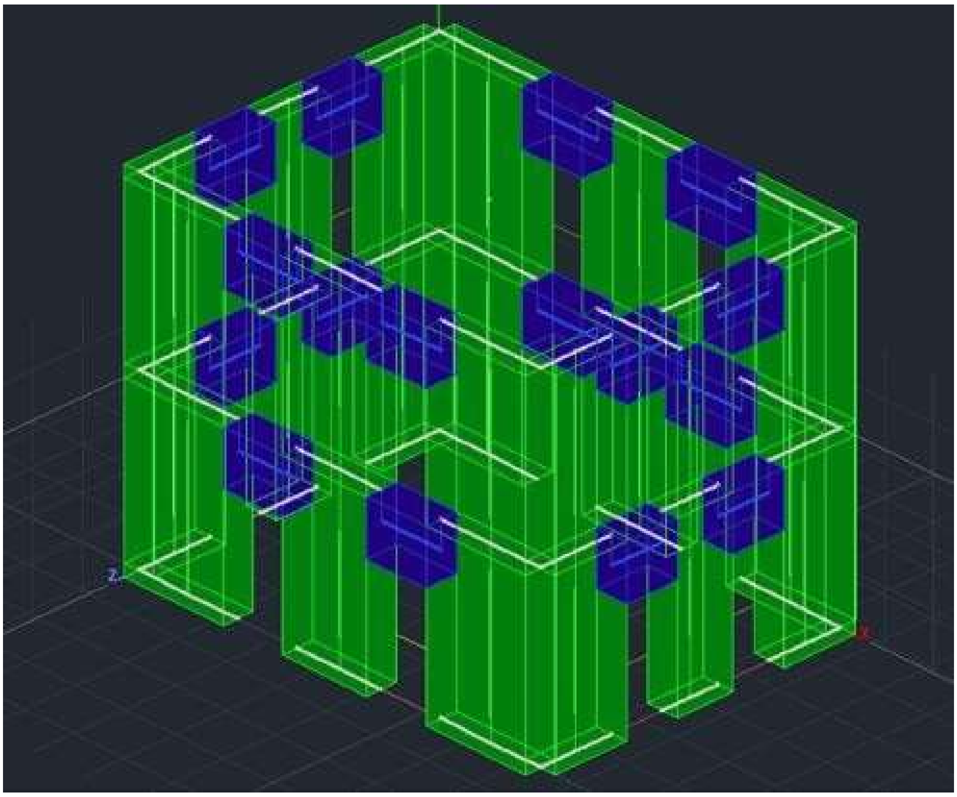
SCADA Pro™
Structural Analysis & Design

Example 11
Masonry with the equal frame method



ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS:

EQUIVALENT FRAME METHOD:

- With the equivalent frame method (EFM), the structure is simulated as a set of linear elements.
- Each wall is considered to consist of 3 types of elements:

  a. **Pillars**

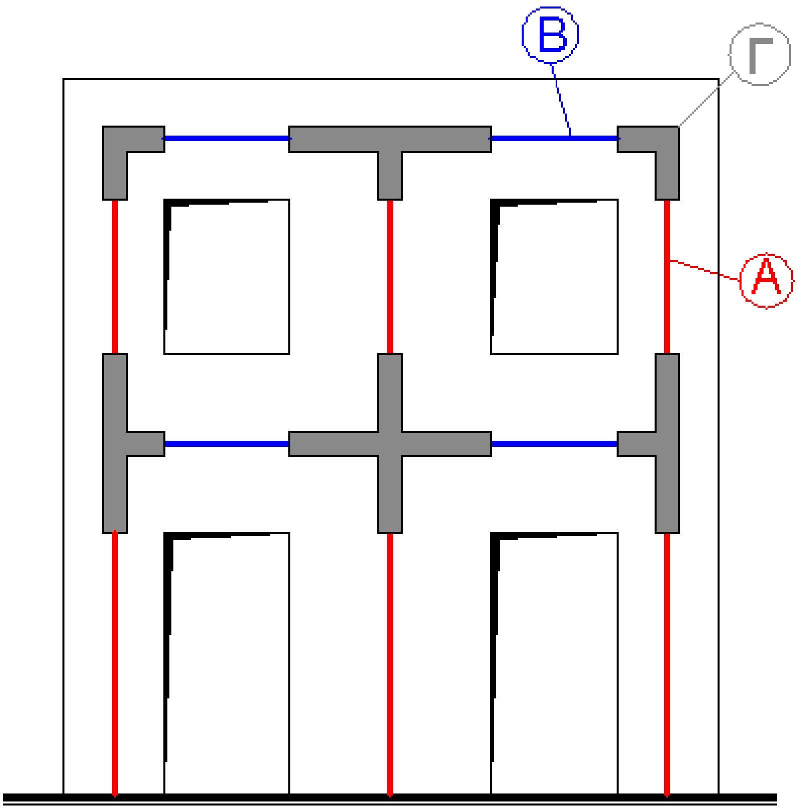
  B. **Hyperthyra - Aprons**

  Γ. **Coupling beams**

  Pillars and lintels are deformable elements, while the coupling beams are undeformable elements connecting the pillars to the transoms - aprons.

- MIP is a popular method for simulating load-bearing masonry structures in professional practice, due to its remarkable simplicity of its application and ease of interpretation of the results.

- The reliability of the results of the method is higher when the analysis of the carrier is carried out by non-linear methods, which allow redistribution of forces for large deformations.



1. CARRIER DESIGN

Our goal is to draw the three-dimensional representation of our vector with lines. We can achieve this in several ways:

Using 2D dwg file:

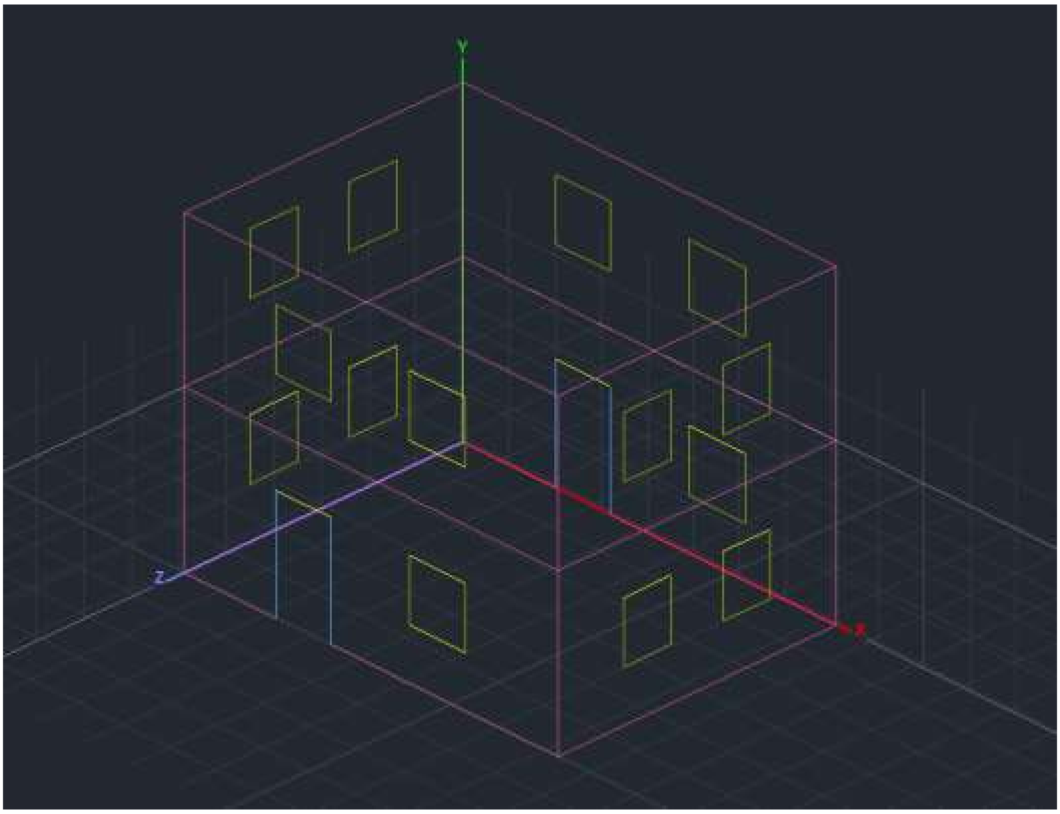
- Import a design into SCADA Pro,
- Select design layer for conversion to SCADA lines,
- Create a new layer for independent selection,
- Opening of Standard Structures,
- Formation of openings,
- Introduction to the SCADA environment,
- Deletion of grids.

Using 3D dwg file:

- Import a design into SCADA Pro,
- Select design layer for conversion to SCADA lines.

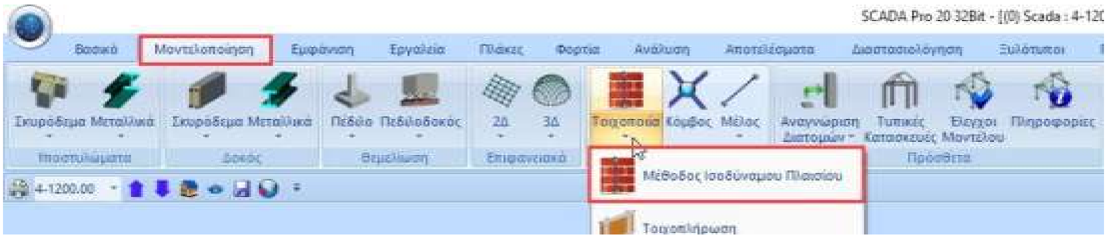
Using SCADA Pro design commands:

- Drawing with relative and absolute coordinates



YouTube GR *Watch on the SCADA Pro YouTube channel the training courses ¨Modeling with linear elements, M.I.P¨ https://www.youtube.com/watch?v=OAVlu-QnVrw&list=PLSYOATQuvG6_OzCHn7-_13c04NayNSLu7l*

2. MASONRY MODELLING WITH M.I.P.



The vector is modelled using the Equivalent Framework Method command.

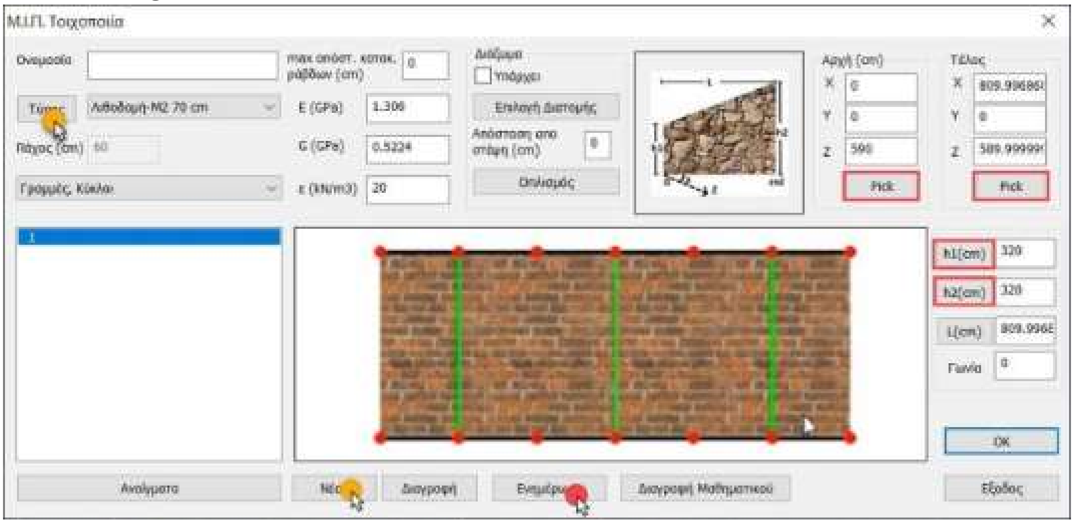
2.1 WALLS:

First the walls are defined:

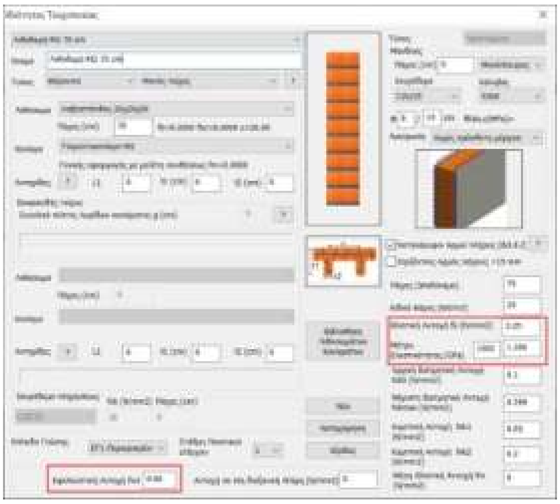
- Press
- Dormitory
- Geometry

The process includes the following steps:

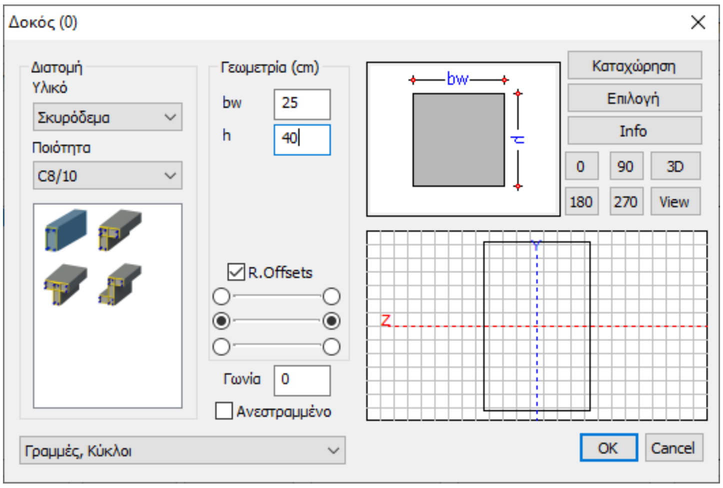
1. We give a name to the wall



2. We select the type of masonry from the pop-up menu or click on to open the masonry library and define a wall of our own, defining the stone, mortar and the compressive and tensile strengths.



