

User's Manual 9.MEMBERS DESIGN







CONTENTS

Ι.	THE	NEW UPGRADED INTERFACE OF SCADA PRO	. 5
11.	DETA	ILED DESCRIPTION OF THE NEW INTERFACE	.6
	Member	s Design	6
	1. Sc	ENARIOS	6
	1.1	New	7
	1.2	LIST	7
	1.3	PARAMETERS	8
	1.3.1	COMBINATIONS	9
	1.3.2	SLABS	15
	ş	More combinations for the slab's design	17
	1.3.3	BEAMS	19
	1.3.4	COLUMNS	21
	1.3.5	FOOTINGS	25
	1.3.6	STEEL REINFORCEMENT	26
	1.3.6.1	SLABS	26
	1.3.6.2	COLUMNS-WALLS	27
	1.3.6.3	BEAMS/FOOTING BEAMS	28
	1.3.6.4	Strip Footing	32
	1.3.6.5	FOOTINGS	33
	1.3.7	CAPACITY DESIGN	33
	1.3.8	STEEL	34
	1.3.9	TIMBER STRUCTURES	41
	Load	Duration Class	43
	1.4	Merge Elements	44
	1.4.1	AUTO MERGE	48
	1.4.2	USERS MERGING	51
	1.4.2.1	Merge concrete columns	51
	1.4.3	VIEW	52
	1.4.4	CORRECTION	53
	1.4.5	SINGLE DELETE	54
	1.4.6	OVERALL DELETE	54
	2. Be	AMS	55
	2.1	CONTINUITY OF BEAMS	55
	2.1.1	MERGE BEAMS / MERGE BEAMS DELETE	55
	2.1.2	SINGLE CONTINUITY/ OVERALL CONTINUITY OF BEAMS	56
	2.1.3	SINGLE/OVERALL DELETION OF CONTINUITY	57
	2.1.4	PREFERENCES OF BEAMS REINFORCEMENT	58
	ş	Mouse right button function	59
	2.2	CHECK REINFORCEMENT	60
	2.2.1	SINGLE	60
	2.2.2	Overall	60
	2.2.3	RESULTS	61
	2.2.3.1	Editor	61
	2.2.3.2	Results	61
	2.2.3.3	EXPLORATION	63
	2.2.3.4	REINFORCEMENT DETAILING	63
	3. CA	PACITY DESIGN	64



3.1	NODE RELEASES	.64
3.1.1	COLUMN	.64
3.1.2	WALLS	.65
3.2	DESIGN	.65
3.2.1	Single	.66
3.2.2	Overall	.66
3.2.3	RESULTS	.66
3.2.4	EXPLORATION	.66
4. C	OLUMNS	.66
4.1	BUCKLING	.67
4.2	CHECK REINFORCEMENT	.67
4.2.1	Single:	.67
4.2.2	Overall	.67
4.2.3	Single Delete	.68
4.2.4	OVERALL DELETE	.68
4.3	RESULTS	.68
4.3.1	Editor	.69
4.3.2	RESULTS	.69
4.3.3	EXPLORATION	.70
4.3.4	DETAILING	.71
4.3.5	STRENGTH CALCULATION (PUSHOVER)	.72
436	CHECK-NODES	73
5 F		74
5.1	CHECK REINFORCEMENT	74
511	Single	74
512		7/
512		75
5.1.5		.75
5.1.4		.75
J.Z		.75
5.2.1	EDITOR	
5.2.2		
5.2.3	EXPLORATION	
6. SI		
6.1	SLAB STRIP CALCULATION	
6.1.1	SINGLE	. 78
6.1.2		. 78
6.1.3	SINGLE (LOAD PATTERN)	. 78
6.1.4	Overall (Load Pattern)	.78
6.2	FLAT SLABS	.79
6.2.1	FLAT SLABS	.79
6.2.1.1	PARAMETERS	.80
ş	Instructions for inserting the support lines in flat slabs	.82
6.2.1.2	CALCULATION OF DESIGN STRIPS	.84
6.2.1.3	Display X, Z	.84
6.2.1.4	DIAGRAMS X, Z	.85
6.2.1.5	RESULTS	.85
6.2.2	PUNCHING SHEAR CHECKS	.87
6.2.2.1	Ву Ріск	.88
6.2.2.2	OVERALL	.96
6.2.2.3	Edit	.96
6.2.2.4	User снеск	.96
6.2.3	COMPOSITE SLABS	.97
6.3	RESULTS	.97



6.3.1	EDITOR	97
6.3.2	SLAB STRIPS	99
6.3.3	M DIAGRAMS	
6.3.4	Q DIAGRAMS	101
6.3.5	M DIAGRAMS (LOAD PATTERN)	101
6.3.6	Q DIAGRAMS (LOAD PATTERN)	101
7. S [.]	TEEL	
7.1.1	CROSS SECTION DESIGN	102
7.1.2	BUCKLING MEMBERS INPUT	110
7.1.3	COLD FORMED SECTIONS	120
7.1.4	CONNECTIONS	121
7.2	TIMBER DESIGN	127
7.2.1	CROSS SECTION DESIGN	127
7.2.2	BUCKLING MEMBERS INPUT	128
7.2.3	CONNECTIONS	131
8. N	ASONRY DESIGN	147
8.1	New masonry building (EC6)	149
8.2	Assessment (EC8-3)	155
8.2.1	MASONRY REINFORCEMENT	157
8.2.1.1	REINFORCEMENT WITH CONCRETE JACKET	158
ş	WALLS WITH CONCRETE JACKET – NOTES:	161
8.2.1.2	Textile Reinforced Mortar (TRM)	161
9. N	1ember Diagrams	164
9.2	Member Diagrams	164
9.3	Nodes Displacements	166
9.4	PLATE ELEMENTS INTERNAL FORCES	167
9.5	SLAB STRIPS DIAGRAMS	168





5



II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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

