckling check	Node Displacement check
Lateral Buckling check	
Lateral Torsional Buckling check	

Selecting the command opens the following window:



esign		•	×		
		2			
Main Beams		N	~		
129 HEA 180	\sim	Param	eters		
Докоі			\sim		
Apply to all members of the Layer					
ng with Min, Max	c of all co	mbinatio	ns		
Layer					
ration of Memb	er Bucklir	ng			
Exploration of Member Servicability					
Member Results Layer Results					
ОК		Cancel			
	esign Main Beams 129 HEA 180 Δοκοί oply to all memb ng with Min, Max Layer Layer ration of Member er Results OK	esign Main Beams 129 HEA 180 ✓ Δοκοί oply to all members of the ng with Min, Max of all con- Layer ration of Member Buckling tion of Member Servicable er Results La OK	esign Main Beams 129 HEA 180 Parama Δοκοί oply to all members of the Layer ng with Min, Max of all combinatio Layer ration of Member Buckling tion of Member Servicability er Results Layer Results Cancel		

Members Design check is by layer. Select the layer from the drop down list and the "Member" list loads all members of this layer and the cross sections. Define the parameters and If you want to set different parameters to some of them, you can create different "Groups" in the same layer. The program contains two default Groups: "Beams" and "Columns".

Membe	r Design 🛛 🗙
Group Name Columnsi Safety Factor 1 Culture and Buckling Direction Y Member's Length Real Coefficient Buckling Lengths 1 1 1 1 1 Coefficient Culture and Culture	New Group Creation Limit of Internal 0.1 Direction Z
Flexural Buckling Ends Constraint Member Loading y Loading Level Lateral Torsional Buckling	Serviceability Check Member Deformation Limits Y 200 Z 200 Node Displacement Limits X 150 Z 150 OK Cancel

Select a "Layer" and click on the "Parameters", and the following dialog box opens:

In the "Group Name" you see the name of the parameter group. If you want to create your group, give a new name and press the button "New Group Creation".



In the "Safety Factor" you can set the limit for the program for the design checks: the intensive forces to the respective strength of the member. The default value is 1.

The "Limit of internal forces" is the limit that the program uses to take into consideration (or to ignore) the intensive sizes.

The rest of the form is divided into four parts, one for each check:

For Lateral Buckling check: Because of the "Merging" of the elements, there is no need anymore to define the Member's Length. The program will consider the length resulting after merge.

The parameter "Buckling Lengths" depends on the support conditions of the member.

Click on the following button to open the following list and select the appropriate conditions so that the program automatically inserts the corresponding factor.

Buckling Lengths Y	
	e

The icons are divided into two groups: The first group includes icons with a specific factor depending on the member support conditions

Flexural Buckling resistance check:

 Flexural Buckling Ends Constraint Member Loading Loading Level 		
• - 1 ² - 1 - 1	Ends Constraint	++

Activate the checkbox and press

The "End Constraints" window, containing the various types of constraints opens.

Press one of the first four buttons to automatically calculate the flexural buckling factor:

End Constraints	×



			Mombor Londi
	Member Loading		
The next parameter		refers to the load	
type of the member	er at the local	axis y, and z respectively. By	
selecting the corresp	onding icon, th	e following options appear:	
Where you choose th	ne type of Mem	ber Loading.	
For Lateral Torsio	n Buckling chec	k: activate the checkbox.	
NOTE : For the l the parameters	ateral buckling are the same.	and the lateral torsion buckling re	esistance check,
 For Serviceabilit "Member Deflection 	y checks: activa ction Limits" an	ate the checkbox "Serviceability Ch d "Node Displacement Limits".	neck" and the checkboxes
Serviceability Chee	ck	Then type the corresponding value	ues in each direction, X and
Member Deformation	on Limits	Z. For example in the figure o	on the left, the limits are
Y 200 Z	200	defined as I/200 and I/150, wher	e l is the member's length.
✓ Node Displacement	Limits		
X 150 Z	150		

Finish the parameters' input and then press the button "OK" to return to the previous dialog box.

