

Example 4 Assessment and redesign according to EC8part3







Contents

I. 11.	PREFACE SCADA PRO	5 5
		5
?	CONDITIONS	5
?	INTRODUCTION	5
III.	GENERAL DESCRIPTION	7
?	Geometry	6
?	Materials	6
?	Design Regulations	6
?	Assumptions about loads - analysis	6
1 st STEP		
1.1	L Creation of a new case study	8
1.2	2 Modelling	8
1.3	3 Slabs - Loads	8
1.4	4 Analysis	8
1.5	5 Design	9
1.6	5 Interaction curves and surfaces	11
	1.6.1 Calculation of the structural members' resistance (Pushover)	11
	1.6.2 Detailing	12
	1.6.3 Calculation of the interaction curves/surfaces M-N	13
2 nd STEI	Ρ	
2.1	L Preface	23
2.2	2 Introduction	23
2.3	8 Preliminary analysis	24
3 rd STEF	P	
3.1	L Nonlinear Analysis Scenarios	28
3.2	2 Run scenarios	28
3.3	8 Nonlinear Analysis (PUSHOVER) Combinations	34
3.4	Nonlinear Analysis (PUSHOVER) Results	35
	3.4.1 Capacity Curve	38



	3.4.2 Idealized Capacity Curve	38
	3.4.3 Target Displacement	39
	3.4.4 Representation of the structure	40
3.5	Non Linear Analysis (PUSHOVER) Checks	43
3.6	Seismic Force	46

4thSTEP

4.1	Colu	mns-walls reinforcement	49
	4.1.1	Rehabilitation of columns-walls	49
	4.1.2	Concrete jacket for columns-walls	51
	4.1.3	FRPs – laminates of columns-walls	57
	4.1.4	Protection of columns-walls	61
4.2	Bear	ms reinforcement	63
	4.2.1	Additional concrete layers – Concrete jackets	65
	4.2.2	Steel laminates – FRPs	69



I. PREFACE

The analysis methods applied for the assessment and the seismic redesign of existing reinforced concrete structures are the elastic methods; static or dynamic, as well as the inelastic methods (material nonlinearity); static or dynamic.

The elastic methods are based on the linear relationship between stresses and deformations for the structural elements, while the nonlinear behavior of the structure is taken into consideration through approximate methods (e.g. global behavior factor q or local ductility factors m). The elastic analysis methods is simpler in implementation, however are less accurate compared to the inelastic analysis methods.

On the other hand, the inelastic analysis methods contribute to a better supervision and assessment of the structural behavior of the structure, by identifying not only the failure mechanisms but also the progressive failure. So, the deformation of the structural members and their remaining overstrength can be checked, as well as the structural ductility of the structure. So, the nonlinear analyses lead to a more reliable and safe structure. The nonlinear dynamic analysis (meaning the time-history analysis with direct integration of the nonlinear equations of motion) is the most reliable analysis method of structures.

In the nonlinear analysis the seismic action is described by real seismic records or artificial accelerograms of peak ground acceleration [1]. However, the simulation of the cyclic post-elastic behavior of the structural members is under research and experimental verification. Also, the accelerograms used for the analysis must be selected as appropriate to represent the seismicity of the region reliably.

Consequently, the engineer that conducts the assessment and the redesign of an existing structure applying nonlinear dynamic analysis must be experienced. Since the nonlinear analysis is complex and computationally time consuming, especially for 3D multi-storey structures (notice that since that method is nonlinear, the mode superposition principles is not applicable), the nonlinear dynamic analysis is not a method of general use.

Instead, the nonlinear static analysis leads to results that are less reliable that the one of the nonlinear dynamic analysis, but more representative about the nonlinear behavior of the structure, that cannot be considered with accuracy, when linear methods are applied. In case that an incremental later load is applied of a certain pattern until collapse, the nonlinear static method is considered as Pushover analysis. The Pushover analysis is not as complex as the nonlinear dynamic analysis.

The nonlinear static analysis is not a modern analysis method. However the last decades, the researchers developed structural models that can simulate the nonlinear behavior of the critical regions of the RC structural members. This is the reason why the nonlinear analysis is so widely applicable for the assessment and the redesign of the existing structures.

II. SCADA PRO

SCADA Pro incorporates the nonlinear static analysis. Considering the default parameters, all demanding nonlinear static analyses are applied, as well as, all the appropriate design checks are performed for each performance objective according to EC8 part 3 and the corresponding results are presented with a direct and supervisory way.



The nonlinear static analysis provides information and supervision of the order of occurrence of the plastic hinges in the critical regions of the structural members. The engineer can study the gradually deformed shape of the structure step by step and so can easily detect the weak regions or under failure regions of the structure.

That method is used for the assessment of the structural behavior and the comparison of the capacity of the existing structures with the structural demand defined in the selected performance objective, according to EC8 part 3. This method can also be applied for the redesign of existing structures or the calculation of the ratio a_u/a_i according to EC8 for new structures. This ratio is calculated through the capacity curve and it is used for the definition of the behavior factor q.

• CONDITIONS

Important precondition for applied the nonlinear analysis scenario is the existence of steel reinforcement in the structural members. The steel reinforcement is calculated, when the structural members are designed ONLY according to Eurocode 2, considering the materials of steel and concrete with modified strength. The modified concrete and steel material correspond to the type of the old materials of the existing structure. The materials that will be used to approach the properties of the existing materials should NOT be type B or STI (old type of materials) but the properties of the new materials should be modifies with the corresponding safety factors as appropriate.

• INTRODUCTION

The user manual contains guidelines for the application of a nonlinear analysis with SCADA Pro.

The test case presented in the current manual is a three storey reinforced concrete (RC) frame structure, considered as existing structure. The main objective is the assessment and the redesign of the structure according to the selected performance objective. The presentation of this test case is the guide for a new user to become familiar with the software SCADA Pro.



III. GENERAL DESCRIPTION

• Geometry

The reinforced concrete (RC) building examined is a three storey frame structure with basement. The foundation is composed by strip foundation and a single footing under the stair core.



• Materials

The material of the structural elements is reinforced concrete of type C20/25 with steel of type B500C for the steel reinforcement.

In case the type of materials of the existing structure is B and STI (old type of material), then they will not be used with their strength values considered, but they will be adjusted to the strength values of the new type of materials modified with the corresponding safety factors.

• Design Regulations

Eurocode 8 for the seismic loads considered. Eurocode 2 for the structural design. Code of Structural Interventions (KAN.EPE.2013) for the retrofit of the structural elements.

- Assumptions about loads analysis
- Dynamic Response Spectrum analysis method

The load cases according to this analysis method are the following:

(1) G (permanent)

(2) Q (imposed)

- (3) EX (seismic loads in X direction derived from dynamic analysis).
- (4) EZ (seismic loads in Z direction derived from dynamic analysis).



(5) Erx \pm (torsional moments derived from the seismic loads imposed with an accidental eccentricity versus the nominal location of the mass of the storey $\pm 2e\tau zi$).

(6) Erz \pm (torsional moments derived from the seismic loads imposed with an accidental eccentricity versus the nominal location of the mass of the storey $\pm 2e\tau xi$).

(7) EY (vertical seismic load in Y direction derived from dynamic analysis).

- Nonlinear static analysis (Pushover analysis)

The structure is subjected to an incremental later load of a certain pattern until collapse.



1. st STEP: PRELIMINARY PROCEDURE

The input data of an existing structure are inserted with the same procedure as for a new designed structure.

The analytical description about the drawing, the analysis and design of a RC structure is presented in the manual entitled "Case study: Reinforced Concrete Structure".

The main steps preceded the Pushover analysis are presented summarily:



Insertion of the structural elements in the interface by using the appropriate tools (dxf/dwg, templates, draft, modelling).

1.3 Slabs - Loads

	i 🛱 🏀	• 🕇	₽ ‰ ;		- 6	•								Untitled ·
-	Basic	Mod	leling	View	Tools	Slabs	Loads	Analysis	Post	t-Processor	Memb	ers Desig	n [Drawings-De
Parame	eters Iden	Q tification	Zoellner	Delete	لوچی Renumbe	ring Mo	deling	Model	Supports	ALL Thickness	Properties	x-x Insert	Real Street	Checks
	In	* sert	Slab				* cori	espondence Edit	*	*		- Str	• ins	

Insertion of slabs and loads, using the corresponding tools.

1.4 Analysis

	🏹 🖓 📚 🗢 👔 🖡 🔚 🎞 2.600.00 🔹 🕢 🕫											TestK41	- ScdaPro	o 32Bit				
		Basic	Modeling	View	Tools	Slabs	Loads	Analysis	Post-Proc	essor N	lembers Des	ign [Drawings-D	etailing	Addons			
Z,	E	C-8_Gre	ek Dynamic (7)		G	\	1		Ī	•	ľ	Z	Ľ	Z	x	z	• • ••	• !
New	/ `	Ac	tive Scenario		Run	Combinati	ons Check	s Seismic Force	Mass Distribution	Mass Center Deviation	s Bending Stiffness X	Bending Stiffness 2	Shear Z Stiffness)	Shear Stiffness Z	Seismic Forces Distribution X	Seismic Forces Distribution Z	Po Centers Deviation	Po - CM Deviation
			Scenarios				Results						1	View				

A preliminary analysis of the structure must be applied according to **Eurocode 8 (static or dynamic EC8 scenario)**.





- As it was aforementioned, important prerequisite for the implementation of the nonlinear analysis scenario is the existence of the steel reinforcement, which is calculated only when the cross-sections are designed according to Eurocode 2 considering the modified strength value of the materials.
- As the materials of the existing structure are considered the modern materials with modified properties concerning the corresponding safety factors. The materials of the existing structure are not type of B and STI (old type of materials).

If the structure studied has materials of type **B** and **STI**, then for the definition of the materials in the design parameters field and before the preliminary design, <u>use the properties of the modern</u> <u>materials modified with the safety factors according to EC8 part3</u>.

Steel Reinforcement Capacity Design Steel									
Combinations	Slabs	Beams	Columns	Footings					
Concrete : C20/25	Co	ncrete ×	Ste	el (Main) 🛛 🗡					
Checks Bending ✔ Axial Force par Shear Conn. Angle a Capacity Design ☐ Shear Ampli	Type Constants Fck (Mpa) ycu ycs Fctm (Mpa)	20 1.5 1 2.2	Type Constants Es (Gpa) Fyk (Mpa ysu yss	B500C V 200 500 1.15 1					
Torsion Check	TRd (Mpa)	0.25	Max Defo	nmations					
Serviceability Crack control Deflection contro	ες (N,M) ες (N)	0.0035	ОК	Cancel					
Soil Failure (Beams	ОК	Cancel	√ ofr. 22	5 (kN/M2)					
Check			✓ ofr. 22	5 (kN/M2)					

The values of the compressive strength Fck of the concrete $\kappa \alpha \iota$ the yield strength Fyk for the steel are the characteristic values.

- According to the *Code of Structural Interventions* (KAN.EPE.2013), the strength value are experimentally defined median values of fcm and fym for concrete and steel, respectively.
- Especially for the existing materials when the design checks are regarding deformation the characteristic values are equal to the median values (Fck=fcm and Fyk=fym). So, in the above fields you will type the corresponding values for concrete and steel for beams and columns.
- As it was aforementioned, the *Code* gives in tables the values of partial safety factors $\gamma_m (\gamma_{cu} and \gamma_{su})$ for the existing materials and the design checks regarding deformation.



EXAMPLE:

Assume that the existing structure's construction materials are B160 and STI with median strength values:

Concrete -> fcm=11,0 MPa Steel -> fym=250,0 MPa

> Since the design checks are regarding forces, then: fk=fm-s and fd=fk/γm

In the corresponding fields considering sufficient data reliability level, type: -Concrete

```
s=0.10fcm -> fck=fcm-0.1fcm=0.90fcm
Reliability level -> γc=1.65 so fcd=0.9x11.5/1.65=6.27 Mpa
```

-Steel

```
s=0.15fym -> fyk=fym-0.15fym=0.85fym
Reliability level -> γs=1.25 so fyd=0.85x270/1.25=183.6 Mpa
```

- In the "Fck" and "Fyk" fields type the calculated FINAL VALUES (Resulting after dividing withthe safety factors) and the corresponding fields of the safety factors ycu and ysu type one.
 - Since the design checks are regarding deformation, then: fk=fm and fd=fk/γm

```
- Concrete
Reliability level -> γc=1.20 so fcd=11.5/1.20=9.58 Mpa
```

```
- Steel
```

Reliability level -> γ s=1.20 so fyd=270/1.20=183.6 Mpa

In the "Fck" and "Fyk" fields type the calculated FINAL VALUES (Resulting after dividing withthe safety factors) and the corresponding fields of the safety factors ycu, ycs, ysu and yss type one.

Then continue with the preliminary design and the final step is the adjustment of the existing steel reinforcement to the calculated one on the preliminary design by using the column's and beam's detailing.



Until that step the synoptic procedure is the same followed for the assessment and design of a new RC structure.



1.6 Interaction curves and surfaces

1.6.1 Calculation of the structural members' resistance (Pushover)

Since the preliminary design has been completed and the existing steel reinforcement had been inserted in the structural model and before the creation of the pushover analysis scenario, it is demanding to precede the "**Strength calculation (Pushover**)" by selecting the corresponding command:

Ribbon: "Member Design">"Columns">"Results">"Strength Calculation (Pushover)" Through this command, the program calculates the interaction diagrams between bending moment (M) and axial force (N) for all columns of the structure in all levels.



Select the **Strength Calculation (Pushover)** of the columns and the walls per level or for the total structure.

The program calculates automatically the interaction surfaces N-Mx-Mz of the columns/walls which are displayed directly on the screen.

NOTES:

- In the valuation step, and after reinforcement adapted to existing, for the beams, the calculation of the interaction curves N-M is not necessary, because concerns uniaxial bending and the differences are very small.
- If however, you want the exact calculation of the moments of resistance of the beams, then into the reinforcement detailing of each beam in corresponding field, select the calculation of Ultimate Moment Resistant pressing "Initial".





1.6.2 Detailing

The command "Detailing" opens a new tool with multiple features about modifying the column's steel reinforcement (the analytical description is in the user manual of SCADA Pro), presenting the internal forces diagrams and the corresponding values and deformations, as well as the interaction curves N-M and the interaction surfaces N-Mx-Mz.