Basic	Modeling	View	Tools	Slabs	Loads	Analysis	Post-Processor	Men	bers Design	Drawin	gs-Detaili	ng	Addon	s O	ptimizati	ion
	Mo	mhor		ian												
	IVIC	mber	Des	igii												
	Basic Modeling	View To	ols Slabs	Loads	Analysis	Post-Processor	Members Design	Drawings-De	tailing Addons	Optimiza	ion					
1	KOS 2000-EAK 1 (C 🔻	7. 🤹		摄		4	1 4		L .	*		8			44	1
New '	Active Scenario	Para- Merge meters Elements	Continuity of Beams	Check - Reinforcemen	Results	Node Design Releases	Buckling Check - Reinforcem	Results	Check R Reinforcement *	esults Slab Calcul	Strip Fla ation * slabs	Results	Steel Design *	Timber Design *	Masonry Design *	Member Diagrams *
	Scenarios			Beams		Capacity Design	Columns		Footings		Slabs - Me	sh	Steel	- Timber		

The 9th Unit entitled "Members Design" contains the following eight groups of commands:

- 1. Scenarios
- 2. Beams
- 3. Capacity Design
- 4. Columns
- 5. Footings
- 6. Slabs Mesh
- 7. Steel
- 8. Timber
- 9. Masonry Design 2D Diagrams

A Since model analysis has been completed, the design checks of the structural elements are applied according to the design code provisions, defined in the tab "Member Design". The reinforcement of the structural elements is calculated according to the design checks.

1. Scenarios



The "Scenarios" command group contains the commands for the creation of a new scenario as well as the editing of the parameters of the design checks and reinforcement in every type of structural elements.

Furthermore, a new group of commands is added, concerning merging of elements (steel and timber)



1.1 New New This command is used to create a new scenario. Type a name, select the corresponding New design regulation and then press the button Scenario \times 1 Name Type EKOS 2000-EAK EKOS 2000-EAK New EC2-EC3 NTC_2008 Design De EC2_Italia EC2_Cyprus Concre Greek old 1959-84 Greek old 1984-93 Steel Austria Exit SBC304-306 EC5 EC6-EC8(3)

NOTES:

The selection of the design code corresponds to the design checks of the structural elements and the calculation of the steel reinforcement. Since you choose analysis' scenario and load combinations according to Eurocodes (see Ribbon "Analysis"), you must create the Eurocode scenarios for the design checks, too.

EC2-W/O EC8

To modify an existing scenario press the button "Update"/.

Design Delete	
Concrete	
Steel	Apply

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

1.2 List

The drop-down list includes all created scenarios. Since you select one scenario, it becomes active. This means that the scenario will be used for the design checks.





1.3 Parameters

This command is used for the definition of all design check parameters:

Pa	ra- ters						
Stru	uctural Component Pa	rameters					×
	Steel Reinforcement	Capacit	y Design	Steel	Tir	mber structures	
	Combinations	Slabs	Beams	Colum	ns	Footings	

NOTE

The parameters dialog box in the new SCADA Pro version contains two more commands for saving and reading the design parameters of the active scenario.

Steel Reinforce	ment	Сара	city Lesian		Steel	Timber s	structu	res
Combinations		Slabs	Bea	ms	Columns	Footings		
Combinations of L	oad Sets	(101	.) Ult.	Serv.	+XX	+Z -	-Z	No
Combinations						ULS/SLS	Dir.	^
1(14) +1.35Lc1-	+1.50Lc2					ULS		
2(1) +1.00Lc1+	0.50Lc2					ULS		
3(2) +1.00Lc1+	0.30Lc2+1	1.00Lc3+0.	30Lc4+1.00	Lc5+0.3	30Lc7+0.30Lc9	ULS	+X	
4(2) +1.00Lc1+	0.30Lc2+1	1.00Lc3+0.	30Lc4+1.00	Lc5+0.3	30Lc70.30Lc9	ULS	+X	
5(2) +1.00Lc1+	0.30Lc2+1	1.00Lc30.	30Lc4+1.00	Lc50.3	30Lc7+0.30Lc9	ULS	+X	
6(2) +1.00Lc1+	0.30Lc2+1	1.00Lc30.	30Lc4+1.00	Lc50.3	30Lc70.30Lc9	ULS	+X	
7(2) +1.00Lc1+0.30Lc2+1.00Lc3+0.30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 ULS								
/(2) +1.00Lc1+	0.30Lc21	1.00Lc3+0.	30Lc41.00	Lc5+0.3	30Lc7+0.30Lc9	ULS	X	
7(2) +1.00Lc1+ 8(2) +1.00Lc1+	0.30Lc2: 0.30Lc2:	1.00Lc3+0. 1.00Lc3+0.	30Lc41.00 30Lc41.00	Lc5+0.3 Lc5+0.3	30Lc7+0.30Lc9 30Lc70.30Lc9	ULS	X	
/(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+	0.30Lc2: 0.30Lc2: 0.30Lc2:	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30.	30Lc41.00 30Lc41.00 30Lc41.00)Lc5+0.3)Lc5+0.3)Lc50.3	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9	ULS ULS ULS	X X X	_
7(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1+	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30	30Lc41.00 30Lc41.00 30Lc41.00).30Lc41.0)Lc5+0.3)Lc5+0.3)Lc50.3)OLc50	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30	ULS ULS ULS ULS	X X X	*
7(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1- «	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30	30Lc41.00 30Lc41.00 30Lc41.00).30Lc41.0)Lc5+0.3)Lc5+0.3)Lc50.3)OLc50	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30	ULS ULS ULS ULS	X X X	~
7(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1+ « evel Multipliers	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1	30Lc41.00 30Lc41.00 30Lc41.00 30Lc41.00 .30Lc41.0)Lc5+0.3)Lc5+0.3)Lc50.3)0Lc50	30Lc7+0.30Lc9 30Lc7-0.30Lc9 30Lc7+0.30Lc9 .30Lc7-0.30	ULS ULS ULS ULS	X X X X	•
7(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1- < evel Multipliers Level	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2 X	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1 Y	30Lc41.00 30Lc41.00 0.30Lc41.00 1.30Lc41.00 / (1-0) Z	0Lc5+0.3 0Lc5+0.3 0Lc50.3 00Lc50	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30 .30Lc70.30	ULS ULS ULS ULS	X X X	*
/(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1+ (≤ evel Multipliers Level 0 - 0.00	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2 X 1.000	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1 Y 1.00L 1 1.00L 1	30Lc41.00 30Lc41.00 30Lc41.00).30Lc41.00 / (1-0) Z 1.000	0Lc5+0.3 0Lc5+0.3 0Lc50.3 00Lc50.	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30 .30Lc70.30 Insert Combinatic	ULS ULS ULS ULS ombinations	X X X X	· · ·
7(2) +1.00cc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1- evel Multipliers Level 0 - 0.00 1 - 400.00	0.30Lc2: 0.30Lc2: +0.30Lc2: +0.30Lc2: X 1.000 1.000	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1.00Lc30 1.00Lc30 1.00Lc30 1.000	30Lc41.00 30Lc41.00 30Lc41.00).30Lc41.00 / (1-θ) Z 1.000 1.000	DLc5+0.3 DLc5+0.3 DLc50.3 DLc50.3	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30 Insert Combinatio	ULS ULS ULS ULS ombinations	X X X X	~
7(2) +1.00cc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1+ evel Multipliers Level 0 - 0.00 1 - 400.00 2 - 700.00	0.30Lc2: 0.30Lc2: 0.30Lc2: +0.30Lc2: +0.30Lc2: 1.000 1.000	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1 V 1.000 1.000 1.000	30Lc41.00 30Lc41.00 30Lc41.00 .30Lc41.00 .30Lc41.00 .30Lc41.00 .30Lc41.00 .30Lc41.00 1.000 1.000)Lc5+0.3)Lc5+0.3)Lc50.3)0Lc50	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30 Insert C Combinatio	ULS ULS ULS ombinations	X X X >	~
<pre>/(2) +1.00Lc1+ 8(2) +1.00Lc1+ 9(2) +1.00Lc1+ 10(2) +1.00Lc1+ < evel Multipliers Level 0 - 0.00 1 - 400.00 2 - 700.00 3 - 1000.00</pre>	0.30Lc2 0.30Lc2 0.30Lc2 +0.30Lc2 × 1.000 1.000 1.000 1.000	1.00Lc3+0. 1.00Lc3+0. 1.00Lc30. -1.00Lc30 1.00Lc30 1.00Lc30 1.00Lc30 1.000 1.000 1.000 1.000	30Lc41.00 30Lc41.00 30Lc41.00 .30Lc41.00	DLC5+0.3 DLC5+0.3 DLC50.3 DDLC50.3	30Lc7+0.30Lc9 30Lc70.30Lc9 30Lc7+0.30Lc9 .30Lc70.30 Insert C Combination G	ULS ULS ULS ombinations ns Calculati	X X X >	~

