

User's Manual 7.ANALYSIS







CONTENTS

I.	THE NEW UPGRADED INTERFACE OF SCADA PRO	3
Ш.	DETAILED DESCRIPTION OF THE NEW INTERFACE	4
AN	ALYSIS	4
1.	SCENARIOS	4
1.	. NEW	
1.	ACTIVE SCENARIO	
1.3	5 KUN	11
1.	SCENARIO	13
	§ Scenario Dunamic	15 15
	§ Scenario Dynamic	15 10
	5 Latocoue Scenarios Anglusis and Static Tupa	10 10
	 Σ Δυάλυση EC-8. General και Τύπο Dynamic 	19 20
	δ 18.2 FC-8 General Static Analysis and FC-8 General Dynamic Analysis	
	 5 3 FC-8 General Analysis and Inelastic (Pushover) Type 	20 30
	δ <u>4</u> Analysis SRC301	
	 § 5 FC-8 Greek Analysis and Time History Linear Type 	50
2.	RESULTS.	
2.1	Combinations	
	§ For Linear Analysis	
	§ Combinations for Wind-Snow	
	§ For Nonlinear (Pushover) Analysis	
2.2	2 Снескѕ	60
	§ For Linear Analysis	
	§ For Nonlinear (Pushover) Analyses	61
2.3	SEISMIC FORCE:	65
	§ Seismic Action of Scenarios Of seismic elastic Static analyzes	65
	§ Seismic Action of seismic inelastic analyzes	65
3.	VIEW	67
	§ For Linear Analysis	67
	§ For Nonlinear (Pushover) Analysis	68
	§ 1.Capacity Curve	69
	§ 2.Idealized Capacity Curve	69
	§ 3.Target Displacement	70
	§ Representation of the structure	71
	§ Member's Moment-Rotation Diagram	74
	§ For Time History Analysis	80





THE NEW UPGRADED INTERFACE of SCADA Pro Ι.



II. DETAILED DESCRIPTION OF THE NEW INTERFACE

In the new upgraded SCADA Pro, all program commands are grouped into the 12 Units.

Basic	Modeling	View	Tools	Slabs	Loads	Analysis	Post-Processo	r Members D	esign Drawin	gs-Detailing	Addons	Optimization
							Analys	is				
	2-470.00 🔹 🏀	• 🖬 造	• 1) =					14MATZ - S	cada		
	Basic Modeli	ng Viev	v Tools	Slabs	Loads	Analysis	Post-Processor	Members Design	Drawings-Detailing	Addons		
New	Seismic E.A.K.(Static) (0)	- 6	Combinat		Enirmic						
INCOV	Scene	ries	Kull	Combinat	Results	Force Di	istribution Deviation	n Stiffness X Stiffne	ss Z Stiffness X Stiffne View	ess Z Distribution	X Distribution 2	Deviation Deviation
The 7	7 th Unit c Scena Resu View	alled arios lts	"ANA	LYSIS'	' inclu	udes th	ne followir	ng three g	groups of a	commar	nds:	
nem SCAD and r	Since mo ber, for A Pro co nonlinea	odelin the de ontain r anal	g and esign c is Eurc lysis m	loads of the opean nethoo	' assi struc , Gree ds is p	gnmen ture, fo ek, and propos	it have be ollows, ba d Italian Do sed, as app	en complesed on pr esign Cod propriate.	eted, the ' ovisions o es, in whi	'Analysi f the cur ch the a	s" of th rrent de pplicati	e structural esign codes. on of linear



"Scenarios" commands group allows scenarios' creation (choosing regulation and type of analysis) and implementation.

1.1 New

The command for the scenarios' creation

Scenario				×
Renumbering Nodes Cuthill-McKee(II)	~ 🗹	Advance Multi-Thre	d eaded Solver	
Disable	Name			_
EC8_General Static (0) EC8_General Dynamic (1)	Analysis	EC8_Ge	neral y	~
	Туре	Static	`	~
	Propertie	S		
	Eleme	nts	Nodes	
	Load Ca	ases	Masses	
	New		Update	
		Run all Ar	nalyses	
		Exi	t	

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



The program is now integrated with new rapid analysis algorithms, using more resources, such as the graphic card, resulting in the more rapid implementation of theirs (Parallel Processing). The activation is achieved through the creation of scenarios.



						Analysis	EC8_General	\sim
Analysis	Sciemia	ы				Туре	Static	~
Pendiyala	- Seisinic	*	Analysis	Seismic	~	Properties	Static	
	Seismic EC-8_Greek		Туре	E.A.K.(Static)	~	Elemer	Dynamic Nonlinear	
	NTC_2008 EC8_Italia		Properties	E.A.K.(Static) E.A.K. (Dynamic-eτi)		Load Ca	Elastic Dynamic Preliminary Static	
	EC8_Cyprus EC8_Austrian EC8_General		Elemer	E.A.K. (Dynamic) Greek 1959-84		New	Preliminary Dynamic Time History Linear	

Select the design code from the "Analysis" list and the analysis method from the "Type" list and click New to create a new analysis scenario. Optionally, type a name.

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

0.1 0							
Other Para	meters	Scr	een	Drawin	ng Laint De	Display	
Project	General Info	mation		Мас	enai - ne	gulation	
Regulation	EC					~	
National	General					~	
Standard Ste	el Shapes		Euro	~	Metric	~ ~	
Concrete			Stee	el			
Foundation	C20/25	\sim	Mem	bers - Eleme	ents S27	′5(Fe430) ∨	
Upper	C20/25	\sim	Stee	l Plate	S27	′5(Fe430) ∨	
			Bolts	•	4.8	~	
Steel			Weld	I	S27	/5(Fe430) 🗸	1
Main	S400s	\sim					
Stimups	S400s	\sim	Timb	er	C14	~	
Safety Facto	rs		-MO	JM1	_M2	UM2	
Ultimate	Serviceabilit	t F	1	1	1 25	1 25	٦
γc 1.5	1	Ľ	vM4				
γs 1.15	1	1	1 1	1	1.1		



Select among the possible scenarios provided in SCADA Pro: For Greece:

LINEAR – NON LINEAR METHODS

- EAK Static	Simplified spectral analysis according to EAK
- EAK Dynamic-eti	Dynamic spectral analysis according to EAK
- EAK Dynamic	Dynamic spectral analysis (masses displacement)
	according to EAK
- Old 1959-84	Seismic analysis according to 1959 Regulation
- Old 1984-93	Seismic analysis according to 1984 Regulation
- Static	Static Analysis without seismic actions
- EC 8 Greek static	Static analysis according to Eurocode 8 and the
	Greek Annex
- EC8 Greek dynamic	Dynamic analysis according to Eurocode 8 and the
	Greek Annex
- EC 8 Greek Preliminary Static	Static Preliminary analysis according to KANEPE
- EC8 Greek Preliminary Dynamic	Dynamic Preliminary analysis according to KANEPE
- EC 8 Greek Time History Linear	Static analysis according to Eurocode 8
- EC 8 Greek Time History Non Linear	Dynamic analysis according to Eurocode 8
- EC 8 Greek NonLinear	Nonlinear analysis according to Eurocode 8 &
	KANEPE.

For other countries:

LINEAR - NON LINEAR METHODS

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

Select

Exit

to save the scenarios and move on to the analysis.



PROPERTIES

"**Properties**" field includes the buttons Elements, Nodes and Loads to define the relevant coefficients.

Properties Elements	1	lodes		§ Bo in the	oth pi se Ele	edef ment	ined a ts, Noc	nd ne des an	ew sce Id Load	narios have, by default, filled ls. The user can modify them
Load Cases	M	asses								
Elements										
	Mult	ipliers	of Pro	perty V	alues (Concr	ete)			✓
EC8_General Dynamic		a la la								1.0
Linear Element Prope	rty Value	Multiplier	S AL	Acv	Ac7	-	Īv	Tv	17	E.A.K.
Concrete V			-	~3y						EC
Steel	<u> </u>	1	1	1	1	1	1	1	1	
BEAMS - TRUSS	1	1	1	1	1	1	1	1	1	
BEAMS - B3Def	1	1	1	1	1	1	1	1	1	
COLUMNS - B3D	1	1	1	1	1	1	1	1	1	
COLUMNS - TRUSS	1	1	1	1	1	1	1	1	1	
WALLS - B3D	1	1	1	1	1	1	1	1	1	
WALLS - TRUSS	1	1	1	1	1	1	1	1	1	
Walls Filter (Lmax/Lmir	n) > 4	1			(Ж		Cancel		

Click "Elements" to open the dialog box that contains the multipliers of the characteristic properties of the linear element, considered in the analysis.

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

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

Е	G	Ak	Asy	Asz	ε	Ix	Iy	Iz		E	G	Ak	Asy	Asz	ε	Ix	Iy	Iz
1	1	1	1	1	1	0.1	1	0.5	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	1	0.5	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	1	0.5	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	1	1	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	1	1	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.666	0.666	1	1	1	1	1	1	1	0.1	0.5	0.5
1	1	1	1	1	1	0.1	0.666	0.666	1	1	1	1	1	1	1	0.1	0.5	0.5

- EC8_General is a scenario without an Annex considered. This means that the user has to update the coefficients according to a state Annex.
- Using EC8_Greek, Italy, Cyprus and Austrian all coefficients are automatically filled in.



You can also define the dimensions for vertical elements to qualify "Walls".

You can also u	enne tr		ension	STOP	ertica	reiem	ents to	y quali	iy vva	1115 .		
Walls Filter (Lma	ax/Lmin))> ['	4									
Press Updat	e t	o unda	to tho	scena	rio in	luding	those	chang	700			
11035		o upuu		Jeene		liuume	sincse	. chung	505.			
Nodes												
					Selec	t whe	ther to	consi	der sla	b's M	aster N	lode (FSR) by
	Node	s		×	selec	ting "ו	′es" (d	efault)) or no	t by s	electin	g "No".
EC8_General D	ynamic											
Master Nodes	Yes		~		More	ovor	choos	o whot	bor to		tho co	rrocponding
Springs					displa	aceme	ent or r	otatio	n of th	e fou	ndatior	n's springs or
Dx	Dy		Dz		not (fixed s	uppor	t cond	itions)			0
Yes 🗸	Yes	✓ Ye	s ∀									
Rx	Ry		Rz									
Yes 🗸	Yes	✓ Ye	s ∨									
ок		Cano	el									
Updat	-											
Press Opdat	te te	o upda	te sce	narios	inclu	ding th	e char	nges.				
Load Cases												
			Loa	d Case	e parti	cipatio	n				×	
EC8_General Dyr	namic		_									
Scenario	g(m/sec2)	9.81		Availa	ible Load	d Cases a	and Load	Groups				
G(1) + Q(2) +	LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG10	
	LC1	1.00	1.00									
	102	0.00	0.00									
	<		-			Г					>	
				ОК			Canc	el				

In this form the loads' participation factors are defined, i.e. the participation of each "Load Case of Scenario" (LC) including the load groups (see "Loads">>"Load Groups").



For scenarios considering the seismic actions,

- select "Dead Loads" G(1) and type 1.00 next to LC1, under LG1 or LG2 or both (it depends on your choice to consider all dead loads together or not).
- select "Live Loads" Q(2), and type 1.00 next to LC2, under LG1 or LG2 or both (it depends on your choice to consider all live loads together or not).

G(1) +

"+" sign located next to the load category Q(2) + indicates that there is an indicative multiplier for the participation of the specific load.

Scenarios without considering seismic actions (simple static method),

- Each load case ("Load Cases of Scenario") is displayed with a number (i.e. LC1) and contains a load and its groups (i.e LG1). The load group is taken into consideration when the value in the corresponding cell is set to a value different than 0.00.
- Each Analysis Scenario can contain up to 4 loads.

Load Case participation Х Static Un. Temp. VarStatic Load Cases of 9.81 g(m/sec2) Available Load Cases and Load Groups Scenario 1+ LC LG1 LG2 LG3 LG4 LG5 LG6 LG7 LG8 LG9 LG... ~ 2+ LC1 0.00 3+ 0.00 4 + LC2 5 LC3 0.00 6 LC4 1.00 7 8 9 10 11 12 13 14 15 16 Ý OK Cancel

> Update to apply any performed modifications.



Click

EXAMPLE:

For example, the LC3 is defined as the first load of the Static scenario.



			Load Ca	ase par	ticipati	on	×	
Static Wind OStatic			-					
enario g	(m/sec2)	9.81	Av	ailable Lo	ad Cases	and Load Groups		
	LC LC	G1 LG2	LG3 LG4			Load Case Definit	tion	×
+	LC1 0.	00				Loud Case Denni		
+	LC2 0.	00		Se	lf-weight	Wind 0 Cpe p+Cpi	~	Insert
	LC3 1.	00						Libert
	LC5 0.	00		LC	S.W.	Description	^	el From Lis
	LC6 0.	00		1	Yes	Dead Load		
2	LC7 0.	00		2	No	Live Load		Delete
2	LC8 0.	00		3	No	Wind 0 Cpe_p+Cpi		Loads
	LC10 0	00		4	No	Wind 0 Cpe_p-Cpi		
	LC11 0.	00		5	No	Wind 0 Cpe_n+Cpi		Delete All
~	I C12 0	00		6	No	Wind 0 Cpe_n-Cpi		Loads
				7	No	Wind 90 Cpe_p+Cpi	~	
				ŝ		1051 00 011 1 011	>	ОК
E.A.K.(Static E.A.K. (Dyna eneral Dynam le: first so able c E.A.K.(Stati c E.A.K.(Dyn	elect tl	he sce	Seismic E. Seismic E. EC8_Gene Dynamic nario fr ign to aj Update'	A.K.(Sta A.K. (Dy eral Dyna om th opear ')	tic) namice amic ne list, . Dead	τi) then activate and ctivate "Disable" to	press "U restore	pdate" in order it. (select > dea
"Renumbe ons: choice of Defaul "Cuthi	ering" each c lt choid II-Mck	field i option ce: "C ee" a	ncludes affects Cuthill-N nd "Asc analysis	a dro the c Ickee endin s, whi	ompu ompu (II)". g Ord ile ch	vn list with multiple tational time. er" take more time oosing "No" is no	e Renur Node: t	mbering ^S Cuthill-McKee(II) No Ascending Order Cuthill-McKee Cuthill-McKee(II) Scada
to cor recom	nplete mende	the ed.	,					
to cor recom Run all A new comm	nplete mende Analyses nand a	the ed. s illows	you to	run al	l the s	scenarios in the list	with one	e click.
to cor recom Run all A new comm	nplete mende Analyses nand a re Sce	e the ed. s illows	you to	run al	l the s	scenarios in the list	with one	e click.





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

1.3 Run



Click the "Run" button to open the parameters of the current analysis window which are classified into:

- EAK scenarios(figure 1)
- Linear scenarios (figure 2)
- Non Linear scenarios (figure 3)
- Time History scenarios (figure 4)

Dynam	ic Seismio	c Action Pro	ocedur	e (E.A.K.)		×
Parameters		Mass Centers (cm)			~
Auto Procedu	re	Level	х	γ	Z	^
Procedure		0 - 0.00	0.00	0.00	0.00	
Masses		1 - 300.00	0.00	300.00	0.00	
9-1- D2 d		2 - 600.00	0.00	600.00	0.00	
Static Ψ 3 , Ψ		3 - 900.00	0.00	900.00	0.00	
T > 1		4 - 1200.00	0.00	1200.00	0.00	
Eccentricitie	s					_
Dynamic Analy	rsis					~
Initialize Dat	a ,			Exit		

L EAK scenarios*** are not described in the English version of the manual.



Parameters	Mass Cente	rs (cm)			¥
Automatic Proced	Level	х	Y	Z	^
edure	0 - 0.00	0.00	0.00	0.00	
Mass - Stiffnes	s 1 - 300.00	0.00	300.00	0.00	-
Regularity	2 - 600.00	0.00	600.00	0.00	
Regular	3 - 900.00	0.00	900.00	0.00	
✓ In Plan	4 - 1200.00	0.00	1200.00	0.00	
✓ in Elevation		0.00	.200.00	0.00	
Equivalent					-
Analysis					
Initialize data			Exit		
					Figure 2
f	Pushover Analysis	Execution	on		×
Parameters	Mass Cente	rs (cm)			
Automatic Proced	ure	rs (city			
edure	Level	X	Y	Z	
Mass - Stiffnes	s 0 - 0.00	0.00	0.00	0.00	
Challin Dumania	1 - 300.00	0.00	300.00	0.00	- N
Static-Dynamic	2 - 600.00	0.00	600.00	0.00	
Pushover	3 - 900.00	0.00	900.00	0.00	
	4 - 1200.00	0.00	1200.00	0.00	
					¥
Initializa Data			Evit		
Initialize Data			Exit		Figure 3
					- Igure 3
				Ψ2 Mul	ltipliers
Always define	the live loads'	coeffic	ient ψ2.	Mass C Heights	enters (cm) s (cm)
, Default value:	ψ2=0.30.		·	Ψ2 Mul	tipliers
				Seismic	: Multipliers ions (cm)
Multipliers	· · · · · · · · · · · · · · · · · · ·	*		Masses	s (kN)
		_			
/ei	Ψ2				
0.00	0.30				



	Linear time-	history analy	vsis (L	inear)		×	
[Parameters	Mass Centers (cr	n)			~	
[Automatic Procedure	Level	Х	Y	Z	^	
Proce	dure	0 - 0 00	0.00	0.00	0.00		
	Mass-Stiffness	4 200.00	0.00	200.00	0.00		
	Time History	1 - 300.00	0.00	300.00	0.00		
	Describe File	2 - 600.00	0.00	600.00	0.00		
	Results File	3 - 900.00	0.00	900.00	0.00		
7	h	4 - 1200.00	0.00	1200.00	0.00		
		5 - 1500.00	0.00	1500.00	0.00		
-11		6 - 1800.00	0.00	1800.00	0.00		
	Update Data	1	Ð	kit			Figure 4
First of	f all, press Initializ	e Data to	upda	te the pa	arame	ters c	f the current scenario.
Then p	Parameters	to def	ine th	e param	eters	of the	project.
The pa	rameters' dialog bo	x vary for eac	ch sele	ected sce	enario	, and	so:

- 1. Scenario
- § Scenario Static

Select Analysis Static and Type Static and press New.

Scenario		×
Renumbering Nodes Cuthill-McKee(II)	~	Advanced Multi-Threaded Solver
Disable	Name	
Static (0)	Analysis	Static \checkmark
	Type Propertie	Static ~
	Eleme	nts Nodes
	Load Ca	ases Masses
	New	Update
		Run all Analyses
		Exit

Elements, the values of the multipliers are automatically updated.



	y Values	(Concret	e)						×
Static									~
Linear Element Prope	rty Value I	Multiplier	s						
Concrete ~	E	G	Ak	Asy	Asz	ε	Ix	Iy	[√] Iz
BEAMS - B3D	1	1	1	1	1	1	1	1	1
BEAMS - TRUSS	1	1	1	1	1	1	1	1	1
BEAMS - B3Def	1	1	1	1	1	1	1	1	1
COLUMNS - B3D	1	1	1	1	1	1	1	1	1
COLUMNS - TRUSS	1	1	1	1	1	1	1	1	1
WALLS - B3D	1	1	1	1	1	1	1	1	1
WALLS - TRUSS	1	1	1	1	1	1	1	1	1
Load Cases:									
: LC1 →1 : LC2 →1									
oad Case participatio	n								×
Static Load Cases of Scenario g(m/se	ec2) 9.81	1	Availal	ble Load	Cases and	d Load Gr	oups		
1+ ^ LC 2+ LC 3 LC 4 LC 6	LG1 1 1.00 2 0.00	LG2	LG3	LG4	LG5	LG6 I	.G7 LG	i8 LG9	LG10
7 8 9									
7 8 9 10 11 12 13 14 15 16 V									>
7 8 9 10 11 12 13 14 15 16 V									>

Z,	Static	(0)	•	G.
New	•	Active Scenario		Run
		Scenarios		

The command *Run* opens the following window:



Static - Dynamic An	alysis (c:\io\MyProject2222\scaanal\Scen000\l000S.DAT)	×
Stiffness Matrix	BANDWIDTH	
Elapsed Time		
Reading Input Data		
Creating Block		
Decomposing Block		
Vector Assembly		
Writting Output		
Execute 📐	Stop	

The analysis runs by pressing the Execute button. Then press Exit.

