

User Manual 5.TOOLS







CONTENTS

1.	BUIL	DING BLOCKS	3
	1 1	RENIMBERING	4
	1.1		-
	1.3	BEAM SEGMENTATION	5
	1.4	BEAM UPON BEAM	6
	1.5	BEAM-POST CONNECTION	8
	1.6	BEAM BREAKAGE	9
	1.7	BEAM CONSOLIDATION.	9
	1.8	POLE ADAPTATION.	10
2		N/CS	11
Ζ.	030-	WCS	11
3.	MON	ITENEGRO	12
	3.1	CALCULATION	12
	FINITE E	LEMENTS	14
	3.2	DOKOI->STYLES	16
	3.2.1	Conversion of beams to columns	16
	3.2.2	Model Smith	19
	3.2.3	Model diagonal bars	19
	3.2.4	Change rigid offset	20
	3.2.5	Piles	20
4.	MELL	s	23
		- /	
	4.1.1	Break	23
	4.1.2	Section	24
	4.1.3	Change of reference	24
	4.1.4	Redefining reference	24
	4.1.5	Members' freedoms	25
	4.1.6	Consolidation of members	28
	4.1.7	Surface rod connection	28
5.	COM	BOES	29
	5.1.1	Replace	29
	5.1.2	Identification	29
	5.1.3	Identification of Rod-Surface nodes	30
	5.1.4	Commitment of the Stick-Surface Nodes	30
	5.1.5	Consolidation	31
	5.1.6	Metal Control	31
6	PRFS	S RFI FΔSF	36
0.	C 4 4		
	6.1.1	Finaing Length, Angle	36
	6.1.2	Finaing Area, Perimeter	36
	6.2	PERASIA	36
	6.3	PERFORMANCE OF PROPERTIES	38



Chapter 5: Tools

									SCAE	0A Pro 32Bit - S0221_2		
	Βασικό	Μοντελοποίηση	Εμφάνιση	Επεξεργασία	Εργαλεία	Πλάκες Φι	ορτία Ανάλυση	Αποτελέσματα	Διαστασιολόγης	η Ξυλότυποι	Πρόσθετα	Βελτιστοποίηση
2	3	T 44	=	* 1	: 🔺			L /~	~	₫ 🕻 🕻	2	
Επα θμ	ναρί- Σταθερ ηση σημεία	κ Κατάτμηση Δοκός επι Δοκών δοκού	Σύνδεση Δοκοί Στύλου *	Σπάσιμο Ενοπο δοκού Δοκ	ίηση Προσαρμογι ών στύλου	ή Ορισμός	Υπολογισμός Δοκοί	->Στύλοι Σπάσιμο	Αντικατάσταση	Μήκος Περασιά Απο Γωνία* Ιδια	όδοση πήτων	
		Δ	ομικά στοιχεία	House house	and the second second	UCS - WCS	Μοντέλο	Μέλη	Κόμβοι	Διάφορα		

The 5th Module is called "TOOLS" and includes the following 6 groups of commands:

- √ Structural elements
- √ UCS-WCS
- √ Model
- √ Members
- √ Nodes
- √ Miscellaneous



The "Building blocks" command group contains the commands that allow the user to manage the building blocks of the study.

IMPORTANT OBSERVATION:

A prerequisite for <u>the</u> operation of the "Structural Elements" commands, <u>apart from "Recount"</u> (which applies to both physical cross-sections, members and nodes), is that they are applied <u>to the</u> **physical elements** of the study, i.e., before the **mathematical model** is created.



1.1 Renumbering



Tool for the renumbering of study data. Select the command and in the dialog box,

	select the category of item from the list
Επαναριθμήσεις Στοιχείων 🛛 🗙	Δοκοί Συνδ.Δοκοί
∆окоі ∽	Πεδιλοδοκοί Πέδιλα Στύλοι
Αυτοματη 🗸	ΜΕΛΗ Αυτόματη
Αρίθμηση	and the numbering type [EntAskTikr].
Από Βήμα	Enter the initial number in the "From" field and the "Step".
1 1 Ενιαία επαναρίθμηση κατά Υ-Υ Επίπεδο ΧΖ (Οροφοι) Από 0 - 0.00 0 Σε 7 - 2340.00 δωμα Αpply Αρχικοποίηση	For "Automatic" numbering, select which levels to apply the new numbering to. For "Selective" numbering, select the item category, Apply and select the items in the 2D view. For the category "Pillars" activate the checkbox to get "continuous count" along the height of the pole for the selected levels. To receive the automatic modifications select "Apply", and for "Selective" continue by sequentially selecting the items for renumbering.

advisable to use it BEFORE renumbering and the necessity of its creation has arisen in order to deal with the case where automatic renumbering omits those elements that <u>do not belong to a level (e.g.</u> when the +,- in the levels are not defined). In more detail:

By creating the mathematical model, the program numbers all the mathematical members. However, performing a recount by the designer afterwards, while there are members that do not belong to a level, created duplicate counts for these members. So in this case, an Initialization is done first and then the recounting desired by the scholar is performed. Members not belonging to a level are automatically numbered at the end of the remaining members.

OBSERVATION:

No item should be numbered 0. If a zero count occurs, an error message will appear in the more model checks.



1.2 Fixed points

Σταθερά

σημεία Command to change the fixed top in the columns and the fixed span in the beams.

How to use:

Select the command and then point to the top of the pole you want to remain fixed. The fixed apex defines the point that remains constant in the case of massive dimensional changes in cross-section, material, etc. The initial position of the fixed point in columns is at their center of gravity.