Once you configure the dimensioning parameters, you can now save them to a file to use them in your next projects.

Press "Save" and type a name



	. This BC BOOTCAMB (C)						
⇒ * ↑	This PC > BOUICAMP (C:) > mel	etes > Idokos > scaanal >		~ O S	earch scaanal		P
organize 🔻 Ne	w folder					800 -	0
ConeDrive	^ Name	Date modified	Туре	Size			
This DC	Scen000	10/5/2017 1:18 µµ	File folder				
THISPC	Scen002	10/5/2017 1:43 μμ	File folder				
Desktop	🛓 sbc.sdp	10/5/2017 3:53 μμ	VLC media file (.s	96 KB			
Documents							
- Downloads							
Music							
Pictures							
Videos							
BOOTCAMP (C:)						
New Volume	D						
INTENSO (F:)							
INTENSO (F:)							
File name:	test.sdp						- 7
Save as type:	Design Parameter(*.sdp)						8
				-		-	

The file extension is sdp scenery design parameters. Use "Load" command to apply the parameters that are already saved.



ATTENTION

A precondition for loading a parameter file is that the current design scenario is the same as the scenario of the parameters. Otherwise, you will see the following message:

ERROR		×
8	The parameter file is incompatible with the current design scenario.	
	ОК	

1.3.1 Combinations

Regardless of the material, the calculation of combinations is a condition for designing.
Combinations

The selection of the existing .cmb combinations file is made:



		~		
	default.cmb EC-8_Greek Dynamic (2).c EC-8_Greek Dynamic (3).c	:mb :mb		
-from the dropdown list	EC-8_Greek Static (2).cmb		with automatic calc	ulation or
- Through the command	Insert Combi	nations	that opens the fold	der with the
		Combinati	ons Calculation	

registered .cmb files. Select the file and press

		Structur	al Co	mpo	nent Pa	aramet	ters				
Capacity	Design			Stee			٦	limber s	tructure	es	
Combinations	Slabs	Bea	Beams		Columns		Footings		Steel Reinforce		
Combinations of L	oad Sets	(10	01)	Ult.	Serv.	+X	X	+Z	Z	١	٥V
Combinations								UL	S/SLS	Dir.	^
1(14) +1.35Lc1+1.	50Lc2							UL	S		
2(1)+1.00Lc1+0.5	0Lc2							UL	S		
3(2) +1.00Lc1+0.3	0Lc2+1.00L	.c3+0.30Lc4	4+1.00L	c5+0.30)Lc7+0.30	Lc9		UL	S	+X	
4(2) +1.00Lc1+0.3	0Lc2+1.00L	.c3+0.30Lc4	4+1.00L	c5+0.30)Lc70.30	Lc9		UL	S	+X	
5(2) +1.00Lc1+0.3	0Lc2+1.00L	.c30.30Lc	4+1.00L	c50.3	0Lc7+0.30	Lc9		UL	S	+X	
6(2) +1.00Lc1+0.3	0Lc2+1.00L	.c30.30Lc	4+1.00L	c50.3	0Lc70.30)Lc9		UL	S	+X	_
7(2) +1.00Lc1+0.3	0Lc21.00L	.c3+0.30Lc	41.00L	.c5+0.3	0Lc7+0.30	Lc9		UL	S	X	_
8(2) +1.00Lc1+0.3	0Lc21.00L	.c3+0.30Lc	41.00L	.c5+0.3	0Lc70.30)Lc9		UL	S -	X	_
9(2) +1.00Lc1+0.3	0Lc21.00L	_c30.30Lc	41.00L	.c50.3	0Lc7+0.30)Lc9		UL	S	X	
10(2) +1.00Lc1+0.	30Lc21.00)Lc30.30L	.c41.00	Lc50.	30Lc /0.3	30Lc9		UL	5	X	\sim
<	SUL ADT U	I CATU AND	6/1±1110	CETU	AUL ANTU A				<u> </u>	>	
Level Multipliers			1/(1-6))	EC-8	_Greek 1	ime His	story Dy	namic ((3).cm	nt 🗸
Level	Х	γ	Z		defau	ilt.cmb _Greek T	ime His	story Dy	namic (3).cm	ıb
0 - 0.00	1.000	1.000	1.00	0	EC-8 EC8_	_Greek T General	ime His Time-H	tory Lin	ear (0). ynamis	cmb che A	Analy
1 - 300.00	1.000	1.000	1.00	0	Seisr	nic E.A.K	(Static). En	(0).cmb d Calc			
2 - 600.00	1.000	1.000	1.00	0		Comb	ination (G+ψ2Q	101		
3 - 900.00	1.000	1.000	1.00	0			Au	tomatic	Design	1	
4 10000	1 000	1 000	1.00	•	•						
								01/		~	