§ Scenario Dynamic

Select Analysis **Dynamic** and Type **Response spectrum** and press New.

Scenario			×
Renumbering Nodes Cuthill-McKee(II)	~	Advanced Multi-Threaded Solver	
Disable	Name		
Dynamic (0)	Analysis	Dynamic	\sim
	Type Properties	Response spectrum s	\sim
	Elemer	Nodes	
	Load Ca	Masses	
	New	Update	
	1	Run all Analyses	
		Exit	

In *Elements,* the values of the multipliers are automatically updated.



Multipliers of Property	/ Values	(Concret	te)						×
Dynamic									~
Linear Element Proper	ty Value	Multiplier	s						
Concrete 🗸 🗸	Е	G	Ak	Asy	Asz	3	Ix	Iy	Iz
BEAMS - B3D	1	1	1	1	1	1	1	1	1
BEAMS - TRUSS	1	1	1	1	1	1	1	1	1
BEAMS - B3Def	1	1	1	1	1	1	1	1	1
COLUMNS - B3D	1	1	1	1	1	1	1	1	1
COLUMNS - TRUSS	1	1	1	1	1	1	1	1	1
WALLS - B3D	1	1	1	1	1	1	1	1	1
WALLS - TRUSS	1	1	1	1	1	1	1	1	1
Walls Filter (Lmax/Lmin) > 4	ł			C	Ж		Cancel	

In *Masses*, set the coefficients for calculating the masses from the available loads (LC1 (dead), LC2 (live)).

Masses C	alculatio	'n								×
Dynamic	: M	ass Calc	ulation fr	rom Avail	lable Loa	d Cases	and Loa	d Group	s	
LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG10
LC1	1.00									
LC2	1.00									
g(m/sec	9.81	1					ОК		Ca	ncel

When the active scenario is **Dynamic**

Z,	Dynamic (0)	Ŧ	ī,
New	' Active Scenario		Run
	Scenarios		

Since you press the command *Run*, the window for running the scenario opens and by pressing *Initialize data*, the following commands are activated:



)ynar	nic					×	
	Parameters	Mass Cente	rs (cm)			~	
	Automatic Procedure	Level	х	Y	Z	~	
Proc	edure	0 - 0.00	0.00	0.00	0.00		
	Mass - Stiffness					_	
	Analysis					-	
	Initialize data			Exit			
res	s the command P	arameter	r s and	define:	_		
yn	amic Analysis			×			
Pa T	arameter ype Acceleration	✓ Par × x	ticipatio	n		£:	
Ei	Response Spectrur genvalues Accurac 10 0.001	n y :y z	0		De Acc Dis	tine celera placer	the Spectrum Type: tion ment
ζ	(%) 5 Modal Combin	CQC	2	\sim			
	ОК		Ca	incel			
arti	cipation Defi 0 0 0	ne the Se	ismic I	Force P	articipa	ation	Factor per direction.
CQ CQ SR	C De C (10%) SS roc	fine the i mplete Qi ot of the s	metho uadrat um of	d of co ic Coml square	ombinii binatio d (SRS	ng th n CC S) me	e modal responses according to eith C and CQC (10%)(3.6 EAK), or the squa ethod.
ige 10 (%	NVAlues Accuracy 0.001) 5 Modal Combinati	Defir ratio	ne the ζ (%).	numbe	r of th	e Eig	envalues, the Accuracy and the dampi
ick	Response Spectr	um to see	e the s	pectrur	n or ch	ange	e it by changing the values of the table:



A/A	T(s	RdTx	RdTy	RdTz	^	
1	0.000	1.570	1.099	1.570		
2	0.050	1.345	1.334	1.345		
3	0.100	1.121	1.570	1.121		
4	0.150	1.121	1.570	1.121		
5	0.200	1.121	1.570	1.121		W
5	0.250	1.121	1.570	1.121		
7	0.300	1.121	1.570	1.121		
3	0.350	1.121	1.570	1.121		
Э	0.400	1.121	1.570	1.121		
10	0.450	1.036	1.451	1.036	~	
Def	fault	Write	ТХТ	OK		
Read	ТХТ Б			Cance	el	
	Damaged 9	Structures	check			
uildin	as' categor	ry I 🗸	Cor	nstruction p	period b	pefore 1985 EAK ???

The commands Write TXT και Read TXT allow the recording and the opening respectively of a .txt file which contains all the values of the response spectrum. You can define a displacement spectrum:

Type

Displacement \lor

and choose a.txt displacement file to create the Displacement Response Spectrum.

By clicking the Automatic procedure the analysis runs

Static - Dynamic Ana	alysis (C:\MELETES\DEKPOL\3\scaanal\Scen000\1000D	Х
Stiffness Matrix	BANDWIDTH = 756	
Elapsed Time	00:00:03 Processing	
Reading Input Data		
Creating Block	1/2	
Decomposing Block	1/2	
Vector Assembly	1/2	
Writting Output		
Εκτέλεση	Stop	

§ Eurocode Scenarios

Static
Dynamic
Seismic
EC-8_Greek
NTC_2008
EC8_Italia
EC8_Cyprus
EC8_Austrian
EC8_General
SBC 301
Polska-obszar LGOM

SCADA Pro contains Eurocode 8 in its general form (EC-8_General), while it also incorporates the national annexes for Greece (EC-8_Greek), Cyprus (EC-8_Cyprus), Italy (EC-8_Italia) and Austria (EC-8_Austrian).



In the option of scenarios' creation and analysis type "EC8_General", there are the following types of analysis scenarios:



The types:

- Static
- Dynamic

Are used for the analysis of new structures according to EC8.

The types:

- Elastic Static
- Elastic Dynamic
- Preliminary Static
- Preliminary Dynamic

Are used for the evaluation and the redesign of existing structures ONLY for the Greek Regulation

§ 1. EC-8_General Analysis and Static Type

Choose Analysis EC-8_General and Type Static and press the New button.

ATTENTION: The materials must be related to the selected regulation and the cross sections during the data input must have the right qualities (C for EC8 scenarios)



Scenario			×
Renumbering Nodes Cuthill-McKee(II)	~	Advanceo Multi-Thre	d eaded Solver
Disable	Name		
EC8_General Static (0)	Analysis	EC8_Ger	neral 🗸 🗸
	Type	Static	~
	Propertie	s	
	Eleme	nts	Nodes
	Load Ca	ases	Masses
	New		Update
		Run all An	nalyses
		Exit	t

§ 2. Ανάλυση EC-8_General και Τύπο Dynamic

Choose EC-8_General Analysis and Dynamic Type and press the New button.

Scenario		×
Renumbering Nodes Cuthill-McKee(II)	~	Advanced Multi-Threaded Solver
Disable	Name	
EC8_General Dynamic (0)	Analysis	EC8_General ~
	Туре	Dynamic 🗸 🗸
	Propertie	S
	Elemer	nts Nodes
	Load Ca	ases Masses
	New	Update
	I	Run all Analyses
		Exit

§ 1&2. EC-8_General Static Analysis and EC-8_General Dynamic Analysis

All of the following concern the **EC-8_General** both for **Static** and **Dynamic** type so they are described once for both.

In *Members* you have to update the corresponding coefficients according to the annex of your country for Concrete and Steel respectively:



Multipliers of Property	y Values	(Concre	te)							×
EC8_General Dynamic	:							[\sim
Linear Element Proper	rty Value	e Multiplie	rs							
Concrete 🚬 🗸	Е	G	Ak	Asy	Asz	ε	Ix	Iy	Iz	
Concrete 💦	1	1	1	1	1	1	1	1	1	
BEAMS - TRUSS	1	1	1	1	1	1	1	1	1	
BEAMS - B3Def	1	1	1	1	1	1	1	1	1	
COLUMNS - B3D	1	1	1	1	1	1	1	1	1	
COLUMNS - TRUSS	1	1	1	1	1	1	1	1	1	
WALLS - B3D	1	1	1	1	1	1	1	1	1	
WALLS - TRUSS	1	1	1	1	1	1	1	1	1	
Walls Filter (Lmax/Lmir	n) > [4			(ОК		Cancel		

In *Load Cases*, type 1.00 next to LC1 for "Dead Loads" and 1.00 next to LC2 for "Live Loads" Q and press the button Update.

Load Case parti	cipation										×
EC8_General D Load Cases of Scenario	lynamic g(m/sec2)	9.81		Availa	ble Load	Cases a	and Load	l Groups			
G(1) + Q(2) +	LC LC1 LC2	LG1 1.00 0.00	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG10
	<										>
			[OK			Cance	el			

By activating either the EC-8_General Static scenario or the EC-8_General Dynamic scenario,



the command *Run* opens the window for the scenario's run and by pressing **Initialize Data**, the following commands are activated:



Seismio	c Actions Calculation - A	nalysis - Check	s			×
	Parameters	Mass Cente	rs (cm)			~
	Automatic Procedure	Level	Х	Y	Z	^
Proce	dure	0 - 0.00	0.00	0.00	0.00	
l	Mass - Stiffness					
	Regularity					
	Regular					
	✓ In Plan ✓ In Elevation					
ſ	Equivalent					-
ĺ	Analysis					~
[Initialize data			Exit		

For the "EC8_General (static/dynamic)" scenario, the parameters' dialog box is the following:

EC8 Parameters			×
Seismic Area Seismic Areas Zone I v a 0,16 *g Importance Zone II v Yi 1	Characteristic Periods Spectrum Type Horizont Type 1 S,avg 1.2 Soil TB(S) 0.15 B TC(S) 0.5 TD(S) 2	al Vertical 0.9 0.05 0.15 1	Apply seismic actions on Levels XZ Down 0 - 0.00 V Up 0 - 0.00 V Dynamic Analysis Eigenvalı 10 Accuracy 0.001 CQC V Spectrum Participation factors PFx 0 PFy 0 PFz 0
Spectrum Response Spectrum Design ζ(%) 5 Horiz Response Spectrum U Structural Type q Concrete v qx Structural Type X Frame System	 ✓ Ductility Class ontal b0 2.5 Vertical pdate Spectrum Sd(T) >= 3.5 qy 3.5 qz Z Frame System 	DCM b0 3 0.2 a*g 3.5 m	Acc.Eccentricities Sd (T) e πx 0.05 *Lx e πz 0.05 *Lz Bays Setbacks X One Z One Z One Z One Z All the other cases
Fundamental Periods X Concrete Moment Resisting Interstorey Drift Limit Seismic Triangu	Calculation Method Frames 0.005 ar ~	EC8-1 particular de la Concrete	r. 4.3.3.2.2 (3) Moment Resisting Frames Walls KANEPE Default OK Cancel ΚΡΙΤΗΡΙΑ ΑΠΑΛΛΑΓΗΣ ΣΤΑΤΙΚΗΣ ΕΠΑΡΚΕΙΑΣ

Special parameters for a specific analysis are determined in this dialog box (level of seismicity of the area, type of soil, the importance of the structure etc.). By clicking "Seismic areas"

Seismic Areas a file that contains a list taken by the national annex, with the places and their corresponding seismicity zone, pops up.



_	Seism	ic A	rea			
		Sei	ismic	Are	as	
z	one	I	~	а	0.16	*g

Select the considered seismic zone and the coefficient "a" will be filled in automatically.

Characteristic Periods					
Spectrum Type Horizontal Vertical					
Type 1 ∨	S,avg	1.2	0.9		
Soil	TB(S)	0.15	0.05		
B ∀	TC(S)	0.5	0.15		
	TD(S)	2	1		

Define the Spectrum Type (for Greece Type 1) and the Soil Type so that all the coefficients for both horizontal and vertical spectrums are filled in

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

Response Spectrum	Design 🗸 Ductility Cla	ss DCM ~
ζ(%) 5	Horizontal b0 2.5	Vertical b0 3
Response Spectrum	Update Spectrum	Sd(T) >= 0.2 a*g

Choose the "Structural Type

5	Structural Type
	Concrete 🗸 🗸
	Concrete
5	Steel
k	Composite
Ì	Unreinforced masonr
	Confined masonry
5	Reinforced masonry
	Low seismity masonr

The "**Behavior factor q**" of the structure is a result of a computation procedure. Additionally, the "**Structure type**" follows certain criteria

q qx	3.5	qy 🗌 3.5	qz 🗌 3	3.5
Structu	iral Type			
X	Fra	me System	Z	Frame System

SCADAPro gives the engineer the opportunity to get rid of them and follow the procedure described in the next chapter: "How to calculate the behavior factor q" In the field **Structure periods:**

In previous versions, there was the Structure Type X and Z field to calculate the fundamental



period. Now it is replaced by	the section:			
Fundamental Periods	Calculation Method	EC8-1 par. 4.3.3.2.2 (3)		~
X Concrete Moment Resisting Frames	v Z	Concrete Moment Resistir	ng Frames	~
There is now an opportunity EC8-1 par. 4.3.3.2.2 (3)	to calculate the peri	od in three ways.		
EC8-1 par. 4.3.3.2.2 (3) EC8-1 par. 4.3.3.2.2 (5) Modal Analysis				
The first two are the approxi	mate methods of EC	8-1.		
 In the first one EC8-1 To choose, per direct 	par. 4.3.3.2.2 (3) it	s necessary: pe		
X Concrete Moment Resisting Frames	~ Z	Concrete Moment Resisting) Frames	\sim
Steel Moment Resistin Concrete Moment Resistin Steel frames with ecce Structures with concre All the other structure (in case that in X or/and Z d	g Frames sisting Frames entric bracings ete or masonry shear wal s lirection, the structu Bays X One	ls ire only consists c	of one frame you acti	vate the
checkbox in the field "Bays"	Z [] One)			
Afterwards, choose the comminimum length that a vertic a column	mand "Walls" Wal cal member must ha	to assign a va ve to be regarded	alue to the as a wall instead of	



ColumnElementVyVzhw12650.022660.032670.042680.052690.062700.072710.082720.092730.0102740.0102740.0ccancel	Column 1					
$\frac{1}{2} \frac{265}{266} = \frac{0.0}{0.0}$ $\frac{3}{2} \frac{267}{267} = \frac{0.0}{0.0}$ $\frac{3}{2} \frac{267}{269} = \frac{0.0}{0.0}$ $\frac{5}{2} \frac{269}{270} = \frac{0.0}{0.0}$ $\frac{6}{2} \frac{270}{271} = \frac{0.0}{0.0}$ $\frac{9}{2} \frac{273}{272} = \frac{0.0}{0.0}$ $\frac{10}{274} = \frac{0.0}$	1	Element	Vy	Vz	hw ^	
2 266 0.0 3 267 0.0 4 268 0.0 5 269 0.0 6 270 0.0 7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 i Clear All OK Cancel button, and aut the walls are checked in each direction, so as T1 is calculated according to paragrap For the second approximate method EC8-1 par. 4.3.3.2.2 (5) further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods.		265			0.0	
3 267 0.0 4 268 0.0 5 269 0.0 6 270 0.0 7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 ic	2	266			0.0	
4 268 0.0 5 269 0.0 6 270 0.0 7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 10 274 0.0 i Image: Clear All OK Cancel Det the min wall length (cm) and Click the Imin Wall Length (cm) >= be the min wall length (cm) and Click the Imin Wall Length (cm) >= button, and autithe walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction. For the second approximate method EC8-1 par. 4.3.3.2.2 (5) , there is no need to further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods. The third method includes a Modal Analysis to calculate the periods.	3	267			0.0	
5 269 0.0 6 270 0.0 7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 10 274 0.0 Add All Clear All OK Cancel button, and autistic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction. For the second approximate method further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods.	4	268			0.0	
6 270 0.0 7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 ic 0.0 0.0	5	269			0.0	
7 271 0.0 8 272 0.0 9 273 0.0 10 274 0.0 Add All Clear All OK Cancel be the min wall length (cm) and Click the min Wall Length (cm) >= button, and aut the walls are checked in each direction, so as T1 is calculated according to paragrap For the second approximate method EC8-1 par. 4.3.3.2.2 (5) the third method includes a Modal Analysis to calculate the periods.	6	270			0.0	
8 272 0.0 9 273 0.0 10 274 0.0 Add All Clear All OK Cancel be the min wall length (cm) and Click the min Wall Length (cm) >= button, and automic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction. For the second approximate method EC8-1 par. 4.3.3.2.2 (5) , there is no need to further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods. The third method includes a Modal Analysis to calculate the periods.	7	271			0.0	
9 273 0.0 0.0 10 274 0.0 0.0 Add All Clear All OK Cancel be the min wall length (cm) and Click the min Wall Length (cm) >= button, and automatic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the second approximate method further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods.	8	272			0.0	
10 274 0.0 Add All Clear All OK Cancel De the min wall length (cm) and Click the min Wall Length (cm) >= button, and aut the walls are checked in each direction, so as T1 is calculated according to paragraph For the second approximate method EC8-1 par. 4.3.3.2.2 (5) , there is no need to further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods. The third method includes a Modal Analysis to calculate the periods.	9	273			0.0	
Add All Clear All OK Cancel be the min wall length (cm) and Click the min Wall Length (cm) >= button, and aut the walls are checked in each direction, so as T1 is calculated according to paragraph For the second approximate method EC8-1 par. 4.3.3.2.2 (5), there is no need to further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods.	10	274			0.0 ~	
Add All Clear All OK Cancel De the min wall length (cm) and Click the min Wall Length (cm) >= button, and automatic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction, so as T1 is calculated according to paragraphic the walls are checked in each direction. For the second approximate method further action as long as it is selected. EC8-1 par. 4.3.3.2.2 (5) , there is no need to further action as long as it is selected. The third method includes a Modal Analysis to calculate the periods. The third method includes a Modal Analysis to calculate the periods.	<				>	
Add All Cold All	Add All	Clear	۵	OK	Capcel	
The third method includes a Modal Analysis to calculate the periods.	pe the min the walls ar	wall length (c re checked in	m) and Cli each direc	ck the min tion, so as T	Wall Length (cm) >= button, and is calculated according to para	l automa graph 4.3
	pe the min the walls a For the s further a	wall length (c re checked in second appro	m) and Cline each direct	ck the min tion, so as T: thod EC8-1	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee	l automa graph 4.3 ed to do a
e program takes into consideration the period which corresponds to the domination of the domination of the modal which has the biggest percentage of the activated mass)	pe the min the walls a For the s further a The third	wall length (c re checked in second appro action as long d method inc	m) and Cli each direc oximate me g as it is sel ludes a Mc	ck the min tion, so as T thod EC8-1 ected. odal Analysis	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee to calculate the periods.	l automa graph 4.3 ed to do a
e user can increase or decrease the number of eigenvalues in case of dynamic or sta long as the calculation of the eigenvalues with Modal Analysis and the percentage	pe the min the walls an For the s further a The third e program ch directior	wall length (c re checked in second appro action as long d method inc takes into co 1. (the moda	m) and Cli each direc oximate me g as it is sel ludes a Mc onsideration l which has	ck the min tion, so as T thod EC8-1 ected. odal Analysis n the period the biggest	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee to calculate the periods. which corresponds to the dom percentage of the activated ma	l automa graph 4.3 ed to do a ninant mo
hosen.	pe the min the walls an For the s further a The third ch direction e user can in long as the chosen.	wall length (c re checked in second appro action as long d method inc takes into co 1. (the moda ncrease or de calculation o	m) and Cli each direct oximate me g as it is sel cludes a Mc onsideration l which has ecrease the of the eigen	ck the min tion, so as T thod EC8-1 ected. odal Analysis n the period the biggest number of e values with I	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee to calculate the periods. which corresponds to the dom percentage of the activated ma genvalues in case of dynamic of Modal Analysis and the percents	l automa graph 4.: ed to do a ninant mo iss) r static ar age of ac
hosen.	pe the min the walls an For the s further a The third e program ch direction e user can in long as the chosen.	wall length (c re checked in second appro action as long d method inc takes into cc 1. (the moda ncrease or de calculation o	m) and Cli each direct oximate me g as it is sel cludes a Mc onsideration l which has ecrease the of the eigen	ck the min tion, so as T: thod EC8-1 ected. odal Analysis n the period the biggest number of e values with I	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee to calculate the periods. which corresponds to the dom percentage of the activated ma genvalues in case of dynamic on Aodal Analysis and the percenta	l automa graph 4.3 ed to do a ninant mo iss) r static an age of acc
Dynamic Analysis	pe the min the walls and For the s further a The third e program ch direction e user can in long as the chosen.	wall length (c re checked in second appro action as long d method inc takes into co 1. (the moda ncrease or de calculation o	m) and Clin each direct each direct each direct eximate me g as it is sel cludes a Mo onsideration l which has ecrease the of the eigen	ck the min tion, so as T: thod EC8-1 ected. odal Analysis n the period the biggest number of e values with I	Wall Length (cm) >= button, and is calculated according to para par. 4.3.3.2.2 (5), there is no nee to calculate the periods. which corresponds to the dom percentage of the activated ma genvalues in case of dynamic of Aodal Analysis and the percent	l automa graph 4.3 ed to do a ninant mo ss) r static an age of acc

Also, there is also the opportunity to choose the method of combining the modal responses according to Complete Quadratic Combination CQC and CQC (10%)(3.6 EAK), or the square root of the sum of squared (SRSS) method.