	Col	umn Editor – 🗆 🗙
Geometry		
Main Reinforn	Concrete Cover Insert Dimensions X Y XYZ Detailing Continuity Width (cm) 60 Continuity Continu	
Checks Checks Recalculation Joint Check Y = 300.00 + ?? - Copy Paste OK Cancel	Cover (mm) 25 Cover (mm) 25 Design Scales Cross Section 1: 20 Detailing 1: 50 Name K12 - 598 Type COLUMN Dimensions 35 /35 H - Her (cm) 300 /60 Area (cm^2) 1225.00 / 1225.00 pmax % - cm^2 2 .00 ppcalc % - cm^2 1.01 - 12.32 Rebars 80 14	



For the interaction curves/surfaces click the following button



1.6.3 Calculation of the interaction curves/surfaces M-N

It is about the calculation and the display of the interaction surface of the axial load and the ultimate bending moment. It depends on the geometry of the cross-section, the material and the steel reinforcement. It is a 3D surface and represents the envelope of the ultimate biaxial bending resistance and the axial load (My, Mz, N). Also, the strain-stress relationship diagrams are displayed for the steel, concrete as well as the moment – curvature diagram of the cross-section.

• CALCULATION OF THE INTERACTION SURFACE

For the generation of the interaction surface of the selected cross-section, select the buttons "Calc1" or "Calc2".

The difference between the two diagrams refers to the part with negative values (-N) that represents the tension.

-**Calc1:** This command generates the linear diagram of tension, which means \rightarrow lower resistance in tension \rightarrow less favorable conditions

-**Calc2**: This command calculates the median values of tension with result in the curved form and more reliable results about the tension.

Notice: The upper part of the diagram (Compression) is not affected from the previous selection. Both calculation methods ("Calc1" and "Calc2") generate the same diagrams under compression.

	Calc1	Calc2
--	-------	-------



For the graphical display of the horizontal cutting planes of bending moments considering a consistent value of axial load N, activate the checkbox "Horizontal".









HORIZONTAL BAR OF MAXIMUM VALUES

My=-206.891, 206.891 Mz=134.438, -134.438 N=-791.304, 2690.560

In the horizontal bar you see the six maximum values that are derived from the 3D interaction surface.



These values represent the maximum (+) and minimum (-) value of each internal force and they are the pick values of the surface.

NOTE:

The axes system of the bending moments coincides with the local axes system of the column, if you haven't modified the calculated angle beta for each column when the mathematical model is created. The discontinuous line represents the negative values.



N+

• FIND POINTS ON THE SURFACE

Ν	My	Mz	Angle
0	0	0	0
Step	100	N-	N+

This field has multiple uses:

• For the display of the horizontal cutting planes

Type in the field "Step" a value and click the following buttons



With a single click a horizontal the perimeter of a horizontal plane is displayed (a curve colored magenta). The points of the curve are pairs of ultimate bending moments in both main direction of the cross-section (Mx, Mz) for a single value of the axial load (N=constant) and multiple values of the rotation angle of the neutral axis. The field "Step" represents the step in which the axial load axis is run for the display of the corresponding horizontal curves (horizontal cutting planes). By selecting N+, the horizontal curves for the positive values of the axial load are displayed and by selecting N- for the negative values. Furthermore, for each horizaontal curve, the maximum and minimum values of the bending moments My and Mz are listed for the corresponding specific value of the axial load N.

N	My	Mz	Angle				
200	0	0	0				
Step	100	N-	N+				
0	0	0	???				
RMy=0.00,0.00 RMz=-0.00,-0.00							
My=-185 Mz=121.	.54,185. 69,-121.	54 69					

Calc-N,My,Mz



• For the calculation of the ultimate biaxial bending resistance for specific values of the internal forces N-My-Mz given.

Type the values of the internal forces in the corresponding fields N, My, Mz and click the following button Calc-N,My,Mz. Then the program calculates and displays the following graph:



- The point with coordinates (N, My, Mz) is located on the diagram

- The line that connect the start of the axes (0, 0, 0) with the point (N, My, Mz) is drawn (a line colored orange).

- The curve N* is drawn and the corresponding My, max and Mz, max are calculated.



- The ultimate biaxial bending resistance (RMy, RMz) is calculated for the corresponding internal forces (N, My, Mz). These are the red dots on the horizontal curve.

- - The moment-curvature diagram is drawn.
- The moment-curvature diagram is defined for the specific value of the angle of the neutral axis.





Insert the values of the internal forces in the corresponding fields and click the following button

2??? The program locates the point with the corresponding coordinates and draws the line (orange line) that connects that point with the start of the axes (0, 0, 0). Then, the program projects the line to intersect the interaction surface (cyan color) and record the corresponding values of the biaxial bending moment resistance of the intersection point N=193.52 My=96.76 Mz=58.05 (useful values in the Pushover analysis).



• 3D DISPLAY OF THE INTERACTION SURFACE

Activate the checkbox "3D" and select the internal force from the drop-down list for the display with colored display.

Му	~
N	
My	
Mz	
No	

If you select My the surface is colored along the axis Y. The color gradation define the spacing of the cutting planes according to the colored bar on the right. The horizontal bar on the bottom of the screen records the maximum and minimum values of the internal forces.







The colored interaction surfaces gradually for the internal forces Mz and N, respectively.





Select "No" to display more analytic interaction surface, without color gradation.



2nd STEP: PRELIMINARY CHECK

2.1 Preface

In the section where you create the analysis scenarios, when you select "EC8_General", the following types of analysis scenarios appear:

Static	¥
Static Dynamic Nonlinear Linear	
Elastic Dynamic Precheck Static	
Precheck Dynamic Time History Dynamic	

The types:

- Static
- Dynamic

That types of analysis are used for the analysis of new structures according to EC8 and the national annexes.

All the rest types:

- Nonlinear
- Linear Static
- Linear Dynamic
- Preliminary Static
- Preliminary Dynamic

They are used for the assessment and the redesign of the existing structures according to the *Code* of *Structural Interventions* (KAN.EPE.2013) provisions.

2.2 Introduction

The types of analysis scenarios "Preliminary Static" and "Preliminary Dynamic" are two preliminary analysis that are applied to check, if the conditions for the application of linear analysis (static or dynamic), for the assessment and redesign of the structure, are fulfilled. More specifically, the insufficiency index " λ " are calculated that indicate the resistance of the structure under seismic loads.. It is also examined the in-plan and in elevation regularity of the structure.

The provisions of the *Code of Structural Interventions* (KAN.EPE.2013) sets specific conditions for the application of static or dynamic analysis.



For the **elastic static analysis** provides a set of criteria (among other morphological and regularity), that have been implemented in the program and are presented in the form of checks; only the criteria that contain quantifiable variables and could be implemented computationally.

For the **elastic dynamic analysis** the only criterion is the insufficiency index « λ » to be less than or equal to 2.5.

• However, for both methods, the elastic methods can be applied for assessment purposes only, if the permanent loads are increase 15% ($\gamma_{Sd} = 0.15$). So in the Ribbon "Analysis" you can define a preliminary scenario analysis either static or dynamic, which will be executed with the elastic response spectrum and all checks will be performed on the selection of the analysis, based on what has been mentioned above.

Necessary condition for applying a preliminary analysis scenario is the existence of steel reinforcement and the calculation of the biaxial bending moment resistance.

2.3 Preliminary analysis

In the Ribbon "Analysis" select the command "New Scenario".

	So	cenario		×
Renumber Nodes	ing No	~		
Disable		Name		
EC8_Genera EC8_Genera	l Static I Precheck Static	Analysis	EC8_General	*
EC8_Genera	l Precheck Dynam	Туре	Precheck Static	~
CC0_Genera		Properties Elemer	Static Dynamic Nonlinear Linear	
		Load Ca	Elastic Dynamic Precheck Static Presheck Dynamic	
		New	Time History Dynamic	
			Exit	

You create a new scenario "Preliminary Static" or "Preliminary Dynamic".

• For this analysis scenario the stiffness of the structural elements are defined according EC8

Then, you follow the process for applying an analysis scenario.



In the dialog box "Parameters" define the parameters like in the analysis scenarios of EC8:

	EC8 Parameters	X
Seismic Area Seismic Areas Zone I v a 0.16 Importance Zone II v V ⁱ 1	Characteristic Periods Spectrum Type Horizontal Vertical Type 1 V S,avg 1.2 0.9 Soil TB(S) 0.15 0.05 B TC(S) 0.5 0.15 TD(S) 2 1	Levels XZ Down 0 - 0.00 V Up 0 - 0.00 V Dynamic Analysis Eigenvalues 10 Accuracy 0.001 Spectrum Participation factors PFx 0 PFy 0 PFz 0
Spectrum Response Spectrum Elastic ζ 5 Horiz Response Spectrum U	✓ Ductility Class DCM ✓ zontal b0 2.5 Vertical b0 3 Ipdate Spectrum Sd(T) >= 0.2 ag	Eccentricities Sd (T) e πx 0.05 *Lx sd (TY) 1 e πz 0.05 *Lz Sd (TY) 1
Structural Type q Concrete q Structural Type x Frame System Structural Type	1 qy 1 qz 1 Z Frame System	Openings Setbacks X One X Z One Z All the other cases All the other cases
T1 according to 4.3.3.2.2 X Concrete Moment Resisting Interstorey Drift Limit	.(5) Frames V Z Concrete 0.005 Damaged Buildings	Moment Resisting Frames V Walls KANEPE Default OK Cancel

• The response spectrum for the preliminary checks must be elastic.

In the above dialog box there is added a new button "KAN.EPE." KANERE, which opens a new dialog box presented above:

Elastic Analysis Parameters	×
Calculation of the constant value of shear length L Data Reliability Level Satisfactory V	.s
Damage level for the calculation of the factor γSd (S	.4.2)
Severe & extended Damages-Strenthening	~
Safety Factor ySd (0.15) 0	
Calculation Method -Analysis / Structural Performan	ce
	~
Values of the behaviour factor q'	
	\lor
	\vee
	~
OK SPECTRA Cance	9



In this dialog box you can define the following two parameters (like in the nonlinear analysis scenario):

- The Data Reliability Level
- The Damage Level for the definition of the coefficient $\gamma_{\text{Sd}}.$

The value 0 in the following field/means that the coefficient will take values. In case you want to set a different value, you type it and the calculation are implemented by taking into consideration your value.

- The following fields are inactive because refer to the scenarios of elastic and dynamic analysis, that will be explained later.
- Especially, for the preliminary check, the selection of the calculation method of the shear length Ls does not affect the results.

Then, you perform the analysis scenario, then you save the file with the load combinations and in the section "Checks" the results of the checks that correspond to criteria of the appropriate analysis application are presented.

- The check about the symmetric distribution of the mass and the stiffness.
- The check about the percentage of the shear force against which the wall structural system resists.
- The check of the interstorey drift of storeys or nodes. The first check refers to the interstorey drift between storeys (underlying and overlying floor) and the nodes' interstorey drift refers to the displacement of each node of the storey. Both checks are implemented in each direction.
- The check about the fundamental periods.
- The checks about the insufficiency indices and the morphological regularity. The check of the insufficient index λ is implemented per storey for the beams and the columns separately. It is recorded the number of the beams where the index λ is lower or greater to 2.5 per strorey, as well as the percentage of the total beams and columns. The totals in the bottom are the summations per elements and in total. Finally, the check of morphological regularity contains the criterion of the median insufficiency index $\lambda_{\rm ki}$ per storey.
- If for example the insufficiency index λ is greater than 4.0 for the 30% of the structural elements, the further assessment of the structure is not necessary.

The nonlinear analysis (Pushover) or the elastic analysis (static or dynamic) are applied concerning the previous criteria.



Since the interaction surface of all structural elements are calculated, the model is ready for performing Pushover analysis.

Initially, go back to Ribbon "Analysis" and create a nonlinear analysis scenario.



- For Greece, select the scenario "EC-8 Greek/Nonlinear" as well as the corresponding scenarios for Cyprus, Italy and Austria, for which the National Annexes are incorporated in SCADA Pro.
- EC8_General is a scenario that doesn't contains a National Annex. This means that the user has to update the coefficients according the Annex of his country.
- Using the scenarios EC8_Greek, EC8_Italy, EC8_Cyprus and EC8_Austrian all the corresponding coefficients are filled in the corresponding fields automatically.
- Especially, for the nonlinear static analysis scenario (EC-8_Greek / EC-8_Nonlinear), when EC8 provisions are considered, the multipliers of the inertial properties that will be defined in this step of the procedure, will be taken into consideration in the initial Pushover analysis where only the permanent and imposed loads are active. The default values of the loads are defined in EC8.
- Then, about the parameters of the nonlinear analysis, you choose if they will be kept the same as in the first pushover analysis or they will be reduced step by step concerning the initial value. The reduction can take place either in the beginning of analysis step or after the formulation of the plastic hinge.