In the rafters you select the command and a small triangle marks the fixed axis which is the centrifugal axis. You select with the mouse the perpendicular that you want to remain constant.



OBSERVATION:

This command is very useful in the IMPROVEMENT of structures and allows to keep the points of the columns and the passages of the beams stable during the variations of the cross-sections.

1.3 Beam segmentation



Δοκών When you enter a continuous beam, the program automatically breaks it into individual beams where it meets columns or walls. This is because by default the Auto Trim switch is active.

OBSERVATION:

The command "Partition beam" has meaning only when when when is inactive. It is applied to the **physical elements** of the beams, i.e. before the **mathematical model** is created. In this case select the command and then the beam.

Suppose you have a row of five (5) columns as shown in the picture





You place a beam with a start post at (K1) and an end post at (K5). The program (under Auto Trim inactive conditions) understands that the beam rests on the (K1) and (K5) substructure. But it has ignored the intermediate columns K2,K3,K4.

To enable it to understand that there are columns at the given locations, you select the command "*Beam Partition*" and then select any point on the beam. The program recognizes the columns at the locations, cuts the beam into sections and makes the connection between the resulting beams and the columns.

1.4 Beam on beam



δοκού Instruction to create *indirect supports on the beams*.

OBSERVATION:

It is applied to the physical elements of the beams, i.e. before the mathematical model is created.

Select the command and then point the mouse at the first beam of the indirect support and then at the second beam.

The individual cases are discussed in the following examples:



Purpose : to create indirect support between beam 1 and D2.



Select the command and then the beams 1 and 2. When creating the mathematical model, the node of the indirect support will be created at position A.



Purpose : to create indirect T-form support.

Draw beams 1 and 2. Select the command and show beams 1 and 2 in sequence. The order does not matter. Then and by creating the mathematical model, the node of the indirect support will be created at position A and beam D2 will be cut into two sections 2a and 2b. $\sim 2^{-1}$ **EXAMPLE 3**



Purpose: creation of indirect T-form support

Draw beams 1 and 2 so that beam 1 ends up to the centreline of beam 2. Select the command and point to beams 1 and 2 in sequence. The order is not important. You will then notice that a distance is created between beams 1 and 2. Then use the command again and again point to beams 1 and 2. Then and by creating the mathematical model, the node of the indirect support will be created at position A and beam 2 will be cut into two sections 2a and 2b.



EXAMPLE 4





Purpose : to create indirect form + support.

Draw beams 1 and 2 as shown in the picture, then select the command and select the beams in sequence regardless of order. When creating the mathematical model, a node is created at position A and the beams are broken into sections 1a, 1b, 2a, 2b.



Purpose: to create multiple indirect support

Suppose you want to place three or more beams which are in an indirect support node. First you form the node of the indirect support between beams 1 and 2 (cross). Then and after placing beams 3 and 4 you call the command "*Beam on Beam*" again and select with the mouse successively these beams as if you had made an indirect support between these beams.

1.5 Beam-post connection

Σύνδεση Δοκού Στύλου physi

Command to connect beams to columns, even when they are not physically connected, so that, after the mathematical model is calculated, the connection between them is established.

OBSERVATION:

It is applied to the physical elements, that is, before the mathematical model is created.



Select the "*Beam - Column Connection*" command and the sub-post to which you will connect the beam(s) you wish to connect.

Select the first beam to be connected to the support (by clicking on a point from the middle to the edge near the post), repeat by selecting in the same way all the beams you want to connect to the support.

Press the right button to complete the process.



▲ If you have connected the end of a beam to a support and try to connect the other end to the same support, the program will not make this connection (otherwise a member with the same start and end node would result).



The command is similar to the previous one, except that the linking does not require manual selection of elements, but is done automatically by the program, which uses predefined criteria to link beams with poles of the active floor.

1.6 Beam breakage



This command allows you to break a beam into individual segments either by specifying the number of segments or the length of each segment.

OBSERVATION:

It is applied to the physical elements, that is, before the mathematical model is created.

By calling the command from the toolbox or from the menu, the following dialog box appears:



where you enter the number of segments or the maximum length respectively for the partition. "OK" and left click on the beam.

1.7 Beam consolidation



The command used to consolidate a beam that was "broken" by a previous use of the "*Break Beam*" command.

After calling the command from the toolbox or from the menu, point the mouse at the sections of the beam in sequence, always starting from the first part of the beginning of the beam.

beam.

OBSERVATION:

It is applied to the physical elements, that is, before the mathematical model is created.



1.8 Pole adaptation



This command is used to modify the position and shape of the cross-section of the columns. (This option works in direct conjunction with the parametric column cross-sections).

The final position of the horizontal branch of the Gamma pillar in the picture should come close to the oblique straight line.



Call the command and select the vertex you want to come to pass, which is the one shown in the picture Fig. c.



The final form of the pole is shown in Fig. d.





You can create more UCSs and through the to select, transfer or delete them.





3. Model



The "Model" command group contains the commands that allow the user to create and manage the mathematical model of the study.

3.1 Calculation



With the command "Calculate", the program calculates and produces the mathematical model of the study (nodes and bars). That is, an automatic simulation of the physical model (structural elements: columns, beams, etc.) is performed with linear members connected by nodes.