Moreover, the results of the modal analysis for the static scenarios are included in the results of seismic action.



To modify the coefficients of the eccentricities, select the respective checkbox and type the new value on the right.

centri	icities –	
	0.05	*Lx
	0.05	*Lz
		0.05

In the same way, the engineer can modify the X, Y, and Z spectrums by typing his values in the respective fields,

Sd (T) 1 Sd (TX) 1 Sd (TY) 1 Sd (TZ) 1	
as well as the spectrum participation factors.	
Spectrum Participation factors PFx O PFy O PFz O In the Indents field, select for each direction the case that is appropriate for the particular study and is defined by the Eurocode. Setbacks x All the other cases z All the other cases	DIRECTION X
Seismic Triangular The engineer can al Orthogonal Distribution of the seismic force between two options. Triangular	lso choose the Type of



Method of calculating the behavior factor q

According to the Eurocode, the "Behavior factor q" of the structure is a result of a computation procedure. Additionally, the "Structure type" follows certain criteria.

SCADA Pro calculates automatically the q factor and the type of the structure. To apply the automatic process, you must follow the procedure described bellow:

After having completed all the previously mentioned values, leave the following boxes blank



without any changes.

Choose "Ok" and use the "Automatic procedure" to run an initial analysis.

Parameters	Mass Centers	Mass Centers (cm)				
Automatic Procedure	Level	х	Y	Z	~	
Procedure	0 - 0.00	0.00	0.00	0.00		
Mass - Stiffness	1 - 400.00	1814.27	400.00	907.40		
/ Regularity	2 - 700.00	1863.90	700.00	906.02		
Regular	3 - 1000.00	1845.56	1000.00	906.88		
In Plan	4 - 1300.00	1266.71	1300.00	867.40		
	5 - 1600.00	1240.93	1600.00	885.20		
Equivalent						
Analysis					-	
Initialize data		E	dt			

of the "Results"

Checks

Click "Exit" to close the dialog box and choose the "Checks" command menu at the ribbon, to open the "Seismic analysis control coefficients" dialog box.



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

Walls Shear Force Par 5 1 2. Reference Level. 0 0.000/m)							
n/n Level	Wa:	Ils Shear, (Kn)	/Total Shear = (Kn)	* nvx 	Walls She (Kn)	ear/Total She (Kn)	ear = nvz
1 ** 2 3 4 5	** 25- 18- 25- 5- 5-	621.141 500.708 707.106 148.031 302.049	2025.586 1940.827 2576.280 820.477 1256.858	0.31 NO 0.26 NO 0.27 NO 0.18 NO 0.24 NO	40-1169.33 44- 575.58 44- 443.95 51- 187.46 51- 184.07	32 1737.703 33 1170.361 58 1085.765 50 558.370 78 581.493	0.67 OI 0.49 N 0.41 N 0.34 N 0.32 N
Buildi Buildi	ing sys	stem defin	nition X:	Frame Syst	em	(coupled or	uncoupled)

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



q 2.76 qy 1.38 qz 2.76 The proposed values may be kept or altered (the latter one is an option that could be utilized from the beginning of the procedure, however, in this occasion the software would not propose any values, at all). Image: Click gray from the spectrum from the spectrum by the new values of the q factor and click response Spectrum for see it. Click "Ok" and conduct the analysis one more time, considering the new q values.
 In case of Steel, Composite and Masonry Structures: Structural Type Just select the relative Structural Type: - Steel structures: the procedure is the same. The only difference is located at the definition of the "Structural Type", which is identified according to the type of the structure, thus, the help TXT file is not necessary. The user can select the type from the beginning and continues like before with the definition of the "q" value. - Masonry structures: any "Structural Type" selection. Just select the type of masonry in "Structural Type" and the "q" value is calculated automatically by the program.

BEAMS - TRUSS

BEAMS - B3Def

COLUMNS - B3D

WALLS - B3D

WALLS - TRUSS

Walls Filter (Lmax/Lmin) > 4

COLUMNS - TRUSS

OK Cancel



A prerequisite for per the existance of the calculation	forming all analysis scenarios with Inelasti f reinforcement of the respective ultimate moments of resist	ic Type is: tance.
▲ For Greece, choo consideration the choose the EC-8 G	se EC-8 Greek / Nonlinear and for Cypr Annexes of the Eurocodes, respectively. Fo eneral and type the parameters of the cor	us, Italy and Austria take into or all other European countries, rresponding annexes.
Scenario Renumbering Nodes Cuthill-McKee(II) Disable EC8_General Nonlinear (0)	Advanced Multi-Threaded Solver Name Analysis EC8_General Type Nonlinear Properties Elements Nodes Load Cases New Update	
In <i>Members,</i> you have country for Concrete a Multipliers of Property Values (Con- EC8. General Nonlinear	Run all Analyses Exit e to update the corresponding coefficients and Steel respectively	according to the annex of your



In *Load Cases*, type 1.00 next to LC1 for "Dead Loads" and 1.00 next to LC2 for "Live Loads" Q and press the button Update.

Load Case parti	cipation										×
EC8_General N Load Cases of Scenario	onlinear g(m/sec2)	9.81		Availa	ble Load	l Cases a	and Load	Groups			
G(1) + Q(2) +	LC LC1 LC2	LG1 1.00 0.00	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG10
			[ОК			Cance	el			

By activating the EC-8_General Nonlinear scenario,

New	EC8_General Nonlinear (0) · Active Scenario	Ŧ	Г Run
	Scenarios		

:

the command *Run* opens the window for the scenario's run and by pressing **Initialize Data**, the following commands are activated:

Parameters	Mass Cente	rs (cm) 🗟		
Automatic Procedure	Level	Х	Y	Z
e	0 - 0.00	0.00	0.00	0.00
Mass - Stiffness		0.00	0.00	
Static-Dynamic				
Pushover				

The procedure involves three steps, carried out sequentially, either automatically ("Automatic Procedure") or selectively (click the procedure's buttons) and contain:



- Mass and stiffness calculation.
- Static analysis for the internal forces calculation due to permanent and live loads is required for starting a successive pushover analysis.
- Dynamic analysis considering the elastic spectrum of EC8 for the calculation of the eigenperiods and the target displacement.
- Pushover analysis.

Always define the live loads' coefficient ψ 2. Default value: ψ 2=0.30.

Parameters

Initialize Data to update the parameters of the current scenario.

Then	press

Press

to define the parameters of this specific project.

	EC8 -	Pushover	Parameters ×
Sesmic Area Sesmic Areas Zone I v a 0.16	Characteristic Periods Responce Type 1 V S,avg	Horizontal Vertical	Lavels XZ Down 0 - 0.00 Image: Plastic Hinge check under the Reference Level
Importance Zone II V ^{Yi} 1	B TC(S)	0.15 0.05 0.5 0.15 2 1	Dynamic Analysis Eigenvalues 10 Accuracy 0.001 Spectrum participation factors
Spectrum Elastic ζ 5 Horiz Response Spectrum U	Ductility Class zontal b0 2.5 Jpdate Spectrum S	DCM vertical b0 3 id(T) >= 0.2 ag	PFx 0 PFy 0 PFz 0 Eccentricities Sd (T) Sd (TX) 1 e πχ 0.05 *Lx Sd (TY) 1
Seismic Combinations \checkmark Fx +k Fz \checkmark Tria \vdash Fx -k Fz \checkmark Orti \checkmark -Fx +k Fz \checkmark Orti \neg -Fx -k Fz \land Acco \checkmark Fz +k Fx \land Acco \vdash Fz -k Fx \land Bas \checkmark -Fz +k Fx \blacksquare Bas	ngular Disribution hogonal Distribution idental Eccentricities Ex idental Eccentricities Ez e Shear from Responce Spec	trum	e mz 0.05 *Lz Sd (TZ) 1 Check Node 0 ✔ Active Infill Masonry Number of Steps 200 Lamda Range (%) 0 Maximum 3 % of Total Height ✔ Constant Value of the Shear Length LS
Default OK	erse Load Factor (k)	0.3 SPECTRA	Active Stiffness After the plastic hinge V Data Credibility Satisfactory V

In the dialog box of the Pushover Analysis' Parameters, the definition of the parameters outside the borders are defined as in linear analysis, but in this case, the Response Spectrum must be Elastic.

In "Levels XZ" section:

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

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

It is recommended to define the last complete level (no stairwell termination)



as the highest level. 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 lower levels than the reference level are taken into consideration as potential locations of plastic hinges.

In "Seismic Combinations" section:

 Seismic Combinations 		
✓ 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-kFx	Base Shear from Responce Spectrum	
-Fz-kFx	Transverse Load Factor (k)	0.3

Check the load combinations that will be taken into account in the pushover analysis. Each combination implies the application of one seismic force in one direction (X or Z direction) and one seismic force in the corresponding transverse direction multiplied by a factor, defined in

Transverse Load Factor (k) 0.3

. The coefficient k is given 0.3 by default.

Check the seismic forces' distribution along height (Triangular or/and Orthogonal).

Accidental Eccentricities Ex

Check Accidental Eccentricities Ez if you want to consider the accidental Eccentricities due to which moments are developed in x or/and z direction.

The check/means that the base shear is calculated from the dynamic analysis.

Activate all seismic combinations with the accidental eccentricities considered and as a result, 64 load combinations are produced. This means that 64 pushover analyses are conducted, thus, the computational time is increased significantly.

In the last section:

Check Node	0	✓ Active Infill Maso	nry	
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 V		~	
Data Credibility	Satisfactory V		~	

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 which is completed when the structure collapses; when plastic hinges are being developed, a collapse mechanism evolves. 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. At the element with the minimum λ , a plastic hinge will be created.

- **Λ** 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.
- A Setting a different value (i.e. $\lambda min=10\%$): This means that the elements with values of λ between λmin and $10\% + \lambda min$ will be considered in this step with plastic hinges in the corresponding edge (element node).

EXAMPLE:

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 at the same step.

The options about the definition of the "Constant value of the Shear Length Ls" are the following:

- A 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, the internal forces are calculated due to dead and live

New > Elements

loads, whereas the elements' stiffness is multiplied by the factors set in 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 hinges 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. The initial value of the effective stiffness is considered until that step of the analysis.

"Data reliability", is the Knowledge Level. Select between Limited, Normal and Full. The parameters that affect the definition of the knowledge levels are geometry, details, material. The knowledge level influences the partial safety factors.

"Active Infill Masonry": When the checkbox is active Active Infill Masonry and model contains



Masonry Infills added with the corresponding tool **Infill** or **Infill** (look Chapter 2) allow the program to take them into account during the analysis. Otherwise, even if they are modeled, the program will exclude their influence.

Partial Safety Factors

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

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

		PSF
KNOWLEDGE LEVEL	Material	Overstrength
KL1	1.20 (γm)	1.20 (γRd , γov)
KL2	γm as in EN1998-	γRd, γov as in EN1998-
	1	1
KL3	0.80 (γm)	0.80 (γRd , γον)

In the dialog box "**Spectra**" the response spectrum is defined, 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 considered limit states 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 according to 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,

hus the performance objectives proposed by the Eurocodes are depicted in the dialog box.			
Spe	ectra	×	
Design Existence (years 50 V	Exponent k 3	}	
Damage Limitation (DL) Check Soil Acceleration a	ag=AgR.yI.(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) Check Soil Acceleration a	ag=AgR.yI.(TR/TLR)1/k	0.2773	
TR Calculation	TLR Calculation		
Return Period TR (years) 2475	Exceedance Probability PLR%	10	
Exceedance Probability PR% 2	Return Period TLR (years)	475	
Near Collapse (NC) Check Soil Acceleration a	ag=AgR.yI.(TR/TLR)1/k	0.2773	
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% E	C8 20%	
ОК	Cancel		

Use National Annex 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.

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


	Pusho	ver Analysis E	ecutio	n		×
[Parameters	Mass Centers	(cm)			¥
[Automatic Procedure	Level	х	Y	Z	^
Proce	dure	0 - 0.00	0.00	0.00	0.00	
V	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	
						¥
[Initialize Data		E	xit		

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!

Since the Pushover analysis is applied, the structure is pushed with a gradually increased lateral static load (triangular or rectangular distribution along height) till the collapse of the structure. So, plastic hinges are formed gradually in the ends of the structural elements' length (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 a plastic region in the joints of the structure. The plastic deformations indicate that the strength resistance of the structural elements is overtaken thus the structure collapses.

§ 4. Analysis SBC301

Regulation of Saudi Arabia provides four methods of analysis for calculating seismic loads:

- Index Force Analysis Procedure (Section 10.7)
- Simplified Analysis Procedure (Section 10.8)
- Equivalent Lateral Force Procedure (Section 10.9)
- Modal Analysis Procedure (Section 10.10)



The process of creating these four scenarios is the same as the previously described one for the EC.

Press "New" to create the scenarios:

Scenario		Ş	×	
Renumbering Nodes Cuthill-McKee(II)	~	Advanced Multi-Threa	ded Solver	
Disable SBC301 Index Force (0) SBC301 Simpified Analysis (1) SBC301 Static (2) SBC301 Dynamic (3)	Name Analysis Type Properties Elemer Load Ca New	SBC 301 Dynamic s nts ases Run all Anal	Nodes Masses Update yses	
		EXIC		

Select the SBC 301 analysis and then one of the four methods of analysis.

SBC Parameters **x**

The dialog box to import and edit the parameters is the same for all four methods:

Site Class Seismic Region A Ss 0.5 S1 0.1 Fa 0.8 Fv 0.8 ?	Levels XZ Down 0 - 0.00 Up 6 - 1800.00
Building category II Kind of structure Concrete	Dynamic Analysis Eigenvalues 10 Tollerance 0.001
Structural system	Partcipation Factors
Bearing Wall Systems	PFx 🔲 0 PFy 💭 0 PFz 💭 0
Special reinforced concrete shear walls	Eccentricities Sd (T)
	Sd (TX)
Spectrum Type of structure	е тіх 0.03 *Lx sd (ту) 1
T0 0.04 Ts 0.2 T1 2 Response Spectrum Spectrum update ?	e πz 0.05 *Lz Sd (TZ) 1
Fundamental period Calculation method Unfavourable ?	
X Frame systems of concrete	Irregularities
Z Frame systems of concrete	Shear Walls OK Cancel



Site Class
The first parameter concerns the type of soil (Site Class) according to par. 9.4.2, which imposes the Fa and Fv factors (Table 9.4.3a & 9.4.3b) as well.
The next group of parameters regards the selection of the mapped maximum considered earthquake spectral response S1 and Ss according to par. 9.4.1.
Seismic Region Ss 0.5 S1 0.1 Fa 0.80 Fv 0.80 ?
The choice of these two parameters in combination with the type of soil identifies automatically the (user editable) Fa and Fv factor values.
The next group of parameters
Building category I Kind of structure Concrete
conserns the selection of Building Category according to Table 1.6-1 and the Kind of structure selection. Press ? to read about all the Kinds of structures according to the corresponding table of SBC 301.
The next section
Structural system Bearing Wall Systems
 This choice determines: the value of the Response modification coefficient R used in various calculations, the value of the Deflection amplification factor Cd used according to Sections 10.9.7.1 and 10.9.7.2 and the value of the overstrength factor Ωo. Finally, according to the seismic category of the structure, restrictions to the selection of the
structural system as well as to the maximum height of the building are specified.



Th 9.4	e nex 4.5 of	t sectio SBC 30	n conse 1	erns the	paramete	ers of the Design Response Spectrum according to par.
-5	Spectru	ım				
	то	0.07	Te 0	37	т. 1	
		0.07		57	11 1	
	Respo	onse Spec	trum	Spectr	um update	
Th va	e app lues S	olication S1 and S	calcula s. Of co	ntes aut urse, it i	omatically is possible	y the values TO and Ts based on the previously defined e to modify them manually.
lf	any d	changes	are m	ade on	these val	lues, either automatically or manually, to update the
sp	ectru	m, pres	s the bu	tton	Spectrum	update
		Res	nonse Sr	ectrum		
By	pres	sing 🛄	ponse op		the respo	nse spectrum in each direction appears.
R	espons	e Spectru	m			
Г						
	A/A	T(s	RdTx	RdTy	RdTz	
	1	0.000	0.243	0.122	0.243	
	2	0.050	0.357	0.122	0.357	
	3	0.100	0.471	0.122	0.471	
	4	0.150	0.585	0.122	0.585	
	5	0.200	0.608	0.122	0.608	
	6	0.250	0.608	0.122	0.608	
	7	0.300	0.608	0.122	0.608	
	8	0.350	0.608	0.122	0.608	
	9	0.400	0.608	0.122	0.608	
	10	0.450	0.608	0.122	0.608	
	De	fault			ОК	
	TX	T File			Cancel	
	CT.	Damaged	Structures	che		
	Buildir	nac' cetean	ev I 🔹		nstruction peri	riod before 1985 EAK 222
	Dallal	igs catego				
		Seismic ma	agnification	coefficien		Spectrum Calculation
L						
Ne	ext, tl	ne auto	matic c	or manu	al definit	tion of the type of structure according to par. 10.3 is
se	, lecter	۲.				
T	ype of	structure	-			
C						
A	utoma	tic	•			
	ſ	2				
	l	:				
Th	ere a	re three	choice	s:		
	•	Automa	atic			

- Flexible
 - Rigid



The first choice identifies the type of structure (Flexible or Rigid) automatically according to par.10.3.1.3 and figures 10.3-1. The user selects the type through the next two choices.

The following parameters concern the calculation of the Fundamental Period.

FL	undamental period		_
(Calculation method	Unfavourable 👻	?
x	Frame systems of steel		•
Z	Frame systems of steel		•

The par. 10.9.3 of SBC 301 provides three methods for the calculation of this size. For each method to be applied, certain conditions must be met.

The program options are:

- Unfavourable
- Average
- Method 1
- Method 2
- Method 3

Through the first option, the most unfavorable period of those calculated from the three methods, will be taken into account.

Through the second option, the value is the average of the three values of the three methods. The next three choices concern the value of the specific method.

In each case, two values are calculated, one for each direction of the earthquake.

The next option is the type of building based on Table 10.9.3.2 and concerns the first method calculation (Eq. 10.9.3.2-1).

The next section concerns the determination of the lower and upper level to be considered for the seismic load application.

	Down 0 - 0.00	▼ Up	3 - 1100.00 🔻			
٦ f	The following pa fields:	rameters ar	e related to the dy	namic analysis. N	lore specifically in	the following

Dynamic Anal	ysis			
Eigenvalues	10	Tollerance	0.001	

The user determines the number of Eigenvalues to be calculated through the performance of the modal analysis and the respective tolerance.



Then you can define the participation Factors for the seismic forces in each direction.

Partcipation Factors



Activate the corresponding checkbox and type the factor for the seismic load. The initial spectrum (for the considered direction) will be multiplied by this factor. The default value is 1, while a value of PFx=2 will double the values of the spectrum along the X direction.

Accordingly, to modify the coefficients for calculating the eccentricities

е тіх	0.05	i *Lx
е тіz	0.05	i *Lz

Activate the corresponding checkbox and type in the value. Finally under the participation rates of design spectra in each direction

Sd (T) Sd (TX)	1
Sd (TY)	1
Sd (TZ)	1

Activate the coefficient for the respective spactra and type in the value. The design spectra acceleration (for the considered direction) will be multiplied by this factor. For example, if you activate the Sd(TX) and enter a value of 2, the design acceleration along the X direction, will be multiplied by 2.