3. We define the geometry of the wall graphically by selecting Pick and h1, h2 and showing the start, end points and heights at the 2 edges of the wall, using the pulls and left click. The fields are automatically filled in, and manual typing of values is also allowed.
4. In addition, the fields of the constants E, G, e are filled in automatically, with the possibility of modification by the user, as well as the values L and Angle.
5. ¨Max sender. Cat. The maximum distance between the bars is determined by the maximum distance between the vertical ribbed elements for the simulation of the equivalent frame columns. Keeping the value 0, the program calculates it by KADET. The user can specify a value of his own and then the program will take it into account.
6. ¨Diasma¨: I activate the checkbox when there is a tier and via the command ¨Cross-section selection¨ define the cross-section of the partition. Then, we indicate the distance of the diaphragm from the crown of the wall.



7. By selecting thecommand, the specified wall is added to the list. H ¨Delete¨ deletes it, ¨Update¨ informs it of any changes, ¨View¨ marks it graphically for easy identification in the Operator. The ¨Delete Mathematical¨ deletes the mathematical model (after it has been calculated, which is done after the walls and openings have been created).
8. The command opens the window for defining the openings of the selected wall.

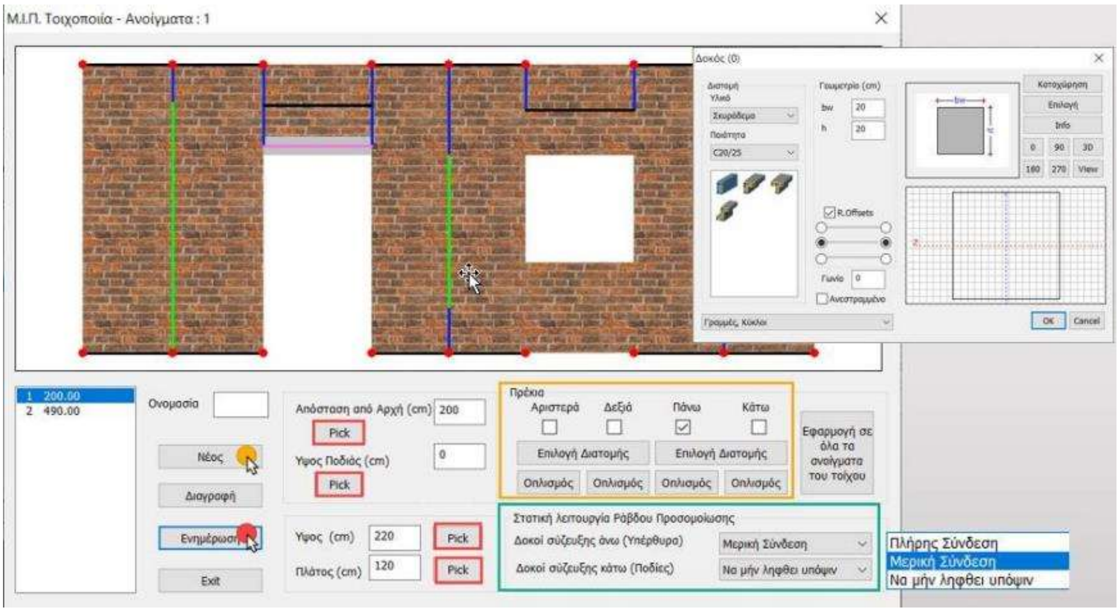
2.2 OPENINGS

Then the openings are defined:

- Geometry
- Pretzels
- Static Rod function

The procedure is similar to that of fixing the walls:

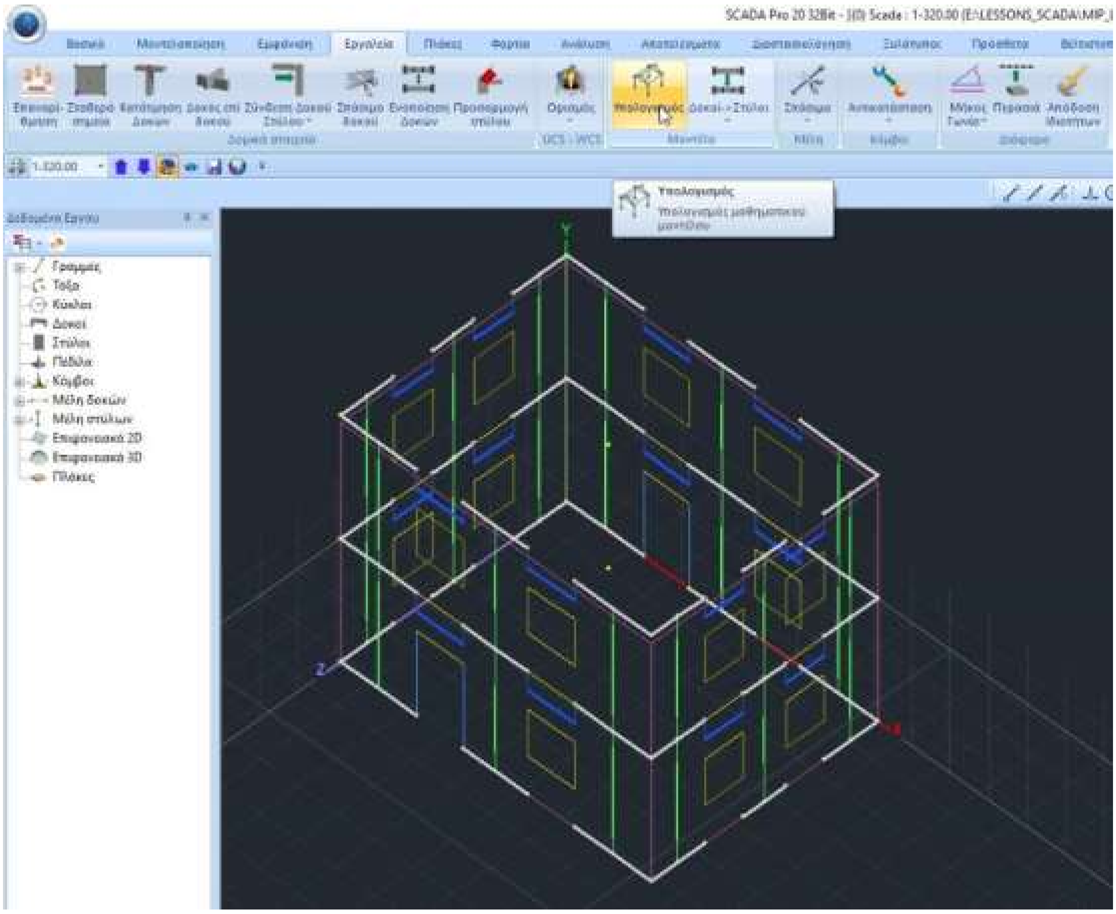
1. We give a name to the opening
2. With the help of Pick we graphically define the geometry of an opening
3. If there are Presses we select their position and their cross-section and if they apply to all openings of the same wall, then we select ¨Apply to all openings of the wall¨, so that the same presses and the same cross-sections are automatically applied.
   other openings that we will make for the same wall.



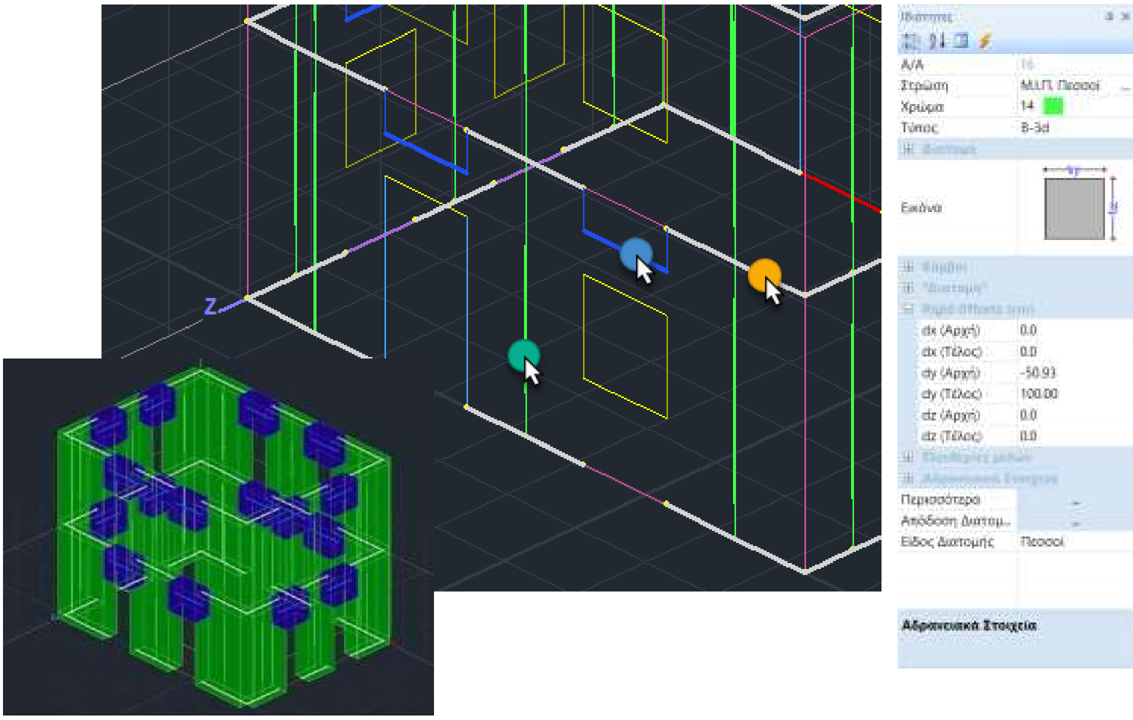
4. In the field ¨Static simulation rod mode¨ we select one of the three ways of participation of the transoms and legs in the mathematical simulation:
   - Full connection
   - Partial connection
   - To disregard
5. By selecting the command, the specified opening is added to the list. H ¨Delete¨ deletes it, ¨Update¨ informs it about possible changes, ¨View¨ marks it graphically for easy identification in the vector.
6. The process is repeated until all openings of the selected wall are defined.
7. ¨Exit¨ to close the opening window and continue with remaining walls and openings.

The modelling is completed when the definition of all walls and openings at all levels is completed and the calculation of the mathematical model is done.