NOTES:

Depending on the case and the fulfilled conditions, you can use either the static or dynamic combination for design. You can also select combinations from different analysis scenarios to check the deviations, on the designing members, between them.

In "Combinations" tab the combinations list is displayed. The first number is the load combination's serial number.

The column "ULS/SLS" indicates the limit state of the combination and the column "Dir." indicates the direction of the participation for the specific capacity design combination.



By using the following bar, you can modify both the limit state and the direction by pressing the corresponding button.

In the column "ULS/SLS" that indicates the limit state of the combination, in case you want to change the status of the combination, first select it and then press the respective button Ult. Serv.

In the column "Dir." That indicates the direction of the participation for the specific capacity design combination, by selecting the corresponding button $+\times$, $-\times$, +Z, -Z you can define the direction of the participation. The label "No" means that the specific combination is excluded from the capacity design.

NOTES:

The regulation concerns the capacity design and its necessity for execution, as long as it is applied per earthquake direction and not per column direction.

Therefore, the exclusion of one direction from the capacity design check for one or more columns is implemented in SCADA, by setting a zero value to the incremental coefficient acd for the seismic combinations in which the seismic force, along with the particular direction, participates with a unit. The characterization of the combinations, which appears in members' design, has that meaning too.

Strue	ctural Compor	nent Parar	meters						×
	Steel Reinforce Combinations	ement	Capa Slabs	icity Desig B	in Jeams	Steel Columns	Timber s	tructur Footing	es 1s
c	ombinations of I	Load Sets	(10)	1) Ult.	Serv.	+XX	+Z	ZN	lo
	Combinations						ULS/SLS	Dir.	^
	1(14) +1.35Lc1	+1.50Lc2					ULS		
	2(1) +1.00Lc1+	-0.50Lc2					ULS		
	3(2) +1.00Lc1+	+0.30Lc2+	1.00Lc3+0.	30Lc4+1	.00Lc5+0.	30Lc7+0.30Lc9	ULS	+X	
	4(2) +1.00Lc1+	-0.30Lc2+	1.00Lc3+0.	30Lc4+1	.00Lc5+0.	30Lc70.30Lc9	ULS	+X	
	5(2) +1.00Lc1+	-0.30Lc2+	1.00Lc30.	30Lc4+1	.00Lc50.	30Lc7+0.30Lc9	ULS	+X	
	6(2) +1.00Lc1+	+0.30Lc2+	1.00Lc30.	30Lc4+1	.00Lc50.	30Lc70.30Lc9	ULS	+X	
	7(2) +1.00Lc1+	-0.30Lc2	1.00Lc3+0.	30Lc41	.00Lc5+0.	30Lc7+0.30Lc9	ULS	X	
	8(2) +1.00Lc1+	-0.30Lc2	1.00Lc3+0.	30Lc41	.00Lc5+0.	30Lc70.30Lc9	ULS	X	
	9(2) +1.00Lc1+	-0.30Lc2	1.00Lc30.	30Lc41	.00Lc50.	30Lc7+0.30Lc9	ULS	X	
	10(2) +1.00Lc1	+0.30Lc2-	1.00Lc3(0.30Lc4	1.00Lc50	.30Lc70.30	ULS	X	*
Le	evel Multipliers		1	/ (1-0)					\sim
	level	X	Y	7		Insert Co	mbinations		
	0 - 0 00	1.000	1.000	1.000		Combination	ns Calculatio	on	
	0 - 0.00	1.000	1.000	1.000					_
	1 - 300.00	1.000	1.000	1.000	- I		_		_
	2 - 600.00	1.000	1.000	1.000		Combination G-	+ψ2Q 1	01	
	3 - 900.00	1.000	1.000	1.000		Automa	tic Design		
	4 - 1200.00	1.000	1.000	1.000		Pecalculate K		luee	
	4500.00	4.000	4 000	4 000	*	Recording to R		aca	
	Save		Load				ОК	Car	ncel



A combination is defined per x or z if the corresponding seismic force has a unit coefficient.

In conclusion, we would assume that in case we want to exclude one direction from one column to avoid the capacity design check, we move to the definition and choose the direction of the local axis that is parallel to the direction of the earthquake we want to exclude.

In case that either the column or the fictitious axes are twisted, we choose the local axis with the smallest angle from the corresponding seismic axis we want to exclude. In this way, the program will calculate the acd just for the particular seismic direction (apparently also for the two local axes of the column), while it will not calculate the acd for the seismic combinations of the other directions.