The next section concerns the determination of Plan and Vertical Irregularities of the structure. This definition, among others, determines the choice of the analysis method based on the table 10.6.1.

The SBC 301 with the Table 10.3.2.1 provides 6 Plan Irregularities.

In the following dialog box:



Vertical Irregularities	
$\begin{array}{c} \mathbf{L}_{1} \\ \hline \\ \hline \\ \\ \hline \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ $	ОК
	Cancel
In-Plane Discontinuity in Vertical Lateral Force-Resisting Elements	
Plan Irregularities	
$ \begin{array}{c} \overbrace{\begin{array}{c} \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\$	Out-of-plane offset
Torsional Irregularity Re-entrant Corners Out-of-Plane Offsets	
$ \begin{array}{ c c } \hline & & & & \\ \hline & & & & \\ \hline & & & & \\ \hline & & & &$	Nonparallel system
Extreme Torsional Irregularity Diaphragm Discontinuity Nonparallel Systems	

The user determines whether Plan Irregularities meet the relevant criteria by checking the corresponding option.

Regarding Vertical Irregularities, six criteria are met as well.

SCADA Pro checks 5 of them and the user activates the 6^{th} by pressing the respective checkbox if the criteria are fulfilled.

Option/concerns the calculation of sizes which are necessary for determining the eigenperiod with the third method (Eq. 10.9.3.2-3).

Press to open the dialog box.

min Column	s Length (cm) >=	200		
Column	Element	Vy	Vz	hw
1	634			0.0
2	635			0.0
3	636			0.0
4	637			0.0
5	638			0.0
6	639			0.0
7	640			0.0
8	630			0.0
9	631			0.0
10	632			0.0
•				Þ



Type the min Column Length (cm) and press the button "min Column Length" for automatically defining the walls in each direction. The program automatically calculates the required for calculating the eigenperiod values.

NOTE:

1 The definition of the walls is a required step for calculating the eigenperiod using the third method.

After setting the parameters, in the run analysis dialog box

Sei	ismi	Actions Calculation - Ana	lysis - Checks				x
		Parameters	Mass Centers	(cm)			•
		Automatic Procedure	Level	Х	Y	Z	*
F	Proce	dure	0 - 0.00	0.00	0.00	0.00	
2	1	Mass - Stiffness	1 - 425.00	982.61	425.00	501.92	
	1	Regularity	2 - 780.00	1348.52	780.00	478.83	=
		Regular	3 - 1100.00	1378.92	1100.00	660.22	
		In Elevation					
		All allowed					
	1	Analysis					-
Initialize data Exit							

press Automatic Procedure and the program automatically performs the analysis process while

making all necessary checks.

Particular attention should be paid to regularity checks

Regular In Plan In Elevation	
All allowed	

where before performing the final analysis the program gives the result of regularity in plan and height, based on the selection and seismic category in the initial parameters.

Then, based on the results, the allowed analysis or analyses are suggested (Table 10.6.1).

- On the regularity results, the user is free to select or deselect if he wants, one or two categories of regularity.
- The program changes respectively the prompt type of permitted analysis.

In the previous case, through the automatic control, the building met the criteria for regularity in plan and in elevation, which allows the use of any of the four methods.



Regular In Plan In Elevation	
All allowed	

However, if both checkboxes are deselected, which means that the building is considered irregular in plan and elevation, then the program proposes the dynamic method for analysis.

Regular In Plan In Elevation	
Dynamic	

"**Checks**" command displays the results of the checks through the automatic procedure. The first section of the checks concerns the Irregularity criteria:



CHECKS REPORT ACCORDING TO THE MAIN DIRECTIONS OF THE BUILDING SIMPLIFIED STATIC ANALYSIS (SBC 301) VERTICAL STRUCTURAL IRREGULARITIES (TABLE 10.3.2.2) Check for Stiffness Irregularity per Building Story (Table 10.3.2.2 (1a & 1b)) n/n | Height (m) | Stiffness X (*10^3 KNM) | Stiffness Z (*10^3 KNM) | Story| Total|Story| Ki | Ki+1 |Ratio| Ki | Ki+1 |Ratio|Result ___ -+---+---+----+----1 | 4.25| 4.25| 9661.63| 5077.48| 1.90| 9914.30| 4708.94| 2.11|No 7.80| 3.55| 5077.48| 4894.28| 1.04| 4708.94| 4010.75| 1.17|No 2 1 3 | 11.00 | 3.20 | 4894.28 | | 4010.75 | | _____ n/n | Height (m) | Stiffness X (*10^3 KNM) | Stiffness Z (*10^3 KNM) | Story Total Story Ki | KAvg |Ratio Ki | KAvg |Ratio Result 1 | 4.25| 4.25| 9661.63| 4985.88| 1.94| 9914.30| 4359.85| 2.27|No 7.80| 3.55| 5077.48| 4894.28| 1.04| 4708.94| 4010.75| 1.17|No 2 1 3 | 11.00 | 3.20 | 4894.28 | | 4010.75 | | _____ Soft Story (i) : Ki/Ki+1<0.7 or Ki/Avg((Ki+1)+(Ki+2)+(Ki+3))<0.80 Extreme Soft Story (i) : Ki/Ki+1<0.6 or Ki/Avg((Ki+1)+(Ki+2)+(Ki+3))<0.70 Check for Weight (Mass) Irregularity per Building Story (Table 10.3.2.2 (2)) _____ n/n | Height (m) | Mass (kN/g) | Mass (kN/g) | Story| Total|Story| Mi | Mi-1 |Ratio| Mi | Mi+1 |Ratio|Result 1 | 4.25| 4.25| 295.68| 0.00| 0.00| 295.68| 331.70| 0.89|No 2 | 7.80| 3.55| 331.70| 295.68| 1.12| 331.70| 65.07| 5.10|Yes 3 | 11.00| 3.20| 65.07| 331.70| 0.20| 65.07| 0.00| 0.00|No Irregular Story (i) : Mi/Mi+1>1.5 or Mi/Mi-1>1.5 Check for Vertical Geometric Irregularity per Build.Story (Table 10.3.2.2 (3)) n/n | Height (m) | Plan Dimensions X (m) | Plan Dimensions Z (m) | Story| Total|Story| Li | Li-1 |Ratio| Li | Li-1 |Ratio|Result ----+----+-----+------+----+----

 1
 4.25
 4.25
 19.60
 19.40
 1.00
 13.70
 13.60
 1.00
 No

 2
 7.80
 3.55
 13.20
 19.60
 1.00
 9.30
 13.70
 1.00
 No

 3
 11.00
 3.20
 9.80
 13.20
 1.00
 9.00
 9.30
 1.00
 No

 _____ _____ Irregular Story (i) : Li/Li+1>1.3 or Li/Li-1>1.3 Presented data and test results of each Vertical Irregularity criterion. At the end of this section, all the above results are presented in summary as well:

(1a)	Stiffness Irregularity (Soft Story)	з.	NO
(1b)	Stiffness Irregularity (Extreme Soft Story)	:	No
(2)	Weight (Mass) Irregularity	:	No
(3)	Vertical Geometric Irregularity	:	No
(4)	In-Plane Discontinuity in Vertical Lateral Force-Resisting Elements	:	No
(5)	Discontinuity in Lateral Strength (Weak Story)	:	No
Verti	ical Structural Irregularity for the total building	:	No

Next, the results of the Plan Irregularities according to the user's choices on the parameters of the analysis, are reported:



PLAN STRUCTURAL IRREGULARITIES (TABLE 10.3.2.1)	
(1a) Torsional Irregularity	: No
(1b) Extreme Torsional Irregularity	: No
(2) Re-entrant Corners	: No
(3) Diaphragm Discontinuity	: No
(4) Out-of-Plane Offsets	: No
(5) Nonparallel Systems	: No
Plan Structural Irregularity for the total building	: No

The next check indicates the percentage of the seismic forces undertaken by the walls of the structure to determine whether the static system of the structure belongs to one of the following two categories:

- Dual Systems with Special Moment Frames Capable of Resisting at Least 25% of Prescribed Seismic Forces.
- Dual Systems with Intermediate Moment Frames Capable of Resisting at Least 25% of Prescribed Seismic Forces.

So if the rate of seismic forces in the walls is at least 0.25, then the user can select the corresponding static system in one of the above two categories.

Concrete	Walls Shear Force	e (TABLE 10.2)	Reference	ce Level: 0	0.000(m)
n/n	Walls Shear/Total	L Shear = nvx	Walls Shear/1	[otal Shear =	nvz
Story	(Kn)	(Kn)	(Kn)	(Kn)	
1 ***	7- 155.362 459	0.459 0.34	42- 236.143	624.820	0.38
2	4- 155.835 270	0.241 0.58	40- 398.700	493.496	0.81
3	5- 32.667 157	7.022 0.21	35- 492.585	553.121	0.89
Recommne All othe	nded Basic Seismic r Types	: Force-Resisitng	System :		

The next check is related to the control of the P-Delta effects according to par.10.9.7.2 The verification is done in each direction.

The next check concerns the Drift limits according to par. 10.12.1



Drift Limits - Par. 10.12.1 - TABLE 10.12 Direction X _____ n/n | Height (m) |Story Drift| Allowable | Story | Total|Story | (Δ) (mm) | (Δa) (mm) | Result _____+ 1 | 4.25| 4.25| 2.46 | 106.25 | 0.0231 2 | 7.80| 3.55| 1.83 | 88.75 | 0.0206 3 | 11.00| 3.20| 1.45 | 80.00 | 0.0181 _____ Drift Limits - Par. 10.12.1 - TABLE 10.12 Direction Z n/n | Height (m) |Story Drift| Allowable | Story | Total|Story | (Δ) (mm) | (Δa) (mm) | Result _____+

 1
 |
 4.25|
 4.25|
 1.59|
 106.25|
 0.0149

 2
 |
 7.80|
 3.55|
 2.37|
 88.75|
 0.0267

 3
 |
 11.00|
 3.20|
 4.18|
 80.00|
 0.0523

The next check concerns the Building Seperation according to par. 10.12.2

The last two checks concern the classification of diaphragm's flexibility of each level of the building.

Diaphragms can be designated as Rigid or Flexible. The classification of the structure in total is determined from the majority of the levels' classification. In the scenarios' parameters, there is an option for choosing to characterize the building either manually or automatically so the program checks the following criteria according to par. 10.3.1.2 & 10.3.1.3

```
Diaphragm Flexibility
Rigid Diaphragm Condition (par. 10.3.1.2)
             _____
n/n | Height (m) | Plan Dimensions | Ratio |Result
Story | Total|Story|
                        (m)
                                    |Lmax/Lmin|
1 | 4.25| 4.25| 19.60 | 13.70 | 1.43 |Yes
   2 | 7.80| 3.55| 13.20 | 9.30 | 1.42 |Yes
  3 | 11.00| 3.20| 9.80 | 9.00 | 1.09 |Yes
_____
Rigid Diaphragm : Diaphragms of concrete slabs or concrete filled
metal deck with Lmax/Lmin <3 and with no plan irregularities
Diaphragm Flexibility
Calculated Flexible Diaphragm Condition (par. 10.3.1.3)
      _____
n/n | Height (m) | Maximum Diaphragm | Average Drift |Result
Story | Total|Story| Deflection (mm) | (mm)

      1
      |
      4.25|
      4.25|
      0.04|
      0.06|
      0.04|
      0.05|No

      2
      |
      7.80|
      3.55|
      0.11|
      0.14|
      0.07|
      0.08|No

      3
      |
      11.00|
      3.20|
      0.18|
      0.21|
      0.07|
      0.07|Yes

  3 | 11.00| 3.20|
                                                 0.07|Yes
         _____
Flexible Diaphragm : Maximum Diaphragm Deflection > 2*(Average Drift)
```



Finally, by choosing "Seismic Force", all the data, related to the seismic analysis and loads distribution, is reported SCENARIO : 5 - Modal Analysis Procedure (Section 10.10) DATA FILE LOAD CASES Load Case 1 (Dead-G) Load Case 2 (Live-O) MASSES CALCULAT. FROM : G+\2*Q RESULTS FILE - INTERNAL FORCES Load Case 1 (Dead- ΣG) Load Case 2 (Live- ΣQ) Load C. 3 (Horizontal Seismic Force x) Load C. 4 (Horizontal Seismic Force z) Load C. 5 (Eccentricity of seism. force x from maxez) Load C. 6 (Eccentricity of seism. force x from minez) Load C. 7 (Eccentricity of seism. force z from maxex) Load C. 8 (Eccentricity of seism. force z from minex) L. Case 9 (Vertical Seismic Force y) GENERAL DATA - CALCULATION PARAMETERS _____ Occupancy Category (TABLE 1.6-1) : I : S1= 0.200 Ss= 0.800 Ground Motion Parameters (%g) Site Class (par. 9.4.2) : B Site Coefficients (TABLE 9.4.3a & 9.4.3b) : Fa= 1.000 Fv= 1.000 Spectral Accelerations (par. 9.4.3) (%g) : Sms= 0.800 Sm1= 0.200 Design Spectral Accelerations (par. 9.4.4) (%g) : Sds= 0.533 Sd1= 0.133 Occupancy Importance Factor (TABLE 9.5) : 1.000 Seismic Design Category (TABLE 9.6a £9.6b) : A (A - A) Reliability factor (par. 10.3.3) : ρ= 1.000 : R= 4.000 Qo= 2.500 Cd= 5.000 Structural System (TABLE 10.2) Bearing Wall Systems Special reinforced concrete shear walls Fundamental Periods of Vibration _____ Direction Ix : T (sec)= 0.49228 Direction IIz: T (sec)= 0.49228 Direction y : Tv (sec)= 0.00000 n/n Level Plan Dimensions Coef. #2 Acc. Eccenticities _____
 0
 0.000
 19.400
 13.600
 0.300
 0.970
 0.680

 1
 4.250
 19.600
 13.700
 0.300
 0.980
 0.685

 2
 7.800
 13.200
 9.300
 0.300
 0.660
 0.465

 3
 11.000
 9.800
 9.000
 0.300
 0.490
 0.450
 ---------eτix = 0.05 *LIx , eτiz = 0.05 *LIIz



:

§ 5. EC-8_Greek Analysis and Time History Linear Type

The application of Time History analysis in SCADA Pro starts by defining the respective Analysis Scenario:

2	Scenario				x		
Renumbering							
Nodes Cuthill-McKee(II)	~						
Disable	Name						
EC8_General Time History Linear	Analysis	EC8_0	General		~		
EC8_General Static (2)	Туре	Time	History Line	ear	~		
EC-o_Greek Time History Linear	Properties	;					
	Eleme	ents	Noc	des			
	Load Ca	ases	Mas	ses			
	New		Up	date			
		E	Exit				
							-
E	C8_General T	ime His	tory Linea	r (0 🔻			6
Select the Active Scenario	Active	Scenario	D		and	press	Run
Linear time-h	istory analy	/sis (L	inear)		×		
Parameters	Mass Centers (c	m)			~		
Automatic Procedure	level						
		X	Y	7			
Procedure	0 - 0.00	X 0.00	Y 0.00	Z 0.00	Î		
Procedure Mass-Stiffness	0 - 0.00 1 - 300.00	X 0.00 0.00	Y 0.00 300.00	Z 0.00 0.00			
Procedure Mass-Stiffness Time History	0 - 0.00 1 - 300.00 2 - 600.00	X 0.00 0.00 0.00	Y 0.00 300.00 600.00	Z 0.00 0.00 0.00	-		
Procedure Mass-Stiffness Time History Results File	0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00	X 0.00 0.00 0.00 0.00	Y 0.00 300.00 600.00 900.00	Z 0.00 0.00 0.00 0.00	-		
Procedure Mass-Stiffness Time History Results File	0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 4 - 1200.00	x 0.00 0.00 0.00 0.00 0.00	Y 0.00 300.00 600.00 900.00 1200.00	Z 0.00 0.00 0.00 0.00 0.00			
Procedure Mass-Stiffness Time History Results File	0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 4 - 1200.00 5 - 1500.00	X 0.00 0.00 0.00 0.00 0.00	Y 0.00 300.00 600.00 900.00 1200.00 1500.00	Z 0.00 0.00 0.00 0.00 0.00 0.00			
Procedure Mass-Stiffness Time History Results File	0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 4 - 1200.00 5 - 1500.00 6 - 1800.00	X 0.00 0.00 0.00 0.00 0.00 0.00	Y 0.00 300.00 600.00 900.00 1200.00 1500.00 1800.00	Z 0.00 0.00 0.00 0.00 0.00 0.00			
Procedure Mass-Stiffness Time History Results File	0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 4 - 1200.00 5 - 1500.00 6 - 1800.00	X 0.00 0.00 0.00 0.00 0.00 0.00	Y 0.00 300.00 600.00 900.00 1200.00 1500.00 1800.00	Z 0.00 0.00 0.00 0.00 0.00 0.00			

The total procedure contains three steps:

Definition of the analysis parameters

Parameters



- Calculation of the masses and stiffness for all members
- Run dynamic analysis for the selected accelerograms

Steps 2 and three are operated either sequentially by pressing the buttons "Mass – Stiffness" and "Time History", or automatically by selecting the button "Automatic Procedure".

The first step of the process before the analysis is to define its parameters.

	EC8 Parameters - Linear time-history and	alysis (L	inear) 📃 🎽		
alysis T	ype Direct Integration V				
genvalu	10 Toleranc 0.001 Accelerograms	~	Data		
elerog	rams				
x					
e			Browse		
its	m/Sec^2 Value Fixed time step (sec)	V 0.01	View		
] Y					
e			Browse		
its	M/Sec*2 Value Fixed time step (sec)	V 0.01	View		
] Z					
e			Browse		
its	Value Pixed time step (sec)	• 0.01	View		
nping (er Defir	Damping Coefficients	Exc	itation Duration		
or bein	T(sec) ω(rad/sec) Mass proportional (a)		30		
t C	0 0 0 0.22	A	nalysis Time		
ond C	0 0 0 Stiffness proportional (β)		0.01		
nping C	coefficient % 5				
	Tradicalization (2) of the arrest dense to 0.1		OK Cancel		
Norma	lized inclination (d) of the second branch				
sele	ect Analysis Type between:	An	alvsis Type	Direct Integration	~
• [Direct Integration and			Modal	-
. 1	Modal	Ei	genvalu ¹⁰	T Direct Integration	
alect	ting Modal vou must determine tl	he nur	nber of Eige	envalues to be taker	n into accor

Analysis Type	Modal	~
Eigenvalu 10	Tolerano	0.001



Under "Accelerograms" the user can choose the directions of the seismic excitation, with the possibility of choosing from one to three directions by activating the corresponding checkbox in "X", "Y" or "Z".

Then the user must import the corresponding record file of the seismic stimulation through Browse

Accelerogra	ams	
✓ X File	C:\Athens_7_9_1999\Athens_7_9_1999_L.txt	Browse
Units	cm/Sec^2 ∨ Value Fixed time step (sec) ∨ 0.005	View
Y		
File	C:\Athens_7_9_1999\Athens_7_9_1999_V.bxt	Browse
Units	cm/Sec^2 \checkmark Value Fixed time step (sec) \checkmark 0.01	View
✓ Z		
File	C:\Athens_7_9_1999\Athens_7_9_1999_T.txt	Browse
Units	cm/Sec^2 ∨ Value Fixed time step (sec) ∨ 0.005	View

This file must be in a .txt format containing a column with the values of ground acceleration for each time step.

The user must also select the respe	ective Units Units	cm/Sec^2 ∨	of the ground acceleration
and the time stop of the recording	Fixed time step (sec)	~	0.01
and the time step of the recording			

If the imported file contains both seismic stimulation and time step, select

Define time and function Define time and function Fixed time step (sec)

and the program will read the time step values.

Finally, it is also possible to display each accelerogram by using the "View" button.



			View accel	erogram (g)			×
0.2 -				, ,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	, 	, ////////////////////////////////////		
0.3- 7816 / 39.075 (5 (sec) / -0.25995,	0.26360	15	20	25	30	35 Ca	39 -
Damping User De	g (Rayleig fined	h)	~	- Dampii Mas	ng Coefficien	nts ial (a)		
First	0	T(sec)	ω(rad/sec)		0.22	(-)		
Second	0	0	0	Stiffne	ess proportio	onal (β)		
Damping	Coefficie	nt %	5					

In the "Damping Rayleigh" section the user must select the parameter values for the Rayleigh registry.