Selecting the command opens the dialog box:

Μαθηματικό Μοντέλο 🛛 🗙					
Επιλογή Κανονισμού (Αδρανειακά)					
EC2 ~					
Μετατροπή κανονι ΕΚΩΣ ΕC2 SBC					
 Υπολογισμός Αδρανειακά Ενημέρωση 					
Υπολογισμός Αδρανειακών - Επιφανειών με την μέθοδο των συνοριακών στοιχείων					
OK Cancel					
	αθηματικό Μοντέλο × Επιλογή Κανονισμού (Αδρανειακά) ΕC2 Μετατροπή κανονι ΕC2 Κατατροπή κανονι ΕC2 Κατατροπή κανονι ΕC2 Κατατροπή κανονι ΕC2 Κατατροπή κανονι ΕC2 Κατατροπή κανονι ΕC2 Κασ Κασ Κασ Κασ Κασ Κασ Κασ Κασ				

The first time you do the mathematical model calculation, you select the regulation for the inertial calculation and OK. In case you want to modify the already calculated inertial by changing the regulation, simply select the other regulation and "Convert regulation".

You can create and delete the Mathematical Model as many times as you want.



The deletion is done through the Edit Layers window (see Basic Usage Menu)

πεξεργασία Στρώσεων Χ						
Εργασίας Γραμμές, Κύκλοι				Επίπεδα XZ - Οροφοι		
Νέο Μαθηματικό Μοντέλο	Μαθηματικό Μοντέλο Update					
Αριθμός	Ορατό	Επεξεργάσιμο	Χρώμ 🐴	Επιλογή όλων		
Πέδιλα Μεταλλικα Υπ/τα	a a	 	12 34	Αποεπιλογή όλων		
Μεταλλικές Δοκοί Πλέγμα Επιφάνειας	0 0	₽ ₽	34 7	Ορατό		
Μαθηματικό Μοντέλο	a ~	1	2	Μη ορατό		
Πλέγμα 3D	ă	- -	36	Επεξεργόσιμο		
Πλέγμα 2D <	Ø	₽°	20 ~	Μη Επεξεργάσιμο		
Διαγραφή Δεδομένων						
Μοντέλο Συνολικά Βάσει επιπέδου ΧΖ Βάσει Στρώσης Μόνο Μοντέλο ΟΚ Cancel						

Ο Αδρανειακά

Ο Ενημέρωση

• Activate "Calculate" and "OK" to automatically generate the mathematical model of the study. First to create the mathematical model of the study and then to populate it with the new members.

OBSERVATION:

The same command can be used as many times as you want, either because you deleted the mathematical model, or because after calculating it you insert additional physical members into the study, etc.

- Activate "Update" and "OK" to update the mathematical model for possible modifications to already existing data to update the inertial of the cross-sections (e.g. displacements of beams or columns, geometric changes of a cross-section).
- In case the calculation of the mathematical model has been previously performed and modifications have been made to the Rigid Offsets, activate "Inertial" so that a new calculation does not affect these modifications.

Υπολογισμός Αδρανειακών -Σπιφανειών με την μέθοδο των συνοριακών στοιχείων

Activating the command, allows the calculation of inertial with the method of boundary elements.



FINITE ELEMENTS

The types of finite elements you can use in **SCADA Pro 21** are generally grouped into **1D elements, 2D elements and 3D elements** and are identified their shapes. For example, elements can take the form of a straight line or curve, triangle or quadrilateral, tetrahedron, and many others.

The simplest element is a line consisting of two nodes. All line, straight line or curve elements are called **1D elements** and are capable of displacements and rotations. Examples of 1D elements are the truss element (truss) and the beam element (beam3d)

SCADA Pro 21 includes :

- Ribbed (linear) Network elements with function in space
- Ribbed (linear) beam-supporting elements with spatial function
- Ribbed (linear) elements of a pedestal on elastic ground with operation in space
- Boundary utilities for the simulation of elastic supports at the nodes of linear elements .



2D elements are usually surface elements triangular or quadrilateral. Examples of 2D elements are 3node triangular element, 6-node triangular element and many others. These surface elements can have regular or irregular shapes as shown in the figure

2D elements are flat elements. Therefore, the linear approximation of the displacements considered are u (x, y) and v (x, y) while the rotations are θ (x, y). Since they correspond to plane stress and simple strain, they are often used to solve 2D elasticity problems.

SCADA Pro 21 includes :

- Finite surface shell elements (quadrilateral or triangular)
- Finite surface shell elements on elastic ground (quadrilateral or triangular)
- Finite surface flat deformation elements
- Finite plane intensity surface elements for simulating surfaces generated by rotation.
- Finite surface elements of flat intensity.
- Boundary ancillary elements for the simulation of elastic supports at the nodes of surface elements





3D Elements. 3D elements are commonly used to simulate volumes. They are derived from 2D elements and are used in more complex simulation problems 3D solid elements have only displacements and no rotations. The three unknown displacement functions are u (x, y, z), v (x, y, z) and w (x, y, z). Examples of 3D solid elements are 4-node tetrahedral element, 10-node tetrahedral element, 8-node isoparametric element, etc.

SCADA Pro 21 includes :

- Three-dimensional, hexahedral, isoparametric finite elements, with varying intensity along their thickness (8-21 nodes)
- Boundary auxiliary elements for the simulation of elastic supports at the nodes of solid elements.





3.2 Beams->Poles



The command group "Structures-> Pillars" group contains the commands that allow the user to:

- to simulate **basement walls** and
- to change the rigid offset of the members.
- to import **Passover**

3.2.1 Conversion of beams to columns

For the simulation of the basement walls (level 0) using the method "Converting beams to columns":

- At the ceiling level of the basement, insert, in place of the wall, a beam as thick as the thickness of the wall.