To apply the parameters that you set to all members of a layer, select the command "Apply to all members of the Layer".

Activate Checking with Min, Max of all combinations and click the button "Check in Layer" to check all members of the current layer, with Min and Max of all combinations. The results of the design checks are displayed in the black window that becomes green if it the checks are satisfied of all members of the active layer and red, if not.



Member D	esign		×			
Layer	Steel_Columns	;	~			
Member	1075 HEB 500	Ň	Parameters			
Group	Columns	~	~			
A	Apply to all members of the Layer					
Checki	Checking with Min, Max of all combinations					
Check in Layer Mem: 36/36 Comb: 4/4						
Explo	Exploration of Member Buckling					
Exploration of Member Servicability						
Member Results Layer Results						
	OK Cancel					

Activating the option:

Checking with Min, Max of all combinations, in checks will be taking into account only the maximum and minimum values of the intensive forces resulting from all combinations, excluding the intermediate values so that the process will be completed at noticeably shorter times.

Double click ok the colored window, opens the dialog box containing members check summary results:

Member	Cross Section	Lateral	Side	Lat.Torsional	Serv.Def	Serv.Displ	
1075	HEB 500	37/0.00	37/0.00			99/0.01	
1076	HEB 500	1/0.01	39/0.00			99/0.01	
1077	HEB 500	69/0.00	69/0.01			99/0.02	
1078	HEB 500	1/0.01	64/0.01			99/0.03	
1079	HEB 500	69/0.00	62/0.00			99/0.01	
1080	HEB 500	1/0.01	64/0.01			99/0.01	
1081	HEB 500	1/0.00	62/0.00			99/0.01	
1082	HEB 500	1/0.01	64/0.01			99/0.01	
1083	HEB 500	37/0.00	37/0.00			99/0.01	
1084	HEB 500	39/0.15	64/0.01			99/0.01	
1085	HEB 500	37/0.00	62/0.00			99/0.01	
1086	HEB 500	1/0.01	39/0.01			99/0.01	
1087	HEB 500	1/0.01	37/0.00			99/0.02	
1088	HEB 500	39/0.00	39/0.00			99/0.01	
1089	HEB 500	1/0.01	62/0.01			99/0.03	
1090	HEB 500	1/0.00	64/0.00			99/0.02	

The first column indicates the number of the member, the second column indicates the cross section and in the next five columns the least favorable ratio of strength and the combination number from which this ratio was resulted is displayed.

Greens are the ratios below unity and red the ratios above it.

Check all the Layers.



7. Connections

7.1 How to perform steel members' connection design:

The last command of the group command "Steel Members Design" is the "Steel Connections", used for the steel connections' design. Select the command and choose one of the following steps:

A) Right click on the screen to open the library that contains all the available steel connections and select the appropriate one. Click on the button "Next Connection Group" to see more connections.



B) Select with left click the members that you want to connect. Then right click to open a library that contains only the suitable connections for the selected members.



Steel connections	×
Decent to Beam Type Beam to Column Beam to Column	Next Connection Group Connection Name dok_styl_asthenis
B (Range) Г (Web) Г	Member Group Definition (Node)
	Edit Connection (Geometry/Checks)
	Cancel

Left click to select the column member and then the beam member, and right click to open the library with the four possible types of connection. Select the last one "Beam to Column (Web) Γ " along the main axis. Next set the name of the current connection.

1 The name must contain only characters from the Latin alphabet and no spaces between the words are allowed.

Then, select the "Member Group Definition (Node)" command and in the dialog box you can add more groups of members with the same connection features (i.e. column – beam) or type your values for the stress resultants N, M, V for the existing groups.

To add groups of members, click into the field "Lower Column" and pick the column 24. Then click into the field "Right Beam" and pick the beam 153 (or just enter the numbers in the corresponding fields) and then click the button "Add".