«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»



	S	Scenario			Selec	t the ne	ew so	enarios "l
Renu	mbering				Dyna	mic an	u defii	ne the " Pr
Node	s No	*			Prop	erties		
🗌 Disab	ble	Name			E	lements		Nodes
Seismic Seismic	E.A.K.(Static) E.A.K. (Dynamic-eti)	Analysis	EC8_Genera	l v				
EC8_Ge	eneral Dynamic	Туре	Dynamic	~	Loa	ad Cases	s	Masses
		Propertie	s	Vadaa				
		Eleme	ints	vodes				
		Load C	ases	lasses				
		New	1	Update				
			Fxit					
	Ρι	ushover /	Analysis E	xecutio	on		×	
2 Г	Pu Parameters	ushover /	Analysis E	xecutic	on		×	
2 [3 [Parameters Automatic Procedur	ushover A	Analysis E ass Centers	(cm)	on v	7	×	
2 3 Procedu	Parameters Automatic Procedur ire	re L	Analysis E ass Centers evel - 0.00	(cm) X 0.00	рп У 0.00	Z 0.00	×	
2 3 Procedu	Parameters Automatic Procedur ure Mass - Stiffness	re L	Analysis E ass Centers evel - 0.00 - 400.00	(cm) X 0.00 0.00	V V 0.00 400.00	Z 0.00 0.00	×	
2 3 Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic	re L 1 2	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00	(cm) X 0.00 0.00 0.00	Y 0.00 400.00 700.00	Z 0.00 0.00 0.00	× •	
2 3 Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover	re L 1 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	(cm) X 0.00 0.00 0.00 0.00	Provide a constraint of the second	Z 0.00 0.00 0.00 0.00	× •	
2 3 Procedu	Parameters Automatic Procedur rre Mass - Stiffness Static-Dynamic Pushover	re L 1 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	(cm) X 0.00 0.00 0.00 0.00	Provide a constraint of the second	Z 0.00 0.00 0.00 0.00		
2 3 Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover	re L 1 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	x cm X 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0	Provide a constraint of the second	Z 0.00 0.00 0.00 0.00		
2 3 Procedu	Pu Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover	re L 1 2 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	x cm) X 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0	Provide a constraint of the second	Z 0.00 0.00 0.00 0.00		
2 3 Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover	re L 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	x cutic X 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0	 Y 0.00 400.00 700.00 1000.00 0 0 	Z 0.00 0.00 0.00 0.00		
2 2 3 2 Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover Initialize Data	re L 1 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	x c c x C x 0.00 0.00 0.00 0.00 0.00 0.00 E E E	Y 0.00 400.00 700.00 1000.00	Z 0.00 0.00 0.00 0.00		
2 _ 3 _ Procedu	Parameters Automatic Procedur ure Mass - Stiffness Static-Dynamic Pushover	re L 1 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	(cm) X 0.00 0.00 0.00 0.00 0.00 E	Y 0.00 400.00 700.00 1000.00 2 2 5	Z 0.00 0.00 0.00 0.00		
2 3 Procedu	Parameters Automatic Procedur Ire Mass - Stiffness Static-Dynamic Pushover	re L 1 2 3	Analysis E ass Centers evel - 0.00 - 400.00 - 700.00 - 1000.00	x cm) X 0.00 0.00 0.00 0.00 0.00 E E E E E E E	Y 0.00 400.00 700.00 1000.00 2 2 3 5	Z 0.00 0.00 0.00 0.00		



• Parameters		
	EC8 - Pushover Para	meters
Sesmic Area Sesmic Areas Zone I v a 0.16 Importance Zone II v Yi 1	Characteristic PeriodsResponceHorizontalVerticalType 1S,avg1.20.9SoilTB(S)0.150.05BTC(S)0.50.15TD(S)21	Lavels XZ Down 0 - 0.00 V Up 4 - 1200.00 V Plastic Hinge check under the Reference Level Dynamic Analysis Eigenvalues 10 Accuracy 0.001 Spectrum participation factors
Spectrum Response Spectrum Elastic ζ 5 Horiz Response Spectrum U	✓ Ductility Class DCM ✓ ontal b0 2.5 Vertical b0 3 pdate Spectrum Sd(T) >= 0.2 ag	PFx 0 PFy 0 PFz 0 Eccentricities Sd (T) Sd (TX) 1 e πιχ 0.05 *Lx Sd (TY) 1 e πιz 0.05 *Lz Sd (TZ) 1
\checkmark Fx +k Fz \checkmark Tria $Fx - kFz$ \checkmark Orti \neg Fx + k Fz \neg Acci \neg Fx + k Fx \neg Acci \checkmark Fz + k Fx \neg Acci \neg Fz + k Fx \neg Base \checkmark Fz + k Fx \neg Base \neg Fz - k Fx \neg Transvo Default \bigcirc K	ngular Disribution logonal Distribution dental Eccentricities Ex dental Eccentricities Ez e: Shear from Responce Spectrum erse Load Factor (k) 0.3	Check Node 0 Number of Steps 200 Lamda Range (%) 0 Maximum 3 % of Total Height ✓ Constant Value of the Shear Length LS Active Stiffness After the plastic hinge ✓ Data Credibility Satisfactory ✓

In the dialog box of Pushover Analysis Parameters, the parameters outside the borders are defined as for a linear analysis. In that case the Response Spectrum must be **<u>Elastic</u>**.

In "Levels XZ" section:

Lavels	XZ				
Down	0 - 0.00	~	Up	4 - 1200.00	~
Plas	tic Hinge chec	k unde	r the R	eference Level	

Select the lowest and the highest level where the seismic actions will be imposed for the analysis' purposes.

It is recommended to define as the highest level the last complete level (no stairwell termination). This level contains the Check Node which is the diaphragm node or another node, in the same level but the outer perimeter of the building.

The activation of the following checkbox/means that the elements that belong in levels under the reference level, are taken into consideration as potential locations of plastic hinges.

In "Seismic Combinations" section:



 Seismic Combinat 	tions					
Fx +k Fz	 Triangular Disribution 					
Fx - k Fz	 Orthogonal Distribution 					
✓ -Fx + k Fz						
-Fx - k Fz	Accidental Eccentricities Ex					
✓ Fz + k Fx	Accidental Eccentricities Ez					
Fz - k Fx	Base Shear from Responce Spectrum					
-Fz - k Fx	Transverse Load Factor (k)	0.3				
 Check the the applic in the 	e load combinations considered in cation of one seismic force in one of corresponding transverse dire	the pushover direction (X o ction multip	analysis. Ea r Z directior blied by a	ach combin n) and one a factor,	ation impl seismic fo defined	lies rce in
Transverse	Load Factor (k) 0.3 . The coefficie	nt k is given a	is 0.3 by def	ault.		
Check the	e seismic forces' distribution along	height (Trian	gular or/anc	l Orthogon	al).	
✓	Accidental Eccentricities Ex					
	Accidental Eccentricities Ez					
Check	if you wan	it to consider	the acciden	ital Eccentr	icities due	e to
which mo	oments are developed in x or/and z	direction.				
• The check	k/means that the base shear is calc	ulated from t	he dynamic	analysis.		
			,	,		
Activate all seis combinations consequently the second sec	smic combinations with the accide are produced. That means tha he computational time is increasec	ental eccentri t 64 pusho l significantly.	cities consid ver analyse	dered and es are co	then 64 lo nducted a	oad and
In the last section	on:					
Check Node	36 Active Infill Masonry					

Check Node Number of Steps	36 200	Active Infill Masonry	• On "Ac
Maximum [☐ Constant Value of	3 the Sł	% of Total Height hear Length Ls	account in ana in our constru
Effective Stiffness	Calo	ulation in every step \sim	• Active consider the
Data Reliability	Satis	sfactory \checkmark	necessary.
Check of 2nd ord	er effe	ects (θ)	

• On "Active Infill Masonry" choose whether we want to be fully taken into account in analysis the Masonry Infill included in our construction.

• Active "Check of 2nd order effects" to consider the 2nd order effects when is necessary.

- In "Check Node", type the number of the node for calculation of the Capacity Curve. This is the diaphragm node or another node in the outer perimeter of the last complete level of the building.
- In "Number of Steps", define the maximum number of steps (analyses) for each Pushover analysis.

Pushover is an iterative analysis procedure that terminates when the structure collapses; when the sequence of the developed plastic hinges evolves a collapse mechanism.

The "Number of Steps" is an upper limit, to avoid an extraordinary number of analysis steps needed until the structural collapse. Steps default value is 200.



- "Maximum displacement" in percentage (%) of the total height of the building is another way of setting an upper limit in the analysis steps. The Pushover iterative process ends when the displacement of the "Check Node" versus the height of the structure gets over the specified percentage. By default the maximum displacement's upper value is 3% of the total height of the building.
- "Lamda Range (%)" is about λ load factor. In each step and for each element, the λ factor is calculated. In the element with the minimum λ, a plastic hinge will be created.
 The default value 0: the program considers only the minimum λ value, which means that in each step only one element is considered for the formation of a plastic hinge, even if there

are elements with λ values very close to that.

The setting a different value (i.e. $\lambda_{min}=10\%$): This means the elements with values of λ between λ_{min} and $10\% + \lambda_{min}$ will be considered in this step with plastic hinges in the corresponding edge (element node).

Suppose that, on the first step of the pushover, minimum λ value is 1.0 and corresponds to a certain element with a plastic hinge. Defining 10% Lamda Range, all elements with λ between 1.0 and 1.1 will develop plastic hinges together in the same step.

- The options about the definition of the "Constant value of the Shear Length Ls" are the following:
- The shear length Ls is calculated as a fixed value based on the length of each element in all steps of the analysis
- Otherwise, the shear length is calculated on each step, based on the internal forces resulting, with Shear Length = M/V in the end section of the element, i.e. the distance of the end section from the point of zero moments.
- "Effective Stiffness" is about the calculation of the elements' stiffness. On the first step of the nonlinear analysis calculated the internal forces due to dead and live



loads and the stiffness considered on the elements, are multiplied by the factors set in

→ Elements

during scenario's creation.

On the second step of the nonlinear analysis considering the seismic loads, you have more than one options for the stiffness calculation:

- Initial: the stiffness remains the same as in the first step and invariable during the analysis procedure

- **Step by step calculation**: the stiffness is calculated in each step, whether plastic hinge are created or not. Stiffness is decreased, compared to the original value.

- **After plastic hinge development**: the stiffness is calculated as previously, with the difference that the value is decreased since the first plastic hinge has been developed. Until that step of analysis the initial value of the effective stiffness is considered.

- "Data reliability", is the Knowledge Level. Select between Limited, Normal and Full. The aspects that affect the definition of the knowledge levels are: geometry, details, material. The knowledge level influences the partial safety factors.
- a. Partial Safety Factors



Based on the knowledge level achieved through the different levels of survey, inspection and testing, the following set of partial safety factors (PSF) as shown in Table 3.3 are used in the verifications.

 Table 3.3: Partial safety factors (PSF) used for verification, according to the different knowledge levels

 (KL).

		PSF
KNOWLEDGE LEVEL	Material	Overstrength
KL1	1.20 (γ _m)	1.20(γ _{Rd} , γ _{ov})
KL2	γ _m as in EN1998-1	γ_{Rd}, γ_{ov} as in EN1998-1
KL3	0.80 (γ _m)	0.80 (γ _{Rd} , γ _{ov})

EXAMPLE:

In the dialog box "Spectra" is defined the response spectrum which indicates the structural demand of the structure. The structural demand in combination with a limit state (LS) composes a performance objective. In the dialog box:

Three limit states are considered for the structural design of the structure. The states of damage related to the limit states considered are presented below:

- -LS of near collapse (NC)
- -LS of significant damage (SD)
- -LS of damage limitation (DL)

National Authorities define the appropriate performance level, which means they propose a performance level (limit state) against a seismic event. They usually propose more than one performance level (a combination of a limit state with a seismic event).

The seismic events or hazard levels (HL) are described with a return period and a peak ground acceleration. The peak ground acceleration and the incidence of the seismic events are characteristic of the seismicity of the region. The return periods established by the National Authorities are usually the following:

The suggested values for the return periods of Eurocode are:

- HL with return period 2475 years, corresponding to a probability of exceedance 2% in 50 years.
- HL with return period 475 years, corresponding to a probability of exceedance 10% in 50 years.
- HL with return period 225 years, corresponding to a probability of exceedance 20% in 50 years.

Selecting one of the three default commands EC8 2% EC8 10% EC8 20%, the dialog box is filled in automaticaly, with the values of the spectra parameters related to a specific limit state, thus the performance objectives proposed by the Eurocodes are depicted in the dialog box.

«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»



SPECTRA

Design Existence (years 50 Exponent k 3 Damage Limitation (DL) Image Limitation (DL) 0.277 If Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Image Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 IR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Image Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 IR Calculation TLR Calculation TLR Calculation 475 Near Collapse (NC) Image Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 IR Calculation TLR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PLR% 2		Sp	ectra	
Damage Limitation (DL) Image Limitation (DL) Image Limitation (DL) Soil Acceleration ag=AgR.γL.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Image Check Soil Acceleration ag=AgR.γL.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 0.277 TR Calculation TLR Calculation 10 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Image Check Soil Acceleration ag=AgR.yL.(TR/TLR)1/k 0.277 IR Calculation TLR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default ECS 10% ECS 20% ECS	esign Existence (years 50	~	Exponent k 3	
Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 0.277 TR Calculation TLR Calculation 10 2475 Exceedance Probability PLR% 2 Return Period TLR (years) 475 Near Collapse (NC) Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation 475 Near Collapse (NC) Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 D	Damage Limitation (DL)			
TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Return Period TLR (years) 475 Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Return Period TLR (years) 2475 Exceedance Probability PLR% 10 Image: Collapse (NC) Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 0.277 TR Calculation TLR Calculation 10 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default EC8 10% EC8 10% EC8 20%	✓ Check So	oil Acceleration	ag=AgR.γI.(TR/TLR)1/k	0.2773
Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Image: Constraint of the second secon	TR Calculation		TLR Calculation	
Exceedance Probability PR% 2 Return Period TLR (years) 475 Severe Damage (SD) Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 0.277 TR Calculation TLR Calculation 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Image: AgR.yI.(TR/TLR)1/k 0.277 IR Calculation Image: AgR.yI.(TR/TLR)1/k 0.277 Return Period TLR (years) 2475 Exceedance Probability PLR% 10 IR Calculation TLR Calculation Image: AgR.yI.(TR/TLR)1/k 0.277 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default Image: Agr.yi EC8 10% EC8 20% EC8 20%	Return Period TR (years)	2475	Exceedance Probability PLR %	10
Severe Damage (SD) ✓ Check Soil Acceleration ag=AgR.γI.(TR/TLR)1/k 0.277 TR Calculation Return Period TR (years) 2475 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) ✓ Check Soil Acceleration ag=AgR.γI.(TR/TLR)1/k 0.277 TR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PLR% 2 Return Period TLR (years) 475 Default GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Exceedance Probability PR%	2	Return Period TLR (years)	475
Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) In Calculation In Calculation 10 Image: Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation 10 Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default Image: Soil Acceleration EC8 10% EC8 20%	Severe Damage (SD)			
TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Return Period TLR (years) Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% Exceedance Probability PR% 2 Return Period TLR (years) Exceedance Probability PR% 2 475 Default EC8 2% EC8 10% EC8 20%	Check So	oil Acceleration	ag=AgR.γI.(TR/TLR)1/k	0.2773
Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image: Collapse (NC) Image:	TR Calculation		TLR Calculation	
Exceedance Probability PR% 2 Return Period TLR (years) 475 Near Collapse (NC) Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation Return Period TR (years) 2475 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Return Period TR (years)	2475	Exceedance Probability PLR%	10
Near Collapse (NC) ✓ Check Soil Acceleration ag=AgR.γI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default GREEK 50% EC8 2% EC8 10% EC8 20%	Exceedance Probability PR%	2	Return Period TLR (years)	475
Check Soil Acceleration ag=AgR.yI.(TR/TLR)1/k 0.277 TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Near Collapse (NC)			
TR Calculation TLR Calculation Return Period TR (years) 2475 Exceedance Probability PR% 2 Default Return Period TLR (years) GREEK 10% GREEK 50%	Check So	oil Acceleration	ag=AgR.γI.(TR/TLR)1/k	0.2773
Return Period TR (years) 2475 Exceedance Probability PLR% 10 Exceedance Probability PR% 2 Return Period TLR (years) 475 Default	TR Calculation		TLR Calculation	
Exceedance Probability PR% 2 Return Period TLR (years) 475 Default GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Return Period TR (years)	2475	Exceedance Probability PLR%	10
Default GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Exceedance Probability PR%	2	Return Period TLR (years)	475
GREEK 10% GREEK 50% EC8 2% EC8 10% EC8 20%	Default			
	GREEK 10% GREEK 5	0% EC	8 2% EC8 10% EC	08 20%
OK Cancel		ОК	Cancel	

Use National Appendix to define the appropriate performance levels; the structural capacity (limit state) related to the seismic demand (hazard level).