-

- Select the command "Beams in columns" and in the dialogue box:

activate "Number of columns" or "Max length of column" and enter the corresponding number

Μετατροπή Δοκού σε Στύλους	\times	Μετατροπή Δοκ	ού σε Στύλους 🛛 🗙
 Πλήθος Στύλων Μαχ μήκος στύλων (cm) 		 Πλήθος Στύλων Μαχ μήκος στύ 	/ 100 λων (cm)
OK Auto	Cancel	OK At	uto Cancel

- Left-click on the beam that will be automatically transformed and broken into as many parts as the "Number of columns" or "Max length" you set.





۰ŏ



-You can repeat the same procedure at the foundation level or select the pole sections and, using the Copy command, copy them to level 0 at the Αντιγραφή corresponding position.

- Continue with "Calculate" the mathematical model, which means creating nodes that are disconnected from each other.
- Connect the nodes with linear members of high stiffness, similar to that of the walls and zero specific weight. After defining the membership properties, it is sufficient to select the first node and window all the others and the program places a member from node to node.



EXAMPLE: Simulation of basement walls and insertion of footings at the base.

1) Enter the physical model of the columns and beams at level 1 (above the foundation level).

2) Select the command "Beams in columns" to convert the beams that will become the walls of the basement.

3) Select the command "Basic">>"Copy layer", the arrow for level change 🖄 and go to the foundation layer where you select the command "Basic">>"Paste layer",

4) Select "Tools">>"Model">>"Calculation";

5) Return to level 1 where you have to join the new nodes with linear members of high stiffness, similar to that of the walls and zero specific weight.

6) Select the command "Modeling">>"Member">>"Mathematical" and in the dialog box:



Γραμμικό μέλος			×	
A/A 0 Túnoc B-3d 💌	A(m^2) 0.75	Asz(m^2)	0.625	
Κόμβοιί Ο j Ο	Ak(m^2) 0.75	beta	0	
Υλικό Σκυρόδεμα 💌	Ix(dm^4) 148.0	4534 E(GPa)	29	
Појотпта С20/25	Iy(dm^4) 39.06	25 G(GPa)	12.0833	
Απόδοση Διατομής Δοκός Διατομή Ο 25/300 Υποστυλώμσ Μέλος Δοκού Μεγάλης Ακαμψίας Rigid Offsets (cm) Αρχή i Τέλος j dx 0 0 0 dy 0 0 0 dz 0 0	- Саларана 55-35 Аскойс (0) - Саларана - Саларана - Саларана -	Teauropia (cm) bw 25 h 300 F⊽ R.Offsets C C C C Fueia 0 Fueia 0 Averopaueho	z	X Karaxienan Enhorth Info 0 90 30 180 270 View Xew CX Cancel

7) Select the button "High stiffness beam member" Μέλος Δοκού Μεγάλης Ακαμψίας that automatically fills in the field of the parameters, cross section 25x300, with zero specific weight and without cross sectional performance.

8) Left-click to join the nodes one by one. You can also use the windowed

R.Offsets

option for convenience and speed.

- 9) Go to level 0 (Foundation):
- 10) Select "Appearance">>"Switches" and disable "Auto Trim"
- 10) Select "Modeling">>"Pedestal" and in the dialog box enter the geometry and

- turn them off

11) Insert the pedestal from node to node.





3.2.2 Model Smith

Based on this method, in place of the wall, two bars are placed crosswise between the two columns, simulating the equivalent wall.

The simulation using the "Smith Model" method is performed as follows:

- 1. At level 1: insert a beam as thick as the wall in place of the wall,
- 2. At level 0: insert the pedestal or the connecting beam,
- 3. Do the Mathematical Model Calculation,
- 4. Select the "Smith Model" command,
- 5. Open the 3D view and point to the beam member above.

OBSERVATION:

The program inserts two chiasmus bars between the two columns. At the same time the A, Ak, Asy, asz, and Iz of the members defining the boundaries of the wall i.e. the top and bottom beams are modified.

The illustration you will see is the following.



3.2.3 Model diagonal bars



Follow the same procedure of the "Smith Model" method.

Based on the "Diagonals" method, in place of the wall, two bars are again placed crosswise between the two columns, simulating the equivalent wall.

OBSERVATION:

The main difference with the Smith method is that it does not follow an automatic modification of the inertial of the members defining the boundaries of the wall, i.e. the top and bottom beams, as before. You can achieve this yourself by changing the dimensions of the superstructure beam e.g. from 25/50 to 25/300 (i.e. giving the height as the height of the basement).



OBSERVATION:

The basic requirement for use of these last two wall simulation methods is: <u>The presence of the</u> <u>mathematical model</u> and the <u>existence of the beams</u> that will then become the crossbars. The beams must have a thickness equal to that of the walls.

In the cross membersrigid offsets are automatically calculated so that the elastic part of the bars starts from the shoulder of the adjacent columns.

3.2.4 Change rigid offset

This command determines the new position of the elastic node at the beginning or end of a mathematical member and automatically modifies the rigid offset of that member. **DEFINITIONS:**

• <u>A beam elastic node</u> is defined as the point of intersection of the beam axis with contour of the column on which it rests.

• <u>An elastic node of a column</u> is defined as the node of the centre of gravity of the crosssection at the beginning or end of the column.

You select the mathematical member with the mouse by pressing on a point near the edge of which you want to modify the rigid offset. The program selects the elastic node at the end of the member. You then use the mouse to select the new position of the elastic node.

3.2.5 Piles

The foundation piles included in the new version of SCADA Pro are circular reinforced concrete piles.

The way the load is transmitted is through the pile tip to the ground while at the same time the lateral friction works (driven piles).

The superstructure loads are transferred through a head bond (simulated in SCADA Pro with finite surface elements) to the top of each pile and then to the ground.