Group N	/lembers					Х
				N(kN)	M(kNm)	V(kN)
Lower Column	346	HEA 220	2.80	0	0	0
Right Beam	555	HEA 200	0.95	0	0	0
	0			0	0	0
	0			0	0	0
	0			0	0	0
228: 346,	555.			^		Add
224: 342,5	224: 342,559,					
223: 341.5	558.				U	pdate
214: 332,	549. 461.)elete
198.316.4	408. 102					5
193:3113	357			~		Exit



Use this dialog box for the design of steel connections with the same type and the same crosssections in total.

The program calculates automatically the forces and proceeds with connection's design, based on the less favorable load combination. So you don't have to guess the point of your structure, where the less favorable beam - column connection in the minor axis will be developed. Furthermore, if this connection is satisfied, then all the other connections with the same type will be automatically satisfied, too.

At the end, click "Exit" and select the command "Edit Connections (Geometry/Checks)". In the new dialog box you can define the type and the geometry of the specific connection. Select the type and enter the geometrical parameters of the cross-section or create your connection.

First the program performs the geometrical checks of the connection (e.g. if the bolts are located too close to the edge of the plate). If there is a problem, the corresponding error message appears in the field on the right. In the specific connection, change the distance e1 from 10 to 15 cm and then click again the button "Calculation (Combinations)".



Click the button "3D" to see a three-dimensional representation of the connection that is updated as you change the parameters.

The buttons "1", "2", "3" are used for the display of the two side views (1 & 2) and the plan view (3). The button " Σ/K " is used for the display of the three-dimensional representation of the welds and bolts.



H - I Beam Column Connection (Secondary Axis of Co	olumn)					×
Type Beam - Column (Г) ~ Type With End Plate ~	Node 1) 22 Max	Bolts S54 = 0.01 S54 = 0.01 (1)	Weld S68 = 0.00 S68 = 0.00 (1)	Section S372 = 0.00	End-Plate/ S374 = 0.00	Max S54 = 0.01 S54 = 0.01 (1)
Beam 100 22	Calculation (Combinations)	Calculations (User)	Simplifie	d Ex	ploration	Save Exit
End-plate (mm) Material 5235 v Image: Section in the section of the section						1 2 3 Σ/Κ 0
Thidness 6 STIFFENERS			•			3D

If the geometrical checks are satisfied, the program calculates and displays all Eurocode 3 design checks for the connection. Click "Simplified" to see the results. Green fonts means adequacy and red failure. If all checks are satisfied the program will be able to save the connection and generate the drawings automatically. Otherwise the procedure will stop and you need to change some values of the connection to continue. To read the main results click the button "Results" and for all the results, click the button "Exploration". The displayed *txt files are those generated by the program for the printout.

Click "Save" and then "Exit" to return to the connections' window.

8. FOOTING DESIGN

8.1 How to perform footing design:

As soon as you complete the connection design, you can move on to the 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 perform the design checks for all the footings on the current level.

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

were <u>satisfied</u> or 😵 <u>failed</u>.

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



9. BILL OF MATERIALS



The "Bill of Materials" command group contains the commands related to the estimation of the materials' quantities and the corresponding cost.

Steel Cross Sections: It calculates the quantity of the structural steel.

"Analytical": per element and cross section concerning the length (m), weight in Kg (per m or in total); "Summary".

Bill of Materials - St	I of Materials - Steel X				
	2				
Element	Cross-Section	Length (m)	Weight/m (Weight (Kg)	^
K1075 / 1075	HEB 500	1.51	187.33	283.43	
K1076 / 1076	HEB 500	3.36	187.33	628.87	
K1077 / 1077	HEB 500	1.51	187.33	283.43	
K1078 / 1078	HEB 500	3.36	187.33	628.87	
K1079 / 1079	HEB 500	1.51	187.33	283.43	
K1080 / 1080	HEB 500	3.36	187.33	628.87	
K1081 / 1081	HEB 500	1.51	187.33	283.43	
K1082 / 1082	HEB 500	3.36	187.33	628.87	
K1083 / 1083	HEB 500	1.51	187.33	283.43	
K1084 / 1084	HEB 500	3.36	187.33	628.87	
K1085 / 1085	HEB 500	1.51	187.33	283.43	
K1086 / 1086	HEB 500	3.36	187.33	628.87	
K1087 / 1087	HEB 500	3.51	187.33	658.28	
K1088 / 1088	HEB 500	1.32	187.33	247.28	
K1089 / 1089	HEB 500	3.51	187.33	658.28	
K1090 / 1090	HEB 500	1.32	187.33	247.28	~
OK	Anal	ytical	TOTAL WEIGHT C	OF STEEL (Kg)	46283.63
Cancel	Sum	mary	F	Results File (Bill of	Materials)