2.3 MATHEMATICAL MODEL

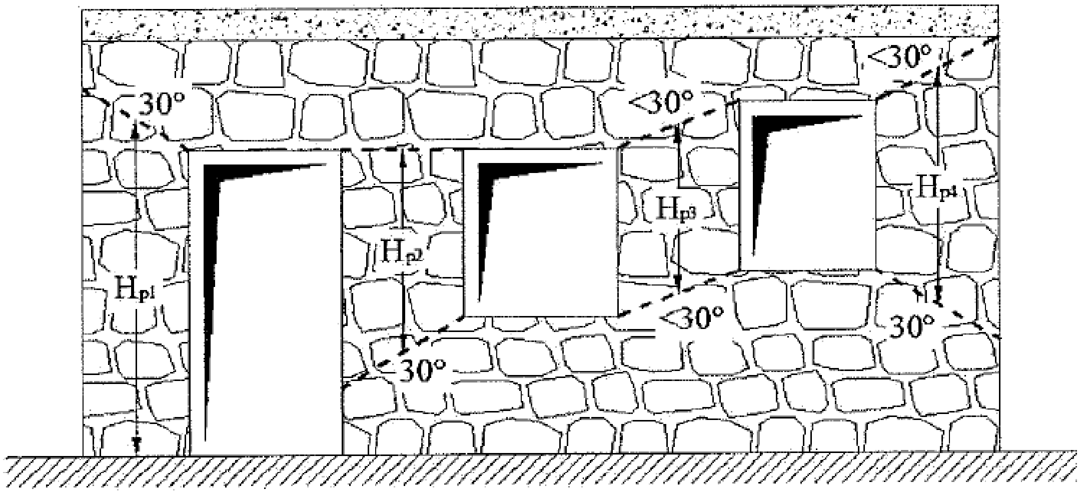


After calculating the mathematical model, the program creates 3 already linear elements:

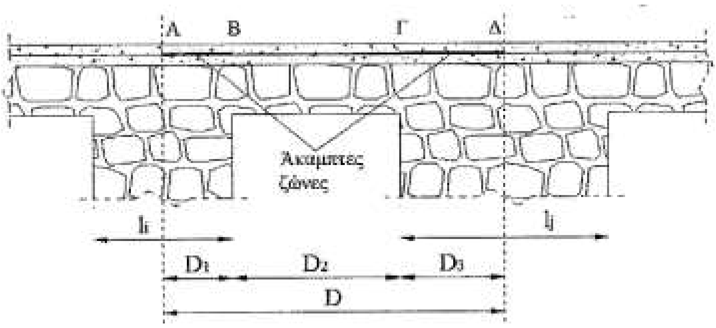


OVERWHELMETS-FOUNDATIONS centripetal

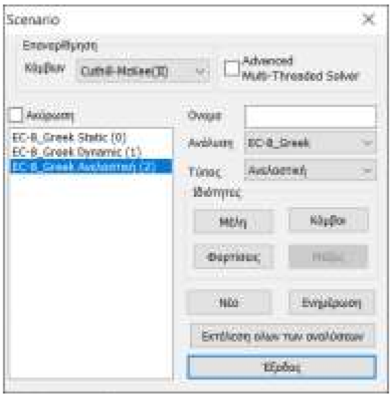
PESCO centrifugal, with rigid offsets depending on the geometry of the wall and openings



CONVERSION DOCKS of great rigidity, weightless, concrete



Then we follow all the known steps for the introduction of plates, loads as in all other linear members made of concrete, metal, etc. In Analysis field we select the scenario ¨EC8_Greek Elasticity¨



3. Elements of Elastic Static Analysis

The simulation of load-bearing masonry with finite surface elements is the most accurate method but it is not suitable for the inelastic analysis of the structure. In this case, the MIP simulation method is recommended.

The following is the description of the procedure as implemented in SCADA for a load-bearing masonry structure with MIP.

First, some information on the analysis and design of these structures.

The checks shall be carried out for static and seismic load combinations as specified in the relevant regulations.
With regard to seismic combinations, the following limit values are distinguished states (performance levels) to control the behaviour of the operator "as a whole" and as individual elements.

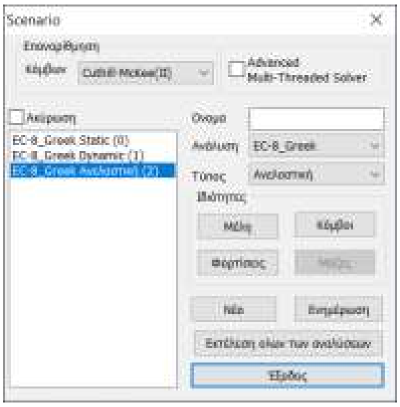
(a) Damage Limitation (DL) Limit State*: the structure is only slightly damaged, with the structural elements retaining a high degree of strength and stiffness. The permanent relative deformations are negligible. It is assumed that the functions of the building are not interrupted during and after the earthquake except, possibly, for secondary functions.*
(b) Major Damage Limit State (SD)*: the entity has suffered significant damage with no collapses, has residual strength and stiffness and the vertical elements are able to take the vertical loads. There are moderate permanent relative deformations. The structure can withstand aftershocks moderate intensity.*
(c) Limit Condition of Quasi-Collapse (NC)*: the entity has suffered serious damage. Its remaining strength and stiffness are low. The vertical elements are still able to bear vertical loads. Large permanent relative deformations have developed. O the carrier is a short distance from collapse and will probably not be able to withstand a subsequent earthquake, even of moderate intensity.*
Checks are carried out at the deformation level both for the whole structure and for the individual members.

Audits at Member State level

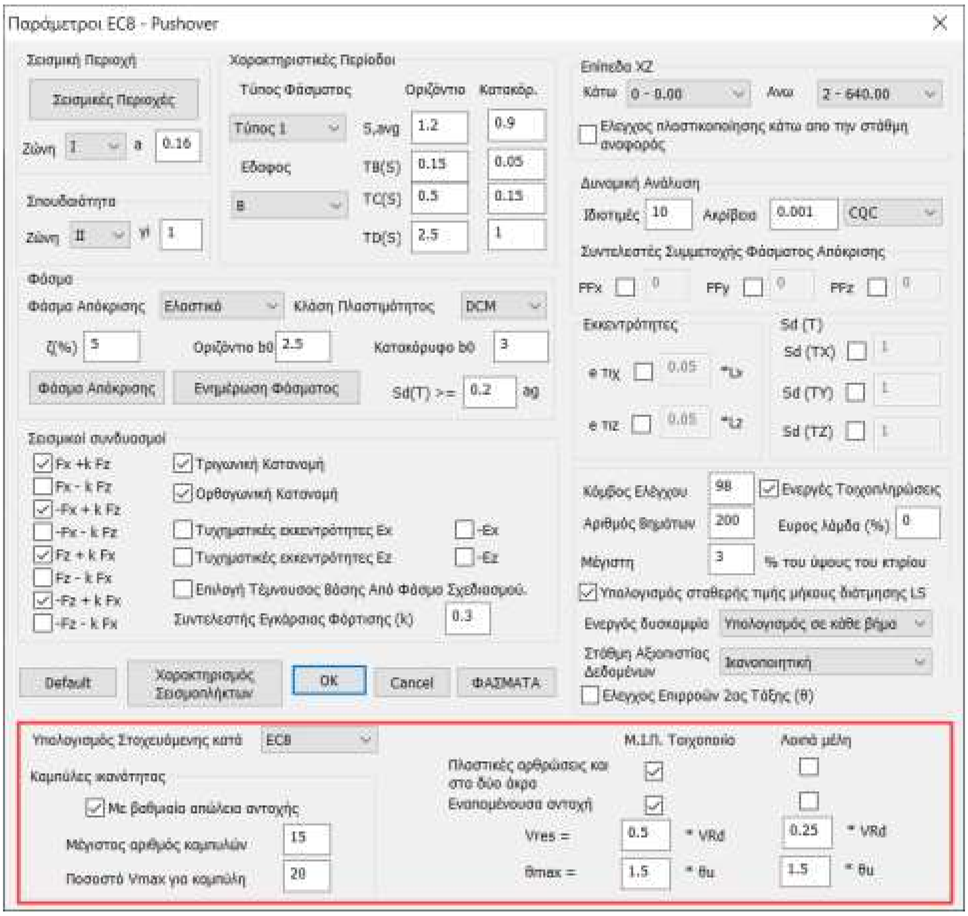
The adequacy checks of individual elements of load-bearing masonry are carried out in terms of deformations and/or forces in accordance with the applicable regulation.

3.1 General overview of the process of the Elastic Static Analysis

- SCENARIO

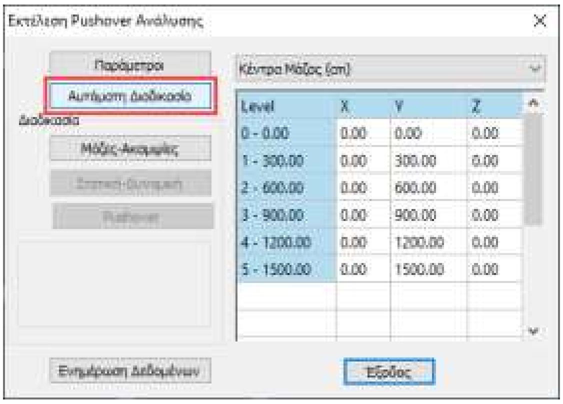


- PARAMETERS



All the parameters of the inelastic analysis are explained in detail in the 8B user manual. Analysis in chapter 1.3.

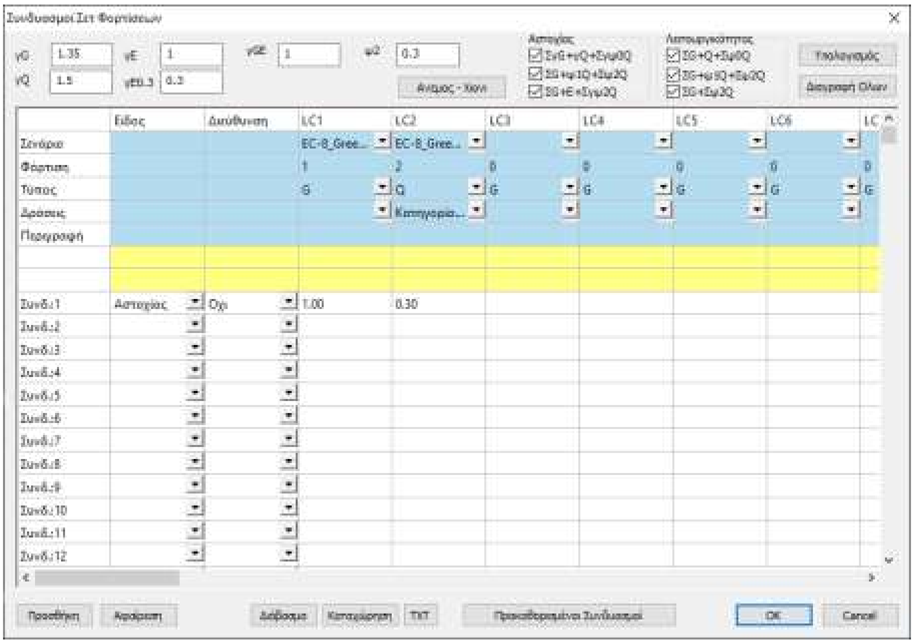
- AUTOMATIC PROCEDURE



During the 'Automatic Process':

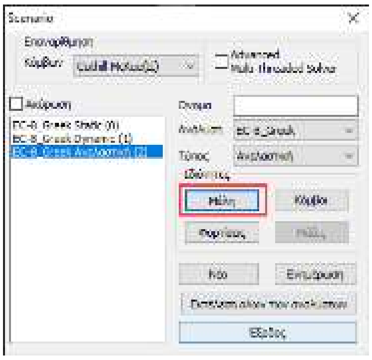
In the solution m=0 only the vertical loads, i.e. the permanent G and mobile Q loads for the combination γ_G $G+\psi_2Q$ (<u>where the value of γG depends on the SWD</u>) are applied in order determine the initial stress of the beam. Subsequently, successive elastic analyses are performed where at each step the external horizontal seismic loads, which it is emphasised remain constant (i.e. their value does not change) in all solutions in the case of the orthogonal or inverted triangular distribution. Thus, according to the acting bending stress and the corresponding available bending strength in the members, the degree of static indeterminacy of the beam is continuously reduced by the creation of plastic joints, until finally the beam is transformed into a mechanism.

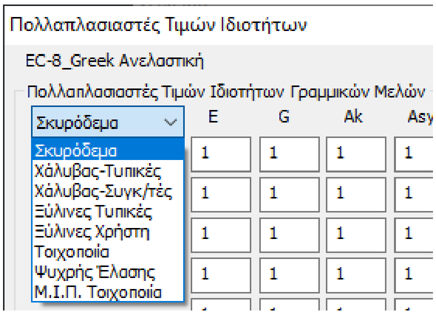
The coefficients of the vertical loads are shown in the following combination.



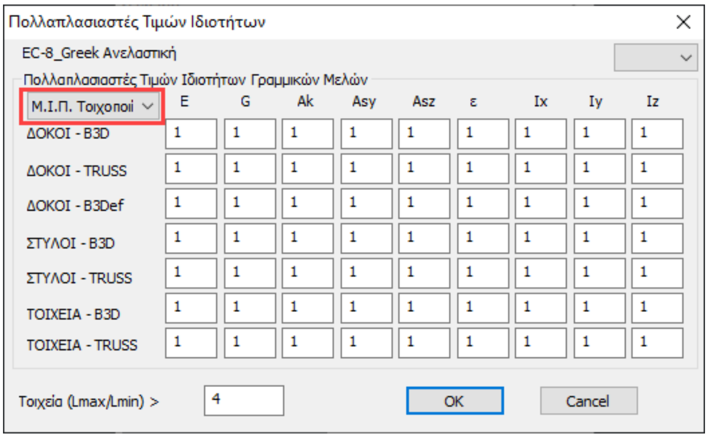
Then the iterative steps of Pushover begin to run. This is followed by an explanation of what happens at a random iteration step *m* and the same applies to the others.