In the program the command to insert the stakes is in the "Tools" section:





													SC	ADA F
	Βασικό	Μοντελοπο	ώηση	Εμφάνιση	Επεξεργο	ασία	Εργαλεία	Πλάκει	Φ	ορτία Ανά	λυση Αποτ	ελέσματα	Διαστασιολόγ	ηση
213 Επαναρ	Σταθερά	Τ Κατάτμηση Δά	κός επί	τύνδεση Δοκού	Σπάσιμο Ε		η Προσαρμ	ονή Οι		Υπολογισμόσ	Δοκοί->Στύλοι	χ Σπάσιμο	Αντικατάσταση	
θμηση	σημεία	Δοκών	δοκού	Στύλου +	δοκού	Δοκών	στύλοι	3	+			+	*	Γω
0-0.0														
Δεδομένα Ση • • • Φ / Γ Φ / Γ	α Εργου Ρ Γραμμές Γόξα	ů >		(1) Scada : 0-0.0	00 (D:\MELE	TES\S02	21_2\\$0221_	2)			Μοντέ	λο smith λο διαγωνίω	ν ράβδων	
	ώκλοι Δοκοί Ξτύλοι Τέδιλα										Πάσσο	λοι	₽	

Selecting the command displays the following dialog box:

Εισαγωγή Πασσάλων 🛛 🗙					
Υλικό	Σκυρόδεμα		\sim		
Ποιότητα	C20/25		\sim		
D <mark>(</mark> m)	1				
L (m)	20	Στοιχεία Εδάφους			
Step (m)	0.5				
Μαθηματικ	κό Μοντέλο		\sim		
l	OK	Cancel			

where you specify: The **material** and **quality**.

· · · · · · · · ·

OBSERVATION:

Theoretically, the pile of circular cross-section can also be attributed to Steel material quality other than concrete. However, the dimensioning will only be done for reinforced concrete piles.

Then you define the **diameter of** the stake and its total **length**.

The **step** has to do with the sections where the overall rod will be segmented in order to create the nodes where the <u>side springs</u> will be placed.

2 transfer springs are placed in each node in the two vertical directions x and z. Finally there is the choice of the Layer to which the rods will belong.

Pressing the top button "Ground data" opens the following dialog box:

CHAPTER 5 "TOOLS"



Στοιχεία Εδάφους				×
Πλήθος εδαφικών ζωνών	1	Ζώνη 1	~	
Στοιχεία Εδαφικής Ζώνης				
Πάχος (m) 10 Ε	ϊδος Μη	-συνεκτικό	~	
Αστράγγιστη διατμητική αντα	οχή αργίλου (Su) kN/m2	0	
Σχετική πυκνότητα άμμου (D	r) %	?	20]
Βάθος υδροφόρου ορίζοντα (ι	m) 0			
OK	Ca	ancel		

where you specify the number of territorial zones and then, for each territorial zone selected from the "Zone" drop down list, its data.

The stake is placed at the node of the header that you point to with the mouse. The position of the piles is chosen exclusively by the designer. Later there will be an optimization process where the optimal combination of number and diameter of piles in a rectangular arrangement will be proposed.



The bars are inserted to the depth corresponding to the total length of the pile. The bars correspond to a circular reinforced concrete post or whatever other material you originally selected.

OBSERVATION:

There is no problem with piles being inserted at a negative altitude.

Then, as usual, the analysis can be run and the intensive quantities can be calculated for the piles as well.

At present, the pile inspection procedure has not been implemented.



4. Members

λ	The " Members " command group includes	the commands that allow the user to manage and modify the mathematical members o				
Σπάσιμο • Μέλη	 Σπάσιμο Τομή Αλλαγή φοράς Αλλαγή φοράς Επαναπροσδιορισμός φοράς Ελευθερίες Μελών Ενοποίηση μελών Ενωση ράβδου επιφανειακού Ενωση ράβδου επιφανειακού (Μέλος) 	 break Section Change of reference Redefining reference Members' freedoms Consolidation of members Union Members linear with surface 				
		-				
	Ενωση ράβδου επιφανειακού (Οροφος)					

OBSERVATION:

The basic requirement for the operation of these commands is that the members come from the "**Modeling**>> **Member**>> **Mathematical**" command with or without a physical cross-section rendering, or from the use of the "**Formal constructions**" command.

4.1.1 Break

This command allows you to "break" a linear mathematical member into individual members based on the number of members or the length of each member.

Select the command and in the dialog box that appears:

Κατάτμηση μελών 🔀					
💿 Πλήθος τ					
🔘 Μах μήκος (cm)		0			
ОК	Auto	Cancel			

set either the number of segments or the maximum length of each segment. Then press the OK button and point the mouse at the member you want to break.



OBSERVATION:

Selecting the "**Auto**" button automatically breaks all the mathematical members of the vector that are crossed.

The option <u>only</u> works <u>with mathematical members</u> and needs care in its use because it breaks all cross members altogether.

4.1.2 Section

This command is used to partition two mathematical members that intersect. Four new members are created with a node at the point of intersection.

Select the command and point the mouse at the two members. The two members "break" into four and a <u>new node</u> is created at their intersection.



4.1.3 Change of reference

Command to change the direction of the members' local axes.

Activate the Local Axes in the "Switches" Select the command and left-click the member, and observe the change of direction.



4.1.4 Redefining reference

This command should be used when one or both of the following messages appear in the general checks:

Error1678: Pole 123 is mounted in the wrong direction There are members with the wrong local axis direction



The first one, which concerns only the poles, has to do with their placement direction (the correct direction is from bottom to top), while the second one is a general message concerning beams and poles and especially for the beams, it appears when they are not placed according to the program's convention which is from left to right and from top to bottom.