Node	= 15							
Col.	bottom =	14						
COMB.	SMRby	SMEby	acdy	acdy	SMRbz	SMEbz	acdz	acdz
			calc				calc	
3	134 000	15.876	10 973	4 000	134 000	2 907	59 929	4 000
4	134 000	15 876	10 973	4 000	134 000	2 907	59 929	4 000
5	134.000	15.569	11,189	4.000	144.800	4,605	40.880	4.000
6	134.000	15,569	11,189	4.000	144.800	4,605	40.880	4.000
7	144.800	15,569	12.091	4.000	134.000	4,605	37.831	4.000
8	144.800	15,569	12.091	4.000	134.000	4,605	37.831	4.000
9	144.800	15.876	11.857	4.000	144.800	2,907	64.759	4.000
10	144.800	15.876	11.857	4.000	144.800	2,907	64.759	4.000
11	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.000
12	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.000
13	134.000	15.876	10.973	4.000	144.800	5.114	36.808	4.000
14	134.000	15.876	10.973	4.000	144.800	5.114	36.808	4.000
15	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.000
16	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.000
17	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.000
18	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.000
19	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.000
20	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.000
21	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.000
22	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.000
23	144.800	14.547	12.941	4.000	134.000	2.907	59.929	4.000
24	144.800	14.547	12.941	4.000	134.000	2,907	59.929	4.000
25	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.000
26	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.000
27	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.000
28	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.000
29	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.000
30	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.000
31	144.800	14.853	12.673	4.000	134.000	3.416	50.993	4.000
22	144.800	14.000	12.073	4.000	144 800	5.114	36 000	4.000
24	144.800	14.547	12.941	4.000	144.800	5.114	36.000	4.000
35	134.000	5.228	33,320	0.000	134.000	12.264	14.204	0.000
36	134.000	5.228	33.320	0.000	134.000	12.264	14.204	0.000
37	144.800	4.205	44.761	0.000	134.000	12.774	13.637	0.000
38	144.800	4.205	44.761	0.000	134.000	12.774	13.637	0.000
39	134.000	4.205	41.422	0.000	144.800	12.774	14.736	0.000
40	134.000	4.205	41.422	0.000	144.800	12.774	14.736	0.000
41	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.000
42	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.000
43	134.000	4.921	35.397	0.000	134.000	12.774	13.637	0.000
44	134.000	4.921	35.397	0.000	134.000	12.774	13.637	0.000
45	144.800	3.899	48.283	0.000	134.000	12.264	14.204	0.000
46	144.800	3.899	48.283	0.000	134.000	12.264	14.204	0.000
47	134.000	3.899	44.682	0.000	144.800	12.264	15.349	0.000
48	134.000	3.899	44.682	0.000	144.800	12.264	15.349	0.000
49	144.800	4.921	38.250	0.000	144.800	12.774	14.736	0.000
5.0	144 000	4 661	00.050	0.000	144 000	10.004	14 004	0.000

Indicatevely, in the following printout:



You can see that the acd have been calculated for combinations till the 34^{th} one (combinations +x and -x), while post the 35^{th} one the acd have not been calculated (combinations +z and -z)) Another way in order not to do the capacity design check in one direction is to modify the combinations' definition in members' design through the above tools.

Combinations	Slabs	Beams	Columns	F	ootings
Combinations of Load S	Sets (101)	Ult. Serv.	+XX	+ZZ	. No
Combinations				ULS/SLS	Dir. ^
1(14) +1.35Lc1+1.50	Lc2			ULS	
2(1) +1.00Lc1+0.50L	c2			ULS	
3(2) +1.00Lc1+0.30L	c2+1.00Lc3+0.30	Lc4+1.00Lc5+0.3	0Lc7+0.30Lc9	ULS	+X
4(2) +1.00Lc1+0.30L	c2+1.00Lc3+0.30	Lc4+1.00Lc5+0.3	0Lc70.30Lc9	ULS	+X
5(2) +1.00Lc1+0.30L	c2+1.00Lc30.30	Lc4+1.00Lc50.3	0Lc7+0.30Lc9	ULS	+X
6(2) +1.00Lc1+0.30L	c2+1.00Lc30.30	Lc4+1.00Lc50.3	0Lc70.30Lc9	ULS	+X
7(2) +1.00Lc1+0.30L	c21.00Lc3+0.30	Lc41.00Lc5+0.3	0Lc7+0.30Lc9	ULS	X
8(2) +1.00Lc1+0.30L	c21.00Lc3+0.30	Lc41.00Lc5+0.3	0Lc70.30Lc9	ULS	X
9(2) +1.00Lc1+0.30L	c21.00Lc30.30	Lc41.00Lc50.3	0Lc7+0.30Lc9	ULS	X
10(2) +1.00Lc1+0.30	Lc21.00Lc30.3	0Lc41.00Lc50.	30Lc70.30	ULS	x 🗸
<					>

You can also select one or more combinations, depending on the seismic direction and to characterize it by "No". In this way, they are not going to be taken into account in the capacity design check.

Finally, the last way in order not to do the capacity design check in one or more directions is to set the acd=0 limit in the field Structural Component Parameters. The same result will be accomplished.

Combinations	Slabs	Beams	Column	ns	Footings
Steel Reinforcement	Capacit	ty Design	Steel	Т	imber structures
Direction x = a Edge 3 Middle 3 Fixed 1 Free 3	icd <= .5 .5 .35 .5		Direction z Edge Middle Fixed Free	= 7 7 7	acd <= 0 0 0
		I			
			Z		

Level	х	Y	Z
0 - 0.00	1.000	1.000	1.000
1 - 400.00	1.000	1.000	1.000
2 - 700.00	1.000	1.000	1.000
3 - 1000.00	1.000	1.000	1.000
4 - 1300.00	1.000	1.000	1.000
5 - 1600.00	1.000	1.000	1.000



"Level Multipliers": In this field, you can increase or decrease the seismic actions in any direction and level, by typing different factors.

Press the button $1/(1-\theta)$ to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, in which 0.1 < θ <0.2.

ATTENTION:

For modification purposes, press the following button

Combinations Calculation

▲ The following field/concerns only the Greek EKOS.

Automatic Design:

Automatic Design

This command offers the possibility for an automatic application of the appropriate design checks and the automatic designing of all structural elements, just by pressing the corresponding button. Set the parameters in the various tabs and then press the button "Automatic Design" or follow step by step the procedure to design the structural elements concerning the fulfillment of the design checks.