Press OK to save the Parameters and close the dialog box.

Automatic Procedure

• Automatic Procedure

Then select Automatic Procedure and the program will conduct the analysis and the corresponding calculations, automatically:

	Parameters	Mass Centers	(cm)			
	Automatic Procedure	Level	Х	Y	Z	,
Proce	dure	0 - 0.00	0.00	0.00	0.00	
\checkmark	Mass - Stiffness	1 - 300.00	0.00	300.00	0.00	
\checkmark	Static-Dynamic	2 - 600.00	605.04	600.00	822.04	
\checkmark	Pushover	3 - 900.00	604.46	900.00	824.72	
		4 - 1200.00	590.37	1200.00	888.12	
						-
						-



Allow the program to calculate Mass and Stiffness, Static and Dynamic analysis and complete the Pushover analysis. The default parameters include four combinations with two types of load distribution and 200 steps for each Pushover analysis, thus 1600 analyses in total!

When the Pushover analysis is applied, the structure is pushed with a gradually increased lateral static load (triangular or rectangular distribution along height) until the collapse of the structure. So, plastic hinges are formed gradually in the edges of the structural elements (beam, column and wall). Along with the plastic hinges' formation, the stiffness of the structural elements and the structure in total is decreased step by step. Finally, a plastic mechanism is generated due to the presence of plastic region in the joints of the structure. The plastic deformations indicate that the strength resistance of the structural elements is overtaken and so the construction collapses.



3.3 Nonlinear Analysis (PUSHOVER) Combinations



Activated the following analysis scenario, press the button "Combinations" to open the window "Load Group Combination", where you can create your combinations or select the default ones by clicking on the command "Default Combinations".

			Συ	νδυα	ασμοί Σε	τΦ	ορτίσεων										×
G 1.35	γE 1	YGE 1					Αστο: Σγ	xίας G+γ 3+ψ΄	/Q+Σγψ0Q 1Q+Σψ2Q		\ει τουργικότητα ✓ ΣG+Q+Σψ0Q ✓ ΣG+ψ1Q+Σψ2	s Q		Υπολ	ογισ	μ ός	
1.5	γE0.3 0.5						ΣC	à+E+	⊦ Σγψ2Q	E	✔ ΣG+Σψ2Q			Διαγρο	αφή	Ολων	<u> </u>
	Είδος	Διεύθυνση	LC1	LC	2		LC3		LC4		LC5		LC6			LC 1	^
Σενάριο			EC-8_Gree	▼ EC	-8_Gree	-	EC-8_Gree	•	EC-8_Gree	•	EC-8_Gree	•	EC-8_0	Gree	-	EC	۰.
όρτιση			1	2			0		0		0		0			0	
ύπος			G	- Q		-	G	•	G	•	G	•	G		•	G	
.ράσεις				▼ Ko	τηγορία	-		•		-		•			-		
εριγραφή																	
						_											
ονδ.:1	Αστοχίας 💻	Οχι 💌	1.10	0.3	0							_					
υνδ.:2	-	-	1	_		_						_					
υνδ.:3	<u> </u>	<u> </u>		_		_						_					
υνδ.:4	-	· ·		_								_					
υνδ.:5	-	-	1	_		_						_					
υνδ.:6	-	· ·	1	_		_						+					
υνδ.:7	-	-	-	_		_						+					
νδ.:8		· ·		_		_						+					
.νδ.:9	-		1	_		_						+					
.v8.:10		-	1	-								+					
vð.:11			1	-		_						+					
vo.:12	-	ļ —	4	_								-			_		*
																1	
5																	

the

Press Save to save the cor

to save the combinations' file as CMB file format in the project's folder.



You will need this file for checks applied for the retrofitting methods.

0		Save A	\ s			×
⊙ ⋺ ⊤ 1 🎚	« pusł	nAMA2 → pushAMA2 →	~ ¢	✓ C Search pushAMA2		
Organize 🔻 New	v folder					0
Libraries Documents Music Pictures Videos Computer	^	Name	[ate modified	Туре	-
		퉬 scaanal	1	2/11/2013 4:00 µµ	File folder	
		scades_c	1	2/11/2013 3:28 µµ	File folder	
		🌗 scades_Sid	1	2/11/2013 3:28 µµ	File folder	
		퉬 scades_Synd	1	2/11/2013 3:28 µµ	File folder	
		퉬 scades_Toixo	1	2/11/2013 3:28 µµ	File folder	
		퉬 scainp	1	2/11/2013 3:21 µµ	File folder	
		퉬 scaPush	1	2/11/2013 3:25 µµ	File folder	
		퉬 tmp	1	2/11/2013 3:28 µµ	File folder	
HP_RECOVERY	,	pushover.cmb	1	4/11/2013 10:55 πμ	CMB File	
	(r	stat.cmb	1	2/11/2013 3:27 µµ	CMB File	
-	~ <					>
File name: pushover.cmb						~
Save as type:	Scada C	ombination(*.cmb)				~
Hide Folders				Save	Cance	

3.4 Nonlinear Analysis (PUSHOVER) Results

In Pushover analysis, the structure is subjected to monotonically increasing lateral forces (uniform or triangular distribution) that represent the seismic inertia forces until collapse. Plastic hinges are developed gradually at the edges of the structural elements (beams, columns, walls).

The plastic hinges lead to the reduction of the stiffness at the start and end length of the structural elements, gradually. The analysis is completed, when a failure mechanism is developed based on the formed plastic hinges in the structural elements. The corresponding structural elements cannot receive further tension due to the excessive plastic deformations developed and the structure is led to failure.

• The user can see both the pushover analysis results in the form of a capacity curve that correspond to a specific load combination and the deformed shape in each point of the capacity curve (analysis steps):

-Select the three dimensional view of the structure

-Select one of the commands in the ribbon "Analysis" and the command group "View"



On the interface, the three dimensional undeformed and deformed shape of the structure is displayed and the dialog box "Report" appears.





In the field "Check Node" define the serial number of the node that will be used for the derivation of the capacity curve.

- The check node is usually master node of the rigid diaphragm in the top floor of the structure. If a diaphragm doesn't exist, select a node in the perimeter of the top floor.
- You can select a different check node to see the corresponding results, without perform again the pushover analysis. The corresponding results are updated automatically.

Check

Node

63

• In this example the Check Node is the one with the label "63".

In the top of the window select one of the distributions set previously



The list contains the steps of the selected nonlinear analysis.
«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»



		Report	:			
Triangular 🗸	Fx+0.3	80*Fz		~	Spec	trum
Step Vb(kN) (λ)					Paran	neters
1. 45.24714 (0.43339)	~	>>	Succes the Pla	sional Disp stic Hinges	lay of	DL
2. 250.40681 (1.96507) 3. 270.06343 (0.18828)	Î				۷	NC
4. 374.40096 (0.99937) 5. 396.80721 (0.21461)						
6. 401.95696 (0.04933) 7. 450.70286 (0.46690)						•
8. 460.33169 (0.09223) 9. 557.85572 (0.93411)						
10. 640.23870 (0.78909) 11. 651.54392 (0.10828) 12. 665 17321 (0.13054)						
13. 817.37946 (1.45787) 14. 822.87042 (0.05259)						
15. 837.11834 (0.13647) 16. 883.55086 (0.44474)						
17. 918.68367 (0.33651) 18. 926.62514 (0.07607)						
19. 997.21076 (0.67609) 20. 1163.37178 (1.59153)						
21. 1206.93775 (0.41729) 22. 1244.18182 (0.35673)						
23. 1247.41822 (0.03100) 24. 1264.49404 (0.16356)						
25. 1264.53895 (0.00043) 26. 1310.44560 (0.43971)		무	13	5	4	
27. 1317.28069 (0.06547) 28. 1317.66863 (0.00372) 29. 1319 18336 (0.01451)				ci	6	6
30. 1324.95729 (0.05530)	~	ks creat	on for Ou	φut		

Each step provides the corresponding shear value Vb(kN) and the λ load factor, while at the same time are presented are presented in the corresponding graph the following data:

- Capacity Curve
- Idealized Capacity Curve
- Target Displacement

Capacity Curve	¥
Capacity Curve	
Idealized Capacity Curve	
. Target Displacement	



3.4.1 Capacity Curve

The capacity curve represents the nonlinear relationship between the base-shear force and the displacement of the control (Check) node.

The points depicted on the capacity curve are the "Steps" of the pushover analysis. The selected step is displayed with pink color and represents the time of a plastic hinge creation.



In the following field Node you can change the check node and derive the results for a different check node, without repeating the analysis. The presentation of the results is updated automatically.

3.4.2 Idealized Capacity Curve

It is the linearized capacity curve according to the procedure given in Annex B of EN 1998-1. The idealized elasto-plastic force- displacement relationship is calculated.

«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»





3.4.3 Target Displacement

The target displacement of the check node, for the tree different Limit States, is determined according to the Displacement Coefficient Method given in Annex B of EN 1998-1.





Spectrum

Spectrum: is like the Spectrum command explained in the section about the scenarios parameters. Note that these parameters are considered in the calculation of the target displacement related to the seismic demand and not to the structural capacity. For that reason these parameters are irrelevant with the capacity curve and so they can be defined or modified before or after the implementation of the analysis procedure.

In Parameter select the method for the derivation of the capacity curve ("Idealization Method")

Parameters

EC8 Parameters						
Idealization Method of the SDOF system						
Equal Areas Calculation 🗸 🗸						
Formula B.6 EC8 ANNEX B Equal Areas Calculation						
Target Displacement Calculation with Iterative Procedure(EC8 ANNEX B.5)						
OK Cancel						

between the two methods proposed in Annex B of EN 1998-1.

✓ Target Displacement Calculation with

Activate Tterative Procedure (EC8 ANNEX B.5) and the program uses the Iterative Procedure proposed in EC8 for the calculation of the target displacement.

3.4.4 Representation of the structure

The deformed shape of the structure is depicted in real time and the location of the plastic hinges formed in the edges of the structural elements for each step. There are two methods:

1. Select the step from the list



in the selected step and the location of the plastic hinges.

The undeformed shape of the structure is displayed together with the deformed one, in gray and red color respectively.

Colored dots indicate the edges with the plastic hinges and the different colors the damage levels.











The graphics generated per distribution (Rectangular, Triangular) and seismic combination.

The Graph - Checks Creation for Output command is necessary for the syntax of the documents, which contain the graphs and the design checks of the project, and their update in case of modifications.

3.5 Non Linear Analysis (PUSHOVER) Checks



First press Graph - Checks Creation for Output and then select "Checks" command. The following window is displayed:

Checks									×				
	Analysis Type - Distribution		DL			SD			NC		Print		^
		в	с	т	в	С	т	в	С	т			
1	Fx+0.30*Fz - Triangular	84	51	135	1	15	16	0	2	2	Yes 🗸		
9	-Fx+0.30*Fz - Triangular	83	38	121	0	0	0	0	0	0			
17	Fz+0.30*Fx - Triangular	90	48	138	0	0	0	0	0	0	Yes		
25	-Fz+0.30*Fx - Triangular	79	35	114	0	0	0	0	0	0	-		
101	Fx+0.30*Fz - Orthogonal	85	52	137	0	6	6	0	0	0	-		
109	-Fx+0.30*Fz - Orthogonal	83	34	117	0	0	0	0	0	0	-		
117	Fz+0.30*Fx - Orthogonal	91	49	140	1	23	24	0	28	28	-		
125	-Fz+0.30*Fx - Orthogonal	81	35	116	0	0	0	0	0	0	-		
											-		
											-		
											-		
											-		
											-		~
✓ Ir	ndude Total Table in Output Select Analysis for Intervention	che	ck						D	iplay	/ Checks		
F	x+0.30*Fz - Triangular			¥					ок		Ca	ncel	

49

50

51

52

53

54

55

56

57

58

For Help, press F1



NUM

A table with the total number of beams (B) and columns (C) which have lower capacity than the demanded, for each pushover and each LS. "T" means total. Print Yes 🗸 Yes Indicate with YES the pushover analysis to include in the printout or just check Include Total Table in Output to consider all of them. Diplay Checks The button opens the TXT file that contains all members' capacity regarding deformation for the pushover analysis a "YES" indicator in the "Print" column: Fx+0.30*Fz - Triangular 84 51 135 1 15 16 0 2 2 Yes 🗸 1 CheckPS 1.txt - WordPad Edit View Insert Format Help File 5 4 R. **B** ΔÅ. ~ MEMBER CAPACITIES IN DEFORMATION TERMS Analysis Type - Distribution : Fx+0.30*Fz - Triangular (1) Target Dispacements: Damage Limitation (DL) 0.052(m) Severe Damage (SD) 0.057(m) Near Collapse (NC) 0.062(m) Beams (Fx+0.30*Fz - Triangular) (1) |Damage Limitation | Severe Damage | Near Collapse (DL) (SD) (NC) _____ 1 Memb. Node |ysd*0sd|0pl/yrd| |ysd*0sd|0pl/yrd| |ysd*0sd|0pl/yrd| 1|0.00000|0.00000|Yes|0.00000|0.01117|Yes|0.00000|0.02234|Yes 46 2|0.00011|0.00000|No |0.00007|0.01367|Yes|0.00007|0.02734|Yes 47 2|0.00000|0.00000|Yes|0.00000|0.01329|Yes|0.00000|0.02658|Yes 3|0.00000|0.00000|Yes|0.00000|0.01093|Yes|0.00000|0.02186|Yes 48 1|0.00000|0.00000|Yes|0.00000|0.01032|Yes|0.00000|0.02064|Yes 13|0.00000|0.00000|Yes|0.00000|0.01243|Yes|0.00000|0.02485|Yes