SCADA Pro gives you the ability to have analytic bills of materials for each steel cross section per member or aggregated bills per section category.



ОК	Ana	lytical	TOTAL WEIGHT O	F STEEL (Kg)	46283.63
	CHS 193,7X10	302.11	45.30	13685.43	
	IPE 200 IPE 300	288.00	22.40 42.20	6451.20 12806.35	
	SHS 100x8,0	72.00	22.90	1648.80	
	SHS 100x5,0	53.02	14.80	784.72	
Element	Cross-Section	Length (m)	Weight/m (Weight (Kg)	

Press "Results File (Bill of Materials)" to attach the Calculation Printout.

10. 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.



(A) 🛱 0.00 ·	🛊 🖡 🍓 🔻						
Basic Mod	ling View To	ools Slabs Loads	Analysis Post-Processor	Members Design	Drawings-Detailing	Addons	
File Import Template	Parameters Lines - A - Circles	Arcs Other s Shapes -	tent Dimensions Style/Labels Rei	nforcement	ious View	lumns	
Files	D	Irawing	Formworks	Edit	View	Level	
21100	×∕×⊥∕×	🖓 🖓 🗚 🖬 🖬 🏹	8 2 € € € € €	💐 🗹 \land 🔽 🙆	€ ♪ ∢ ×		
Project Data	ф х						<u>^</u>
Image: Columns -C Image: Columns Image: Columns <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td>							
🙀 Project Parame 🗟 P	oject Data <						>

10.1 How to import the detailing drawings:

The automatically created drawings of the created connections are located inside the folder of the project:

C:\scadapro\ "project1" \scades_Synd\sxedia



Use the command to open these drawings inside the SCADA Pro environment: Selecting "Import" command opens the following dialog box for choosing the project's folder. Then select:

In List files of Type: select "Scada connection *.con" (In Directories: find the pathC:\scadapro\"STEEL" \scades_Synd\sxedia)



Open		X
Look in:	📙 steel15	∽ 🎯 🎓 🔛 🕶
4	Name	Date modified Type
	Ec1	10/12/2015 10:41 File folder
Quick access	📜 postpro	10/12/2015 10:41 File folder
	📜 scaanal	10/12/2015 10:41 File folder
·	scades_c	7/9/2016 11:39 πμ File folder
Desktop	📒 scades_FlatSlab	7/9/2016 11:38 πμ File folder
-	📒 scades_Jylina	10/12/2015 10:41 File folder
	scades_Sid	7/9/2016 11:38 πμ File folder
Libraries	scades_Sid1	10/12/2015 10:42 File folder
	scades_Synd	10/12/2015 10:42 File folder
	scades_Toixo	15/10/2015 11:56 File folder
This PC	scainp	10/Search Files
	scaPush	15/ C:/PELATES/UNI/acades_Synd/undar/0_1.co
		15/
Network	<	
	File name:	
	Files of the set	
	Files of type: Scada connection(*.c	on)
Scale Fac	tor 1.0 Level 1	Find
		OK Cancel

Next choose the considered name of the connection (so that it turns blue), click "ok" and finally click inside the desktop at the desired insertion point. In this way three views of the selected connection are created.




Follow the previously described procedure to create and import over 120 different type of connections that the program covers.

To create the respective views of the model in total you must follow a different procedure.