1.3.2 Slabs

	Capa	citv De	sian				Steel				٦	imber str	uctures				
Combi	nations	1	Slabs		Beam	ns	Colu	umns	F	ooting	gs	Steel	Reinforcem	ent			
Co	ncrete :	C20/2	5		Ste	el (Ma	ain) ·BE	000			Ste	el (Stirrup	s) ·B5000				
Cheeler		52012	-		0.6	(ivid					018	o. (ounup	0,.00000				
Load	, combin	ations	for strip	calcu	ation												
LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL							
LC1	1.35									0							
LC2	1.50									1							
She	ar																
✓	Check	c															
Serv	viceabi	lity															
~	Crack	contro	I	С	ack wi	dth (mr	n)	0	.3								
	Defler	rtion or	ntrol		lal -												
	Delle	Jaon Co	muor	[I	raj a												
Die	agrams	scale 1	m =		5			(kN / I	(Nm)								
Die	granie	ioulo i						(
										_			1				
												014	-				
												ОК	Canc	el			
												ОК	Canc	el			
					6	porato	. C20/	25		Gtor	J (M-		Canc	el	(Stimus	-) ·DE00	<u>_</u>
sthe	follo	wing	; but	tons	Co	ncrete	: C20//	25		Stee	el (Ma	ОК in) :B5000	Canc	el Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; buti al of	tons the s	Co labs:	ncrete	:C20//	25 e and	the	Stee stee	el (Ma el re	ок in) :B5000 bar's ty	Canc	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; buti al of	tons the s	Co labs:	ncrete	: C20//	25 e and	the	Stee	el (Ma el re	ок in) :B5000 bar's tv	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; buti al of	tons the s	Co labs:	ncrete	: C20//	25 e and	the	Stee stee	el (Ma el re	ок in) :85000 bar's tv	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; butt al of	tons the s	Co labs:	ncrete	:C20//	25 e and	the	Stee stee	el (Ma el re	ок in) :B5000 bar's tv	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; buti al of	tons the s	Co labs:	ncrete : con	:C20//	25 e and	the	Stee	el (Ma	ок in) :B5000 bar's tr	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; butt al of	tons the s	Co labs:	ncrete CON	:C20//	25 e and	the	Stee	el (Ma el re	ок in) :В5000 bar's tv	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; buti al of	tons the s	Co labs:	ncrete : con	:C20/2	25 e and	the	Stee	ا (Ma	ок in) :В5000 bar's t	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing ateria	; butt al of	tons the s	Co labs:	ncrete	:C20/	25 e and	the	Stee	। (Ma	ок in) :B5000 bar's tr	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; buti al of	tons the s	Co labs:	ncrete : con	:C20/3	25 e and	the	Stee	el (Ma	ок in) :В5000 bar's tr	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; butt	cons the s	<u>Co</u> labs:	ncrete	:C20//	25 e and	the	Stee	al (Ma	ok in) :B5000 bar's ty	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; butt	cons the s	Co labs:	ncrete	:C20/J	25 e and	the	Stee	ા (Ma	ок in) :B5000 bar's t	ype.	Steel	(Stirrup	s) :B500	C
s the ge th	follo ne ma	wing	; butt	cons the s	Co labs:	con	:C20/J	25 e and	l the	Stee	Ma P	ок in) :B5000 bar's tr	ype.	Steel	(Stirrup	s) :B500	С
s the ge th	follo ne ma	wing	; butt	cons the s	Co labs:	ncrete con	: C20/J	25 e and	l the	Stee	el (Ma	ок in) :В5000 bar's t	ype.	Steel	(Stirrup	s) :B500	C
s the ge th	follo ne ma	wing	; butt	cons the s	Co labs:	ncrete CON	: C20/J	25 e and	the	Stee	કો (Ma	ок in) :В5000 bar's t	ype.	Steel	(Stirrup	s) :B500	C
s the ge th	follo ne ma	wing	; butt	cons the s	Co labs:	ncrete con	:C20/J	and	l the	Stee	비 (Ma	ок in) :B5000 bar's t	ype.	Steel	(Stirrup	s) :B500	C



Quality Constants Fck (Mpa) ycu	C20/25	×	By cl paran Ma "F _{ck} ": "γ _{cu} ":	hoosing neters a ore spe the cha the pa	g a c are upo ecificall aracter rtial fao	differer dated a y, istic cy ctors fo	it cor utoma linder	ncrete itically. strengt rete in	type, th in M the ult	the Pa. imate	corresponding e limit state
γcs Fctm (Mp TRd (Mpa Max Defo εc (N,M) εc (N)	1 2.2 0.26 mations 0.0035 0.002 Can		"γ _{cs} ": Fctm: Trd: In the concre axial I You ca	the pa the ter the sh e field ete is d oad (N an also	rtial fac nsile st ear str "Max lefined) or jus type a	ctors fo rength ength c Deforr for the t for th differe	r conc of con of conc nation e simul e axial ent valu	rete in crete in s" the taneou load. ue from	the ser n MPa MPa maxin is actio n the de	vicea num n of t efault	bility limit state. deformation of bending (M) and cone.
Ster Quality Constant Es (Gpa) Fyk (Mpa ysu ysu yss Max Defc Es OK	el (Main) B500C S 200 500 1.15 1 rmations 0.02 Ca	×	In St the s "Es" reint "Fyk" in M "γsu" state "γss" serv In th	eel (M steel re forcem ': the c Pa. ': the p : the iceabili ne "Ma steel re	ain & S inforce Aore sp design ent in haract bartial f e part ty limit ix Defo inforce	Stirrups ement. Decifica value GPa. eristic y factors tial fac tstate. ormatio ement a) dialo Ily, of m yield st for ste ctors ns" fie are ind	odulus trength eel rein for th eld the licated.	there a of ela of the of the stee maxim	re the astici e stee nent i el re num o	e parameters of ty of the steel I reinforcement in ultimate limit inforcement in deformations of
In the "Che Loads comb	cks" field nation for str	the fol ip calcul	lowing ation	dialog	box caı	n be ed	ited as	s appro	priate.		
In the "Che Loads comb	cks" field nation for str i1 LG2	the fol ip calcul LG3	lowing ation LG4	dialog LG5	box cai	n be ed LG7	ited as	appro	priate. PL		

LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL
LC1	1.35	0.00								0
LC2	1.50									1



- The live and dead loads' factors considered during the strip slab calculation are displayed and can be modified as appropriate.
- PL column refers to the **adverse loads of slabs**. 0 for not considering the corresponding line's loads and 1 to consider them.

Shear Check		
Serviceability Cracking	Crack width (mm)	0.3
Deflection chec	[l/a] a	
Diagram scale 1 m =	5	(kN / kNm)

Activate the relative checks for the slab verification. Define the value in the field "Crack width" for the concrete cracking check. Define scale for diagrams' display.