OBSERVATION:

So when the above messages appear, by using the "Redefine times" command, the times are automatically corrected for the whole vector, and so there will be no problems arising in the dimensioning of the perimeters.

4.1.5 Members' freedoms

The Member Freedoms mandate is a new mandate to massively change the degrees of freedom of the limbs of members.

When you call it, the following dialog box appears

Ελευθ	Ελευθερίες Μελών 🛛 🗙					
N	Vy	Vz	Mx	My	Mz	
		Πάκτ	τωση			
		Αρθρ	ωση			
		Ελευ	θερία			
	OK		(Cancel		

where you are given the ability to easily and quickly modify the degrees of freedom of the limbs of the members that you wish.

This command can be ideally combined with the graphical display of the degrees of freedom for direct monitoring of the changes that take place.







	Ελευθερίες Μελών 🛛 🗙					
	N	Vy	Vz	Mx	My	Mz
			Πάκτ	യന		
			Αρθρ	ωση		
			Ελευ	θερία		
Packing		OK		(Cancel	
	Ελευθ	θερίες	Μελι	ύν		×
	N	Vy	Vz	Mx	My	Mz
			Πάκτ	τωση		
			Αρθρ	οωση		
			Ελευ	θερία		
Link		ОК		(Cancel	
	Ελευθ	ερίες	Μελό	ύν		×
	N M	Vy 🗹	Vz ☑	Mx	My	Mz
			Πάκτ	τωση		
			Αρθρ	ωση		
	Ελευθερία					
Freedom		ОК		(Cancel	

Freedom

Or you can select which degrees you want to free and then, by pressing ok, show the ends of the members that will get the specified degrees of freedom.

This command can be ideally combined with the graphical display of the degrees of freedom for direct monitoring of the changes that take place.

CHAPTER 5 "TOOLS"





Finally, it should be mentioned that new control messages have been created to prevent uncontrolled changes to members' degrees of freedom, resulting in errors and the analysis not running. All of the following deterrents lead (not always) to errors in the analysis). The cases with corresponding messages are :

- N at both ends or Vy at both ends or Vz at both ends or Mx at both ends.
- Member instability (number). Release of (N) (Vy) (Vz) (Mx) of both limbs is not allowed.
- My at both ends and Vz at one of two ends or Mz at both ends and Vy at one of the two ends.
- Member instability (number). When (My) (Mz) is released at both ends, (Vz) (Vy) is not allowed to be released at either end.

OBSERVATION:

Based on above messages, the initial degrees of freedom were adjusted to the cross bars of the smith model and the wall fill. Generally where the program used bifacial or axial-only members from the outset, only My and Mz at both ends are freed.



4.1.6 Consolidation of members

This command unifies two or more mathematical members that are placed in sequence. The resulting member has the inertial elements of the first member in the order of the members (Fig. a).



Select the command and show the mathematical members in sequence, always starting from the first one. The resulting mathematical member has the inert elements of the first member. Then delete the intermediate nodes (Fig. b1,2).



0

4.1.7 Surface rod connection

A surface that has been simulated with surface elements and enclosed by linear elements (e.g. slabbeams), creates the need for interconnections between them.

The command works as follows: it first breaks the linear member into individual parts (members and nodes), as many as the surface elements it is adjacent to. Then, it connects with rigid offsets the nodes of the linear members and those of the nearest surface members.

- Select the command, left-click on the linear member and the nodes of the surface one by one successively or with a window.

Otherwise, for greater convenience and as , choose between:

- Section "**Tools**" >> "**Members**" >>" ^{Ενωση ράβδου επιφανειακού (Μέλος) " where you select the beams one by one and the connection is made automatically.}

Section "Tools" >>" Members ">>" ^{Ενωση ράβδου επιφανειακού (Οροφος)} ".
 Where the active floor members enclosing the surface ones break and are automatically connected.



5. Nodes



5.1.1 Replace

This command replaces one node with another deletes the original node. Select the command and the node you want to replace, then point to the replacement node (Fig. a) The original node is deleted and the member is rigidly offset to the new node (Fig. b).





⊚≣€

Select the command and point to two or more nodes. The program creates a <u>new node in the</u> <u>geometric locus of the selected nodes</u>, erases the previous nodes and the elements connected to them are now connected by rigid offsets to the new node.

Select the command, point to the nodes and finish with a right click.



5.1.3 Identification of Rod-Surface nodes

 \triangleleft

Select the command and point to the node of one or more linear members and finally to the node of the surface to make the match. The program erases the member nodes and connects them with rigid offsets to the surface node.

Select the command, point to the nodes and finish with a right click.



5.1.4 Commitment of the Stick-Surface Nodes

Command to bind a node of a linear member (e.g. a column) to the <u>nearest node</u> of a surface element (e.g. a pavement).

Select the command and point to the linear node and then to the surface node to which it will be bound.

(see: Section "Basic" >>" Layers - Layers " >>"Edit XZ Layers")



Τρόπος Σύνδεσης Κόμβων Στύλων με Πλέγμα Επιφανειακών		
Εξάρτηση στον πλησιέστερο κόμβο του επιφανειακού	\sim	

A SCADA Pro allows the collaboration of linear and surface elements in the same interface. The need for binding between them is therefore born.

5.1.5 Consolidation

Command to unify nodes that are very close .

Select the command and set a distance value. Nodes that are less than or equal to this distance will be consolidated, resulting in a single node.

Offset		×
		ОК
Αποσταση (cm)	200	Cancel

5.1.6 Metal Control

In the new version of the program, the possibility of an initial specification of metal columns and metal beams was added.