More specifically the Damping Coefficient % must be defined, along with two modes that will be imposed on this coefficient.

Using these parameters the program calculates the values of the Damping Coefficients (a) and Stiffness proportional (b).



Finally, the user must define the Excitation Duration and the Analysis Time to be considered.

It's not necessary that the Analysis Time coincides with those of the accelerograms. In case that the time step of the analysis is smaller than the Accelerograms' step, the appropriate value is derived from a linear interpolation between the nearest two points.

In case that the time step of the analysis is bigger than the Accelerograms steps, then the structure will perform free oscillation for the remaining time.

After completing the input of the parameters, the user returns to the Run analysis window to proceed to the next steps



2. Results



"Results" command group contains commands about the generation of the load combinations, the design checks, the analysis results and the seismic forces.

2.1 Combinations

SCADA Pro contains all the combination files for all Static and Dynamic scenarios of Elastic and Inelastic Analysis, as "**Defalut Combinations**".

Name	Date modified	Туре	Size
eak-dyn.cmb	23/3/2010 1:27 μμ	CMB File	55 KB
📄 eak-dyn-et.cmb	11/1/2010 5:12 µµ	CMB File	48 KB
📄 eak-static.cmb	11/1/2010 5:11 μμ	CMB File	53 KB
📄 Ec8-dyn.cmb	23/3/2010 1:22 µµ	CMB File	48 KB
Ec8-dyn-cypr.cmb	23/3/2010 1:22 µµ	CMB File	48 KB
Ec8-PushOver.cmb	13/5/2013 11:44 πμ	CMB File	7 KB
Ec8-static.cmb	23/3/2010 1:21 µµ	CMB File	53 KB
Ec8-static-cypr.cmb	23/3/2010 1:21 µµ	CMB File	53 KB
📄 ita-dyn.cmb	23/3/2010 1:09 µµ	CMB File	48 KB
📄 itaEc8-dyn.cmb	23/3/2010 1:18 µµ	CMB File	48 KB
📄 itaEc8-static.cmb	23/3/2010 3:12 µµ	CMB File	53 KB
📄 ita-static.cmb	23/3/2010 1:06 µµ	CMB File	53 KB
📄 pal-static.cmb	27/2/2018 11:35 πμ	CMB File	3 KB
📄 sbc-000.cmb	5/5/2017 4:35 μμ	CMB File	91 KB
📄 sbc-001.cmb	5/5/2017 4:35 μμ	CMB File	91 KB
📄 sbc-002.cmb	5/5/2017 4:15 μμ	CMB File	91 KB
📄 sbc-003.cmb	5/5/2017 4:25 μμ	CMB File	91 KB

Default combinations concern seismic scenarios. To create combinations of non-seismic scenarios, you can use both automatic and manual mode.

Since you perform a seismic analysis scenario, its combinations are generated by the program automatically. By selecting the command "Combinations" the window with the combinations of the active seismic scenario opens.

You can achieve the same thing by selecting the command "Default combinations", that is, the program will import the combinations concerning the active seismic analysis scenario.

Predetermined combinations of the "executed" seismic analysis scenarios are automatically recorded by the program.

§ For Linear Analysis

Combinations: creating load combinations for Post-Processor and Members Design as well.



							Lo	ad Gro	ups Combin	nati	ions						x
YG YQ	1.35 1.5	γE γE0.3	1 0.3	Y	GE 1		ψ	2 0.3 V	Vind - Snow	[Ultimate ▼ ΣγG+γQ+Σγψ0Q ▼ ΣG+ψ1Q+Σψ2Q ▼ ΣG+E+Σγψ2Q		Serviceability ΣG+Q+ΣψΩ SG+ψ1Q+Σ SG+Σψ2Q	iQ iψ2Q	Calco Dele	ulation ete All	
		Туре		Dire	ection		LC1		LC2		LC3	LC4	1	LC5		LC6	^
Scer	nario						EC-8_G	reek 💌	EC-8_Greek	•	EC-8_Greek	EC	-8_Greek 💌	EC-8_Greek	-	EC-8	
Load	d Case						1		2		3	4		5		6	
Load	d Type						G	-	Q	•	Ex	Ez	<u>-</u>	Erx	-	Erx	
Acti	ons							-	Category A:	•	<u> </u>	·	-		-		
Des	cription																
Com	nb.:1	Ultima	te _	No		•	1.35		1.50								
Com	1b.:2	Ultima	te	No		-	1.00		0.50								
Con	nb.:3	Ultima	te	Dir.	+X	•	1.00		0.30		1.00	0.3	0	1.00			
Con	nb.:4	Ultima	ite _	Dir.	+X	•	1.00		0.30		1.00	0.3	0	1.00			
Com	nb.:5	Ultima	ite _	Dir.	+X	•	1.00		0.30		1.00	-0.	30	1.00			
Con	nb.:6	Ultima	te	Dir.	+X	•	1.00		0.30		1.00	-0.	30	1.00			
Com	nb.:7	Ultima	te	Dir.	-X	•	1.00		0.30		-1.00	0.3	0	-1.00			
Con	nb.:8	Ultima	te	Dir.	-X	•	1.00		0.30		-1.00	0.3	0	-1.00			
Con	1b.:9	Ultima	te	Dir.	-X	•	1.00		0.30		-1.00	-0.	30	-1.00			
Con	nb.:10	Ultima	te	Dir.	-X	•	1.00		0.30		-1.00	-0.3	30	-1.00			~
<				1												>	
	Add	Remove			Read		Save	ТХ	Т	Def	fault Combinations			ОК		Cancel	

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

The same results are derived from the "**Default Combination**" button, which fills in the table with the combinations of the active scenario analysis.

The default combinations of the executed analysis, are automatically saved by the program. You can create your combinations without using the "Default", or add more loads of other scenarios and calculate the new combinations either by modifying the defaults or deleting all "Delete All" and typing other coefficients.

						Load	Groups Co	ombina	tions		×
γG	1.35	γE	1	γGE	1	ψ2	0.3		Ultimate ✓ ΣγG+γQ+Σγψ0Q	Serviceability ✓ ΣG+Q+Σψ0Q	Calculatio
γQ	1.5	γE0.3	0.3			_	Wind - Sn	ow	 ΣG+ψ1Q+Σψ2Q ΣG+E+Σγψ2Q 	 ΣG+ψ1Q+Σψ2Q ΣG+Σψ2Q 	Delete All

Furthermore, you can type the factors, select the combinations and then press 'Calculation" to fill in the table.

The tool "Load Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.

§ Combinations for Wind-Snow

The tool "Laod Groups Combinations" works like an Excel file offering possibilities like copy, delete using Ctrl+C, Ctrl+V, Shift and right click.



Default combinations concern seismic scenarios. To create combinations of scenarios without seismic loads you can use both automatic and manual mode.

- The automatic mode:

requires that the automatic procedure for the calculation and distribution of loads of wind and snow as well as the automatic creation of the loads and combinations (see Chapter 6) is already done.

011	I	\$	-		Ĩ.		Load Attributi Wind	on O	90	180	270	Snow		Acci
arameter	s Eo	dit View	Member	Post	-Processor		Cpe_p+Cpi	3	7	11	15		Туріса	dent
•		• •	Corresponder	ice			Cpe_p-Cpi	4	8	12	16	Case i	19	22
		Wind	- Snow Loads				Cpe_n+Cpi	5	9	13	17	Case ii	20	23
							Cpe_n-Cpi	6	10	14	18	Case	21	24
		Load C	ase Definitio	n	×			то	otal Load	Deletio	n (Snow-V	Vind Loads)		
Self-we	eight	Wind 0 Cpe	_р+Срі	¥	Insert			Load At	ribution	in Mem	bers(from	Wind and S	now)	
LC S.	W. D	escription		^	el From Lis	1	Scenarios							
1 Ye	s D	ead Load					Wind 0		Stat	tic Wind	0(1) 🗸		Re	sults
2 No	b Li	ive Load			Delete		Wind 90		Stat	tic Ανεμα	oς90 🗸			
3 No	o V	Vind 0 Cpe_p+	-Срі		Loads		✓ Wind 180	1	Stat	tic Ανεμα	oς 18(Υ			
4 No	o V	Vind 0 Cpe_p-	Срі				✓ Wind 270		Stat	tic Ανεμα	oç 27(∨			
5 No	0 V	Vind 0 Cpe_n+	-Срі		Delete All		✓ Typical S	now	Stat	tic Xióvi	Типік 🗸			
0 N		Vind 0 Cpe_n-i	-Cpi		Loads		Accidenta	I Snow	Nev	v Scena	rio 🗸			
/ IN		······································	- C-:	\sim										
o				N	OK		An	alvsis S	cenario (Creation			_	

Concerning the above conditions, it is possible to automatically create wind and snow combinations by using the command Wind - Snow.

After running the seismic scenario and all the static scenarios of wind and snow, activate the seismic scenario and choose the command "Combinations". The combinations of the active seismic scenarios are completed automatically. To create automatically the combinations of all

wind and snow loads, press Wind - Snow . Automatically the coefficients of all wind and snow

scenarios will be filled, offering a complete loads combinations file. Press save to save the file.

- The manual way :

Except for the "Default Combinations," you can add others with loads from other scenarios.



G 1.35 Q 1.5	γE γE0.3	1 0.3	γGI	1					Failure ΣγG- ΣG+ ΣG+	∙γQ+Σγψ0Q μ1Q+Σψ2Q Ξ+Σγψ2Q		Serviceability ▼ ΣG+Q+Σψ0 ▼ ΣG+ψ1Q+Σα ▼ ΣG+Σψ2Q	Q #2Q	Caicu Delet	e All
	Kind		Direction		LC1		LC2	LC3		LC4		LC5		LC6	L
Scenario					EC-8_Gree	•	EC-8_Gree	EC-8	Gree	EC-8_Gree	<u></u>	EC-8_Gree	-	EC-8_Gree	E
Load Case					1		2	3		4		5		6	5
Туре					G	•	Q _	ExD	-	EzD	•	Erx	•	Erz	- Ey
Actions					G		Category A	•	-	<u>.</u>	-]	•		•
Comb.:1 Comb.:2 Comb.:3 Comb.:4	Failure Failure Serviceabi Failure Failure	ility	No Dir. +X DirX Dir. +Z DirZ	•	Ez ExD EzD EyD Erx Erz Erz ErzD ErzD ErzD		Category B:Of Category C:At Category D:St Category E:St Category F:W Category H:Rt Snow 1000m< Snow H<=100	fice are reas whi topping trage us eight<3 kN <we toofs (0.0 H (0.70 00m (0.5</we 	as (0.70, (ere people Areas (0. se areas (1 0kN (0.70 eight<160 00, 0.00, (, 0.50, 0.1 0, 0.20, (0.50 , 0 e cong 70 , 0.7 .00 , 0. , 0.70 , kN (0.7 0.00) 20) 0.00)		1.00		0.30	0.
Comb.:6	Failure	•	Dir. +X	-	NULL		Temperature	.20,0.0 (0.60,0) .50 , 0.00)			1.00		-0.30	-(
Comb.:7	Failure	•	Dir. +X	-	1.00		0.30	1.00		0.30		-1.00		0.30	0.
Comb.:8	Failure	•	Dir. +X	•	1.00		0.30	1.00		0.30		-1.00		0.30	-(
Comb.:9	Failure	•	Dir. +X	-	1.00		0.30	1.00		0.30		-1.00		-0.30	0.
Comb.:10	Failure	-	Dir. +X	•	1.00		0.30	1.00		0.30		-1.00		-0.30	-(
Comb.:11	Failure	-	Dir. +X	-	1.00		0.30	1.00		-0.30		1.00		-0.30	0.
Comb.:12	Failure	-	Dir. +X	-	1.00		0.30	1.00		-0.30		1.00		-0.30	-(
<				_											>

First type in the partial safety factors γG , γQ , γE , and check the Failure and Serviceability equations you want to be considered.

Then fill in the "LC" columns:

- Select the previously created Scenario from the scenarios list.
- Select the Load Type between Dead, Live, Seismic, or Null. Null is selected for another type of loads like Wind, Temperature, etc.
- To add to the "Default Combinations," those of snow loads choose LC10, load case 1, type Null, action Snow and "Calculate".

Save of the combinations file.

- In the Actions, line select the corresponding action from the list.
- In the Description line, you can type in a description. This is optional.

Do the same thing for all Load Cases and then click Calculation. Combinations' lines are filled in with the appropriate coefficients automatically.

"Type" column indicates which type of limit state is examined through the defined Failure

combination Serviceability

No
Dir. +X
DirX
Dir. +Z
DirZ

"Direction" column indicates in which direction the capacity of the structure will be examined during the Capacity Design procedure for the specific combination.

Add Remove buttons allow to add or remove lines and columns since they have been selected, as in an excel file.

You always have to Save the combination as a CMB file in the project's folder. You will need this file during "Post-Processor" and "Members Design".



Press Read to open a previously saved CMB file, or TXT to open a TXT file that contains the load combinations.

§ For Nonlinear (Pushover) Analysis

Nonlinear analysis combination is a unique combination of Dead and Live loads. It is displayed by Opening the "Combinations" when a Nonlinear Active Scenario

EC-8_Gree	k Nonline	ar (1)	-	ic acti		tod										
	_			is acti	va	tea.						1				-
	Kind	Directio	n	LC1		LC2	_	LC3		LC4		LC5	_	LC6	_	LC ^
Scenario				EC-8_Gree	-	EC-8_Gree	-	EC-8_Gree	-	EC-8_Gree	-	EC-8_Gree	-	EC-8_Gree	-	EC
Load Case				1		2	_	0		0		0		0	_	0
Туре				G	-	Q	-	G	-	G	•	G	-	G	-	G
Actions					-	Category A	-		-		-		-		-	
Description																
Comb.:1	Failure	▼ No	-	1.10		0.30										
Comb.:2		-	-]												
Comb.:3		-	-]												
Comb.:4		•	-													_
Comb.:5		•	-													
Comb.:6		•	-													
Comb.:7		•	-													
Comb.:8		-	-													
Comb.:9		•	-													
Comb.:10		-	-													
Comb.:11		-	-	1												
Comb.:12		-	-	1												
<				ż												>
In the fie	Id " <i>Seis</i> ad Combin	mic Loa	nd Co	mbinat	ior	ns"	la		y S	is scena		J				
Fx +k	Fz	🗹 Tri	angula	ar Disribut	ion	n										
Fx -k	Fz		form	Dietributie												
	• k Fz	⊻ un	norm	Distribute	л											
-Fx -	k Fz	Ac	cident	al Eccentr	ricit	ties Ex										
	k Fx		cident	al Eccentr	ricit	ties Ez										
🗹 Fz +																
✓ Fz +	Fx			-												
✓ Fz + I Fz - k ✓ Fz + k	Fx	Ba	se She	ear from R	les	ponce Spe	ect	rum								

We define the combinations for which inelastic analyzes will be performed. Each combination means that a seismic force will be applied along the specified direction (x or z) by a factor 1 as well as a vertical seismic force in the vertical direction by a factor defined in the field "Transverse Load Factor".

<u>The default value is 0.3.</u>



Moreover, we define the type of height distribution of seismic force (Triangular or Orthogonal). KANEPE demands both types of seismic force distribution.

Also, in case we want, except for the seismic forces, the moments resulting from the accidental eccentricities to be taken into consideration, then we activate the fields "Accidental eccentricities Ex and Ez".

Then, to design the reinforcement, you should define another combination as well as the distribution, selecting the command "**Checks**" in the field "**Analysis method for the Reinforcing Model Design**" (you can see §2.2 "**Checks**")





2.2 Checks

§ For Linear Analysis

Press "Checks" and in the dialog box:

- Type in the minimum length for defining the walls and click the corresponding button,
- set limits on the mass and the stiffness considering the regularity conditions of the building,
- press "OK".

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

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

eismic Aı	nalysis Co	ontrol	Co ×						
Angula	ar Disortion	γ i <=	0.005						
min Wall	Length (cm)>=	200						
Column	Element	Vy	Vz ^						
1	1		✓						
10	2								
18	3	 Image: A start of the start of							
2	4								
3	5								
11	6		✓						
19	7								
12	8								
4	9								
5	10		~						
Add A		Clea	ar All						
Mass - Sti	ffness Limits								
Masses		Stiffne	ss						
Reducti	0.5	Reduct	ti 0.5						
Increase	0.35	Increa	s€ 0.35						
Wall Adequacy explorer (nv)									
File with internal forces from combinations (combin.txt)									
OK Cancel									

J	check.txt - WordPad – 🗆	
ile Edit	t View Insert Format Help	
D ≊ 		
	CHECKS REPORT ACCORDING TO THE MANIN DIRECTIONS OF THE BUILDING SIMPLIFIED STATIC ANALYSIS (EC8)	^
	(
Che	eck for mass and stiff.differences per build.level (par.4.2.3.3.)	
n/n	Total Tot.Mass Total Stifness Differneces Mass - Stifness	
Level	Heig(M) KN/g Ki*10^3(KNM) (Mi+1-Mi)/Mi - (Ki+1-Ki)/Ki	
1	(ΔM1)*-(ΔK1-X)*(ΔK1-Z)*(ΔM1)*-(ΔK1-X)-*(ΔK1-Z) 4.500 483.0631 10808.7701 12329.8051 1	
2	7.750 472.885 10808.770 12329.805 red 0.02 inc. 0.00 inc. 0.00	
3	11.000 321.950 10808.770 12329.805 red 0.31 inc. 0.00 inc. 0.00	
	*	
Masses	s : The increase must be <=0.35 - The reduction must be <=0.50	
Stifne	ees . Increase must be /=0.35 - Deduction must be /=0.50	
Stifne	ess : Increase must be <=0.35 - Reduction must be <=0.50	
Stifn Check	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Stifn Check	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Stifn Check	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Check Check	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Check Cente	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Check Cente n/n Level	ess : Increase must be <=0.35 - Reduction must be <=0.50 	
Cente n/n Level	ess : Increase must be <=0.35 - Reduction must be <=0.50	
Cente n/n Level	ers : Increase must be <=0.35 - Reduction must be <=0.50 	
Cente Cente n/n Level 1 2 3	ers : Increase must be <=0.35 - Reduction must be <=0.50 	
Stifn(Check Cente n/n Level 1 2 3 	ers : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height(m) X Coor.(m) X Coor.(m) Z Coor.(m) C.W-C.S(m) 	
Stifne Check Cente n/n Level 1 2 3 	ers : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height(m) X Coor.(m) X Coor.(m) Z Coor.(m) C.W-C.S(m) 	
Cente Check Cente n/n Level	<pre>ess : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height(m) X Coor.(m) X Coor.(m) Z Coor.(m) C.W-C.S(m)</pre>	
Cente Cente n/n Level 2 3 	<pre>ess : Increase must be <=0.35 - Reduction must be <=0.50 "satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height(m) X Coor.(m) Z Coor.(m) Z Coor.(m) C.W-C.S(m) 4.500 17.4196 6.3322 16.8076 6.9811 0.8919 7.750 17.4230 6.3216 16.3556 6.6830 1.1269 11.000 17.5070 6.4098 15.9014 6.4230 1.6057 Shear Force Par. 5.1.2. Reference Level: 0 0.000(m) </pre>	
Cente Cente n/n Level 1 2 3 Walls	<pre>ess : Increase must be <=0.35 - Reduction must be <=0.50</pre>	
Cente Check Cente n/n Level Walls n/n Level	<pre>ess : Increase must be <=0.35 - Reduction must be <=0.50</pre>	
Cento Check Cento n/n Level 	ers : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height (m) X Coor. (m) Z Coor. (m) X Coor. (m) Z Coor. (m) C.W-C.S (m) 4.500 17.4196 6.3322 16.8076 6.9811 0.8919 7.750 17.4230 6.3216 16.3556 6.6830 1.1269 11.000 17.5070 6.4098 15.9014 6.4230 1.6057 Shear Force Par. 5.1.2. Reference Level: 0 0.000 (m) Walls Shear/Total Shear = nvx Walls Shear/Total Shear = nvz (Kn) (Kn) (Kn) (Kn) ***********************************	
Cento Check Cento n/n Level 1 2 3 Walls n/n Level 	ers : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height (m) X Coor. (m) Z Coor. (m) X Coor. (m) Z Coor. (m) C.W-C.S (m) 4.500 17.4196 6.3322 16.8076 6.9811 0.8919 7.750 17.4230 6.3216 16.3556 6.6830 1.1269 11.000 17.5070 6.4098 15.9014 6.4230 1.6057 Shear Force Par. 5.1.2. Reference Level: 0 0.000 (m) + Walls Shear/Total Shear = nvx Walls Shear/Total Shear = nvz (Kn) (Kn) (Kn) (Kn) *** 3- 0.000 1310.019 0.00 NO 35- 0.000 1310.329 0.00 NO 3- 0.000 1136.063 0.00 NO	
Center Check Chech	ers : Increase must be <=0.35 - Reduction must be <=0.50 satisfy the regular.in elevation criteria er Weight - Center of Stiff Total CENTER WEIGHT CENTER OF STIFF Distance Height(m) X Coor.(m) Z Coor.(m) Z Coor.(m) C.W-C.S(m) *	~



§ For Nonlinear (Pushover) Analyses

The precondition in order the checks of Inelastic Analyzes to open is that since the analysis is over, select the command Mass Distribution so as the window **Report** opens and then press the button Graph - Checks Creation for Output

	Analysis Type - Distribution		DL			SD			NC		Prin	t	^
		в	с	т	B	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	Yes	•	
17	Fz+0.30*Fx - Triangular	90	48	138	0	0	0	0	0	0	Vac	•	
25	-Fz+0.30*Fx - Triangular	79	35	114	0	0	0	0	0	0	Tes		
101	Fx+0.30*Fz - Orthogonal	85	52	137	0	6	6	0	0	0	Yes	÷	
109	-Fx+0.30*Fz - Orthogonal	83	34	117	0	0	0	0	0	0	Yes	<u> </u>	
117	Fz+0.30*Fx - Orthogonal	91	49	140	1	23	24	0	28	28	Yes	-	
125	-Fz+0.30*Fx - Orthogonal	81	35	116	0	0	0	0	0	0	Yes	-	
												•	
											1	•	
												•	
												-	
											-	•	
In	clude Total Table in Output	- 14	atha	d Dec						Dipla	ay Ch	ecks	

The table above contains the total number of beams (B) and columns (C) which have lower capacity than the demanded. For each pushover and each LS. The symbol "T" means total.