> 13|0.00000|0.00000|Yes|0.00000|0.01058|Yes|0.00000|0.02115|Yes 14|0.00000|0.00000|Yes|0.00000|0.00991|Yes|0.00000|0.01981|Yes 14|0.00000|0.00000|Yes|0.00000|0.01048|Yes|0.00000|0.02097|Yes

> 8|0.00000|0.00000|No |0.00001|0.00944|Yes|0.00001|0.01887|Yes 15|0.00000|0.00000|Yes|0.00000|0.01240|Yes|0.00000|0.02481|Yes

7|0.00000|0.00000|Yes|0.00000|0.01344|Yes|0.00000|0.02687|Yes

8|0.00000|0.00000|Yes|0.00000|0.01157|Yes|0.00000|0.02315|Yes 710.0004110.000001No 10.0002510.009171Yes10.0003110.018351Yes

7|0.00000|0.00000|Yes|0.00000|0.01059|Yes|0.00000|0.02117|Yes 10|0.00000|0.00000|Yes|0.00000|0.00904|Yes|0.00000|0.01808|Yes

10|0.00000|0.00000|Yes|0.00000|0.01347|Yes|0.00000|0.02694|Yes 11|0.00000|0.00000|Yes|0.00000|0.01347|Yes|0.00000|0.02694|Yes 11|0.00000|0.00000|Yes|0.00000|0.00904|Yes|0.00000|0.01808|Yes

9|0.00000|0.00000|Yes|0.00000|0.00992|Yes|0.00000|0.01983|Yes

9|0.00000|0.00000|Yes|0.00000|0.01075|Yes|0.00000|0.02151|Yes 6|0.00000|0.00000|Yes|0.00000|0.01367|Yes|0.00000|0.02734|Yes

6|0.00000|0.00000|Yes|0.00000|0.01261|Yes|0.00000|0.02523|Yes 5|0.00000|0.00000|Yes|0.00000|0.01140|Yes|0.00000|0.02280|Yes

5|0.00000|0.00000|Yes|0.00000|0.01243|Yes|0.00000|0.02485|Yes



Checks document help you to assess in which Pushover analysis, the structural elements present lower capacity than the one defined in the considered LS. It means that it can be easily observed in which Pushover analysis, the defined performance level is not satisfied. In that case the structure must be modified, for example through the reinforcement of some structural elements and be redesigned.

First decide the analysis, which indicates the redesign of the existing structure by selecting one from the list.

Select Analysis for Intervention check	
Fx+0.30*Fz - Triangular	×
Fx+0.30*Fz - Triangular	
-Fx+0.30*Fz - Triangular	
Fz+0.30*Fx - Triangular	
-Fz+0.30*Fx - Triangular	
Fx+0.30*Fz - Orthogonal	
-Fx+0.30*Fz - Orthogonal	
Fz+0.30*Fx - Orthogonal	
-Fz+0.30*Fx - Orthogonal	

The Interventions procedure is explained on "Members Design" chapter.

In the bottom of the text file, the Shear resistance check appears only for the structural members under shear failure.

CheckPS_9.txt - WordPad	- 🗆 🗙
File Edit View Insert Format Help	
	^
SHEAR CHECKS	
Beams (-Fx+0.30*Fz - Triangular) (9)	
MEMB.NODES Vrd, s Vrd, max Vr Ved RATIO A - DL B - SD F - NC	
25 3 y: 300.74 328.50 117.41 198.47 1.6905 NO NO NO	
4 y: 300.74 328.50 116.07 207.28 1.7858 NO NO NO NO	
26 1 [Y: 240.59] 328.50] 95.18] 202.98] 2.1326] NO NO NO	
1 31 510, 370 141 328 501 155 961 156 021 1 00041 NO 1 NO 1 NO 1	
32 9 Y: 240.59 328.50 94.18 103.74 1.1014 NO NO NO	
5 y: 0.0000	
35 7 y: 641.57 722.70 271.92 436.54 1.6054 NO NO NO	
8 y: 641.57 722.70 265.75 476.86 1.7944 NO NO NO	
36 5 y: 801.96 722.70 320.52 416.26 1.2987 NO NO NO	
6 Y: 801.96 722.70 323.68 461.28 1.4251 NO NO NO	
38 6 y: 641.57 722.70 263.95 455.90 1.7273 NO NO NO	
2 10 10 10 10 10 10 10 10 10 10 10 10 10	
1 39 10/17: 641.57/ 722.70/ 243.96/ 500.91/ 1.2500/ NO / NO / NO /	
40 7/10 641.571 722 701 263 961 448 581 1 6941 NO 1 NO 1	
3 y: 641.57 722.70 262.85 479.75 1.8252 NO NO NO	
41 11 V: 641.57 722.70 243.92 306.17 1.2552 NO NO NO	
7 y: 641.57 722.70 248.37 337.35 1.3582 NO NO NO	
57 31 y: 963.90 904.87 360.99 364.84 1.0107 NO NO NO	
27 y: 0.0000	
Columns (-Fx+0.30*Fz - Triangular) (9)	
Μέλος Κόμβ. Vrd,s Vrd,max Vr Ved Λόγος Α - DL Β - SD Γ - NC	
5 J5 Y; 556.53 642.40 2/0.52 2/1.55 1.0031	
	~
For Help, press F1	NUM //



3.6 Seismic Force

Activated the nonlinear analysis scenario and then press "Seismic Force".

In the TXT file, first the load case data are presented, then, the spectrum parameters, the performance level and the damage level. Then, for each analysis, the maximum base shear, the corresponding maximum displacement, the overstrength ratio and the minimum overstrength ratios presented in each direction.





4 th STEP: RETROFITTING METHODS

A structural member need to be reinforced when it can not carry the vertical loads and the loads resulting from the design earthquake. The need for reinforcement of the concrete structural elements is determined based on the options:

- Performance objective
- Load distribution along height of the horizontal/seismic loads considered in the design checks of the retrofitting methods.

If you have select, for example, performance objective "B" and load distribution triangular, then you have to access the report file of the study and do the following:



Select the seismic load distribution along height considered in the design checks of the retrofitting methods and also the performance objective. Then examine each step of the capacity curve of the structure to locate the column, where is formulated the first plastic hinge.

In this way, an iterative process starts, where you reinforce and then recheck some structural elements, starting with that column, until the structure's capacity satisfies the defined performance objective.

You return to the Ribbon "Members Design" where:

Use the command "Detailing" for beams and columns to apply the rehabilitation and reinforcing methods according to the requirements of the Code of Structural Interventions.



Important prerequisite for the design of the retrofitting methods is the calculation of the load combinations of the nonlinear static analysis saved in the corresponding step of the analysis/design procedure.

	-820.00 👻 🕥 🔻				
Basic Modeling View	w Tools Slabs Loads	Analysis Post-Processor	Members Design	Drawings-Detailing Addons	
EC2 Eurocode (0) Vertical New Active Scenario Parameters	Continuity Check - Results	Node Design Releases	Check - Results Reinforcement	Check Results Reinforcement	Cross-Section D Buckling Memb Connections
Scenarios	Beams	Capacity Design	Columns	Footings SI	abs - Mesh Steel
111000	Structura	I Component Parameters	×		^
ArcSource Epyon A × ✓ Lines ✓ Lines ✓ Arcs ✓ Beams Columns Footings ▲ Nodes ▲ Mathbeams ■ Mathbeams ■ Mathbeams ■ Surf 2D ● Surf 3D ● Slabs	Steel Reinforcement Combinations Slabs Combinations of Load Sets (10) Combinations (14) +1.35Lc1+1.50Lc2 2(1) +1.00Lc1+0.50Lc2 3(2) +1.00Lc1+0.30Lc2+1.00Lc3-0. 3(2) +1.00Lc1+0.30Lc2+1.00Lc3-0. 5(2) +1.00Lc1+0.30Lc2+1.00Lc3-0. 5(2) +1.00Lc1+0.30Lc2+1.00Lc3-0. 5(2) +1.00Lc1+0.30Lc2-1.00Lc3-0. 7(2) +1.00Lc1+0.30Lc2-1.00Lc3-0. 9(2) +1.00Lc1+0.30Lc2-1.00Lc3-0. 9(2) +1.00Lc1+0.30Lc2-1.00Lc3-0. 10(2) +1.00Lc1+0.30Lc2-1.00Lc3-0.	Capacity Design Beams Column 11) Ult. Serv. +X -X 30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 30Lc4-1.00Lc5+0.30Lc7+0.30Lc9 30Lc4-1.00Lc5+0.30Lc7+0.30Lc9 30Lc4-1.00Lc5+0.30Lc7+0.30Lc9 30Lc4-1.00Lc5+0.30Lc7-0.30Lc9 30Lc4-1.00Lc5+0.30Lc7+0.30Lc9	Steel s Footings +Z -Z No ULS/SLS Dir. 0 ULS/SLS VL VL ULS ULS VL ULS +X VLS ULS +X VLS ULS +X VLS ULS -X VLS ULS -X VLS ULS -X VLS	Store As Source As S	SendypubMM2 P Des molified Type SendypubMM2 P Des molified Type SendopubMM2 P SendopubMM2 P
	Level Multipliers	1 / (1-θ) Ins	ert Combinations	Computer ScaPut ScaPuth ScaPuth ScaPuth ScaPuth mp mp Montemp	12/11/2013-32:3 µµ File folder 12/11/2013-32:5 µµ File folder 12/11/2013-32:5 µµ File folder 14/11/2013-30:5 ¶µ CMB File 12/11/2013-32:7 µµ CMB File ▼
	0 - 0.00 1.000 1.000 1.000 1.000	1.000 Comb	inations Calculation	File name Save as type: Save as type:	* * *
	2 - 820.00 1.000 1.000	1.000 Combin	ation G+w2Q 101	Hide Folders	Save Cancel
		A	utomatic Design		
			OK Cancel		
🗔 Παράμετροι 🗠 Δεδομένα Ε	<				× × × ×



4.1 Columns-walls reinforcement

4.1.1 Rehabilitation of columns-walls

The section "Rehabilitation" contains tools for the rehabilitation of the columns according to the Code of Structural Interventions (KAN.EPE.2013).

	Colu	mn Editor – 🗖 🗙
Rehabilitation		
FRP FRP	Rehabilitation Corrosion Protection Materials used on the surface that act as corrosion inhibitors for the steel reinforcement of the reinforced concrete structures and applied by impregnation. Concrete Repair Repairing mortars for the structural rehabilitation of concrete members. Crack Filling Cementitious binding materials for the structural rehabilitation of the concrete cracking, welded and / or infilled.	
< >	Printout Add Delete	
Recalculation	Protection	
Joint Check	Fire Protection Layers Fire resistant mortars applied by using epoxy resins.	
Y = 820.00 + ?? -	Concrete layers or coating Repairing mortars of one or more components for final protective coating.	© 112004(72.00 (10-101) @ 112004(72.00 (10-004)) @ 17004(72.00 (10-004)) 1(0)-2.05 (0)
Copy Paste	Paint Protection Plastic-elastic paint protection for concrete and coatings	
OK Cancel	Sika	31 21 Stars(7.1.0.6) ⊗ 1 Stars(7.1.0.6) 100 - 1.0.1) S 1 Stars(7.1.0.6) 100 - 1.0.1) S 1 Tars(7.1.0.6) 100 - 1.0.1) S 1 Tars(7.1.0.6) 100 - 1.0.1)
۲he user ca	n select one of the three rehabilita	tion methods by activating the corresponding

Printout Add

and the rehabilitation methods will be

Sika

included in the final report.

checkbox. Then, select the command "Add"

Select the command "Delete", the rehabilitation methods are excluded from the report.

Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the company Sika A.E.

The user has direct access in the library of Sika materials by pressing the button which appear in the dialog boxes about the beam reinforcement.

Sika Select the command "Sika" and then select the appropriate material for each

rehabilitation method. Also, select the following button and a PDF file is downloaded automatically with an analytical description of the material properties as well as information of its use.

«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»



Sika

Concrete Repa	air		×
Corrosion Protection	-		
Sika® Ferrogard®-903+ 🗸 🗸	?		
* *	?		
To Sika® FerroGard®-903+ είναι επιφανειακής εφαρμογής αναστολέας διάβρωσης για το χαλύβδινο οπλισμό κατασκευών 2.Σ., σχεδιασμένος για χρήση με εμποτισμό. To Sika® FerroGard®-903+ βασίζεται σε οργανικά και ανόργανα	^ ~		
Concrete Repair			Sika" FerroGard"-903+ Avarrania; 6d0pwrry; rdh.do III. do umrinouni
Sika® MonoTop®-910 S	?	ř.	(Benautring operation)
Sika® MonoTop® Dynamic 🗸 🗸 🗸	?		Appendix hypotenty h
επισκευαστικό κονίαμα δομητικής αποκατάστης σκυροδέματος	^	Uch C	All the second secon
Crack Filling	Ŷ	ustr -	Expension Foreign and distance and the control of the con
ikaDur@_31	2	- S	Box Real-Approx * Resolution of the All Hort of the Till State Recolleging * Resolution of the All Hort of the Till State
ikaDur@-52	2	75	 All sciences ins septem ing analysis playable pro- al analysis ins second building to deman Many Manage Warning and All sciences
- νεματώσεις δομητικής αποκατάστασης	< _ >		resultati menancia, ser instrumente lasse toter eta aprilar acciona parte della menanzia este este este acciona e este acciona este este este acciona e este acciona este este este acciona e este acciona este este este acciona este este este este este este este acciona este este este este este este este est



4.1.2 Concrete jacket for columns-walls

The section "Jacketing" contains tools for the reinforcement of the columns according to the Code of Structural Interventions (KAN.EPE.2013).