New	Click the in a *.dw and give a the 3D_d	"Export" g format. a name to wg Files (command of In the "Save A the exported (*.DWG) form	the As" file at.	e main men field select e and in the	iu to e the fo "save	xport your drawing Ider of your project as type" field select	
<u>Open</u>	Save As				×			
	Save in:	STEEL		~	G 🌶 📂 🖽 -			
<u>S</u> ave	(Pa)	Name	*		Date modified	Type ^		
		MyProject			16/1/2014 9:35 πμ	File fol		
	Recent places	퉬 postpro			16/1/2014 9:35 πμ	File fol		
Save As		퉬 scaanal			16/1/2014 11:18 πμ	File fol		
		퉬 scades_c			16/1/2014 9:35 πμ	File fol		
	Desktop	scades_Sid			16/1/2014 9:35 πμ	File fol		
Close Project	<u> </u>	scades_Sync	1		16/1/2014 9:35 πμ	File fol		
	6-1	scades_Toixo		16/1/2014 2:15 μμ	File fol			
	Libraries	scainp		16/1/2014 9:35 πμ	File fol			
		je scamel		21/10/2010 12:43 πμ	File fol			
Import		scaoptim			16/1/2014 9:55 πμ	File fol		
	Computer	scatron			21/10/2010 12:42 min	File fol		
		tmn			16/1/2014 0-35 mil	File fol Y		
		<			10/1/2011/5/05/10	>		
Export	Network	File name:			×	Save		
		Save as type:	3D_Dwg Files (*.DWG)		~	Cancel		

Next, if you open the exported *.dwg file using the autocad you'll notice that the whole structure is exported as a 3D model including the name of each cross-section. Since now you are working in AutoCAD environment you can create any view of your model that you want and even apply photorealism commands.

11.PRINTING

11.1 How to create the report:

To create the report, open the unit "ADDONS" and select the command "Print". In the "Calculation's Printout" dialog form, on the left there are the available for printing units. To add a unit to the printing list (located on the right) double click on it.

For this example, select the units that you wish to print and click the "Project Report" button. The preview printing file is automatically opened.

EXAMPLE 3: "Steel Structure"



Iculation's Printout					
Availiable Chapters		Printout	Number of Pages :		
🖃 General	^	Cover		Building Data	
Cover		Short Descriptio	n	Move Up	
		Regulations			
Regulations		Solemn Declarat	tion	Move Down	
		Program Assum	ptions	Delete	
		Seismic Analysis	Parameters (EC)	i Delete	
		Material Descrip	tions (EC)	Delete All	
Table of Contents		Combinations (E	C)		
		Sc2 Seismic Dist	ribution	Insert File	
Charges Constant C		Sc2 Seismic Acti	on	Error Correction	
		Cross section D	esign:HEA 220 Steel	End conceasi	
		Cross section D	esign:HEA 200 Steel		
Steel_Columns		Cross section D	esign:IPE 200 Steel	Format Page	
		Buckling:Steel_(Columns		
HEA 220		Buckling:Steel_E	Beams	Paging 0	
Steel_Beams		Short-Steel			
HEA 200					
IPE 200				Export Printout	
Buckling				0.11	
Steel_Columns				Print	
Steel_Beams				Project Report	
Gonnections				Save	
Serviceability				Suve	
	~			Cancel	

Through this environment, you can save your report under .pdf, or .doc, .excel, or .xml format and modify further the exported result if you wish.

Export to PDF		Export to Rich Text	х
Export	Page range	Page range	
Information	O Current page	• All	
Constitu	C Pages:	Current page	
Security	Enter pagenumbers and/or pageranges, separated by commas, For example, 1,3,5-12	O Pages:	
Viewer	Options Compressed I Background Embedded Fonts I Print Optimized OK Cancel	Enter pagenumbers and/or pageranges, separated by commas. For example, 1,3,5-12 Options Wysiwyg Page breaks Pictures None	
		🔽 Open after export	
		OK Cancel	

Through this simple example you got familiar with some of the most important commands of SCADA Pro. Working with the program you will discover that it has unlimited modeling and design potentials to perform even the most complicated analysis.

EXAMPLE 3: "Steel Structure"