💶 Έλεγχος Μεταλλικών ΣΜRc > 1.3ΣMRb

This specification is based on the requirement in paragraph 4.4.2.3. of EC8-1 for general and local ductility conditions.

This specification is done in the form of checking the strength moments of the beams and columns that exist at the node.

More specifically, in order to avoid a floor mechanism, the following relationship shall be satisfied for nodes where primary seismic columns and primary or secondary seismic beams are present

$\sum M_{\rm Rc} \ge 1.3 \sum M_{\rm Rb} \tag{4.29}$

Or else, the ratio of the sum of the strength moments of the columns to the corresponding sum of the strength moments of the beams at the node shall be greater than 1.3.

This check, like all the checks, is done <u>on a directional basis</u>.

Based on this requirement, the designer can adjust the cross-sections of the columns accordingly.

OBSERVATIONS:

CHAPTER 5 "TOOLS"



The primary seismic beams in iron are the head beams and the rest are secondary. So the program checks at a node where at least one subcolumn and at least one beam exist. The substructure and the beam must be identified as such and it is not sufficient that they belong to this layer.

ραμμικό μ	ιέλος					×
A/A	29	Túnoç B-3d 🗸 🗸	A(m^2)	0.0076835	Asz(m^2)	0
Κόμβοι ί	44	j 43	Ak(m^2)	0.0076835	beta	90
Υλικό	Χάλυβας-Τυτ	1ικές V	Ix(dm^4)	0.0041551	E(GPa)	210
Ποιότητα	S275(Fe430)) ~	Iy (dm^4)	0.7763171	G(GPa)	80.7692
Απόδοση	Διατομής		Iz (d m^4)	0.2768805	ε (kN/m^3)	78.5
Ynoστυ	λώμα 🗸 🗹	Διστομή	Asy(m^2)	0	at*10^-5	1.2
H	EA 240	Υποστυλώμα 🗸	Δείκτης Εδο	άφους Ks (MP	a/cm)	0
Μέλο	ς Δοκού Μεγά	λης Ακαμψίας				
Rigid Off	sets (cm)		Ελευθερί	ες μελών		
A	φχή ί	Τέλος j		N Vy	Vz Mx M	ly Mz
dx 0		0	Αρχή i Τέλος i			
dy 0		0	TENOGJ			
uy -			Μεταλλικο	α Υπ/τα		\sim
dz 0		0	ОК	Can	cel	Info

Also the program will not take into account any antifoams, lintels, or tectiles that exist since they do not have this designation (Beams, columns).

The strength moments are calculated on the basis of the following relationships in EC3-1



6.2.5 Bending moment

(1) The design value of the bending moment M_{Ed} at each cross-section should satisfy:

$$\frac{M_{Ed}}{M_{c,Rd}} \le 1.0 \tag{6.12}$$

where $M_{c,Rd}$ is determined considering fastener holes, see (4) to (6).

(2) The design resistance for bending about one principal axis of a cross-section is determined as follows:

$$M_{c,Rd} = M_{pl,Rd} = \frac{W_{pl} f_y}{\gamma_{M0}}$$
 for class 1 or 2 cross sections (6.13)

$$M_{c.Rd} = M_{el.Rd} = \frac{W_{el.min} f_y}{\gamma_{M0}}$$
 for class 3 cross sections (6.14)

$$M_{c,Rd} = \frac{W_{eff,min} f_y}{\gamma_{M0}} \qquad \text{for class 4 cross sections} \qquad (6.15)$$

where Welmin and Weffmin corresponds to the fibre with the maximum elastic stress.

In order to select the type for which the corresponding resistance moment will be calculated, the program now classifies the cross-section in the data input and the corresponding relation is selected accordingly.

OBSERVATION:

However, for cross-sections of category 4, it was not possible to calculate the required Weff, min size in the data input, so the check for these cross-sections is not performed and a corresponding information message is displayed.

Let's take a closer look at the use of the command:

The command can be executed either per level in 2D or for all nodes that "look" in 3D.

PREREQUISITE: Nodes should be "visible", because, as is well known, nodes that are not embedded in a floor are not shown. So for them, the specification will not be done, either in 2D or in 3D.

So when the command is executed, the results are displayed at the bottom





The results apply to all nodes and for each node are listed:

- The number of the node and then and per direction the ratio of the strength moments and the corresponding result are indicated.

- The word error in front of the message is only displayed when one or both of the two reasons are less than 1.3.

OBSERVATION:

When even one member that exists on the node is category 4, the check is not performed and the following informational message is displayed:

"There was no satisfactory audit. There are category 4 members."

Clicking on a line turns the corresponding node in the vector model red.

Because the ratios for both directions always occur when no beams occur in one direction, the denominator of the fraction becomes 0 and the ratio becomes 0, but obviously, in that direction there is no problem.

Regarding the y-y and z-z directions, these refer (as in the concrete content material) to the local axes of the column ending at the node.



For example, at the following node inside the red circle:



• In the y-y direction (green axis) of the post that ends at the level, there is a beam that participates with its y-y (local beam) resistance moment and the post (ATTENTION) participates with its z-z resistance moment.

• In the z-z direction (always of the column) 2 beams participate which participate with the yy local moment and the column with the corresponding y-y one.