According the Code, the concrete jacket is a uniform concrete layer that surrounds the column cross-section in a closed form. Otherwise, when the concrete layer in a side is not connected to the layer of the adjacent side, then the reinforcing method is considered as additional concrete layers.

	Colur	mn Editor 🛛 🗖 🗙
Rehabilitation		●● ● ● ● ● ● ● ● ● ● ● ●
FRP FRP Protection	Placement Cover (mm) 0 Thidmess (cm) Length (cm) Thidmess (cm) 2 0 0 Side 0 0 Cross-section U-shaped Jacket Materials Concrete : C20/25 Steel (Main) :8500C Sika Bolts - Hangers :8500C Steel (Stirrups) :8500C Sika	
< > Recalculation Joint Check Y = 820.00 + ?? Copy Paste OK Cancel	Design Crecks Performance Level A - DL Side selection Anchorage length (cm) 0 Total calculation Friction mechanism participation percentage(%) 0 Suspensors Stirrups 0 Diameter (mm) 14 show 6 Diameter (mm) 14 show 7 10 cm Dowels Diameter (mm) 14 Number 0 Series 0 Cover (mm) Per (cm) 0 Alternately 0 0 Anchorage length (mm) 0 Calculation Re-check	

Materiale

Define all "Materials" (concrete jacket, main steel reinforcement, stirrups)

Concrete	Steel (Stirrups)
Type C20/25 Υ Constants Fck (Mpa) 20 γcu 1.5 1 γcs 1 1 Fctm (Mpa) 2.2 1	Туре В500С ✓ Constants Es (Gpa) 200 Fyk (Mpa) 500 γsu 1.15 γss 1
TRd (Mpa) 0.25	Max Deformations
Max Deformations	
εc (N,M) 0.0035	OK Cancel
εc (N) 0.002	
OK Cancel	

Herechela		
Concrete : C20/25	Steel (Main) :B500C	C1-
Bolts - Hangers :B500C	Steel (Stirrups) :B500C	ыка



Sika

Furthermore, in SCADA Pro, the techniques and the material considered in each rehabilitation method are enriched with the corresponding material and techniques of the company Sika A.E.

The user has direct access in the library of Sika materials by pressing the button which appear in the dialog boxes about the beam reinforcement.

Select the command "Sika" Sika, and then select the appropriate material for each

rehabilitation method. Also, select the following button and a PDF file is downloaded automatically with an analytical description of the material properties as well as information of its use.

1. Define the "Cover" and "Thickness" of the concrete layer and apply either on the total crosssection as a jacket or on a side by clicking the button "Side" and then selecting with the mouse the corresponding side. In this way, you can define different thickness per side. Therefore, the cover is common in all sides of the cross-sections.

The minimum "Thickness" of the jacket is modified concerning the type of concrete (standard, gunite, special concrete).



When the thickness per side defers, then you select the command "Side" and pick with the mouse the corresponding side. If the thickness is the same in the total cross-section, you select the button "Total cross-section".

Furthermore, there is the option to insert U-shaped Jacket, typing the respective Thickness and Length.



2. Insert the steel reinforcement of the jacket with the commands "Main Reinforcement" and "Stirrups" from the list (Chapter A "Column's Detailing").



- 4. Return to the command "Jacket" for the calculation of the dowels.
- 5. In the field "Stirrups" give the dimeter and the spacing of the stirrups of the jacket.





- Select the appropriate "Performance level"; Damage Limitation-DL (Immediate Occupancy), Significant Damage-SD (Life Safety), Near Collapse-NC (Collapse Prevention).
- Performance Level A DL V
- 7. The compressive force F_{cm} of the jacket is safely transferred as a shear force along the interface through the three following mechanisms:
 - friction
 - welded suspensors
 - dowels

Anchorage length (cm) 0 Friction mechanism participation percentage(%) 0

Which are activated within an available assemblage

length "u_o". The shear resistance along the interface is calculated considering the friction, welded suspensors and dowels mechanisms.

In SCADA Pro the critical mechanism for the transfer of the compressive force is the dowels. The friction and the welded suspensors are option and the user decide if they will be taken into consideration in the calculation of the shear resistance along the interface.

For the welded suspensors define the diameter, the number and the spacing h_s between the new and the existing main steel reinforcement.

Suspensors				
Diameter (m	nm)	14	۷	
Number 0	hs	(mm)	0	

For the friction mechanism you have to define one of the following parameters:

- The assemblage length and then the program calculate the resistance considering the friction coefficient μ =1.0.
- The percentage (%) of the compressive force that will be transferred through the friction mechanism.

In case that the friction and the welded suspensors mechanisms are not taken into consideration, the total compressive force is transferred through the dowels.

8. In the field "Dowels" define the diameter and then the program calculate the number and the spacing of the dowels, as well as the cover in the top, bottom and in both sides:





9. In the field "Design checks", select the following :

Design Checks Side selection Total calculation

- Calculation in total: Select this command and the checks will be performed in all sides of the cross-section (according to KAN.EPE.2013) and the corresponding results will be presented per side.
- Side selection: Select the side in order the checks to be performed per side. Show the corresponding side with the mouse, define the diameter of the dowels and click the button "Calculation". The program calculate the dowels' parameters automatically for the corresponding side.

The command "Recheck" will be activated in a future version of the software.

The results of the design checks are presented in the bottom of the dialog box:

Mz = 63.36 -138.27 y: Vrd,r=753.98 Vrm=603.19 y: (Vrd,r+Vrm)/yR=1043.97 z: Vrd,r=282.74 Vrm=226.19

In the beginning of the design checks, the inertial forces appears in the top and the bottom of the column.

Mx = -0.71 -0.71 My = 14.38 -42.38 Mz = -6.83 15.24

Also, the shear resistance per direction are presented according to the paragraph § 8.2.2.2, KAN.EPE.2013.

y: Vrd,r=331.75 Vrm=331.75		
y: (Vrd,r+Vrm)/yR=510.39		
z: Vrd,r=256.35 Vrm=256.35		_
z: (Vrd,r+Vrm)/yR=394.39		Ŧ
< <u> </u>	•	

At the end, the thickness of the concrete layer is presented for the corresponding side, as well as all the parameters of the dowels.





The program calculates the appropriate number of dowels by comparing the number of dowels based on the value of the compression force and the minimum number of dowels based on the jacket's area and keeps the greater.

▲ In the previous example the minimum number of dowels is 13, while the calculated one is 18, so which is the final number of dowels.

Finally, select the command "Report" to add the design checks' results in the corresponding chapter of the report.

The command "Recheck" will be activated in the future version fo the software.

The analytical printout of the results is defined in the ribbon "Add-ons" in the command "Calculations' Printout".

Basic Modeling View Basic Modeling View Freek Anguages Parameters Bill of Ma	Tools Slabs Loa Steel Cross-Sections terials Calculatio	ds Addons wint Output V Tree V Property View	
	Calculation's	Printout	
Availiable Chapters	Printout	Number of Pages :	Building Data
Analysis	Jacketing	Lev:1	Move Up
Design Jacketing			Move Down
Columns			Delete
Level 1			Delete All
evel 2 Beam			Insert File
Steel Masonry			Error Correction
Bill of Materials			
			Format Page
			Paging
			Export Printou
			Print
			Project Report
			Save
			Cancel

Select the section "Retrofitting methods" and then select a level or levels and the corresponding results of the design checks as well as the calculated number of dowels per level will be recorded.

Material

Steel (Main) (\$275)



4.1.3 FRPs – laminates of columns-walls

The steel laminates or the fiber reinforcing polymers (FRPs) is a reinforcing method that results to the increase of the bending resistance and the application of confinement reinforcement. The laminates in general are used as additional tensile reinforcement due to the inadequate existing steel reinforcement. The laminates strengthen the tension zone against flexural failure.

According to the paragraph §8.2.1.3, KAN.EPE.2013, a reinforced concrete cross-section can be strengthened in bending with steel laminates or FRP fabrics. This reinforcing method is applied mainly in beams and slabs and rarely in columns, because it is not allowed to be applied in regions under compression. As an exception, it can be applied in regions under compression when that regions resist against other type of failures, e.g. local buckling resistance of the rebars by applying confinement.

				Colu	mn Editor – 🗖 🗙
Þ	Rehabilitatior				● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●
	Jacketing FRP Protection	Reinforcing M Material Steel (Ma	Method Steel Plates	~	1
< R	> ecalculation	Performance L A - DL Placement Length (cm) Width (cm) Number of L No bendir S Cross	Level Accessibility (Table S 4.3 Image: Constraint of the state of the stateo	i) *	
+	Y = 300.00 ?? - Copy Paste OK Cancel	Print	tout des 1/1 Automatic Thicken Calculation	255	
1.	Select th	e reinfo	rcing method; Steel Lan	ninat	es or FRPs (Fiber reinforced polymers)
	Reinforcin	ng Method	Steel Plates		×

Fiber Reinforced Polymers



Side

2.	Select the Material.	Material Steel (Main) :S275(Fe430) Type S275(Fe430 V Constants Es (Gpa) 210 Fyk (Mpa) 275 ysu 1.15 yss 1 Max Deformations es 0.02 OK Cancel				
^	Furthermore, in SCAD, method are enriched w The user has direct acc	A Pro, the technic vith the correspon cess in the library	ues and the ding materic of Sika mate	e mater Il and te prials by	ial considere chniques of t pressing the	d in each retrofitting he company Sika A.E. button Sika,
	which appear in the di	alog boxes about t	the beam rei	nforcen	nent	
Sele	ect the command "Sika	a" Sika	and then se	elect th	e appropriat	e material for each
reha auto use	abilitation method. Also matically with an analy	o, select the fol tical description o	lowing butt of the mater	on ? ial prop	and a PDF erties as well	file is downloaded as information of its
3.	Select the Performanc	e Level and the Ac	cessibility le	vel		
P	Performance Level	Accessibility (1	able S 4.3)			
	A - DL \vee	Normal (Usual) ¥			
4	In the field Placement	select.				
P	lacement					
, i	Length (cm) ⁰ Thickness	s 0 Def	fault			
	Width (cm) 0 Anchorag	ge (cm) 0 Si	de *			
1	Number of Layers 0	Strips' Data				
_ •						
Det leng	ault: This command is u gth of the column and th	sed in order the ne width of the	Placement			
lam	inate to be filled in auto	omatically. The	Length (cm) 300	Thickness	0 Default
wid	th of the laminate is eq	uai to the width	Mindala (march	40	Anglassan (m)	0 Side 1

40

Width (cm)

Anchorage (cm) 0

of the corresponding side of the column by

default.



Afterwards, you set the value of the thickness and the length of the laminate in the corresponding fields with two ways:

- 1. For each side: Select the button "Side" and show with the mouse the corresponding side of the column.
- 2. For the total cross-section: Select the button "Cross-section".

The command "Default" sets the data of all sides of the column. If you want to import laminates in all sides with the same thickness, you enter, at first, the thickness and the anchorage length. Then, click the button "Cross-section".

If you want afterwards to change the thickness of the laminates of the cross-section in total, set a new value for the thickness and press the button "Default" without pressing again the button "Total cross-section". The existing laminates change considering the new thickness value.

Side data: This command is used to show the number of the side selected with the mouse and also the data of the reinforcing method applied in the corresponding side.



Layers: In this field set the number of the layers.

0

Spacing (cm)

Strips' Data		The placement of the laminates can be uniform or in strips;
		continuous or discontinuous with intermediate spacing.
Width (cm)	0	Activate the "Continuous Formulation" in the field "Strips' d and define the width of the laminate. For considering the

'ips' data" the discontinuous formulation, deactivate the previous checkbox and define the spacing between the strips.

Activate the checkbox "No participation in bending" No bending participation and the laminate in the corresponding side will not participate in the bending resistance of the reinforced cross-section.

	Printout		
	Checks	1/1	Automatic Thickness Calculation
My Mz My Mz Vy Vz Vy Vz	/ : Msd(72.90 z : Msd(-5.16 / : Msd(-99.1) z : Msd(-7.22) vsd(-7.22) vsd(-57.362 vsd(-57.362 vsd(-57.362 vsd(-57.362	16) <=2/3Mrd'(1 1) <=2/3Mrd'(- 80) <=2/3Mrd'(- 5) <=2/3Mrd'(- < Vrd,c(81.51: 2) < Vrd,c(81.51: 2) < Vrd,c(81.52: 2) < Vrd,c(79.82- 2) < Vrd,c(79.82- 2) < Vrd,c(79.83- 2) <td>37.976 6.227) (-87.09 6.345) 2)(1) : 12)(1) 4)(1) : 24)(1)</td>	37.976 6.227) (-87.09 6.345) 2)(1) : 12)(1) 4)(1) : 24)(1)
Vz «	: Vsd(-57.362	2) < Vrd,c(79.8	24)(1)

By selecting the command "Design checks", the program calculates and presents the minimum thickness t_1 and t_2 per side, in the checks' results concerning the cross-section of the laminate and the type of the material. Then, the thickness t_1 and t_2 are calculated again concerning the minimum



values of t_1 and t_2 and the design checks have to be repeated. Since the calculation of the thickness t_1 and t_2 is an iterative method, select the button "Automatic calculation of the thickness".

Automatic Thickness Calculation

Then the program calculates automatically the final minimum thickness t_2 , which is presented in the window in the bottom of the dialog box. Then, you have to set this calculated value in the corresponding field and repeat the final design checks.

The structural adequacy of the laminate or the FRP is reached with the increase of the thickness or the number of the layers.

In the section of the results of the design checks are presented first the bending resistance checks of the cross-section and the shear resistance check by direction X or Z according to the paragraph § 8.2.1.3 (a), KAN.EPE.2013.