Especially for a Static Elastic analysis scenario, whether **EC-8_General** or EIA (**EC-8_Greek Elastic**), the multipliers of the inertial quantities (within the Members) defined here will be taken into account in the first analysis of the
Pushover involving permanent and mobile loads with default values those prescribed by EC8.



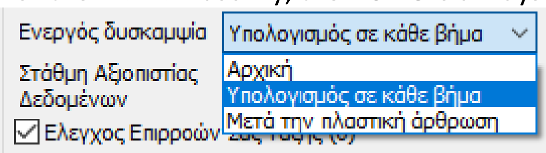


There are separate options within the Membersfor Concrete and M.I.P.

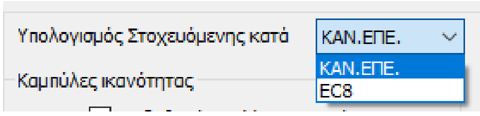


The parameter "***Active stiffness***" concerns the way of calculating the stiffnesses of the elements of the structure. The first step of the inelastic analysis calculates the intensive quantities from the permanent and mobile loads of the structure. The stiffnesses taken into account for these magnitudes are multiplied by the coefficients specified in the scenario parameters in the '**Members**' option.

For the M.I.P. masonry, the **Home** is always taken regardless of the from option.

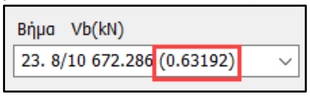


The calculation of the **targeted movement** is done either by CANEP (method of coefficients) or by EC8.



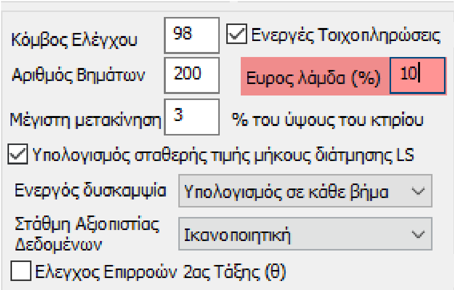
It should be noted that these parameters, because they relate to the calculation of the targeted movement, can be set or modified and after run the inelastic analysis without the need to re-run it. The same applies to the control node.

At each step, **the incremental load factor λ**, defined as the product of of the worst strength (the smallest of the 4 resulting from the 3 in-plane and 1 out-of-plane failure modes) to the corresponding stress magnitude.



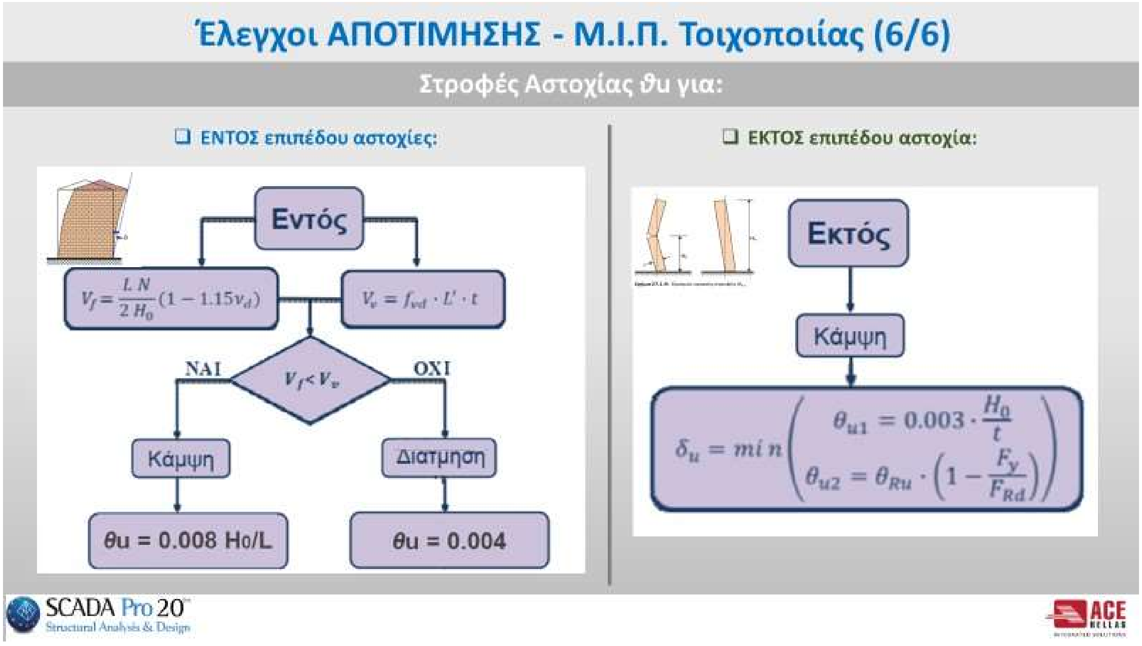
With the default value of 0 in the Lambda Range (%) parameter, the program selects a minimum value, i.e. only one element, even if there are values from other elements that are very close to it.

Setting a value other than 0 e.g. 10% means that those λ values that are less than or equal to the minimum value λ plus 10% will be taken into account in this step with result in more than one plastic joint being created at the same time.



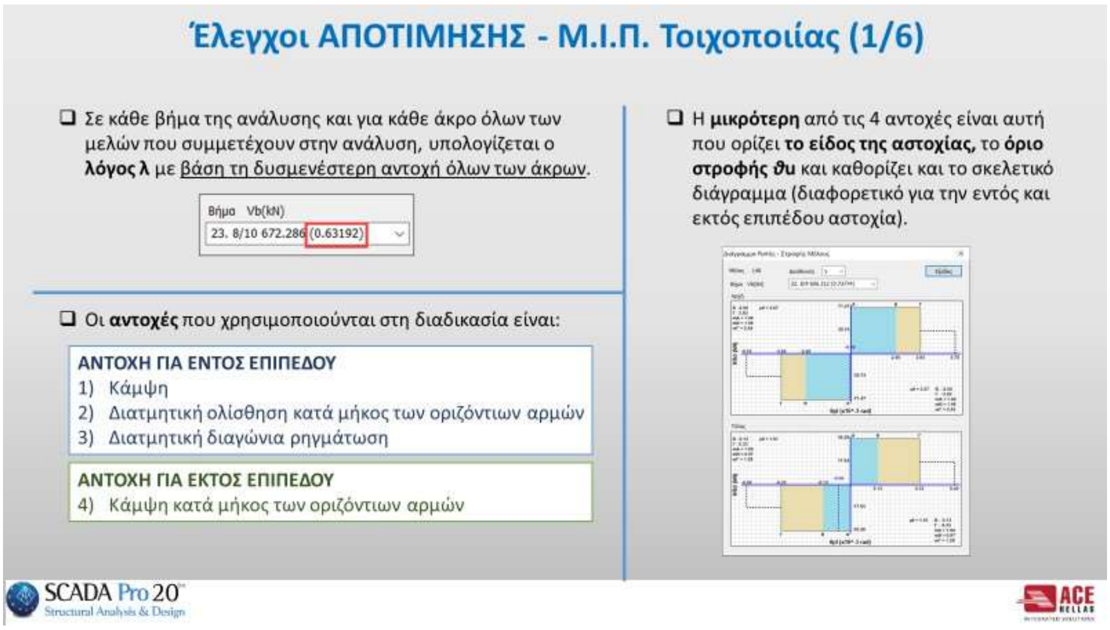
Assume that in the first step of the pushover the minimum value λ is 1 and corresponds to a the specific structural element on which the plastic joint will be created. By setting a value of 10% in this parameter, elements with λ values from 1 to 1.1 will also have plastic joints created in them, simultaneously with the first element.

The smallest of the 4 strengths is the one that defines the type of failure, the limit of rotation **θu** and determines the skeletal diagram (different for in-plane and out-of-plane failure).



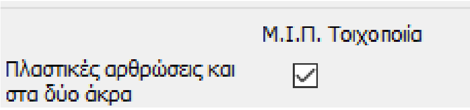
In load-bearing masonry for the calculation of λ**, 4 checks** are carried out as prescribed by the KADET.

RESISTANCE TO IN-FIELD	RESISTANCE TO OUT-OF-FIELD
1) Bend 2) Shear slip along the horizontal joints 3) Shear diagonal cracking	4) Bending along horizontal joints



Also, after the calculation of λ, the new intensive magnitudes are calculated for each end, which are the intensive magnitudes of the previous step+ λ times the intensive magnitudes of the current. These sizes are the final step sizes at each end. Of course they are not recalculated λ because then we would have an endless process.

From all the λ of each step, the largest is selected and at the end plastic joint (internal release) is activated in NEXT STEP, i.e. the corresponding moments are released at the ends of the member. If the option is activated:



then the moments at the other end are also released, regardless of λ.

Depending on the range λ given to the parameters, the plastic joints are activated in the respective limbs. These limbs are the "candidates" for plastic joints to be implemented in the next step.

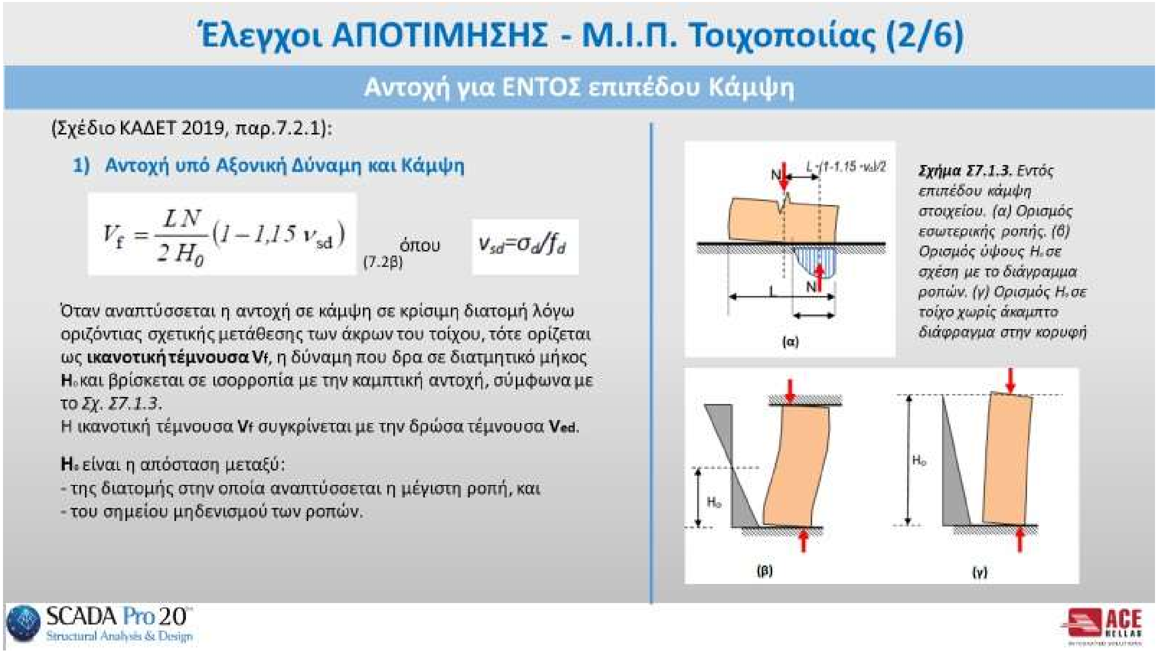
In the next step, the analysis is performed on the modified beam and the known dots are placed at the ends of the previously "candidate" members, which are painted in a colour determined by comparing the measure of the rotation of the plastic joint and the limits as defined below for in-plane and out-of-plane respectively:

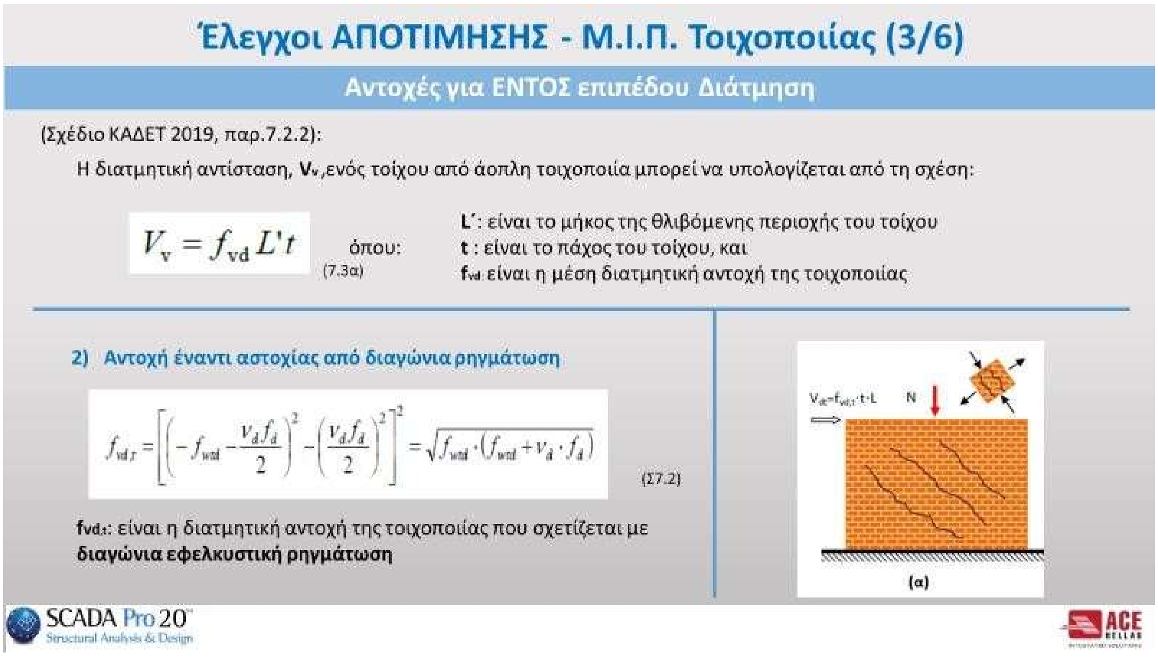
- $\theta DL=0$ (once it is exceeded, i.e. once it is activated, it turns blue)
- $\theta SD=\theta u/\gamma Rd$ (once it is exceeded, it turns yellow)
- $\theta NC=4/3*\theta u/\gamma Rd$ (once it exceeds it, it turns red)

for in-plane $\gamma Rd=1.5$ for out-of-plane $\gamma Rd=2$
The way of calculating θu is different for in-plane and out-of-plane failure.

PDF *Refer to the pdf file of the presentation entitled ¨Part 3º : Valuation of an existing building made of load-bearing masonry with linear finite elements (L.F.I.P.); and to KADET for more.*

YouTube GR *Watch on the SCADA Pro channel on YouTube the webinar entitled "[SCADA Pro - 10/6/20 - Evaluation of masonry with linear elements M.I.P.¨](https://www.youtube.com/watch?v=m-1J3Ok_-_0&list=PLSYOATQuvG68jro3H29zOfZy6y1AB2p9G&index=6)* https://www.youtube.com/watch?v=m-1J3Ok_-_0&list=PLSYOATQuvG68jro3H29zOfZy6y1AB2p9G&index=6





Έλεγχοι ΑΠΟΤΙΜΗΣΗΣ - Μ.Ι.Π. Τοιχοποιίας (4/6)

Αντοχές για ΕΝΤΟΣ επιπέδου Διάτμηση

(Σχέδιο ΚΑΔΕΤ 2019, παρ.7.2.2):

3) Αντοχή υπό διατμητική αστοχία ολίσθησης κατά μήκος των οριζόντιων αρμών

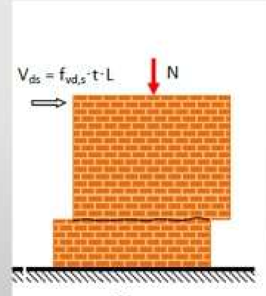
$$f_{vd} = f_{vm0} + 0{,}4\, N/L\,'t \leq 0{,}065 f_b, \quad (7.3\beta)$$

f_{vd}: είναι η μέση διατμητική αντοχή της τοιχοποιίας η οποία **συνυπολογίζει και την παρουσία κατακόρυφου φορτίου**

όπου:

f_{vm0}: μέση διατμητική αντοχή απουσίας κατακόρυφου φορτίου.

f_b : είναι η κανονικοποιημένη θλιπτική αντοχή του τοιχοσώματος



(β)

SCADA Pro 20 Structural Analysis & Design

Έλεγχοι ΑΠΟΤΙΜΗΣΗΣ - Μ.Ι.Π. Τοιχοποιίας (5/6)

Αντοχή για ΕΚΤΟΣ επιπέδου Κάμψη

(Σχέδιο ΚΑΔΕΤ 2019, παρ.7.3):

4) Αντοχή υπό κάμψη κατά μήκος των οριζόντιων αρμών

Η ροπή κάμψεως την οποία μπορεί να αναλάβει η διατομή ελέγχου εξαρτάται από την θλιπτική αντοχή της τοιχοποιίας και από την τιμή του (ευμενούς) αξονικού φορτίου, κατά την ακόλουθη σχέση:

$$M_{Rd1,o} = \frac{1}{2}\ell t_w^2 \sigma_0 \left(1 - \frac{\sigma_0}{f_d}\right) \quad (7.6\alpha)$$

f_c : θλιπτική αντοχή της τοιχοποιίας

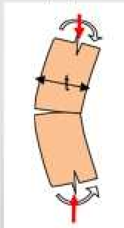
$\sigma_0 = N_{sd} / (l \cdot t_w)$, μέση θλιπτική τάση λόγω αξονικής δράσης στην διατομή ελέγχου,

ℓ : μήκος τοιχοποιίας,

t_w : πάχος τοιχοποιίας

Η ικανοτική τέμνουσα, $F_{\gamma R}$, σε εκτός επιπέδου κάμψη υπολογίζεται από τον λόγο της αντίστοιχης ροπής $M_{Rdl,o}$ δια το μήκος διάτμησης του τοίχου H_0, ήτοι την απόσταση από την κρίσιμη διατομή όπου αναπτύσσεται η μέγιστη ροπή, μέχρι το σημείο μηδενισμού της ροπής.

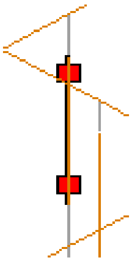
- **Παράλληλα στον οριζόντιο αρμό** (περί τον οριζόντιο άξονα)



Σχήμα Σ7.1.6: Υπολογισμός αντοχής τοίχου σε κάμψη (εκτός επιπέδου δράση) σύμφωνα με την θεώρηση ανενεργού περιοχής

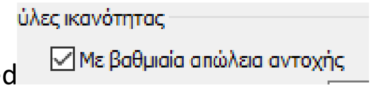
SCADA Pro 20 Structural Analysis & Design

A red square may also be displayed, indicating that the end has failed in tension.



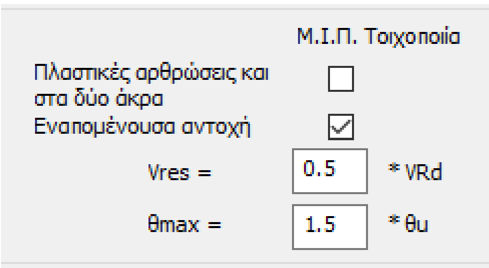
In the tensile test, the tensile strength given in the masonry library is taken into account, i.e. the pile and the lintel do not fail immediately but first exhaust their tensile strength.

Now, the measure of the rotation of the plastic joint is obtained as the absolute value of the algebraic difference of the rotation of the auxiliary node (the node created at a distance of 1 cm from the node of the end of the member) minus the rotation of the node of the end of the member.

When I have selected 

in this step the curve stops and starts a new one.

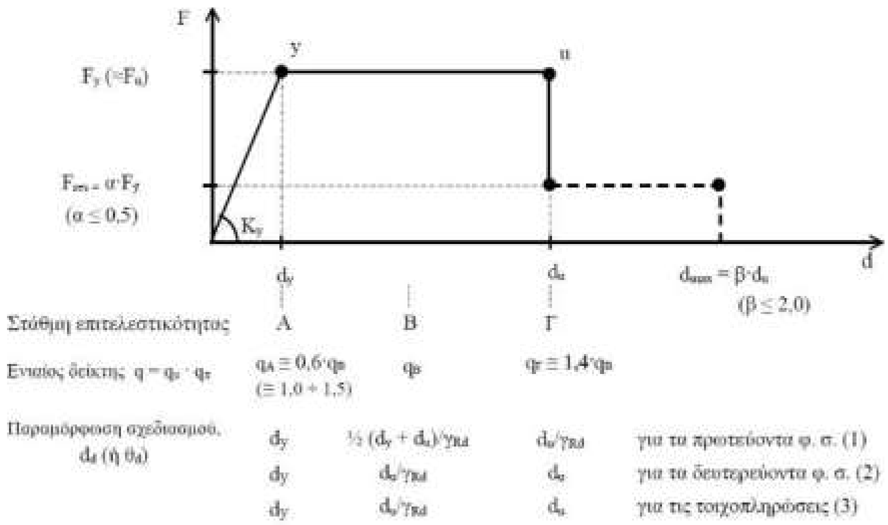
The vector has now been modified with the corresponding degrees of freedom at the ends of the members, i.e. joints have been applied to these ends if we have NOT checked the "Residual Strength" option



If we have checked it, then these members in the new curve participate normally without joints, but with two basic modifications in their skeletal diagram:

1. Their strength, which, as we said, is the smallest of the 4 (3 in-plane and 1 out-of-plane) and calculated in the λ procedure, is reduced by the percentage indicated in the Vres field and is the remaining strength after failure of the end of the element.
2. The limit θu is incremented by the value indicated in the θmax field.

The fact is that the above is not provided for by the KADET but by the KANEPE as follows



Σκελετικό Διάγραμμα Συμπεριφοράς
(για τα επιμέρους δομικά στοιχεία, ή το δόμημα – ως σύνολο)

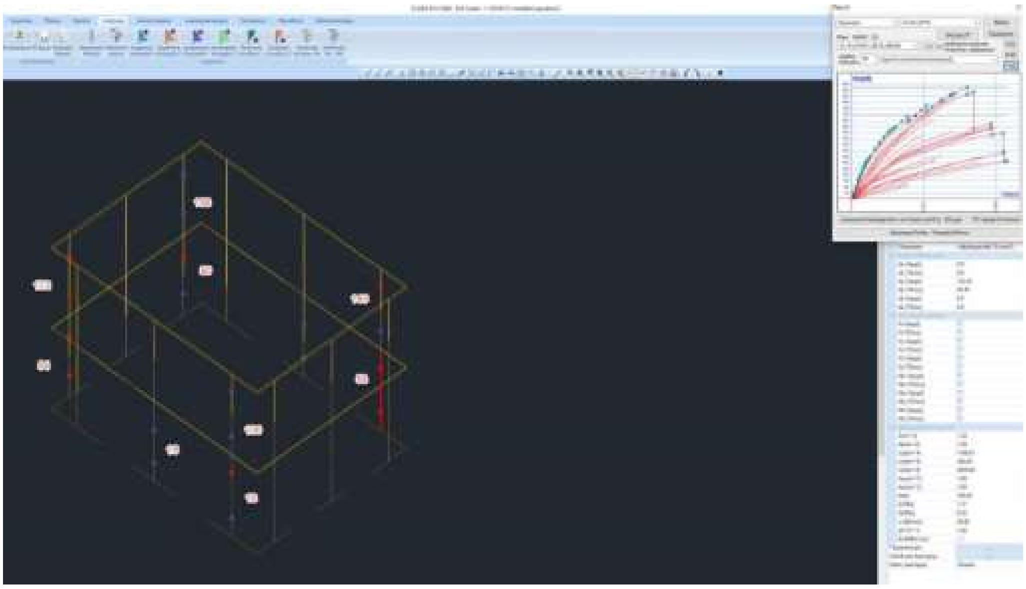
Anyway, and because the limits are not strictly defined, that's why it is as an option and as a function and as a price.

The results, as they appear in the printout, include only those elements (passes or transoms) in which a plastic joint has been created in one of its ends in a step.