Finally, it is recalled that EC8 excludes the roof from the above requirement:

 $\sum M_{\rm Rc} \geq 1.3 \sum M_{\rm Rb}$

(4.29)

όπου

- ΣΜ_{Re} είναι το άθροισμα των τιμών σχεδιασμού των ροπών αντοχής των υποστυλωμάτων που συμβάλλουν στον κόμβο. Στη σχέση (4.29) θα πρέπει να χρησιμοποιείται η ελάχιστη τιμή της ροπής αντοχής των υποστυλωμάτων μέσα στο εύρος διακύμανσης των αξονικών δυνάμεων των υποστυλωμάτων που αντιστοιχούν στη σεισμική κατάσταση σχεδιασμού και
- ΣΜ_{Rb} είναι το άθροισμα των τιμών σχεδιασμού των ροπών αντοχής των δοκών που συμβάλλουν στον κόμβο. Όταν χρησιμοποιούνται συνδέσεις μερικής αντοχής, οι ροπές αντοχής αυτών των συνδέσεων λαμβάνονται υπόψη στον υπολογισμό του ΣΜ_{Rb}.

ΣΗΜΕΙΩΣΗ: Η αυστηρή ερμηνεία της σχέσης (4.29) απαιτεί τον υπολογισμό των ροπών στο κέντρο του κόμβου. Οι ροπές αυτές αντιστοιχούν στην ανάπτυξη των τιμών σχεδιασμού των ροπών αντοχής των υποστυλωμάτων ή των δοκών στις εξωτερικές παρειές του κόμβου, καθώς και σε μία κατάλληλη πρόβλεψη για τις ροπές που οφείλονται στις τέμνουσες δυνάμεις των παρειών του κόμβου. Εντούτοις, η απώλεια στην ακρίβεια είναι μικρή και η απλοποίηση είναι σημαντική εάν αγνοηθεί η επίδραση των τεμνουσών. Επομένως η προσέγγιση αυτή θεωρείται τότε αποδεκτή.

(5) Η σχέση (4.29) θα πρέπει να ικανοποιείται σε δύο κάθετα κατακόρυφα επίπεδα κάμψης, τα οποία, σε κτίρια με πλαίσια διατεταγμένα σε δύο κάθετες διευθύνσεις, ορίζονται από τις δύο αυτές διευθύνσεις. Θα πρέπει να ικανοποιείται και για τις δύο κατευθύνσεις (θετική και αρνητική) της δράσης των ροπών δοκών περί τον κόμβο με τις ροπές των υποστυλωμάτων πάντα αντίθετες προς τις ροπές των δοκών. Εάν το δομικό σύστημα είναι πλαισιωτό ή ισοδύναμης πλαισιακής λειτουργίας σε μια μόνον από τις δύο κύριες οριζόντιες διευθύνσεις του δομικού συστήματος, τότε η σχέση (4.29) θα πρέπει να ικανοποιείται μόνον σε κατακόρυφα επίπεδα σε αυτή την διεύθυνση.

(6) Οι κανόνες (4) και (5) της παρούσας δεν έχουν εφαρμογή στο δώμα πολυώροφων κτιρίων.

6.



Miscellaneous



6.1.1 Finding Length, Angle

To find length, relative distances by x, y and z and angle of inclination, select the command and the first point that defines the origin. Then, by moving the mouse pointer, you can see the distance L, the relative coordinates Dx, Dy and Dz and the angle of inclination in the bottom right of the status bar

L=800.00 Dx=-800.00 Dy=0.00 Dz=0.00 Angle=0.00 . By selecting the second point you can see the items

you want.

6.1.2 Finding Area, Perimeter

To find the area and perimeter of a surface, after selecting the command, select the vertices or lines that delimit the surface you want to measure. Complete the selection by pressing the right mouse button and in the status bar you will see the area, the coordinates of the centre of gravity of the surface and its perimeter

Area=153500.00 Xkb=601.43 Zkb=1046.82 P=1600.25

6.2 Perasia



To bring one element into line with another. Select the command, the entity (e.g. a pole) that will be passed and then specify the line (or half-plane or circle or point) with respect to which the entity will be passed.



EXAMPLE 1 Assume line (e) and the 80x50 pole. Activate the command.



Select the pole by left-clicking on the side (1) and after line (e). The post is placed as in figure 1.

Selecting the pole by left-clicking on the side (2) and then the line (e), The pole is placed as in figure 2.

EXAMPLE 2 Let's take the beam (T1) and two 30x60 poles.

Activate the command.



By selecting the beam by left-clicking on a central point on the top side, and then the top side of the two poles, the beam is positioned as in Figure 1.

By selecting the beam by left-clicking on the top side but at a point near the left post, and then clicking again on the beam at a point near the right post, you get the result shown in Figure 2.

EXAMPLE 3

Suppose you have placed two circular columns and a beam connecting them. You now want to bring the passing beam towards the circular column, as shown in the second figure.



- Select the command "Passage" and then the beam D1 by pressing with the mouse on the edge (a).

SCADA Pro 25

Structural Analysis & Desi

- Then press with the mouse on the K2 column from the line (e') upwards.

- The beam comes to end (a) in the final position you want.

- Then use the mouse to press the beam D1 at its end (b) and then the support K1 below the straight line (e'').

- Now the beam came and was placed exactly as you wish.

6.3 Performance of Properties

A command that allows the properties of the selected object to be mapped to other similar objects.

Select the command and left-click a model object to open the corresponding window containing the individual properties.

Check the properties you want to assign and OK to close the window. Then, select (using one of the Select tool modes) the similar objects to which the selected properties of the first onewill be assigned

Απόδοση Ιδιοτήτων 📃
Στρώση Χρώμα Υλικό Διατομή Συμετοχη Εδάφους Αδρανειακά Ak A Ix Iy Ix Asz beta G
Ελευθερίες Μελών Βαθμοί Ελευθερίος Κόμβου

CHAPTER 5 "TOOLS"