Furthermore the results are presented by side and the value of the parameter ΔM ; the difference between the design bending moment and the moment resistance of the initial cross-section, is calculated. If the parameter ΔM is positive (the initial cross-section should be reinforced) the thickness values t_1 and t_2 are calculated as described above. The thickness is defined by the user. SIDE: 1

∆M=45.86 ojd1 = 293995.859 ojd2 = 447795.526 min T(mm) : t=0.400 t1=0.693 t2=0.455

In the previous example, the thickness t is less than the appropriate t₁ και t₂. So the thickness t must be equal to 0.7. If you preserve the thickness value t=0.4 then should be used two layers. The corresponding results are presented below:

```
SIDE: 1

\Delta M=45.86

\sigma jd1 = 293995.859

\sigma jd2 = 316639.253

min T(mm): t=0.400 t1=0.347 t2=0.322
```

So, if you use two layers, then a minimum thickness t=0.35 is needed.

If $\Delta M=0$, then there is no need for reinforcement, so $t_1=t_2=0$.

Finally, the shear resistance check is presented according to § 8.2.2.2 (iii) KAN.EPE.

Printout Select the command "Printout" to add the results in the corresponding chapter of the report of the study:



4.1.4 Protection of columns-walls

The section "Protection" contains the tools for the application of protection methods in columns according to KAN.EPE.2013.

		Column Editor – 🗆 🗙
Rehabilitat	tior	●● ● ● ● 🗶 肆 啭 慰 💁 🔤
Jacketing	Parket day and	
FRP	Corrosion Protection	\bigcirc 1¢
Y Protection	Materials used on the surface that act as corrosion inhibitors for the steel reinforcement of the reinforced concrete structures and applied by impregnation.	\forall
	Concrete Repair Repairing mortars for the structural rehabilitation of	
	concrete members.	
	Cementitious binding materials for the structural rehabilitation of the concrete cracking, welded and / or	
	infilled. Sika	
	Printout	
<	> Add Delete	
Recalculation	Fire Protection Layers	
X = 200.00	reresistant mortars applied by using epoxy resins.	
+ ?? -	Concrete layers or coating Repairing mortars of one or more components for final protoching coating	
Сору	Paint Protection	
Paste	Plastic-elastic paint protection for concrete and coatings	1 4
OK	Sika	\ominus
The use checkbe methoo	er can select one of the th ox. Then, select the comn ds will be included in the f	ree rehabilitation methods by activating the corresponding Printout Add Delete and "Add" Tinal report.
▲ Fu me	rthermore, in SCADA Pro ethod are enriched with th	, the techniques and the material considered in each retrofitting be corresponding material and techniques of the company Sika A.E.
wł	hich appear in the dialog b	poxes about the beam reinforcement.
Select t	he command "Sika"	Sika , and then select the appropriate material for each
rehabili automa use.	itation method. Also, sele atically with an analytical	ct the following button and a PDF file is downloaded description of the material properties as well as information of its
IMPOR	TANT NOTE:	
The rei	nforced parts of the colur	nns and walls are pointed out on the screen:

- 1. In plan view: The node is colored yellow
- 2. In 3D view: The structural element is colored yellow.





Also, according to the type of the reinforcing method, an indicative letter appears of the reinforcing method:

- Concrete Jacket: "J"
- Laminate: "L"
- ✤ FRP: "F"

Prerequisite for the appearance of the label is that you have first selected the button "Report" in the dialog box of the corresponding beam reinforcing method.



4.2 Beams reinforcer	nent	
	Beams Editor	- 🗆 🗙
5 C C C C	韓 孽	OK Cancel
		REINFORC. TABLE
4.60 4.60 0.2 0.2 0.2 0.2 0.2 0.2 0.2 0.	40 4.60 46 0.4 1.20 4 9 0.4 1.20 8 1.20 5.44 1.20 0.44 1.60 9 0.44 1.20 8 (1.20 1.20 9 0.44 1.20 9 0.44 1.20	Image of the second s
Geometry Span Main Reinforcement Support Reinforce Number of Span 2 General Data Type Concete Jacketing Uniform reinforcement in the total length Cover Slab thickness Accessibility (mm) (cm) (Table S4.3) 0 0 T Normal (Usual)	ement Stirrups Additional Crack control Diagrams Retrofitting method Material Concrete : C8/10 Steel (Main) :S220 Steel (Main) :S220 Bolts - Hangers :S220 Performance level A - DL V	

The section "Retrofitting methods" contains the tool for the seismic reinforcement and rehabilitation of the beams according to the Greek Code of Structural Interventions (KAN.EPE.).

First the calculated steel reinforcement is adjusted to the existing one, then you proceed with the definition of the structural reinforcing techniques.

Select the span graphically or by left click on the span in the cad interface. Otherwise, set the

number of the span in the field "Number of Span"

In the "Beams editor", the beam is depicted in the direction at the input step. To locate quickly the beam of interest among the rest structural elements, it is suggested, the numbering and the local axes of all beams to be displayed and then, the corresponding beam to be selected by setting its serial number in the editor utility. To define the left and right support of the beam, use the direction of the local axis xx'. The start and the end of the beam element in the editor's interface is defined concerning the local axis xx' direction, without considering the beam's orientation in the plan view.

"General Data"



Select from the drop-down list the structural reinforcing method applied to the selected beam.

Concete Jacketing Steel Plates

- Fiber Reinforced Polymers
- The activation of the checkbox "Uniform reinforcement in the total length" Uniform reinforcement in the total length means that the reinforcement, applied in each critical crosssection, will be designed (supports, span) considering the less favorable values of the inertial forces. The less favorable inertial forces are derived from the comparison of the inertial forces in all crosssections of the beam. Otherwise, if the checkbox remains inactive, then the inertial forces of each critical region examined will be used for the design of the corresponding retrofitting method.
- In both options above, the structural reinforcement must be applied in the three critical sections of the beam (left/right support and span).
- In the case of the concrete jacket and the additional concrete layers, type a value in the field "Cover (mm)" to define the concrete cover of the cross-section.
- Activate the checkbox "T" and type the thickness of the plate in the field "Plate thickness (cm)" in case of a T-shaped cross-section. For a Γ-shaped cross-section, type the thickness of the plate without activating the checkbox "T".
- ▲ For zero value for the thickness of the plate, a rectangular cross-section will be considered, either the checkbox "T" is active or inactive.

"Performance level": Define the performance level of the structure.

		A - DL
		B - SD
Performance level	A - DL 🗸 🗸	Γ-NC

"Accessibility": Define the accesibility of the area where the reinforcement is applied according to the § 4.5.3.2 in KAN.EPE.





4.2.1 Additional concrete layers – Concrete jackets

"Materials": Select the type of material for each structural component of the reinforcing method.

Material	Concrete	
Concrete : C8/10	Туре С20/25 🗸	
Steel (Main) :S220	Constants Eck (Mpa) 20	
Steel (Stirrups) :S220	γcu 1.5	Steel (Stirrups)
Bolts - Hangers :S220	γcs 1	Type S220 ✓ Constants
	Fctm (Mpa) 2.2	Es (Gpa) 200
	TRd (Mpa) 0.25	Fyk (Mpa) 220
	Max Deformations	γsu 1.15
	εc (N,M) 0.0035	yss 1
	ες (N) 0.002	Max Deformations
		εs 0.02
	OK Cancel	OK Cancel

"Input Data": Define the input data of the concrete jacket in the two supports and the span of the beam.

Left support The same in both sides The fade rebars are taken into consideration The side rebars are taken into consideration The side rebars are taken into consideration Do not participate in the bending Rebasilitation Right support Left side Left side Rebasilitation Protection Left side Left side Left side Rebasilitation Protection Do not participate in the bending resistance check Rebars Rebars Rebars Corner 0 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance check Rebars Corner 0 0 0 Do not participate in the bending resistance c	Input data		Beam structural reinforcement	×
Left side Length (m) 120 Thickness (m) 0 Do not participate in the bending resistance check Rebars Corner	Left support Span Right support	The same in both sides The side rebars are taken into consideration The additional rebars are taken into consideration Default Default	Top flange Length (cm) 120 Thickness (cm) 0 Do not participate in the bending resistance check Rebars 4 0 \sim 0 0 Φ \bullet d1(cm) 0	Sika OK Cancel Rehabilitation Protection
		Left side Length (cm) 120 Thickness (cm) 0 Do not participate in the bending resistance check Rebars Corner ϕ 6 \checkmark Intermediate 0 ϕ 6 \checkmark Dowels - Suspensors Diameter (nm) 6 \checkmark Anchorage Length (mm) 0 Stirrups ϕ 6 \checkmark / 0 cm Ultimate Moment Resistance Initial Reinforced	Bottom flange Length (cm) 120 Thickness (cm) 0 Do not participate in the bending resistance check Rebars 4 6 0 6 0 6 0 0	Right side Length (cm) 120 Thickness (cm) 0 Do not participate in the bending resistance check Rebars Corner Design Checks Report



In the dialog box "Beam structural reinforcement" that appears, you put the concrete jacket per side (top, bottom, left side, right side). There is also a preview window part of the dialog box where the settings are depicted.

The calculations and the design checks of the beam are the same with the checks of the column presented in Chapter A.

To be taken into consideration the existing side rebars in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

The side-rebars are taken into consideration

To be taken into consideration the existing additional steel reinforcement in the supports of the beam, in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

The additional rebars are taken into consideration

If you activate the following checkbox The same in both sides then, the field "Right Side" is deactivated and receive the same input parameters as they are defined in the field "Left Side".

Right side				
Length (cm)	120			
Thickness (cm	n) O			
Do not participate in the bending resistance check				
Rebars				
Corner	Φ 6 🗸 🗌			
Internediate 0	Φ 6 🗸			

NOTICE:

If you select the button **"Default"**, then, the corresponding length of the supports and the span are defined automatically. The support's length is the critical length and the span's length is the total length of the beam minus the critical of the two supports. The calculated lengths correspond to the reinforcing length.

Ultimate Moment Resistance		
Initial	Reinforced	

In the field "Ultimate Moments Resistance", if you select the button "Initial" or "Reinforced", then the ultimate bending resistance is calculated of the existing or he reinforced cross-

In the field "Dowels - Suspensors" you define the input parameters of the dowels or the suspensors; diameter and anchorage length.

Dowels - Suspensors			
Diameter (mm)	16	۷	
Anchorage Length (mm)	100		

In the field "Stirrups" set the diameter and the spacing of the stirrups of the concrete jacket.

Stirru	ups -				
Φ	8	~	1	10	cm

section, respectively.



Input parameters in the fields "Top" and "Bottom"

If you activate the checkbox "Not considered in the bending resistance check", then the corresponding concrete jacket or the additional concrete layer are not taken into consideration in the calculation process of the reinforced cross-section.

In the first line of the input parameters of the steel reinforcement, you define the number of the rebars of the first/basic rebars' layer and their diameter. For more than one layers of rebars, you define in the second line, the number of the rebar layers, the diameter of the rebars and the spacing d_1 between the rebar layers.

The rebar layers have always two rebars.

The activation of the checkbox, on the right of the diameter's drop-down list 20, means that the corresponding steel reinforcement is not taken into account in the calculation of the ultimate moment resistance.

Left side Length (cm) 50 Thickness (cm) 10	• •	
Do not participate in the bending resistance check	·	
Rebars	•	
Corner Φ 14 ¥	•	
Intermediate 3 Φ 14 \checkmark		

Input parameters in the fields "Left Side" and "Right Side"

In the field "Rebars" you define the diameter of the corner rebars of the corresponding side, as well as, the number and the diameter of the intermediate rebars. The rest parameters are the same with the parameters of the top and bottom of the cross-section.

In the field "Ultimate Moment Resistance" select the button "Initial" of "Reinforced". Then, a new dialog box opens, where the interaction diagram of the uniaxial bending moment and the axial force is depicted for the existing and the reinforced cross-section, respectively.



«ASSESSMENT AND REDESIGN ACCORDING TO EC8 part3»



	Beam structural reinforcement	×
 ✓ The same in both sides ✓ The side-rebars are taken into consideration ✓ The additional rebars are taken into consideration 	Top flange Length (cm) 0 Thickness (cm) 0 Sika Do not participate in the bending resistance check Rebars	OK Cancel
Default	4 Φ 16 ✓ Rehabit 2 Φ 12 ✓ d1(cm) 3 Prote	litation
Left side Length (cm) 50 Thickness (cm) 10 Do not participate in the bending resistance check Rebars Corner ϕ 14 \checkmark Intermediate 3 ϕ 14 \checkmark ϕ 14 \checkmark	Right side Length (cm Thickness (c Do not partici- bending resis Rebars Corner Internediate 0	$ \begin{array}{c c} \mathbf{n} & 120 \\ \hline \mathbf{m} & 0 \\ \hline \mathbf{p} ate in the tance check \\ \hline \mathbf{\Phi} & 6 \\ \hline \mathbf{\Phi} & 6 \\ \hline \mathbf{\Phi} & 6 \\ \hline \end{array} $
Dowels - Suspensors Diameter (mm) 15 Anchorage Length (mm) 100 Stirrups ϕ 8 / 10 cm Ultimate Moment Resistance Initial Reinforced	Bottom flange Length (cm) 120 Do not participate in the bending resistance check Rebars 4 0 0 0 0 0	Report

Select the command "Design checks" and the program performs all design checks for the concrete jacket in all reinforced sides with concrete jacket (according to the Code of structural interventions) and so the appropriate number of dowels is calculated. Those design checks and their corresponding results are similar to those considered for the columns.

The results of the design checks are presented on the bottom of the window. Finally, select the command "Report" and the results of the design checks will be added in the corresponding chapter of the Report.

After any modification on the parameters of the concrete jacket, select the command "Report", so that the repost to be updated with the new data as appropriate.



4.2.2 Steel laminates – FRPs

For these two retrofitting methods, the sane procedure is followed as previously. For the steel laminates or FRPs, select the corresponding layer from the drop-down list.

General I	Data	
Туре	Steel Plates	¥
🖌 Unifi	Concete Jacketing	
Cover	Steel Plates Fiber Reinforced Polymers	

"Material": Select the type of the steel for the laminates and the fiber reinforced polymers.

	Steel (Stirrups)
Material	Type S235(Fe360 ∨
Concrete : C8/10	Constants
Steel (Stirrups) :S220	Fyk (Mpa) 235
Bolts - Hangers :S220	ysu 1.15
	yss 1 Max Deformations es 0.02

The performance objective and the accessibility level are defined following the same procedure as for the definition of the concrete jacket.

"Input data": Set the input data for the "Steel Laminates" or the "FRPs" for the supports and the sap of the beam.