Indicate with YES the pushover analysis to be included in the prints, otherwise 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 with a "YES" indicator in the "Print" column:

1 Fx+0.30*Fz - Triangular	84 51 135 1 15 16 0 2 2 Yes 🗸
1	CheckPS_1.txt - WordPad –
File Edit View Insert Format Help	
	जि. व
MEMBER CAPACI	TIES IN DEFORMATION TERMS
Analysis Type - Distribution	1 : Fx+0.30*Fz - Triangular (1)
larget Dispacements:	Severe Damage (SD) 0.052(m)
	Near Collarse (NC) 0.062(m)
Beams (Fx+0.30*Fz - Triangul	lar) (1)
	nn Settere Damage Near Collange
	(SD) (NC)
Memb. Node [ysd*0sd]0pl/yrd]	ysd*0sd 0pl/yrd ysd*0sd 0pl/yrd
+++++	+++++
46 1 0.00000 0.00000	Yes 0.00000 0.01117 Yes 0.00000 0.02234 Yes
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
	Yes 0.00000 0.01243 Yes 0.00000 0.02485 Yes
	128 0.00000 0.01058 128 0.00000 0.02115 128
50 1410.0000010.00000	Yes10.0000010.010481Yes10.0000010.020971Yes
810.0000010.00000	No 0.00001 0.00944 Yes 0.00001 0.01887 Yes
51 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
52 8 0.00000 0.00000	Yes 0.00000 0.01157 Yes 0.00000 0.02315 Yes
7 0.00041 0.00000	No 0.00025 0.00917 Yes 0.00031 0.01835 Yes
53 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
54 10 0.00000 0.00000	Yes 0.00000 0.01347 Yes 0.00000 0.02694 Yes
	Tes 0.00000 0.0134/[Tes 0.00000 0.02694 Tes
910.0000010.00000	Yesin 0000010.00904[123]0.00000[0.01808]123
56 910.0000010.00000	Yes10.0000010.010751Yes10.0000010.021511Yes
610.0000010.00000	Yes 0.00000 0.01367 Yes 0.00000 0.02734 Yes
57 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
58 5 0.00000 0.00000	Yes 0.00000 0.01243 Yes 0.00000 0.02485 Yes 🗸
For Help, press F1	NUM

At the bottom of the file, the Shear Resistance Check appears only for members that fail in shear.



SHEAR RESISTANCE CHECK											
<pre>3eams (Fx+0.30*Fz - Triangular) (1)</pre>											
El	em.	Node	Vrd,s	Vrd,max	Vr	Ved	Ratio A	– DLI	B - SI	D F - 1	NCI
		+	+	++	+	+-	++-	+		+	
Columns (Fx+0.30*Fz - Triangular) (1)											
01	umns	s (Fx+0.3	0*Fz - T	riangular)	(1)						
01 E1	em.	8 (Fx+0.3 Node	0*Fz - T Vrd,s	Vrd,max	(1) 	Ved	Ratio A	- DL	B - SI	D F - 1	
01 E1	em.	8 (Fx+0.3 Node	0*Fz - T Vrd,s	Vrd,max	(1) Vr	Ved	Ratio A	- DL	B – SI	D F - 1	NC
01 E1	em. 4	Node 42 y:	Vrd,s + 1369.81	Vrd,max 2227.03	(1) Vr 171.52	Ved 171.71	Ratio A	- DL	B – SI	D F - 1	NC
01 E1	em. 4	Node 42 y: 5 y:	0*Fz - T Vrd,s + 1369.81 1369.81 276.60	Vrd,max 2227.03 2227.03	(1) Vr 171.52 171.51 251.99	Ved 171.71 171.71 2.68	Ratio A	- DL 	B - SI	D F - 1 -+	NC
01 E1	umn: em. 4 18	Node 42 y: 5 y: 56 y:	0*Fz - T Vrd,s + 1369.81 1369.81 276.60 276.60	Vrd, max 2227.03 2227.03 128.48 128.48	(1) Vr 171.52 171.51 251.99 254.75	Ved 171.71 171.71 2.68 2.68	Ratio A 1.0011 1.0011 0.0209	NO	B - SI NO I	D F - 1 NO NO	NC
01 E1	em. 4 18	Node 42 y: 5 y: 56 y: 19 y:	0*Fz - T Vrd,s + 1369.81 1369.81 276.60 276.60 276.60	<pre>rlangular) Vrd,max 2227.03 2227.03 128.48 128.48 128.48 </pre>	(1) Vr 171.52 171.51 251.99 254.75 284.94	Ved 171.71 171.71 2.68 2.68 54.80	Ratio A 1.0011 1.0011 0.0209 0.0209 0.4265	NO NO	B - SI NO NO NO	 	NC

At the end of this file and if the parameters of the Masonry Infills scenarios are active, the checkbox/displays the results of masonry infills capacities in deformation terms for each infill element.

For tensioned members, results are not displayed because these effects are not taken into account in the project's model.

MASONRY INFILLS CAPACITIES IN DEFORMATION TERMS											
	Damage	Limitati	on	Severe	Damage		Near Collapse				
	(D)	L)		1 (3	5D)		1	(NC)			
Member	ysd*ef	εγ	l -	γsd*ɛf	εu/γrd	I	γsd*εf	εu	l.		
	-+	+	+	+	+	+	+	+	+		
121 Comp	. 0.00440	0.00150	No	0.00493	0.00269	No	0.00534	0.00350	No		
122 Tens	-	l i i i i i i i i i i i i i i i i i i i	1	1			I		1 1		
123 Comp	. 0.00262	0.00150	No	0.00278	0.00269	No	0.00291	0.00350	Yes		
124 Tens	- 1	l i i i i i i i i i i i i i i i i i i i		1	1		I				
125 Comp	. 0.00426	0.00150	No	0.00474	0.00269	No	0.00511	0.00350	No		
126 Tens	- 1	I.	1	1			I		L L .		
127 Comp	. 0.00291	0.00150	No	0.00306	0.00269	No	0.00317	0.00350	Yes		
128 Tens	-1	l i i	1	l i	l i	1	I		I I		

Check ^{Include Total Table in Output} to include the table in the print output.

- Checks document help you to assess in which Pushover analysis, the structural elements present lower capacity than the one defined in the considered LS, i.e. it can be easily observed in which Pushover analysis, the defined performance level is not satisfied. In such case, the structure must be strengthened, for example through the reinforcement of some structural elements, and be redesigned.
- First select from the list the analysis, which indicates the redesign of the existing structure.