NEW FEATURES

§ More combinations for the slab's design

In the new version of SCADA Pro, there is the possibility of introducing more combinations for the slab's design. In the design parameters window, inside Slabs:

Steel F	Reinforce	ment		Capacity	Design		Steel		Timber	structure
Comb	inations		Slabs		Bea	ims	Col	umns	mns Footing	
Concr	ete : C3	0/37		Steel (Main) :B	500C		Steel	(Stirrups	s) :B500C
hecks oad co	mbinatio	ns for st	rip calcu	lation	Г	1 ~	ULS 🗸	In	sert	Delete
LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL
LC1	1.35									0
LC2	1.50									1
Shea	r Check ceability Crack cor Deflectior	n trol n contro	Cra	ack width a] a	ı (mm)	0.3	3			
Diag	ram scale	e 1 m =		5		(kn / kn	lm)			
								0	К	Can



uad cu	moniauo	ins for si	uip calcu	2			000 -	113	-re	Delete
LC	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL
LC1	1.00									0
LC2	1.00									1

There are two default combinations, one for the Ultimate and one for the Serviceability Limit States.

To create a new combination, press, "Insert". The new combination is the combination number 3 and the coefficients are all 0.

Load co	mbinatio	ns for st	rip calcul	ation		3 ~	ULS 🗸	Ins	ert	Delete
LC	Lor	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	PL
LC1	0.00									0
LC2	0.00									1

Change the coefficients and select the Limit State.

In the same way, you can define as many combinations as you wish, or modify the ones you have already created. The program will use the combination with the worst moment regarding the USL combinations and correspondingly will make deformations checks by the functionality combinations. By using "Delete" button you can delete the created combinations. The 1 & 2 default combinations cannot be deleted.



1.3.3 Beams

This field contains the design checks for the beams verifications.

Capacity [Design		Steel		Timber struct	tures
Combinations	Slabs	Beams	Columns	Footings	Steel Re	einforcemen
Concrete : C20	/25	Steel (Ma	ain) :B500C	St	teel (Stirrups)	:B500C
hecks						
Bending						
Axial Force	e participatio	n	Minimum	steel reinford	cement 0)
Shoor						
Conn Angle	a = 00					
Conn. Angle	a - 50					
Capacity De	esign amplific	ation				
Shear A	mplification					
ShearA	mplification					
Shear A	Amplification					
Torsion	Amplification					
☐ Shear A Torsion ✔ Check	Amplification					
Shear A Torsion Check Serviceability	Amplification					
Torsion Check Serviceability Crack control	Amplification	rack width (mm)	0.3	k1 [0.8 k2	0.5
Shear A Torsion Check Serviceability Crack contro	Amplification	rack width (mm)	0.3	k1 [0.8 k2	0.5
Shear A Torsion Check Serviceability Crack contro	Amplification	rack width (mm) _{/a] a} 250	0.3 k3 3.4	k1 (0.8 k2 0.425 kt	0.5
Shear A Torsion Check Serviceability Crack contro Deflection co Soil Failure (Be	Amplification	rack width (mm) /a] a 250	0.3 k3 3.4	k1 (0.8 k2 0.425 kt	0.5
Shear A	Amplification	rack width (mm) _{/a] a} 250	0.3 k3 3.4	k1 (0.8 k2 0.425 kt	0.5
Shear A	Amplification DI C ontrol [1 ams OEF] Data	rack width (mm) _{//a] a.} 250	0.3 k3 3.4 ral. 0	k1 (k4 (0.8 k2 0.425 kt	0.5 0.4 (kN/M2)
☐ Shear A Torsion ✔ Check Serviceability ✔ Crack contro ☐ Deflection co Soil Failure (Be ☐ Check	Amplification	rack width (mm) /a] a. 250	0.3 k3 3.4 al. 0	k1 (k4 (0.8 k2 0.425 kt	0.5 0.4 (kN/M2)
Shear A	ams OEF)	rack width (mm) /a] a 250	0.3 k3 3.4 al. 0	k1 (k4 (0.8 k2 0.425 kt	0.5 0.4 (kN/M2)

In the "Bending" field activate the checkbox "Axial Force participation" for considering the axial force in the bending design checks.

In the "Shear" field define in the drop-down list "Conn. Angle" as the angle of the stirrups



In the "Capacity Design amplification" field keep activated the following checkbox Axial Force participation, in case that it is necessary to perform capacity design (e.g. DCM and DCH categories, "Capacity Design" is necessary) according to §5.4.2.2, EC8.



IMPORTANT NOTE!!!

 γ_{Rd} strength factor receives the value that corresponds to the ductility class automatically: γ_{Rd} =1,0 for DCM γ_{Rd} =1,2 for DCH

In the "Priority" field activate the checkbox "Check". Then, the program will consider the shear resistance $V_{cd} = 0$ and will calculate the stirrups.

In the "Serviceability" field activate the checks to be considered in the serviceability limit state (i.e. Crack and Deflection control) and modify the values in the "Crack width" and " k1", "k2", "k3", "k4", "kt" (§7.3.4, EC2) fields.

k1: is a coefficient which takes into account the bond properties of the bonded reinforcement:
k1=0.8 for high bond bars
k1=1.6 for bars with an effectively plain surface (e.g. prestressed tendons)
k2: is a coefficient which takes into account the distribution of the strains
k2=0.5 for bending
k2=1.0 for pure tension
k3=3.4
k4=0.425

The "Deflection control" In the beams according to EC2 is made by selecting the corresponding check in the parameters of the beam.

Define the upper limit (I/a) of the deformation.

Serviceability			
Crack control	Crack wi	dth (mm)	
Deflection control	[/a] a	250 0	

The results appear at the end of the exploration file

```
DEFLECTION CONTROL
```

BEAM	1 5 b=0.25	h=0.50 c=0	.03			
COMB	М	N	Du1	Du2		
100	-21.356	-0.000	0.00443	0.00011	0.00454	0.01500
101	-20.315	-0.000	0.00432	0.00012	0.00443	0.01500
102	-23.958	-0.000	0.00471	0.00010	0.00481	0.01500

EXAMPLE:

In the above beam, a check considering the three combinations of functionality (100, 101, 102) is done and the size Du1 is the maximum deformation of the element, as it results from the calculation of the elastic cord.