3.2 EXAMPLE

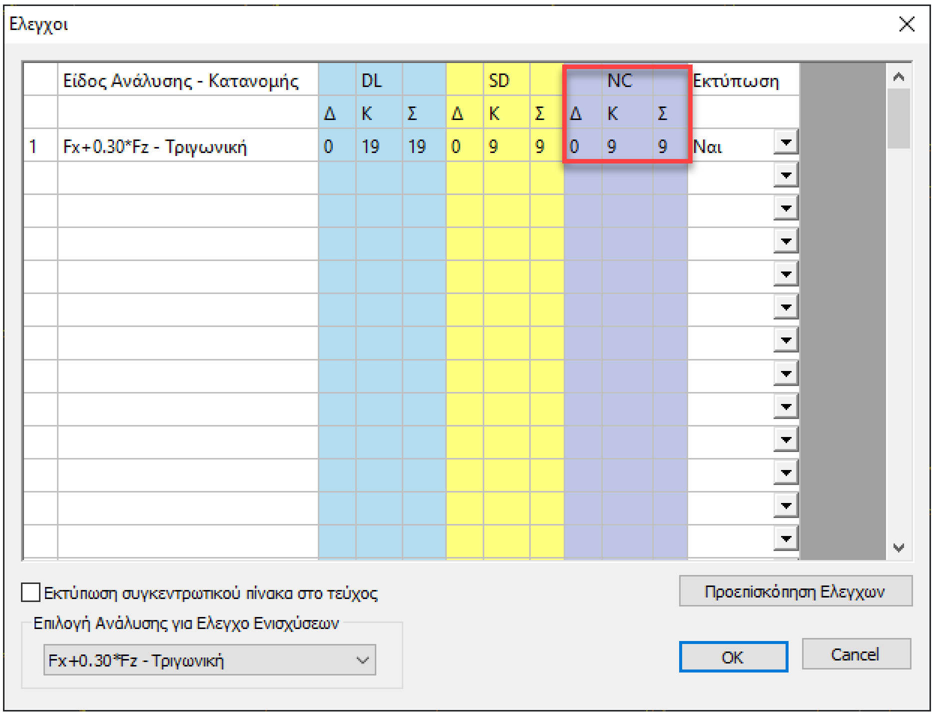
Let's look at an example of one of these results.

For a performance level C corresponding to step 32 the form of the failure of the structure is as follows



In the parameters we have unchecked the option "Plastic joints at both ends" and so at the ends of some elements there are different kinds of failures like in member 80. In total we have 9 pins that fail but in red we have 9 ends which are all square which means they fail in tension.

This is also shown in the summary table



3.3 AUDIT PREVIEW

The printout includes only the items that fail

Μέλος	Κόμβ.	Περιορισμένες Βλάβες (Α - DL)			Σημαντικές Βλάβες (Β - SD)			Οιονεί Κατάρρευση (Γ - NC)		
		θd	θc=θy		θd	θc=θu/γRd		θd	θc= 4/3*θu/γRd	
16	8	-0.59	0.00	Οχι	-0.70	4.64	Ναι	-0.68	5.05	Ναι
				(1)	0.150		(1)	0.134		(1)
	11	0.58	0.00	Οχι	0.50	4.64	Ναι	0.56	5.05	Ναι
				(1)	0.108		(1)	0.112		(1)
18	14	-0.86	0.00	Οχι	-0.94	2.79	Ναι	-0.94	3.69	Ναι
				(1)	0.336		(1)	0.254		(1)
	17	-0.47	0.00	Οχι	-0.52	3.79	Οχι	-0.52	5.03	Οχι
					0.136			0.103		
58	35	-0.77	0.00	Οχι	-0.96	2.28	Οχι	-0.96	3.04	Οχι
					0.420			0.315		
	37	-0.99	0.00	Οχι	-1.05	2.28	Οχι	-1.07	3.04	Οχι
					0.460			0.351		
60	40	0.00	0.00	Ναι	-0.63	2.76	Ναι	-0.63	3.60	Ναι
					0.228		(1)	0.176		(1)
	43	0.81	0.00	Οχι	0.50	2.76	Ναι	0.49	3.60	Ναι
				(1)	0.182		(1)	0.135		(1)
80	51	-0.64	0.00	Οχι	-0.80	4.02	Ναι	-0.80	3.05	Ναι
				(4)	0.200		(4)	0.262		(1)
	53	0.27	0.00	Οχι	0.27	3.27	Οχι	0.32	4.38	Οχι
					0.082			0.072		
82	56	0.00	0.00	Ναι	0.00	0.00	Ναι	0.00	5.33	Ναι
					0.000			0.003		
	59	0.00	0.00	Ναι	0.00	0.00	Ναι	0.64	4.79	Ναι
					0.000			0.133		(4)
84	62	-0.76	0.00	Οχι	-0.86	2.28	Οχι	-0.86	3.04	Οχι
					0.377			0.283		
	64	0.24	0.00	Οχι	0.24	2.28	Οχι	0.28	3.04	Οχι
					0.103			0.093		
106	17	-1.04	0.00	Οχι	-1.04	2.33	Ναι	-1.07	3.11	Ναι
				(1)	0.447		(1)	0.344		(1)
	72	-0.89	0.00	Οχι	-0.93	2.33	Ναι	-0.94	3.11	Ναι
				(1)	0.398		(1)	0.302		(1)
146	37	-1.59	0.00	Οχι	-1.67	2.48	Ναι	-1.68	3.13	Ναι
				(1)	0.675		(1)	0.538		(1)
	82	-1.74	0.00	Οχι	-1.74	3.48	Οχι	-1.70	4.46	Οχι
					0.499			0.382		
168	53	-0.24	0.00	Οχι	-0.16	1.67	Ναι	-0.15	2.13	Ναι
				(1)	0.096		(1)	0.068		(1)
	90	0.11	0.00	Οχι	0.07	1.67	Ναι	0.10	2.13	Ναι
				(1)	0.039		(1)	0.046		(1)
172	64	-0.21	0.00	Οχι	-0.21	3.13	Οχι	-0.28	3.57	Οχι
				(1)	0.067			0.079		
	96	-0.08	0.00	Οχι	-0.12	3.13	Οχι	0.11	3.57	Οχι
					0.037			0.032		

A total of 9 failures. In item 18, at the end the indication is "No" despite the fact that the ratio is 0.315<1. The reason is that its indication in the 3D is red square which means it failed in tension. This is why there is no number under 'No' to indicate the type of failure.

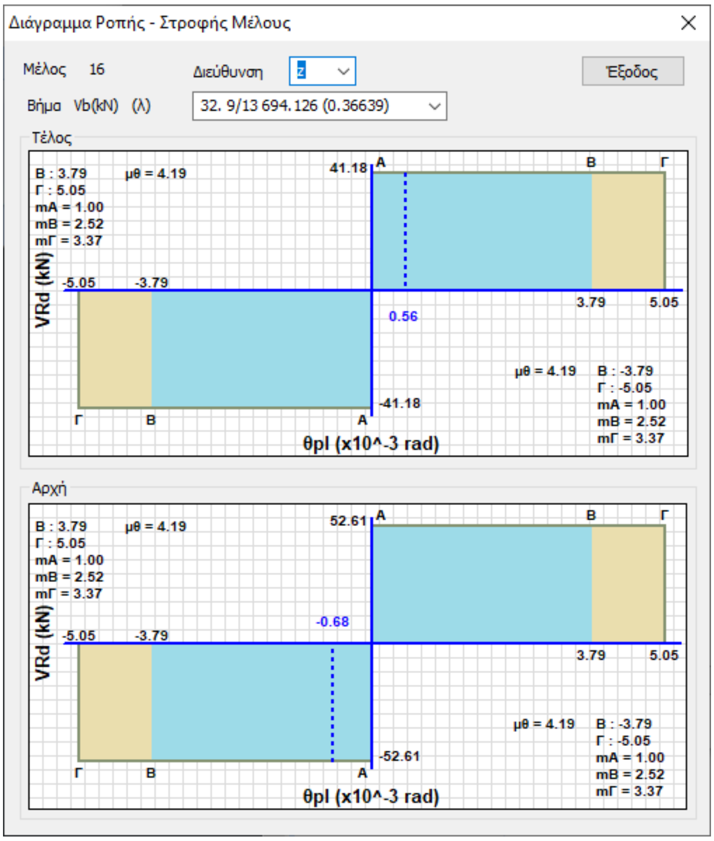
Member 16 which has two blue failures appears to be in the first level performance has failed by in-plane bending, but has not failed in B and C not (ratios less than unity).

Note that the printout now includes (also for concrete) ONLY those elements that up to the step corresponding to performance level C have developed at one or both ends a plastic joint. That is, those which in all steps do NOT develop a plastic joint at either end and those which do, but at a step greater than the step corresponding to performance level C are NOT printed.

3.4 SKELETAL DIAGRAMS

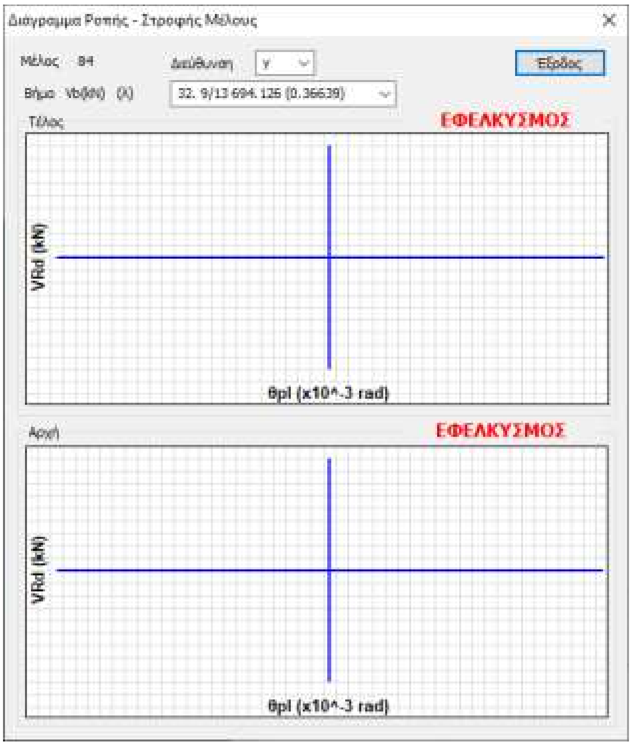
Let's look at the skeletal diagrams of some of the above elements.

For member 16, its skeleton diagram for step 32 (C level) has the following form



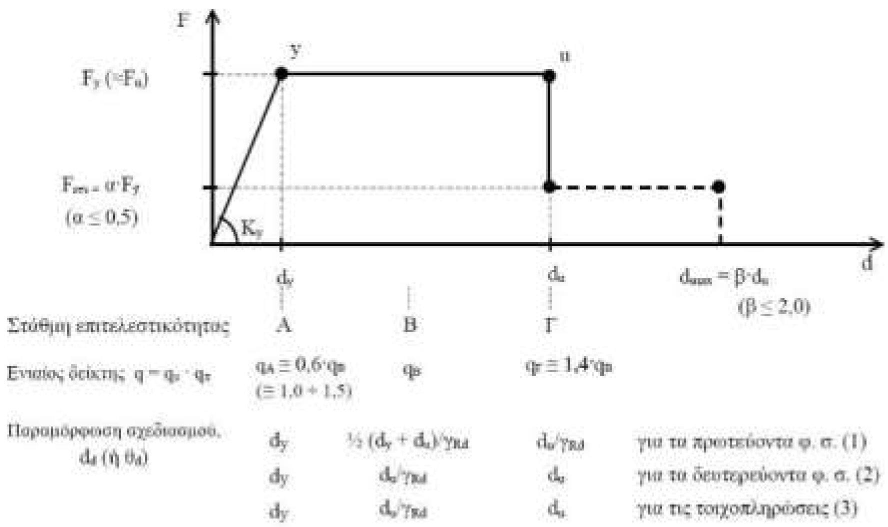
Start price -0.68, end price 0.56. These are the same values shown in the printout. These values are for torque about local z i.e. in-plane bending as indicated by the number 1 in the printout.