Analysis Method for the Reinforcing Method Design



Fx+0.30*Fz - Triangular			V							
-Ex+0.30*Ez - Triangular										
E7±0 30*Ex - Triangular										
Ez LO 20%Ex. Triangular										
-FZ+0.30*FX - Triangular										
Fx+0.30*Fz - Orthogona	·									
-Fx+0.30*Fz - Orthogon	al									
Fz+0.30*Fx - Orthogona										
-Fz+0.30*Fx - Orthogon	al									
1 The Interventions procedure is explained in "Members Design" chapter.										
🔥 In both proces	ses of eva	luati	on and	d interv	ention,	no el	ement	s with lack o	f structural	
integrity must	he met fo	or anv	/ analy	vsis tyne	of the	consi	dered	l imit State		
integrity must	be met re		anary	sis type	. or the	consi	ucreu	Linne State.		
Internal For	ces									
Press the	CO	mma	na to	open tr	e txt fil	le con	taining	:		
NODE DISPLACE	FMENTS		ROTA	TIONS						
			NOTA	nons						
MEMBER INTE	RNAL FOF	RCES								
BEAMS ACTIVE	STIFFNE	SS								
File Edit View Insert Format Help										
	B									
NODE DISP	LACEMENTS / RO	TATIONS								
	NENTS									
				TONC						
Node Num DISPLACE Numb. L. dx(mm) dy(mm)	dz(mm)	rx(ra	d) ry	IONS (rad)	rz(rad)					
Node Num DISPLACE Numb. L. dx(mm) dy(mm) 	dz(mm) -	rx (ra	d) ry	I O N S (rad)	rz (rad)	_				
Node Num DISPLACE Numb. L. dx(mm) dy(mm)	dz (mm)	rx (ra	d) ry	I O N S ((rad) MEMBERS IN	rz(rad) TERN.FORCES	5				
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) 1 0.000E+000 0.000E+00 2 2.409E-001 -3.605E-00 3 3 1.919E-001 -4.964E-00 5 5 2.454E-001 -2.222E-00 5	dz (mm) - Memb. Numb.	rx (ra 	d) ry	IONS (rad) MEMBERS IN	rz(rad) TERN.FORCES	S Tors.	Bend.	Bend.		
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) 1 0.000E+000 0.000E+00 2 2.409E-001 -3.465E-00 3 1.919E-001 -4.964E-00 5 2.454E-001 -2.222E-00 6 2.183E-001 -2.399E-00 7 2.468E-001 -1.98E-00	Memb. Numb. Num. Load	R rx(ra S./E. A Node -	d) ry (xial N(KN)	I O N S 7(rad) MEMBERS IN Shear QY(KN) 	rz(rad) IERN.FORCES Shear QZ(KN)	5 Tors. MX (KNM)	Bend. MY(KNM	Bend. MZ (KNM)		
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) 1 0.000E+000 0.000E+00 2 2.409E-001 -3.605E-00 3 1.919E-001 -4.864E-00 5 2.454E-001 -2.222E-00 6 2.183E-001 -2.399E-00 7 2.448E-001 -1.88E-00 8 2.448E-001 -3.310E-00	dz (mm) Memb. Numb. Num. Load 1	R rx(ra S./E. A Node - 39	d) ry [[] [] [] [] [] [] [] [] [] [] [] [] [I O N S y(rad) MEMBERS IN Shear QY(KN) 3.85 -3.85	rz(rad) TERN.FORCES Shear QZ(KN) 	5 Tors. MX(KNM) -0.02	Bend.	Bend.) MZ(KNM) 4 3.98 31 11 43		
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) 1 0.000E+000 0.000E+00 2 2.409E-001 -3.65E-00 3 3 1.919E-001 -4.964E-00 5 5 2.454E-001 -2.399E-00 6 7 2.448E-001 -3.10E-00 8 8 2.458E-001 -3.130E-00 9 9 2.187E-001 -1.775E-00 10 1.912E-001 -1.25E-00	H L H I S dz (nm) Memb. Numb. Num. Load 1 2	rx(ra 	Axial N (KN) 131.56 -77.56 80.14	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) IERN.FORCES Shear QZ(KN) -9.09 -9.09 -2.2	5 Tors. MX (KNM) -0.02 0.02	Bend. MY (KNM 	Bend. MZ(KNM) 4 3.98 31 11.43 TIVE STIFFNESS		
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn) 0.002+000 0.002+000 2 1.402-001 -3.652-00 3 1.919E-001 -4.65E-00 5 2.454E-001 -2.359E-00 6 2.183E-001 -2.359E-00 7 1.48E-001 -3.188E-00 8 2.448E-001 -3.130E-00 9 2.487E-001 -3.130E-00 9 1.919E-001 -1.175E-00 11 1.125E-00	H L H I S dz (nm) Memb. Numb. Num. Load 1 2 4	rx(ra 	Axial N(KN) 131.56 -77.56 80.14 -54.83 419.61	I O N S ('rad) MEMBERS IN Shear QY(KN) 3.85 -3.85 2.76 2.76 -2.76 1.1.34	rz(rad) TERN.FORCES Shear QZ(KN) -9.091 -9.091 -2.2 2.2 -3.6	5 Tors. MX(KNM) -0.02 0.02	Bend. MY(KNM 2 -34.0 2 -2.3 BEAMS AG Plastic H	Bend. M2 (KNM) 	K (along Z)	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn) 1 0.0002+000 0.0002+000 2 1.492-001 -3.652-00 3 1.919E-001 -3.992-00 5 2.454E-001 -2.3992-00 6 2.183E-001 -3.188E-00 3 7 2.448E-001 -3.130E-00 9 8 2.492E-001 -3.130E-00 9 10 1.919E-001 -1.775E-00 10 11 1.919E-001 -3.456E-00 12 12 2.448E-001 -4.567E-00 13 13 1.919E-001 -5.456E-00 13	Memb. Numb. Num. Load 1 2 4	rx (ra 	Axial (1) N(KN) 131.56 -77.56 80.14 -54.83 419.61 -385.86	I O N S ((rad) MEMBERS IN Shear QY(KN) 3.85 2.76 11.34 -11.34	rz(rad) TERN.FORCES Shear QZ(KN) -9.09 -9.09 -2.2 2.2 -3.6 3.6	5 Tors. MX(KNM) -0.02 0.02 (Member Num.	Bend. MY(KNM 2 -34.0 2 -2.3 BEAMS AG Plastic H Start 1	Bend. MZ(KNM) 	K (along Z) kN和2	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn) 1 0.0002+000 0.0002+000 2 2.4092-001 -3.6052-00 3 1.919E-001 -4.964E-00 5 2.454E-001 -2.3992-00 6 2.183E-001 -2.3992-00 7 1.48E-001 -1.188E-00 1.188E-00 9 2.461E-001 -3.130E-00 9 2.187E-001 -1.775E-00 10 1.919E-001 -1.3456E-00 11 1.919E-001 -3.456E-00 12 2.448E-001 -4.567E-00 13 1.919E-001 -5.456E-00 14 2.192E-001 -5.456E-00 14 2.192E-001 -5.456E-00	dz(mm) Memb. Numb. Num. Load 1 2 4 4 5	rx(ra 	Axial N (KN) 131.56 -77.56 80.14 -54.83 419.61 -385.86 677.46	I O N S /(rad) MEMBERS IN Shear QY(KN) 3.85 -2.76 -2.76 11.34 -11.34 0.50 -0.50	rz(rad) TERN.FORCES Shear QZ(KN) -2.2 2.2 -3.6 3.6 -0.2 0.2	S Tors. MX(KNM) -0.02 0.02	Bend. MY(KNM 2 34.0 2 2.3 BEAMS A BEAMS A Plastic H: Start 1 	Bend. MZ(KNM) 	K (along Z) kNm2 10156.94561	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn) 1 0.0002+000 0.0002+000 2 1.4052-001 -3.6652-00 3 1 9192-001 -4.9642-00 5 2.4542-001 -2.222E-00 6 2.1832-001 -1.3102-00 8 2.4482-001 -3.1882-00 8 2.4482-001 -3.1825-00 10 1.9192-001 -3.4562-00 10 1.9192-001 -3.4562-00 11 1.9192-001 -3.4562-00 11 1.9192-001 -3.4562-00 13 1.9192-001 -3.6692-00 13 1.9192-001 -3.6692-00 14 2.1925-001 -3.7572-00 14 2.1925-001 -3.6692-00 16 2.1572-001 -7.3572-00	dz(mm) Memb. Numb. Num. Load 1 2 4 4 5 5 6	rx(ra rx(ra Node - 39 21 40 31 42 51 43 6 44	C T A T d) ry 	I O N S ('rad) MEMBERS IN Shear QY(KN) 1 Shear 1 QY(KN) 1 -3.85 1 -3.85 1 -3.85 1 -2.76 1 -2.76 1 -2.76 1 -2.76 1 -11.34 0 -0.50 1 -1.89 1 -0.50 1 -0.80 1 -0	rz(rad) TERN.FORCES Shear QZ(KN) -2.2 2.2 -3.6 3.6 -0.2 0.2 -1.1 1	5 Tors. MX (KNM) -0.02 0.02 0.02 (Member Num. 1 44 45	Bend. MY(KNM 2 -34.0 -2.3 BEAMS A(Plastic H. Start 1 No N(No N(Bend. MZ(KNM) 	K (along Z) kkm2 	
Node Num D I S P L A C E Numb.L. dx (mm) dy (mn) 1 0.0002+0001 0.0002+001 2 1.919E-0011+3.655E-00 2 3 1.919E-0011+3.655E-00 3 5 1.2.454E-0011+2.222E-001 6 6 1.2.454E-0011+3.188E-00 8 9 1.2.48E-0011+3.130E-00 9 9 1.2.48E-0011+3.130E-00 9 10 1.919E-0011+3.148E-00 1.125E-00 11 1.919E-0011+3.456E-00 1.318E-00 12 1.2.48E-0011+3.669E-00 1.318E-00 13 1.919E-0011+3.669E-00 1.41 14 2.199E-0011+3.669E-00 1.41 14 2.199E-0011+3.669E-00 1.618E-00 16 1.2187E-0011+7.2108E-00 1.818E-00 18 1.4073E-0011+3.748E-00 1.418E-00 19 4.873E-0011+7.2108E-00 1.61743E-00 201 4.738E-0011+7.2108E-00 1.81948E-0011+7.2108E-00	dz(mm) 	rx(ra rx(ra Node - 39 2 40 3 42 42 5 42 5 43 6 71 75	A C T A T d) Fy d) Sy	I O N S ('rad) MEMBERS IN Shear QY(KN) 1 Shear 1 QY(KN) 1 -3.85 1 -3.85 1 -3.85 1 -2.76 1 -2.76 1 -2.76 1 -2.76 1 -11.34 0 -0.50 1 -1.89 1 .89 1 .89 -2.08	rz(rad) TERN.FORCES Shear QZ(KN) -9.09 -2.2 2.2 -3.6 -0.2 -0.2 -1.1 1.1 -4.0	5 Tors. MX(KNM) -0.02 0.02 0.02 1.02	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along 2) kkm2 	
Node Num D I S P L A C E Numb.IL. dx (mm) dy (mm) 1 0.0002+0001 0.0002+0001 2 1.919E-0011+3.65E-00 2.4092-0011+3.65E-00 3 1.919E-0011+4.964E-00 5 5 1.2.454E-0011+2.222E-00 6 6 1.2.454E-0011+3.188E-00 8 9 1.2.487E-0011+3.130E-00 9 9 1.2.487E-0011+3.138E-00 1.1.25E-00 10 1.919E-0011+3.4567E-00 1.318E-00 11 1.919E-0011+3.669E-00 1.318E-00 12 1.2.482E-0011+3.667E-00 1.318E-00 13 1.919E-0011+3.669E-00 1.418E-00 14 1.919E-0011+3.669E-00 1.418E-00 14 1.2199E-0011+3.669E-00 1.618E-00 16 1.2187E-0011+7.357E-00 1.669E-00 16 1.2187E-0011+7.210E-00 2.190E-001+5.748E-00 20 4.673E-001+3.074E-00 2.190E-001+3.074E-00 20 4.673E-001+3.074E-00 2.1	dz(mm) dz(mm) Memb. Numb. Num. Load 1 2 4 5 6 7 8	rx(ra rx(ra S./E. JA Node - 39 2 40 40 31 42 51 43 51 51 51 51 51 51 51 5	Lxial 131.56 -77.56 80.14 -41.83 419.61 -385.86 677.46 -626.83 148.65 -127.05 394.78 -373.18 -373.18 216.71	I O N S ('rad) MEMBERS IN Shear QY(KN) Shear QY(KN) 3.85 -3.85 -3.85 -2.76 -2.76 1.34 0.50 -11.34 0.50 -1.89 1.89 1.89 1.89 1.89 -2.08 2.08 -0.24 -0.	rz(rad) TERN.FORCES Shear Q2(RN) -9.09 -9.09 -2.2 2.2 2.2 2.2 -3.6 -0.2 0.2 -1.1 1.1 1.1 -4.0 4.0 0.4 0.4 1.1 1.1 1.2 1.4	S Tors. MX (KNM) 	Bend. MY(KNM 2 -34.0 2 -	Bend. MZ(KNM) 	K (along 2) kkm2 	
Node Num D I S P L A C E Numb.L. dx (mm) dy (mm) 1 0.0002+0001 0.0002+0001 2 1.919E-0011+3.65E-00 2.4092-0011+3.65E-00 3 1.919E-0011+4.964E-00 5 5 1.2.454E-0011+2.222E-00 6 6 1.2.454E-0011+3.188E-00 1.1.88E-00 9 1.2.48E-0011+3.130E-00 9 11 1.919E-0011+1.25E-00 111 11 1.919E-0011+3.456E-00 131 12 1.919E-0011+3.656E-00 131 13 1.919E-0011+3.656E-00 131 14 2.199E-0011+3.657E-00 141.82E-00 13 1.919E-0011+3.657E-00 141.82E-00 14 2.199E-0011+3.657E-00 1569E-00 16 2.187E-0011+7.210E-00 191 14 2.199E-0011+3.657E-00 211.91.91.91.91.91.91.91.91.91.91.91.91.9	dz(mm) dz(mm) Memb. Numb. Num. Load 	K TX (TA Node - 39 2 40 31 42 5 42 43 43 44 71 45 8 46 9	Lxial 131.56 -77.56 80.14 -54.83 419.61 677.46 -626.83 148.65 -127.05 394.78 -373.18 216.71 -195.11	I O N S ('rad) MEMBERS IN Shear QY(KN) Shear QY(KN) 3.85 -3.85 -3.85 -2.76 -2.76 1.34 0.50 -11.34 0.50 -1.89 1.89 1.90 2.08 1.90 1.90 1.90 1.89 1.90 1.90 1.89 1.90	rz(rad) TERN.FORCES Shear Q2(RN) -9.09 -9.09 -2.2 2.2 2.2 2.2 -3.6 -0.2 0.2 -1.1 1.1 1.1 -4.0 4.0 -0.4	S Tors. MX(KNM) -0.02 0.02 0.02 1.02 1.02 1.02 1.02 1.02	Bend. MY(KNM 2 -34.0 2 -34.0 2 -2.3 BEAMS A/ Plastic H: Start 1 No N(N No N(N) No N(N) No N(N) No N(N) No N(N)	Bend. MZ(KNM) 	K (along 2) kkm2 10156.94561 13278.84313 13008.70413 10156.94561 18160.65744 14494.69739 12916.76626	
Node Num D I S P L A C E Numb.lL. dx (mm) dy (mm) 1 0.0002+000 0.0002+000 2 1.912-001 -3.65E-00 2.4032-001 -3.222E-00 3 1.9192-001 -4.964E-00 5 5 2.454E-001 -2.222E-00 6 6 1.2532-001 -3.233E-00 8 9 2.482E-001 -3.188E-00 8 10 1.9192-001 -1.125E-00 11 11 1.9192-001 -3.6692-00 12 12 2.488E-001 -4.567E-00 13 13 1.9192-001 -3.6692-00 14 14 2.1992-001 -5.4692-00 16 15 1.9192-001 -5.4692-00 16 14 2.1992-001 -5.4692-00 16 15 1.9192-001 -7.357E-00 18 14 2.1992-001 -7.4382-00 19 16 1.2187E-001 -7.2102-00 21 17 5.6932-001 -3.647E-00 21 18 1.9192-001 -3.647E-00 21	dz(mm) dz(mm) Memb. Numb. Num. Load 	K TX(TA) Node - 39 2 40 31 42 5 42 43 44 71 45 8 46 74	Unit and a set of the	I O N S ((rad) MEMBERS IN Shear QY(RN) 1 Shear QY(RN) 1 -3.85 1 -3.85 1 -2.76 1 -2.76 1 1.34 0 -50 1 -11.34 0 -50 1 -2.08 1 .89 1 .	rz(rad) TERN.FORCES Shear QZ(KN) -9.09 -9.09 -2.2 2.2 2.2 2.2 2.2 -3.6 -0.2 0.4 -1.1 -4.0 4.0 -5.6 -6.8 0.8 -0.8	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 2) -34.0 2) -34.0 2) -2.3 BEAMS & Plastic H. Start 1 No N. No N.	Bend. MZ(KNM) 	K (along Z) kkm2 10156.94561 13278.84513 1308.70413 10156.94561 18160.65744 1494.69739 12916.76626 12970.37008 18171.38223	
Node Num D I S P L A C E Numb.lL. dx (mm) dy (mm) 1 0.0002+000 0.0002+000 2 1.912-001 -3.65E-00 2.4022-001 -3.65E-00 3 1.9192-001 -4.964E-00 5 5 2.454E-001 -3.222E-00 6 6 1.255E-001 -2.222E-00 8 9 2.448E-001 -3.10E-00 8 9 2.448E-001 -3.10E-00 8 10 1.919E-001 -4.567E-00 1.25E-00 11 1.919E-001 -4.567E-00 1.2 12 2.448E-001 -4.567E-00 1.4567E-00 13 1.919E-001 -3.669E-00 1.4 14 2.199E-001 -3.669E-00 1.6 15 1.912E-001 -3.649E-00 1.5 16 2.187E-001 -7.210E-00 2.190E-001 -5.67E-00 20 6.622E-001 -1.612E-00 2.2 16 6.622E-001 -3.647E-00 2.2 21 5.6622E-001 -3.647E-00 2.2 22 6.6622E-001	dz(mm) dz(mm) Memb. Numb. Num. Load 	K TX (TA Node 39 21 40 39 21 40 43 51 41 51 41 51 41 51 41 51 41 41 41 41 41 41 41 4	Line (1) (1) (2) (2) (2) (2) (2) (2) (2) (2) (2) (2	I O N S ((rad) MEMBERS IN Shear QY(RN) 1 Shear QY(RN) 1 -3.85 1 -3.85 1 -2.76 1 -2.98 1 -0.24 1 -0.24 1 -0.66 1 -0.64 1 -0.66 1 -0.65 1 -0.64 1 -0.65 1 -0.64 1 -0.65 1 -0.64 1 -0.65 1 -0.64 1 -0.65 1 -0.64 1 -0.65 1 -0.64 1 -0.65 1 -0.6	rz(rad) TERN.FORCES Shear QZ(KN) -0.091 -2.22 2.22 -3.66 -0.22 0.64 -0.22 -1.11 -1.12 -1.11 -4.0 0.4 -0.4 -0.8 -0.	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along Z) kkm2 10156.94561 13278.45413 13008.70413 10156.94561 18160.65744 1494.69739 12916.76626 12870.37008 18171.38223 12556.14660	
Node Num D I S P L A C E Numb.L. dx (mm) dy (mn)	dz(mm) dz(mm) Memb. Numb. Num. Load 	K TX (TA Node 39 40 40 42 42 42 42 42 42 44 7 44 7 44 7 44 7 44 7 44 145 8 44 9 44 145 8 44 145 1	Unit of the first	I O N S ((rad) MEMBERS IN Shear QY(KN) QY(KN) 	rz(rad) TERN.FORCES Shear QZ(RN) -9.09 -9.09 -2.22 2.2 2.2 -3.6 -0.2 0.4 -1.1 1.1 -4.0 0.4 -0.4 -0.4 -0.4 -0.4 -0.5 -1.2 -1.2 -1.2 -2.2 -2.2 -3.6 -0.2 -3.6 -0.2 -1.1 -1.2 -1.2 -1.2 -2.2 -3.6 -0.2 -1.2 -1.4 -1.4 -1.4 -0.4 -0.4 -0.4 -1.4 -0.8 -1.8	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along Z) kNm2 10156.94561 13278.84313 13008.70413 10156.94561 12916.76526 12916.76526 12917.76526 12877.37008 18171.38223 12556.14660 13296.10827	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm)	A L A I S dz (mm) Numb. Numb. Num. Load 1 2 4 4 5 6 7 8 9 1 0 1 1 1 1 1 1 1 1 	IS./E. R INode INode 1 39 21 401 31 421 51 431 61 1 1 41 71 451 91 41 91 471 101 481 111 491 121	<pre>ukial di ryy ukial N(KN) 131.56 -77.56 80.14 -385.86 677.46 77.46 -626.83 148.65 -127.05 394.78 -373.18 216.71 -195.11 141.31 -119.71 409.39 -387.79 540.58 -518.98 -518.98</pre>	I O N S (rcad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear Q2(NN) -9.091 -9.091 -9.021 2.2 2.2 2.2 0.2 1.11 -0.2 1.11 -0.2 0.2 1.11 -0.0 1.11 -0.0 1.11 -0.0 1.11 -0.0 1.11 -1.0 1.17 -1.7 -2.2 -1.1 -1.1 -1.0 -1.1 -1.1 -1.0 -1.1 -1.2 -2.2 -1.1 -1.1 -1.0 -1.1 -1.0 -2.8 -2.8 -2.8 -2.8 -2.8 -2.8 -2.8 -2.8 -2.8 -1.1 -1.1 -1.1 -1.0 -1.1 -1.0 -1.1 -1.2 -1.1 -1.2 -1.1 -1.2 -2.8 -1.1 -1.2 -2.8 -1.1 -2.8 -1.2 -2.8 -1.2 -2.8	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along Z) kNm2 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) dy (mm) dy (mm) 0.000E+000 0.000E+000 2 402 2.409E-001 -3.65E-00 3 1.919E-001 -4.964E-00 5 2.454E-001 -2.399E-00 6 2.183E-001 -2.399E-00 7 2.454E-001 -3.10E-00 8 2.448E-001 -3.10E-00 9 2.187E-001 -1.158E-00 10 1.919E-001 -4.158E-00 11 1.919E-001 -4.018E-00 12 2.448E-001 -4.158E-00 13 1.919E-001 -4.018E-00 14 2.199E-001 -4.018E-00 15 1.919E-001 -5.743E-00 16 1.2.190E-001 -3.074E-00 21 4.673E-001 -3.074E-00 22 6.622E-001 -4.6132E-00 23 6.622E-001 -3.074E-00 24 1.5.705E-001 -1.5.25E-00 23 6.622E-001 -4.6132E-00 24 1.5.705E-001 -1.5.25E-00 25 4.705E-001 -4.6138E-00	A L A I S dz (mm) Memb. Numb. Num. Load 2 4 5 6 1 5 6 1 8 9 1 0 1	IS./E. Z INode Node <td< td=""><td>Line (1) (1) (2) (2) (2) (2) (2) (2) (2) (2) (2) (2</td><td>I O N S (rad) MEMBERS IN Shear QY(KN) </td><td>rz(rad) TERN.FORCES Shear Q2(NN) -9.091 -9.091 -9.021 2.2 2.2 2.2 1.2 0.2 1.11 -1.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -5.8 1.12 -5.6 -1.12</td><td>S Tors. MX (KNM) -0.02 0.0</td><td> Bend. MY(KNM </td><td> Bend. MZ(KNM) </td><td> K (along Z) kNm2 </td></td<>	Line (1) (1) (2) (2) (2) (2) (2) (2) (2) (2) (2) (2	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear Q2(NN) -9.091 -9.091 -9.021 2.2 2.2 2.2 1.2 0.2 1.11 -1.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -4.0 1.11 -5.8 1.12 -5.6 -1.12	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along Z) kNm2 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm)	A E A I S dz (mm) Memb. Numb. Num. Load 2 4 4 5 6 7 8 9 10 11 12 13	IS./E. R INode 39 21 40 31 42 43 44 7 445 81 46 91 47 10 471 121 481 111 491 121 501 131 511 141	UN (RN) (N (RN) (N (RN) (N (RN) (131.56) (-77.56) (80.14) (-77.56) (80.14) (-385.86) (677.46) (419.61) (-385.86) (677.46) (-485.86) (-677.46) (-485.87) (-197.72) (-197.7	I O N S (rad) MEMBERS IN Shear QY(KN) QY(KN) 	rz(rad) TERN.FORCES Shear Q2(NN) -9.091 -9.091 -2.22 2.22 2.2 0.2 -0.22 0.2 -0.22 0.2 -1.11 -0.02 0.2 -1.12 -0.04 -0.05 -0.05 -0.02 -1.11 -0.04 -0.04 -0.04 -0.04 -0.04 -0.05 -0.05 -0.02 -0.02 -0.04 -0.04 -0.04 -0.04 -0.05	5 Tors. MX (KNM) -0.02 	Bend. MY(KNM 	Bend. M2 (KNM) M2 (KNM) 	K (along Z) kNm2 1 0156.94561 13278.4513 1008.70413 10156.94561 18160.65744 1494.69739 12916.76626 12870.37008 12156.14660 12282.65800 13296.10827 13593.50766 9655.59400 12594.92481 10156.94561 13278.4333	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm) dx (mm) dy (mm) 0.000E+000 0.000E+000 2 4000 0.000E+000 0.000E+000 2 1 9 0.000E+000 0.000E+000 2 1 9 2.409E-001 -3.458E-00 3 1 1.919E-001 -3.130E-00 6 1 2.458E-001 -3.130E-00 7 1 2.446E-001 -1.318E-00	A L A I S dz (mm) Memb. Numb. Num. Load 2 4 5 6 1 5 8 9 1 0 1 1 1 1 1 1 3 1 5	IS./E./R INode	UN 14 1 (U) 14	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear Q2(NN) -9.09 -9.09 -2.2 2.2 2.2 1.3.6 -0.2 0.2 -1.1 -1.0 4.0 4.0 4.0 0.4 -0.6 0.4 -0.6 1.7 -3.3 3.0 -3.3 -3.3 -3.3 -3.3 -3.3 -3.3 -3.0 -5.0	5 Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. M2 (KNM) M2 (KNM) 4 3.98 31 11.43 TIVE STIFFNESS Inge! K (along Y) End kNm2 	K (along Z) kNm2 1 0156.94561 13278.84313 1008.70413 10156.94561 18160.65744 1494.69739 12916.76626 12870.37008 12156.14660 12182.65800 13296.10827 13593.50766 9655.59400 12594.92481 10156.94561 13278.84313 13008.70413	
Node Num D I S P L A C E Numb. L. dx (mm) dy (m)	A L A I S dz (mm) Memb. Numb. Num. Load 2 4 5 6 1 5 8 9 10 11 12 13 15 17	IS./E./R INode Inod	UN 1 A 1 () ry ()	I O N S (rad) MEMBERS IN Shear QY(KN) 3.85] -3.72] -5.41] -5.41] -5.41] -5.41] -0.01]	rz(rad) TERN.FORCES Shear 22(KN) -9.091 -9.091 -2.22 2.2 1.3.6 -0.2 -3.6 -0.2 0.2 -1.1 -4.0 4.0 4.0 -6.8 -3.3 -3.3 -3.3 -3.3 -3.3 -5.2 -6.8 -7.8	5 Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM -312.3 EBAMS A BEAMS A BEAMS A NO N NO	Bend. MZ (KNM) MZ (KNM) 4 3.98 31 11.43 TTIVE STIFFNESS ITIVE STIFFNESS INTE STIFFNESS 0 10156.94561 0 10156.94561 0 12916.76626 0 1294.92481 0 1056.94561 0 13278.4313 0 10156.94561 0 10156.9456	K (along Z) kNm2 10156.94561 13278.84313 13008.70413 13008.70413 10156.94561 18160.65744 1494.69739 12916.76626 12870.37008 18271.38223 12556.14660 12282.65800 13296.10827 13593.50766 9655.559400 12594.92481 10156.94561 13278.84313 13008.70413 10156.94561 1326.744	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn)	A L A I S dz (mm) Memb. Numb. Num. Load 2 4 5 6 1 5 6 1 1 10 11 12 13 15 17 17 1 12 11 12 11 12 11 12 11 111 11 1 11 1 1 11 11 11 11 11 11 11 1 1 1 1 1 1 1 1 	IS./E./A INode Inod	UN 14 1 (N (N) 131.56 -77.56 80.14 -54.83 419.61 -385.86 677.46 -626.83 149.65 -127.05 394.78 -373.18 216.71 -195.11 141.31 -197.11 409.39 -387.79 540.58 476.92 -455.364 105.16 -81.54 856.79 -829.45 -85.45 -85.45 -85.45 -85.45 -455.4	I O N S (rad) MEMBERS IN Shear QY(RN) 	rz(rad) TERN.FORCES Shear 22(KN) -9.09 -9.09 -2.22 2.2 -3.6 -0.22 -3.6 -0.2 0.22 -1.1 -1.1 -4.0 4.0 -4.0 -4.8 -4.8 -4.8 -7.7 -3.3 -3.3 -3.3 -6.8 -7.8	5 Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ (KNM) 	K (along Z) kkm2 10156.94561 13278.84313 13008.70413 10156.94561 18160.65744 1494.69739 12916.76626 12870.37008 18171.38223 12556.14660 13296.10827 13593.50766 9655.59400 12294.92481 10156.94561 13278.84313 13008.70413 10156.94561 13278.70413 10156.94561 13778.78915 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn)	Memb. Numb. Num. Load 	IS./E./A IS./I	O T A T d) ry d) ry uill N(RN) 131.56 -77.561 -80.14 -54.83 419.61 -127.051 394.78 -385.86 677.461 -626.83 148.65 -127.051 -127.051 141.31 -19.71 409.39 -518.98 476.92 -455.324 -455.364 105.161 05.161 -455.364 105.161 -455.464 -354.98 -455.464 -65.47.99 -455.464 105.161 -455.464 105.161 -455.47.47 -455.47.47 -455.47 -455.47 -457.47.81 -278.18 -278.18	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear 22(NN) -9.09 -9.09 -2.22 2.2 -3.66 -0.22 0.22 -1.11 -4.0 0.24 -4.0 0.40 0.44 -6.8 -8.8 -6.8 -8.8 -6.8 -8.8 -8.8 -8.8 -6.8 -8.8 -7.8	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along Z) kWm2 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mn)	A L A I S dz (zm) Memb. Numb. Num. Load 	IS./E.IA IS.IA	O T A T d) ry d) ry usial 1 N(KN) 1 131.56 -77.56 -77.56 80.14 -54.83 419.61 -385.86 677.46 677.46 73.18 216.71 -127.05 -127.057 394.78 -373.18 216.71 -195.11 -119.71 409.39 -387.79 540.58 -518.98 477.26 -455.324 -455.324 105.16 05.16 -1514.98 477.26 -455.364 105.16 -15.49 856.79 -85.47 -278.18 218.50	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear (22(NN) -9.09 -9.09 -2.2 2.2 2.2 -3.6 -0.2 0.2 -1.1 1.1 -4.0 4.0 0.4 -0.4 -0.4 -0.4 -0.4 1.7 -3.3 3.3 1.7 -3.3 -3.6 -6.8 -6.	S Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) -	K (along 2) kklm2 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm)	A L A I S dz (mm) Memb. Numb. Num. Load 	K rx(ra rx(ra IS./E. R IS./E. R I.Node 1	<pre>C T A T d) ry IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII</pre>	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear 22(NN) -9.09 -9.09 -2.2 2.2 2.2 -3.6 -0.2 0.2 -1.1 1.1 -4.0 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.5 -1.7 -3.3 3.3 -3.3 -3.3 -3.3 -3.2 -3.6 -0.2 -1.1 -0.4 -0.4 -0.4 -0.5 -6.8 -8.8 -8.2 -0.2	5 Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along 2) kkm2 	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm)	A L A T S dz (mm) Memb. Numb. Num. Load 	IS./E./R IS./I	<pre>core A fraction of a frac</pre>	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear 22(NN) -9.09 -9.09 -2.2 2.2 -3.6 -0.2 0.2 -1.1 1.1 -1.0 -4.0 0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.4 -0.5 1.7 -1.3 -3.3 3.3 -3.3 -3.3 -2.2 -3.6 -2.2 -3.6 -3.8 -3.6 -3.8 -3.6 -3.8 -3.6	5 Tors. MX (KNM) -0.02 0.0	Bend. MY(KNM -31 BEAMS A BEAMS A Plastic H: Start 1 Start 1 No N No	Bend. MZ(KNM) 	K (along 2) kNm2 10156.94561 13278.84313 13008.70413 10156.94561 18160.65744 14494.69739 12916.76626 12870.37008 13296.10827 13593.50766 9655.59400 12594.92481 10156.94561 13278.84313 10056.70413 10156.94561 13278.84313 10056.70413 10156.94561 13278.84313 10056.70413 10156.94561 13278.84313 1005.70413 10156.94561 13278.84313 1005.70413 10156.94561 13278.84313 1005.70413 10156.94561 13178.7815 12870.37008 11016.19213 11311.87931 1296.10827 13296.10827 1329	
Node Num D I S P L A C E Numb. L. dx (mm) dy (mm)	A L A I S dz (mm) Memb. Numb. Num. Load 	IS./E./R IS./I	<pre>core A f d) ryy list(x) l</pre>	I O N S (rad) MEMBERS IN Shear QY(KN) 	rz(rad) TERN.FORCES Shear 22(NN) -9.09 -9.09 -9.22 2.22 -3.6 -0.22 0.4 -0.2 0.4 -1.1 1.1 1.1 4.0 0.4 -0.4 -0.5 1.7 -1.3 -3.3 -3.2 -3.6 -0.2 -1.1 -3.3 -3.3 -3.3 -3.3 -3.3 -3.6 -4.8 -4.8 -4.8 -4.8 -4.8 -5.8	5 MX (KNM) -0.02 0.02	Bend. MY(KNM 	Bend. MZ(KNM) 	K (along 2) kkm2 10156.94561 13278.84313 13008.70413 10156.94561 18160.65744 14494.69739 12916.76626 12970.37008 12182.68000 12956.14660 12954.92481 10156.94561 13278.84313 1008.70413 10256.94561 13278.84313 1008.70413 10256.94561 13278.84313 1008.70413 10156.94561 12778.7815 1277.8915 1277.3708 1101.619213 1101.87931 1259.4007 1296.0371 1296.1027 1329.4037 129.4037 129.40722 149.97.8857 149.787 149.787 149.787 129.40722 149.787 	