	Beam structural reinforcement	×
The same in both sides The side-rebars are taken into consideration The additional rebars are taken into consider Default	Top flange 0 Thickness (mm) 0 Width (cm) 0 Anchorage (cm) 0 Number of layers 1 1 Not participate in bending 1 1	Sika Report Rehabilitation Protection
Left side Length (cm) 0 Thidoness (mm) 0 Width (cm) 0 Anchorage (cm) 0 Number of layers 1 Not participate in bending Strips' data Continuous placement Width (cm) 0 Spacing (cm) 0	· · ·	Right side Length (cm) 0 Thickness (mm) 0 Width (cm) 0 Anchorage (cm) 0 Number of layers 1 Not participate in bending Strips' data Continuous placement Width (cm) 0 Spacing (cm) 0
Ultimate Moment Resistance Initial Reinforced	Bottom flange Length (cm) 0 Thickness (mm) 0 Width (cm) 0 Anchorage (cm) 0 Number of layers 1 Not participate in bending	calculation



In the dialog box "Beam Reinforcement" that appears, the input data per side of the cross-section (top, bottom, left side, right side) are defined. There is also a window in the dialog box, where the results of the design checks are summarized.

The calculations and the design checks of the beam are the same with the checks of the column presented in Chapter A.

To be taken into consideration the existing side rebars in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

The side-rebars are taken into consideration

To be taken into consideration the existing additional steel reinforcement in the supports of the beam, in the calculation of the ultimate moment resistance of the cross-section, you activate the following checkbox:

The additional rebars are taken into consideration

If you activate the following checkbox The same in both sides then, the field "Right Side" is deactivated and receive the same input parameters as they are defined in the field "Left Side".

Right side			
Length (cm)	120		
Thickness (cm)	0		
Do not participate in the bending resistance check			
Rebars			
Corner d	6 🗸 🗌		
Internediate 0 d	6 🗸		

If you select the button **"Default"**, then, the corresponding length of the supports and the span are defined automatically. The support's length is the critical length and the span's length is the total length of the beam minus the critical of the two supports. The calculated lengths correspond to the reinforcing length.

Ultimate Moment Resistance				
Initial	Initial Reinforced			

In the field "Ultimate Moments Resistance", if you select the button "Initial" or "Reinforced", then the ultimate bending resistance is calculated of the existing or he reinforced cross-section, respectively.



Input parameters in the fields "Top" and "Bottom"

Top fange 50 Thickness (mm) 1 Width (cm) 25 Anchorage (cm) 40 Number of layers 1 1 Not participate in bending 1 1				
Length (cm) 50 Thickness (mm) 1 Width (cm) 25 Anchorage (cm) 40 Number of layers 1 Not participate in bending	Top flange		1	
Width (cm) 25 Anchorage (cm) 40 Number of layers 1 . . Not participate in bending 	Length (cm)	50	Thickness (mm)	1
Number of layers 1 Not participate in bending	Width (cm)	25	Anchorage (cm)	40
Not participate in bending	Number of la	iyers	1	
• •	Not parti	cipate in	bending	
· · ·				
· · ·				
		•	•	
			•	

If you activate the checkbox "Not considered in the bending resistance check", then the corresponding concrete jacket or the additional concrete layer are not taken into consideration in the calculation process of the reinforced cross-section.

For the length of the laminate is defined as the length for the concrete jacket. The width of the laminate is defined at first equal to the width of the corresponding side. The definition of the anchorage length is mandatory and the program at first sets a value by default, which can be modified by the user. The number of the layers is the number of the layers of the reinforcement.

Left side Length (cm) 40 Thickness (mm) 1 Width (cm) 50 Anchorage (cm) 33	• •
Number of layers 1	· ·
Continuous placement Width (cm) 0 Spacing (cm) 0	

Input parameters in the fields "Left Side" and "Right Side"

The definition of the reinforcement's geometry is defined with similar way to the one placed on the top and the bottom of the cross-section. The checkbox about the non-consideration of the laminates of the left and right side in the bending resistance of the cross-section is already activated, because they reinforce the cross-section mainly against shear failure.

The placement of the laminates can be uniform or in strips; continuous or discontinuous with intermediate spacing. Activate the "Continuous Formulation" in the field "Strips' data" and define the width of the laminate. For considering the discontinuous formulation, deactivate the previous checkbox and define the spacing between the strips.

calculation



Sika

By selecting the command "Design checks", the program calculates and presents the minimum thickness t_1 and t_2 per side, in the checks' results concerning the cross-section of the laminate and the type of the material. Then, the thickness t_1 and t_2 are calculated again concerning the minimum values of t_1 and t_2 and the design checks have to be repeated. Since the calculation of the thickness t_1 and t_2 is an iterative method, select the button "Automatic calculation of the Automatic thickness

thickness"

Then the program calculates automatically the final minimum thickness t_2 , which is presented in the window in the bottom of the dialog box. Then, you have to set this calculated value in the corresponding field and repeat the final design checks.

A The structural adequacy of the laminate or the FRP is reached with the increase of the thickness or the number of the layers.

Report Finally, select the command "Report" and the results of the design checks will be added in the corresponding chapter of the Report.

- For each modification in the steel laminate or the FRPs, repeat the selection of the command "Report" to be added in the final report of the study.
- Furthermore, in SCADA Pro, the techniques and the material considered in each retrofitting method are enriched with the corresponding material and techniques of the company Sika A.E.

The user has direct access in the library of Sika materials by pressing the button which appear in the dialog boxes about the beam reinforcement.

The buttons "Rehabilitation" and "Protection" correspond to a dialog box each with tools about the rehabilitation and protection of the beam structural elements according to the Code of Structural Interventions (KAN.EPE).

	Rehabilitation - Protection	Rehabilitation - Protection
Rehabilitation	Concrete Repair	Concrete Repair
Protection	Corrosion Protection Materials used on the surface that act as corrosion inhibitors for the steel reinforcement of the reinforced concrete structures and apolied by imprennation.	Corrosion Protection Materials used on the surface that act as corrosion inhibitors for the steel reinforcement of the reinforced concrete structures and acoled by impremention.
roccom	Concrete Repair Repairing mortars for the structural rehabilitation of concrete members.	Concrete Repair Repairing mortars for the structural rehabilitation of concrete members.
	Crack Filing Cementibious binding materials for the structural rehabilitation of the concrete cracking, welded and / or infiled.	Crack Filing Cementibious binding materials for the structural rehabilitation of the concrete cracking, welded and / or infiled.
	Sika	Sika
	Add Delete	Add Delete
	Protection Fire Protection Layers Fire resistant mortars applied by using epoxy resin.	Protection Fire Protection Layers Fire resistant mortars applied by using epoxy resin.
	Concrete layers or coating Repairing mortars of one or more components for final protective coating.	Concrete layers or coating Repairing mortars of one or more components for final protective coating.
	Plant Protection Plastic-elastic paint protection for concrete and coatings.	Paint Protection Plastic-elastic paint protection for concrete and coatings.
	Sika	Ska
	OK Cancel	OK Cancel
The user can select one of the thr	ee rehabilitation and prote	ction methods by activating t

corresponding checkbox. By selecting the command "Report" Report can add the corresponding information in the final report of the study.


The reinforced beams are indicated on the screen like the reinforced columns: The corresponding structural member is colored yellow:





Also, according to the type of the reinforcing method, an indicative letter appears of the reinforcing method:

- Concrete Jacket: "J"
- Laminate: "L"
- ✤ FRP: "F"



The following definition takes place according to the local axes of the beam (according to the direction that it was drawn: from left to right and vice versa). For that reason always activate the view of the local axes before the input of the reinforcing method.

Prerequisite for the appearance of the label is that you have first selected the button "Report" in the dialog box of the corresponding beam reinforcing method.



5 th STEP: ELASTIC ANALYSIS

In the case of an elastic analysis, you create a new analysis scenario; static or dynamic analysis.

So	cenario		×
Renumbering Nodes No	~		
Disable	Name		
EC8_General Static EC8_General Precheck Static	Analysis	EC8_General	*
EC8_General Precheck Dynam	Туре	Elastic Dynamic	~
Co_General Elastic Dynamic	Properties Elemer	Static Dynamic Nonlinear Linear	
	Load Ca	Elastic Dynamic Precheck Static Precheck Dynamic	
	New	Time History Dynamic	
		Exit	

Then, you follow the process of performing a scenario. In the dialog box "Parameters" do the following :

	EC8 Parameters	×
Seismic Area Seismic Areas	Characteristic Periods Spectrum Type Horizontal Vertical	Levels XZ Down 0 - 0.00 V Up 0 - 0.00 V
Zone I v a 0.16	Type 1 S,avg 1.2 0.9 Soil TB(S) 0.15 0.05	Dynamic Analysis Eigenvalues 10 Accuracy 0.001
Importance Zone II v Yi 1	B V TC(S) 0.5 0.15 TD(S) 2 1	Spectrum Participation factors PFx 0 PFy 0 PFz 0
Spectrum Response Spectrum Elastic ζ 5 Hori	Ductility Class DCM v zontal b0 2.5 Vertical b0 3	Ессеntricities Sd (T) е тих 0.05 алх Sd (TX) 1 sd (TY) 1
Response Spectrum	Jpdate Spectrum Sd(T) >= 0.2 ag	e πz 0.05 *Lz Sd (TZ) 1
Concrete v qx Structural Type	▼ 1 qy ▼ 1 qz ▼ 1	X One X All the other cases
X Frame System Structural Type T1 according to 4.3.3.2.2	Z Frame System	Z All the other cases
X Concrete Moment Resisting	g Frames v Z Concret	e Moment Resisting Frames V
Interstorey Drift Limit	0.005 Damaged Buildings	Walls KANEPE Default OK Cancel



First define the corresponding parameters like you do for the analysis scenario of EC8.

Since you have defined the corresponding parameters, perform the scenario for the elastic analysis and then select "Load Combinations" \rightarrow "Default" and save the file of the load combinations that will be used for the design of the reinforcement methods.

If you select "Checks" all the checks appear, with the criteria for the selection of the elastic method included. Nevertheless, the check considering the insufficiency index is the most important.

κρ 	ισιμοι	δεικτες		ανεπο	αρκεια	ςλ (5ομικω	ν στα	ວເχεເພາ	ν (πα _β	p.5.5	.2α(1))		
	α/α	Συν/κο	ī		Δo	κοί	1	1	Υποστυλ	λώματα	x I		Σύνολ	0	1
ĮΣ	τάθμης	Υψός (M)	I	λ<=	=1.0	λ:	>1.0	λ<:	=1.0	λ>:	1.0	λ<=	1.0	λ>:	1.0
1-	1	3.000	+	4	+ 12%1	3	+ 9%1	6	+· 20%1	0	+- 0%1	10	16%	3	 5%
i.	2	6.000	i	6	18%	1	3%	6	20%	ō	0%	12	19%	1	2%
1	3	9.000	I	6	18%	1	3%	6	20%	0	0%	12	19%	1	2%
1	4	12.000	I	4	12%	0	0%	4	13%	0	0%	8	13%	0	0%
1	5	15.000	I	4	12%	0	0%	4	13%	0	0%	8	13%	0	0%
I.	6	18.000	I	4	12%	0	0%	4	13%	0	0%	8	13%	0	0%
	Σύνολα	 D	1	28	85%	5	15%	30	100%	0	+- 0%	58	92%	5	8%)
			-												

Για όλα τα στοιχεία πρέπει λ<=1.0. Εαν λ>1.0 το κτίριο πρέπει να είναι μορφολογικά κανονικό. ---- Ο ελεγχος Δεν Ικανοποιείται

In the table above the structural member in failure are presented that should be reinforced. The check about the insufficiency index is implemented regarding internal forces (bending moments). The program calculates the indices λ due to bending for the structural elements (ductile and brittle). The classification of the structural members in ductile or brittle is done simultaneously. Three criteria of brittle failure mechanisms are implemented according to KAN.EPE.2013. If one of the three criteria is fulfilled, then the structural element is considered as brittle and so, the insufficiency index λ , is calculated in term of shear force. This calculation is made with no respect to the analysis method "m" or "q".

The failed structural elements and the results of the brittle and ductile structural elements are presented below analytically in the section about the print-out files.

When it comes to the import and the design of the reinforcement methods, see the description in the nonlinear analysis section. Pay attention though in the following features:

- Important condition for the design of the reinforcement methods, when elastic analysis is implemented, is the selection and the calculation of the load combinations that you saved in the previous step.
- ▲ When it comes to the columns' concrete jacket, important condition is in the field "Checks" in the drop-down list "Performance level" to click the option with the sign "*******".

Γ-NC



Design Checks		
Side selection	Performance Level	*****: \
	Anchorage length (cm)	A - DL B - SD
Total calculation	participation percentage(%)) F - NC
Suspensors	Sti	In upa
The same option sho	uld be selected for the Ff	RPs – Laminates of the columns
Performance Level	Accessibility (Tal	ble S 4.3)
****** 🗸	Normal (Usual)	~
A - DL		

Thickness

1 The same option should be also selected for the beams in the dialog box "Beams Detailing".

0

Default

Geometry	Span Main Reinforcement	Support Reinforcement	Stirrups	Additional	Crack control	Diagrams	Retrofitting method	
Number o	f Span 1 Data	Ma	terial Concre	te : C8/10	Input	data ft support		
Type	ype Concete Jacketing Uniform reinforcement in the total length Cover Slab thickness Accessibility		Steel (Main) :S220 Steel (Stirrups) :S220 Bolts - Hangers :S220			Span Right support		
0		(Usual) Y Peri	formance le	evel A - C A - C B - S	DL V SD			
				***	VC ****			

Since you have inserted the reinforcement methods and calculated the new ultimate bending moment resistance, go back to the Ribbon "Analysis", perform the elastic analysis scenario and see again the value of the insufficiency indices λ of the structural elements.

In the section of the printout and the field of the analysis scenarios are presented the scenarios that you have already created.

In the preliminary elastic analysis scenario, apart from the already described section, there are also the following options:

- <u>Analysis results (the results of the criteria that were described above)</u>
- Insufficiency indices λ : The corresponding values per structural element are presented analytically in the stage of the preliminary analysis.

It should be mentioned that the upper bound of the index λ is 2.5 for the preliminary analysis scenario.