Member 84 has failed at both ends in tension. Its skeletal diagram is for step 32 this



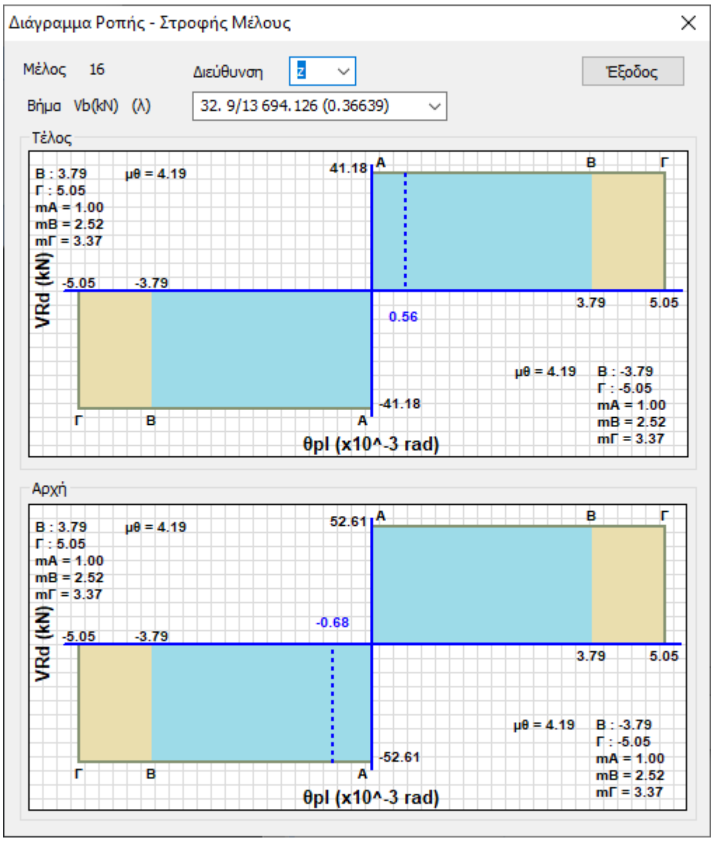
As far as skeletal diagrams are concerned:

- In the skeletons of masonry members and in the skeletons of concrete membersthe skeleton is calculated with the axial of each step.
- The skeletal diagram is a strength diagram of the end of the member. critical sizes to be drawn are Fy, θy and θu, as defined in the CANEP



Σκελετικό Διάγραμμα Συμπεριφοράς
(για τα επιμέρους δομικά στοιχεία, ή το δόμημα – ως σύνολο)

In Scada θy or dy is 0. What is displayed is:



The inclined anion elastic branch does not appear, so θy=dy=0. It was preferred to show, for better overview, the boundaries of the performance stations B and C and which are

- θDL=θy=0
- θSD=θu/γRd
- θNC=4/3*θu/γRd

For an even better overview of the results, detailed results have been created for each step with the λ ratios calculated for each end of the
Membership. Let's look at the example:

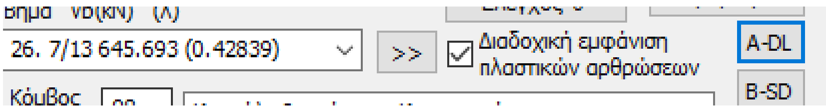
In the previous printout for member 80 at the beginning and for performance level A we had a failure at the end of the

Διατμητική αστοχία λόγω διαγώνιας ρηγμάτωσης (3)
ΑΣΤΟΧΙΑ ΕΚΤΟΣ ΕΠΙΠΕΔΟΥ (γRd=2.0)
Αστοχία από κάμψη κατά μήκος των οριζόντιων αρμών (4)

Μέλος	Κόμβ.	Περιορισμένες Βλάβες (A - DL) θd	θc=θy		Σημαντικές Βλάβες (B - SD) θd	θc=θu/γRd		Οιονεί Κατάρρευση (Γ - NC) θd	θc= 4/3*θu/γRd	
16	8	-0.59	0.00	Οχι	-0.70	4.64	Ναι	-0.68	5.05	Ναι
				(1)	0.150		(1)	0.134		(1)
	11	0.58	0.00	Οχι	0.50	4.64	Ναι	0.56	5.05	Ναι
				(1)	0.108		(1)	0.112		(1)
18	14	-0.86	0.00	Οχι	-0.94	2.79	Ναι	-0.94	3.69	Ναι
				(1)	0.336		(1)	0.254		(1)
	17	-0.47	0.00	Οχι	-0.52	3.79	Οχι	-0.52	5.03	Οχι
					0.136			0.103		
58	35	-0.77	0.00	Οχι	-0.96	2.28	Οχι	-0.96	3.04	Οχι
					0.420			0.315		
	37	-0.99	0.00	Οχι	-1.05	2.28	Οχι	-1.07	3.04	Οχι
					0.460			0.351		
60	40	0.00	0.00	Ναι	-0.63	2.76	Ναι	-0.63	3.60	Ναι
					0.228		(1)	0.176		(1)
	43	0.81	0.00	Οχι	0.50	2.76	Ναι	0.49	3.60	Ναι
				(1)	0.182		(1)	0.135		(1)
80	51	-0.64	0.00	Οχι	-0.80	4.02	Ναι	-0.80	3.05	Ναι
				(4)	0.200		(4)	0.262		(1)
	53	0.27	0.00	Οχι	0.27	3.27	Οχι	0.32	4.38	Οχι
					0.082			0.072		
82	56	0.00	0.00	Ναι	0.00	0.00	Ναι	0.00	5.33	Ναι
					0.000			0.003		
	59	0.00	0.00	Ναι	0.00	0.00	Ναι	0.64	4.79	Ναι
					0.000			0.133		(4)
84	62	-0.76	0.00	Οχι	-0.86	2.28	Οχι	-0.86	3.04	Οχι
					0.377			0.283		
	64	0.24	0.00	Οχι	0.24	2.28	Οχι	0.28	3.04	Οχι
					0.103			0.093		
106	17	-1.04	0.00	Οχι	-1.04	2.33	Ναι	-1.07	3.11	Ναι
				(1)	0.447		(1)	0.344		(1)
	72	-0.69	0.00	Οχι	-0.93	2.33	Ναι	-0.94	3.11	Ναι
				(1)	0.398		(1)	0.302		(1)
146	37	-1.59	0.00	Οχι	-1.67	2.48	Ναι	-1.68	3.13	Ναι
				(1)	0.675		(1)	0.538		(1)
	82	-1.74	0.00	Οχι	-1.74	3.48	Οχι	-1.70	4.46	Οχι
					0.499			0.382		
168	53	-0.24	0.00	Οχι	-0.16	1.67	Ναι	-0.15	2.13	Ναι
				(1)	0.096		(1)	0.068		(1)
	90	0.11	0.00	Οχι	0.07	1.67	Ναι	0.10	2.13	Ναι
				(1)	0.039		(1)	0.046		(1)
172	64	-0.21	0.00	Οχι	-0.21	3.13	Οχι	-0.28	3.57	Οχι
				(1)	0.067			0.079		
	96	-0.08	0.00	Οχι	-0.12	3.13	Οχι	0.11	3.57	Οχι
					0.037			0.032		

The type of failure is (4) i.e. out-of-plane bending. In the file folder and within the subfolder of this analysis scenario, a
an envelope called "RESULTS". There are txt files there, each corresponding to each step of the analysis. The name of the file indicates the number of the
curve and the step corresponding to that curve and is of the format PUS_RESULT2_7.TXT. This means that this file contains the results from the second curve, from its seventh step.

We return to the example and first look at which step and curve the A level corresponds to. It is step 26



representing the 13 step of the 7 curve. So we will open the corresponding file and look for member 80.

The results are these:

Column name [80 9] Recalc

l1= 0.154268 VRd_f= 4.444897 (scale=1.000000)

l2= 2.873666 VRd_v1= 82.798276 (scale=1.000000)

l3= 1.661277 VRd_v2= 47.865983 (scale=1.000000)

l4= 0.137181 VRd_2= 1.859916 (scale=1.000000)

l1= 10000.000000 VRd_f= 0.000000 (scale=1.000000)

l2= 2.699420 VRd_v1= 77.777778 (scale=1.000000)

l3= 1.457687 VRd_v2= 42.000000 (scale=1.000000)

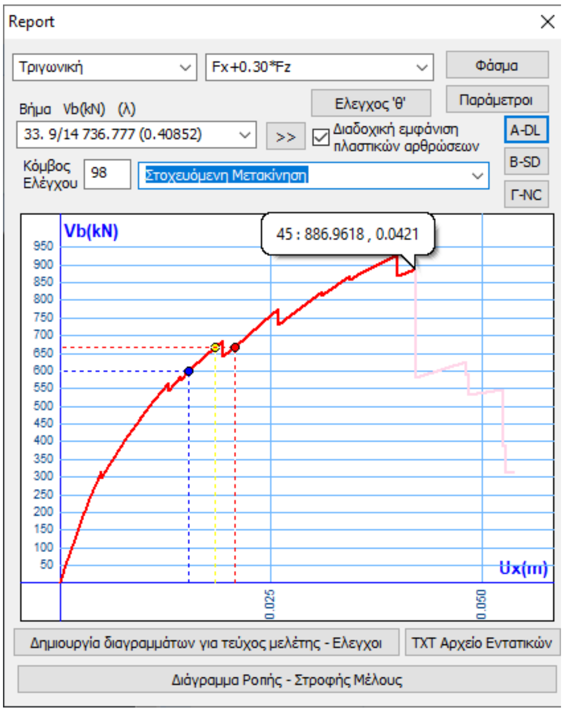
l4= 10000.000000 VRd_2= 0.000000 (scale=1.000000)

The first 4 l's represent the beginning and the next 4 the end. So we see that the smallest λ is λ4=0.137181 which indeed corresponds to the out-of-plane bend.

The value 10000 at the end means that the corresponding strength was not calculated for this end because end has tensile strength and the end actually failed in tension (red square). The strengths that are not calculated when there is
tensile force is the two in-plane and out-of-plane bending forces (1) and (4).

3.5 CONTROL AT THE LEVEL OF THE INSTITUTION

This check is for the whole vector and compares the movement dm which is the movement corresponding to the last step of the pushover

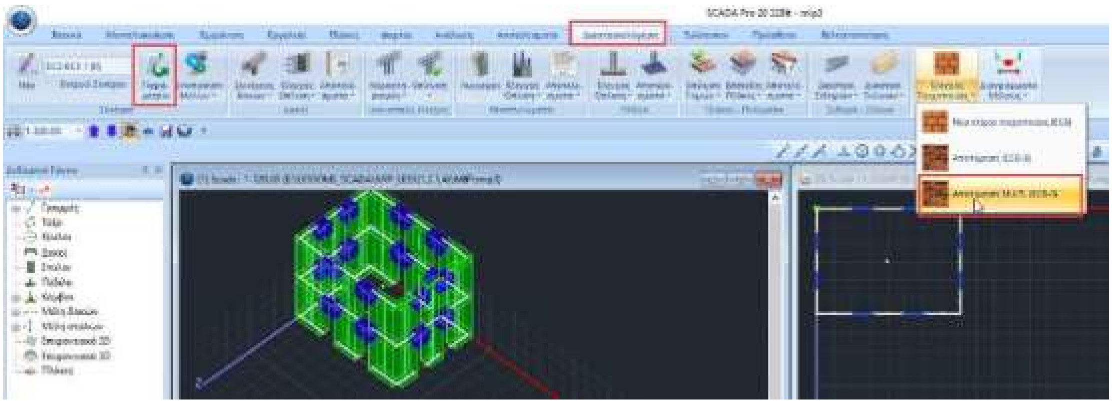


With the targeted movements corresponding to the performance levels.

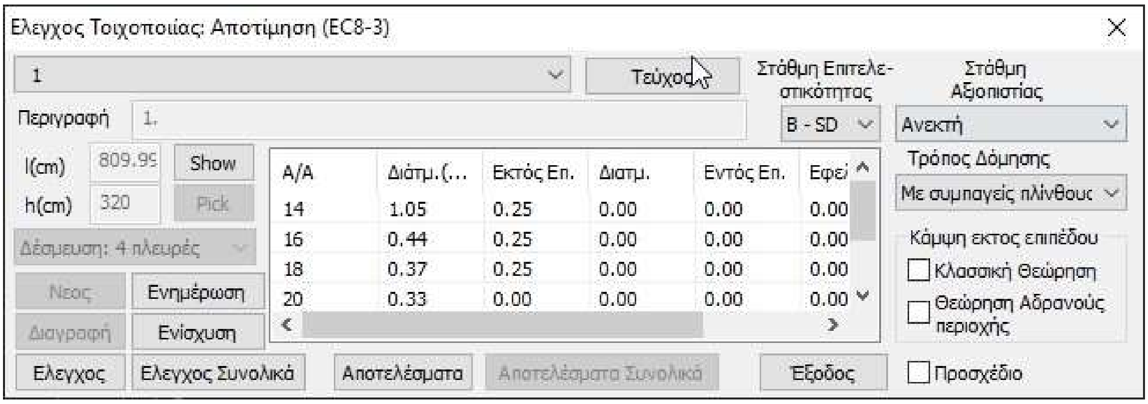
In this example the value is dm=4.21 cm. This is the maximum displacement that the carrier can withstand before it collapses. This is compared to the targeted movement of each performance level dt and must of course be greater, i.e. the requirement (target) is less than the "resistance". In our example

```
              ΕΛΕΓΧΟΣ ΕΠΑΡΚΕΙΑΣ ΦΟΡΕΑ ΣΕ ΟΡΟΥΣ ΠΑΡΑΜΟΡΦΩΣΕΩΝ

                                  C0     C1     C2     C3  Se(T) (m/sec2)  Te(sec)
Περιορισμένες Βλάβες(A-DL)   1.20   1.22   1.00   1.00        4.71        0.30
Σημαντικές Βλάβες     (B-SD)   1.20   1.22   1.20   1.00        4.71        0.30
Οιονεί Κατάρρευση     (Γ-NC)   1.20   1.22   1.35   1.00        4.71        0.30

                                  Στοχευόμενη  Συνολική
                                  Μετακινηση   Μετακινηση      λόγος
                                     dt(cm)      dm(cm)       λ=dt/dm       ΕΠΑΡΚΕΙΑ
Περιορισμένες Βλάβες(A-DL)        1.53        4.21          0.36           Ναι
Σημαντικές Βλάβες     (B-SD)        1.84        4.21          0.44           Ναι
Οιονεί Κατάρρευση     (Γ-NC)        2.07        4.21          0.49           Ναι
--------------------------------------------------------------------------------
```