2.3 Seismic Force:

- **§** Seismic Action of Scenarios Of seismic elastic Static analyzes
- § SCENARIO : 0 DATA AND RESULTS OF SEISMIC FORCE
- § DATA FILE LOAD CASES
- **§** RESULTS FILE INTERNAL FORCES
- § SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF THE BUILDING
- § Calculations Parameters
- **§** Fundamental Periods (Proximity Type)
- **§** Height Distribution Of Seismic Force (Shear-Moment)
- **§** Seismic Action of seismic inelastic analyzes
- **§** SCENARIO : 0 DATA AND RESULTS OF SEISMIC FORCE
- § DATA FILE LOAD CASES
- **§** RESULTS FILE INTERNAL FORCES
- **§** SEISMIC ACTION ALONG THE MAIN DIRECTIONS OF THE BUILDING
- **§** Calculations Parameters
- **§** Fundamental Periods (Modal Resp.Spect.analysis)
- **§** Eigenvalues Participation Factors
- **§** Mass Participation Factors / Direction
- § Active Modal Masses
- **§** Acceleration Response Spectrum Matrix Values
- **§** Limit States Elastic Response Spectra

Moreover, through the window *Report*, the button

Internal Forces

Press the command to open the txt file containing:

- NODE DISPLACEMENTS AND ROTATIONS
- MEMBER INTERNAL FORCES
- BEAMS ACTIVE STIFFNESS



<u> </u>									
File Edit View Insert Format Help									
	B								
NODE DISP	LACEMENTS / ROI	TATION	s						
Node Num DISPLACE	MENTS	I	ROTAT	IONS					
Numb. L. dx (mm) dy (mm)	dz(mm)	rx(ra	ad) ry	(rad)	rz(rad)				
	- -								
1 0.000E+000 0.000E+00				MEMBERS IN	TERN FORCE				
2 2.409E-001 -3.605E-00				MEMBERS IN	ILKN.FORCE	1.5			
3 1.919E-001 -4.964E-00	Memb INumb I	S /F I	Avial	Shear I	Shear	Tore	I Band I	Bend	
5 2.454E-001 -2.222E-00	Num Lond H	Node I	N(EN)	OV (ENI)	OZ (EN)	1 MV (PN	A MY (ENM)	MZ (ENM)	
6 2.183E-001 -2.399E-00			IN (I'LN)		¥2 (IUI)			H2 (RNH)	
7 2.448E-001 -1.188E-00	11	391	131 56	3,851	9.09	-0.1	021 -34 041	3 98	
8 2.448E-001 -3.310E-00		21	-77.56	-3,851	-9.09	. 0.	021 -2.331	11.43	
9 2.187E-001 -1.775E-00	2	401	80.14	2.761	-2.2		BEAMS ACT	VE STIFFNESS	
10 1.919E-001 -1.125E-00	-	31	-54.83	-2.761	2.2		DEALS ACT		
11 1.919E-001 -3.436E-00	41	421	419.61	11.341	-3.6	IMember	IPlastic Hind	el K (along V)	K (along 7)
12 2.448E-001 -4.567E-001		51	-385.86	-11.341	3.6	Num.	Start End	kNm2	kNm2
13 1.919E-001 -4.018E-00	51 1	431	677.46	0.501	-0.2				
14 2.199E-001 -3.669E-001		61	-626.83	-0.501	0.2	44		10156.94561	10156.94561
16 2.187E-001 -7.357E-00	6 1	441	148.65	-1.891	-1.1	45	NO NO	13278.84313	13278.84313
18 2.190E-001 -5.743E-00	i i	71	-127.05	1.89	1.1	1 46	NO NO	13008.70413	13008.70413
19 4.873E-001 -7.210E-00	71 1	451	394.78	-2.08	-4.0	47	NO NO	10156.94561	10156,94561
20 6.703E-001 -3.074E-00	l i i	8	-373.18	2.08	4.0	48	NO NO	18160.65744	18160.65744
21 5.693E-001 -3.467E-00	81 1	46	216.71	-0.24	0.4	I 49	NO NO	14494.69739	14494.69739
221 6.682E-001 -1.612E-00	l i i	9	-195.11	0.24	-0.4	50	NO NO	12916.76626	12916.76626
231 6.682E-001 -4.813E-00.	9	471	141.31	1.86	-0.8	i 51	NO NO	12870.37008	12870.37008
24 5.707E-001 -2.414E-00	i i	10	-119.71	-1.86	0.8	53	NO NO	18171.38223	18171.38223
251 4.705E-0011-1.525E-00	10	48	409.39	1.08	-4.8	54	No No	12556.14660	12556.14660
261 1 4.7052-0011-4.9762-00		11	-387.79	-1.08	4.8	55	No No	12182.65800	12182.65800
2/1 1 0.002E-0011-0.030E-00	11	491	540.58	-3.72	1.7	56	No No	13296.10827	13296.10827
281 1 4.7052-0011-5.9912-00	1 1	12	-518.98	3.72	-1.7	57	No No	13593.50766	13593.50766
201 1 5 7165-0011-0 6005-001	12	50	476.92	2.18	3.3	58	No No	9655.59400	9655.59400
311 1.212E+0001-3.457E-000	1 1	13	-455.32	-2.18	-3.3	59	No No	12594.92481	12594.92481
321 9.861E-001 -3.957E-00	13	51	477.26	-5.41	3.0	60	No No	10156.94561	10156.94561
331 1.208E+0001-5.411E-00	- I I	14	-453.64	5.41	-3.0	61	No No	13278.84313	13278.84313
341 I 7.645E-0011-5.567E-00	15	53	105.16	-0.01	-6.8	62	No No	13008.70413	13008.70413
351 1.208E+0001-7.958E-00	I I	16	-81.54	0.01	6.8	63	No No	10156.94561	10156.94561
361 J 7.645E-0011-7.048E-00	17	55	856.79	0.10	-8.8	64	No No	18160.65744	18160.65744
371 9.987E-001 -5.924E-00	I I I	18	-829.45	-0.10	8.8	65	No No	13778.78915	13778.78915
381 9.912E-0011-1.015E+00	18	56	417.39	-7.80	-49.8	66	No No	12870.37008	12870.37008
391 0.000E+000 0.000E+000		19	-278.18	7.80	49.8	67	No No	11106.19213	11106.19213
401 0.000E+000 0.000E+000	19	5	218.50	11.43	-10.2	68	No No	18171.38223	18171.38223
421 0.000E+000 0.000E+000		201	-193.18	-11.43	10.2	69	No No	11301.87931	11301.87931
431 0.000E+000 0.000E+000	20	6	406.21	-4.43	-0.1	70	No No	11549.68037	11549.68037
441 0.000E+000 0.000E+000		21	-368.24	4.43	0.1	71	No No	13296.10827	13296.10827
	21	71	73.59	-7.74	-7.7	72	No No	13593.46722	13593.46722
or Help, press F1		22	-57.39	7.74	7.7	73	No No	14997.88577	14997.88577

ADA Pro CHAPTER 7 "ANALYSIS" tructural Analysis & Design 3. VIEW X Z z X Ŧ x Z Shear Seismic Forces Seismic Forces Po Centers Po - CM Mass Centers Bending Bending Shear Mass Distribution Deviation Stiffness X Stiffness Z Stiffness Z Distribution X Distribution Z Deviation Deviation View "View" command group is used in a different way for Linear and Nonlinear Scenarios. **§** For Linear Analysis For Linear Scenarios: the commands are used so as the user reviews the data in the following

Mass Distribution Mass Centers Deviation Mass Centers Deviation Bending Stiffness X *10-3 Bending Stiffness Z *10-3 Shear Stiffness X *10-3 Shear Stiffness Z *10-3 Seismic Forces Distribution X Seismic Forces Distribution Z

By each selection, the diagram with the corresponding title opens.

drop-down list.

Report	×	Report X	Report X	Report 🗙	Report X	Report X	Report
Κατανομή Μαζών	•	Απόκλιση Κέντρων Μάζας 📃 💌	Καμπτική Ακαμψία Χ *10-3 💌	Διατμητική Ακαμψία Χ *10-3 💌	Κατανομή Σεισμικών Δυνάμεω 💌	Απόκλιση Κέντρων Ρο 💌	Απόκλιση Ρο - Κ.Μ.
18.00	41.56	18.000.00	18.00 59.73	18.00 1555.56	18.00 0 60.63	18.000.24	18.00
15.00	43.91	15.000.00	15.00 59.73	15.00 1555.56	15.00 · · · · · · · · · · · · · · · · · ·	15.000.00	15.00
12.00	43.91	12.000.00	12.00 59.73	12.00 1555.56	12.00 31.58	12.00	12.00
9.00	77.94	9.00 3.77	9.00 89.60	9.00 2333.33	9.00 + 42.05	9.00 1.76	9.001.87 •
6.00	79.12	6.00	6.00 89.60	6.00 2333.33	6.00 28.45	-6.00	-6.001.98
3.00	79.12	3.00	3.00 89.60	3.00 2333.33	3.00 3 14.23	3.00	3.001.98
0.00	M(kN)	0.00	0.00 H(m) KNm	0.00 H(m) kNm	-0.00 H(m) F(kN)	0.00	0.00
18.00	-5.35%	18.00	18.00 0.00%	18.00 0.00%	18.00 - 60.63	18.000.00+	18.000.00
15.00	0.00%	15.000.00	15.000.00%	15.00 0.00%	15.00	15.000.00	15.000.00-
12.00	-43.66%	12.00	-33.33%	12.00	12.00	12.00	12.000.00-
9.00	-1.48%	9.00	9.000.00%	9.00 0.00%	9.00	9.00	9.000.00-
6.00	0.00%	6.000.00	6.00 0.00%	6.000.00%	-6.00 - 202.19	6.00 0.00	-6.000.00-
3.00	0.00%	3.000.00	3.000.00%	3.00 0.00%	3.00 - 216.42	3.000.00	3.000.00
0.00	DM(%)	0.00	0.00 (%)	0.00(%)	0.00 216.42 H(m) Q(kN)	0.00	0.00



For Nonlinear (Pushover) Analysis §

For Nonlinear (Pushover) Scenarios:

Select one of the "View" commands (for example "Mass Distribution"). The depiction of the structure changes and the corresponding dialog box opens:



1 It's a new tool used for the presentation of the capacity curve of all Pushover analyses and the corresponding view of the deformed and undeformed shape of the structure in each step. In the top of the window select one of the previously defined distributions

			Fx+0.30*Fz	~	
Triangular	¥		Fx+0.30*Fz		
			-Fx+0.30*Fz		
Triangular			Fz+0.30*Fx		
Orthogonal		and a load combination	-Fz+0.30*Fx		

The list contains the steps of the selected nonlinear analysis.

	Rep	ort			2
Triangular V	Fx+0.30*Fz	!	¥	Spe	ctrum
Step Vb(kN) (λ)				Para	neters
1. 45.24714 (0.43339)	✓ >:	Succes	ssional Displ	ay of	DL
1. 45.24714 (0.43339)	^	- uncing	souer miges		SD
2. 250.40681 (1.96507)				~	
3. 270.06343 (0.18828)					NC
5 396 80721 (0 21461)					
6 401 95696 (0.04933)		+		_	
7, 450, 70286 (0, 46690)					
8. 460.33169 (0.09223)	0000		_		
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)					
17 018 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)					UX(III)
26. 1310.44560 (0.43971)		9 9	2	12	5
27. 1317.28069 (0.06547)			-	-	-
29. 1319.18336 (0.01451)	ks Cr	eation for Ou	tput		
30. 1324.95/29 (0.05530)	*			_	

Ti O

Each step provides the corresponding shear value Vb(kN) and the λ load factor, while the following data can be shown graphically:

- § **Capacity Curve**
- § Idealized Capacity Curve
- § **Target Displacement**





§ 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 in 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.

Approaching the mouse to the points of the steps the indication of the step's number and the corresponding values of Vb and Ux is displayed.



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

§ 2.Idealized Capacity Curve

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





The button Parameters is used in the definition of the parameters on how to make the capacity curve bilinear. This bilinear curve is necessary in order the slopes of its two branches to be used to calculate the period and the corresponding spectral acceleration.

§ 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.



Approaching the mouse to the points, the values for the three target displacements are displayed, one for each level of performance and the corresponding shear to the check node.



Spectrum is similar to the Spectrum command explained in the section regarding the parameters of the scenario. 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 to the capacity curve and so they can be defined or modified before or after the implementation of the analysis procedure.

In Parameters select the method for the derivation of the capacity curve ("Idealization Method") between the two methods proposed in Annex B of EN 1998-1.

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								

✓ Target Displacement Calculation with Iterative Procedure(EC8 ANNEX B.5) and the program uses the Iterative Activate Procedure proposed in EC8 for the calculation of the target displacement.

Representation of the structure δ

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

1. Select the step from the list



and you will see the corresponding deformed shape of the structure in the selected step and the location of the plastic hinges.

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

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








73



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

- By choosing a type of distribution and seismic combination from the list,

Load Step of Vb (kN) 1. 135.441 (1.29729) \checkmark the steps of this inelastic analysis for each step, the Shear Vb(kN) λ load factor and the corresponding minimum (λ) factor are displayed. The corresponding point is also marked on the capacity curve, in pink
At the bottom side of the window Graph - Checks Creation for Output Internal Forces
Member's Moment - Rotation Diagram
 The selection of
Graph - Checks Creation for Output
 documents, which contain the graphs and the design checks of the project, as well as any updates in case of modifications. Press Internal Forces command to open the txt file containing:
NODE DISPLACEMENTS AND ROTATIONS MEMBER INTERNAL FORCES BEAMS ACTIVE STIFFNESS The selection of the
Member's Moment - Rotation Diagram command
displays the member's moment-rotation diagram per member (start-end) as well as direction. § Member's Moment-Rotation Diagram
Selecting/command and pointing by left clicking a beam or column member, the member's moment-rotation diagram is displayed per member (start-end) as well as direction for the selected distribution. Report
Τριγωνική ~ Fx+0.30*Fz ~
A prerequisite to display the member's moment-rotation diagram is that the checks have already be done, that is, the command
Graph - Checks Creation for Output has been selected





This diagram is based on the following assumptions:

- The calculation of moment My is made according to the relation (A.6) of the Kanepe's annex 7A.
- The value of My is different for each step due to the axial force that is implicated in its calculation.

For this diagram, however, a fixed value was used and is the one derived from the axial force of the vertical loads.

- Two values of My (positive and negative) are calculated and two areas with the (different) limits for the performance levels are designed respectively.
 For columns, due to symmetrical reinforcement, the two values will always be the same.
- As well as it is already known the diagram doesn't have an elastic branch, so only the plastic area is displayed.
- The θ values have been divided by the corresponding safety coefficients. The ultimate θ_{Pl} limits corresponding to the respective performance levels have been divided by the safety coefficient γ_{rd} =1.8, while the angles of rotation θ_{sd} have been multiplied by the coefficient.





This is done to make sure that compatibility with the corresponding print results has been

The diagram shows the angle of rotation of the plastic joint (requirement) for the three steps of the analysis corresponding to the three levels of performance:

A: blue B: orange C: red

The values are displayed, depending on the sign of the angle, in the corresponding area.

In the dialog box that appears:

The corresponding diagram for each edge (start-end) is displayed.







The direction is selected through the corresponding direction field

Especially for beams, the predetermined direction is the principal direction z assuming however that the angle of rotation of the plastic joint is the worst magnitude of both directions.

Two colored areas appear, one for the positive and one of the negative values of the axis, where the **blue** represents the **B** performance level, while the **brown** one represents the **C**.

The **black** values are the **limits** for each level of performance.

In the diagram, they appear as an integer. But in the bottom left for the negative and in the upper left for the positive they are written with their decimal digits.

The colors appearing in the circles at the ends of each member in the three-dimensional structure depend on the corresponding angle of rotation of the plastic joint.

More specifically:





The **blue** color means that the respective blue line is inside the blue area, that is, the limit A (which is value 0) is exceeded, but both this and the other two values have not exceeded the boundary of the B (blue area).





The yellow color means that the respective value (orange line) has entered the brown area and the respective red line has not gone out of the brown area.



Finally, the red color means that the respective red value has gone out of the brown area.



All of the above are valid as long as the structure is on the step that corresponds to the performance level C, so as all of the above are applied.

Also, the ductility indices regarding angle of rotation $\mu\theta$ for each level of performance are also given. First, the required is indicated and then the one available is written in brackets. The values are displayed with red color if the first value is bigger than the second one. For the first performance level $\mu\theta$ A=1.



§ For Time History Analysis

	EC8_General Time History Linear (2 🔻
"View" commands activated	' Active Scenario

After completing the analysis, press a command from the "View" menu that opens the following window:



where the user can choose the direction of the seismic action (X, Y, Z or XYZ) and the scale that the result of the analysis will be visualized. Type the number of the node to see the response.

The graph of the response of the selected node over time with the maximum and minimum value is automatically displayed.

At the same time in the upper window, the selected Accelerogram of the selected seismic action is displayed.

Finally, it is possible to view the deformed state of the structure for each time step of the analysis. For this purpose, the model appears in the following three-dimensional representation, where the undeformed shape is presented in parallel with the movement of the deformed shape.