4. VALUATION OF THE M.I.P.

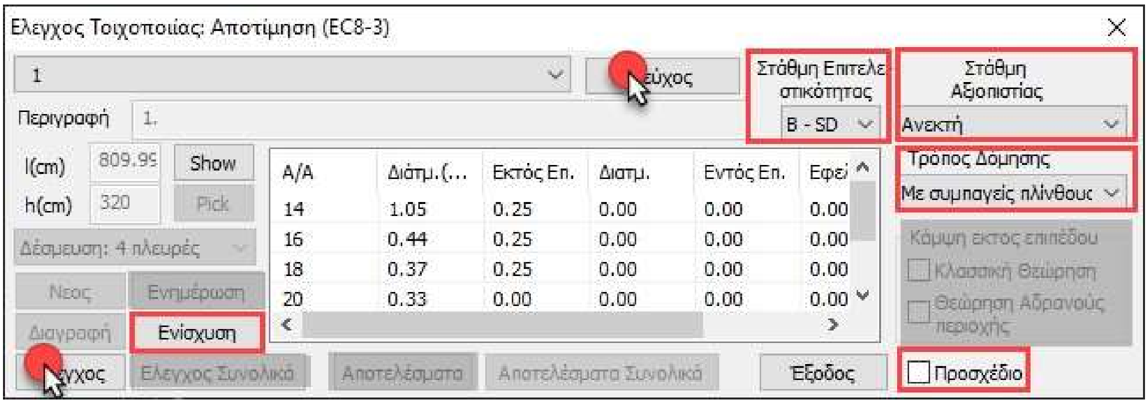


Within the field of sizing and after calculating the combination of the inelastic analysis in Parameters, through the command Valuation of M.I.P.the possibility of placing reinforcements on walls that have been simulated with the equivalent frames.

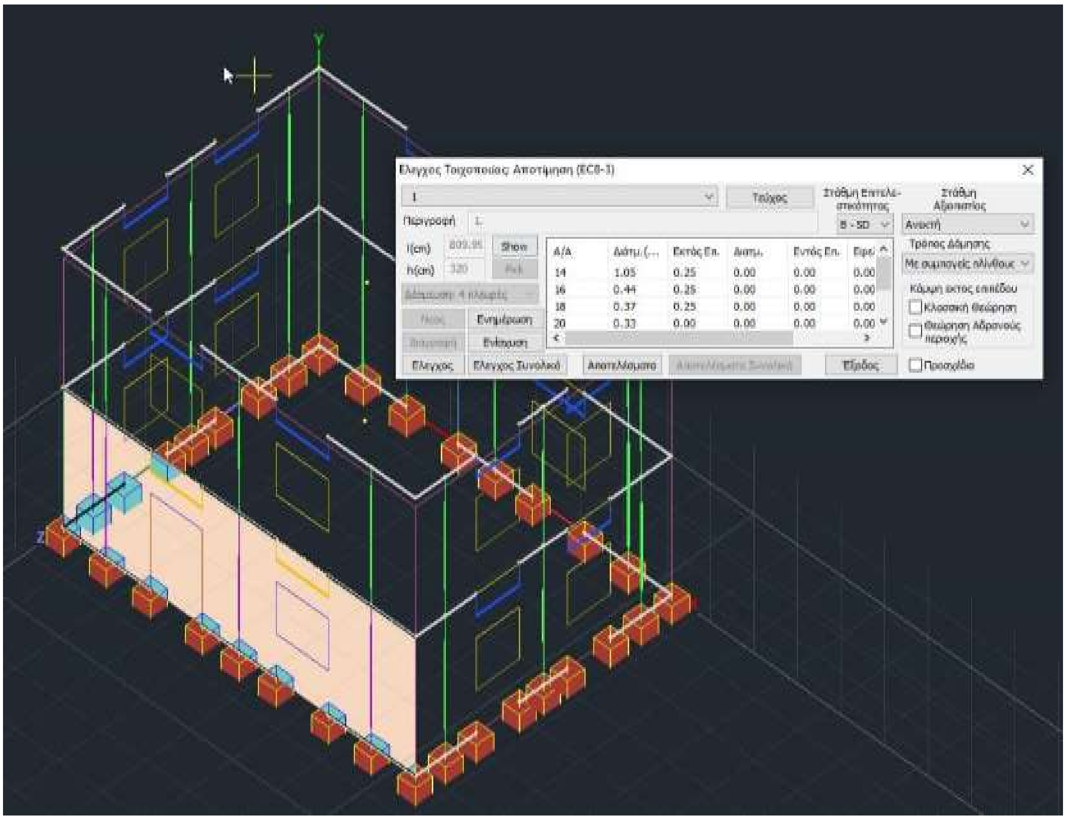


The dialog box is similar to the one for load-bearing masonry with finite surface elements.

The walls are now already defined and the user is asked to select only as many as are shown schematically in the image below:

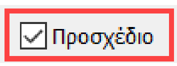


You can select one of the walls in the list and then "Show" to display it in the 3-dimensional view.



You select the Performance Level, the Data Reliability Level and the Building Mode (by CDET).

SCADA Pro offers the possibility to evaluate the masonry according to the draft of the KADET.

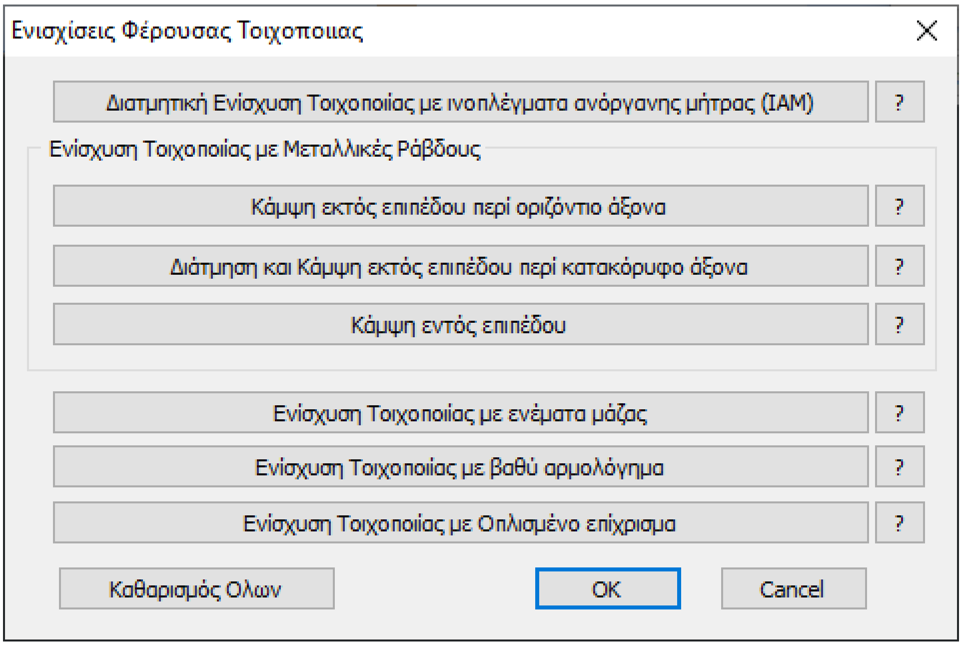


If the option "Draft CADET" is checked and all checks are done based on the CADET.

Then select the Aid command to enter the amount needed redesign your operator.

4.1 AIDS - M.I.P.

Selecting the Boost command opens the window of possible boosts.

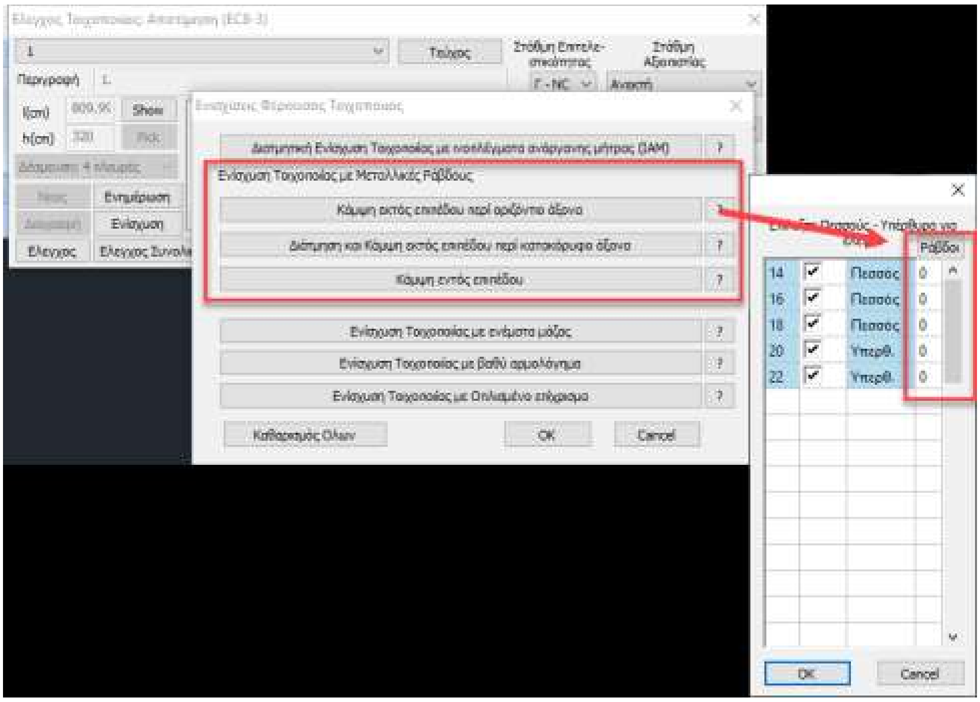


Everything mentioned for masonry with finite surface elements listed in Manualapplies. Dimensioning¨ in chapter 2.2.4 on p.33 and presented in the webinar video entitled: ¨SCADA Pro - 3/6/20 - Valuation of masonry with surface elements¨ which can be found at https://www.youtube.com/watch?v=ro7w7MOxMhY&list=PLSYOATQuvG68jro3H29zOfZy6y1AB2p9G&index=5

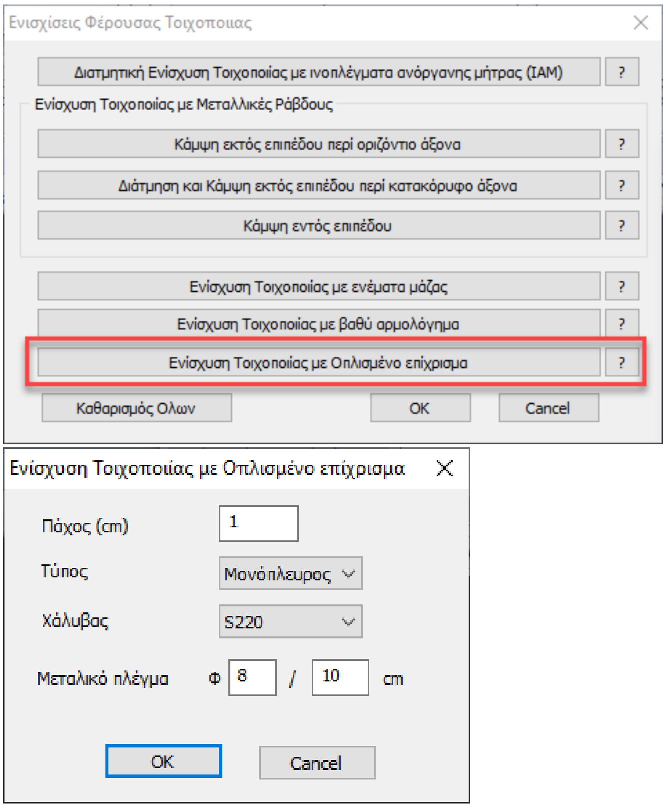
In addition you will find:

1. In reinforcements with metal bars the possibility to set the number of bars for pickets and lintels different from the number of modeling. In case you do not intervene manually, the program will take the number of bars of the modeling.

YouTube GR *Watch on SCADA Pro's YouTube channel the training course entitled4 - M.I.P : Masonry reinforcement with M.I.P¨ https://www.youtube.com/watch?v=JRwOZTbizng&list=PLSYOATQuvG6_OzCHn7-13c04NayNSLu7l&index=4*



2. Reinforcement with reinforced coating



You set its attributes and continue the procedure as described in the example on page 42 in Manual ¨10d. Sizing¨.