Du2 is the deformation as calculated based on the EC2 relationship 7.18. Then comes the sum (Du1+Du2) and the last column is the upper limit I/a.

It should be (Du1+Du2) < I/ a, so as the check to be achieved.



Soil Failure (Beams OEF): In this field, there are design checks related to beams on elastic foundation.

Soil Bearing Capacity (EC7)Calculation MethodInternal friction angle φSoil cohesion C (kN/m2)Without Water Pres.Shear Stength Su (kN/m2)OKCancelActivate the checkbox "Check" and click the button "Data". A dialog box opens entitled "Soil Bearing Capacity (EC7)" where you enter the soil data, in case you have a geotechnical study. Select a calculation method from the drop-down list and define an internal friction angle φ, the soil cohesion and the shear strength Su. Then, the program will calculate the stresses.Alternatively, activate the checkbox next to each stress and enter a value.Soil Failure calculation for Beams OEF, according to the selected calculation method, is based on the EC7 provisions.

The parameters in the field "Soil Failure" regarding the soil resistance for beams OEF are calculated using EC7 methods. Otherwise, you can activate the checkbox Menar the relative stresses and type the value for the nominal stress σ_{al} (KN/m²) and the stress fracture σ_{fr} (KN/m²).

The soil failure check based on EC7 is shown in the results and the most unfavorable reason is displayed.

Analytical results of the checks for each combination are also shown in the Exploration file.

1.3.4 Columns

In this field, there are the design checks related to columns and shear walls verification.



Capacity I	Design		Steel	Т	imber structures
Combinations	Slabs	Beams	Columns	Footings	Steel Reinforcemen
Concrete : C20/	25	Steel (Main) :B500C	St	teel (Stirrups):B500C
Checks Shear - Bending Capacity Amplifi	cation		Moment Amplif	ication	
Confinement Confinemen Buckling Dir.Y-Y Limit	nt a	0	Wall critical Coordinate In this field the critical I according f A) a specifi B) If you do C) If you ha ATTENTIC	Length y (m) you can define height of the wa to the following: ied length which n't have a base ve basement (c)N	3
-Torsion ✔ Check			Bars Calo Beam-Co Minimum ste	culation Method olumn Joint Che el reinforcemer	ck nt 0

In the field "Capacity Amplification" activate the appropriate checkbox, in case that the capacity design is necessary (e.g. According to EC8, in §4.4.2.3 and §5.4.2.2, for DCM and DCH, "Capacity Design" is necessary).

 γ_{Rd} overstrength factor receives the related to the ductility class value automatically: γ_{Rd} =1.1 for DCM γ_{Rd} =1.3 for DCH

In the field "Wall Critical Length" type a value according to the cases in the list below.

A) Type a specific value, regardless of the Design Regulation considered in the calculations.

B) If there is no basement, indicate the height H of the first level. The program compares this value with the parameter I_w =H/6 and keeps the greater one.

 Γ) If there is a basement, define the height H of the level above the basement. The program compares this value to the I_w and H/6 and keeps the greater one.

The definition of a value in this field is mandatory for the calculation of the walls critical length. Activate the checkbox "Confinement" and type a value in the "a" field. Otherwise, keep the checkbox inactive and the program will calculate the stirrups section and the distribution according to the paragraphs §5.4.3.2.2 and §5.4.3.4.2 of EC8^{*1}.



For Columns (§ 5.4.3.2.2, EC8)

$$\alpha \omega_{\rm wd} \ge 30 \mu_{\varphi} v_{\rm d} \cdot \varepsilon_{\rm sy,d} \cdot \frac{b_{\rm c}}{b_{\rm o}} - 0.035 \tag{5.15}$$

Where

*1

 $ω_{wd}$ is the mechanical volumetric ratio of confining hoops within the critical regions; $ω_{wd} = \frac{volume of confining hoops}{volume of concrete core} \cdot \frac{f_{yd}}{f_{cd}}$

 μ_{ϕ} is the required value of the curvature ductility factor;

 v_d is the normalized design axial force ($v_d = N_{Ed} / A_c \cdot f_{cd}$);

 $\varepsilon_{sy,d}$ is the design value of tension steel strain at yield

For Walls (§ 5.4.3.4.2, EC8)

For walls of rectangular cross-section, the mechanical volumetric ratio of the required confining reinforcement ω_{wd} in boundary elements should satisfy the following expression, with the values of μ_{φ} as specified in (2) of this sub clause:

 $\alpha \omega_{\rm wd} \ge 30 \cdot \mu_{\varphi} \cdot \left(\nu_{\rm d} + \omega_{\nu}\right) \cdot \varepsilon_{\rm syd} \cdot \frac{b_{\rm c}}{b_{\rm o}} - 0.035 \tag{5.20}$

Where the parameters are defined in 5.4.3.2.2(8), except ω_v , which is the mechanical ratio of vertical web reinforcement ($\omega_v = \rho_v \cdot f_{yd,v}/f_{cd}$).

In the field "Buckling" activate the checkbox \checkmark referred to the Y or Z direction (along with the local axis Y or/and Z).

(NOTE: View of local axes: Menu>>"View">>"Switches">>"Local Axis")

In the field "Short Columns" activate the checkbox "Check" to perform the required check in DCH cases *².

*2

Beam-column joints (§5.5.2.3)

(1)P The horizontal shear acting around the core of a joint between primary seismic beams and columns shall be determined to take into account the most adverse conditions under seismic loading, i.e. capacity design conditions for the beams framing into the joint and the lowest compatible values of shear forces in the framing elements.

(2) Simplified expressions for the horizontal shear force acting on the concrete core of the joints may be used as follows:

a) for interior beam - column joints

 $V_{jhd} = \gamma_{Rd} \cdot (A_{s1} + A_{s2}) \cdot f_{yd} - V_C$ (5.22)

b) for exterior beam – column joints:

 $V_{jhd} = \gamma_{Rd} \cdot A_{s1} \cdot f_{yd} - V_C$ Where $A_{s1} \quad is the area of the beam top reinforcement;$ $A_{s2} \quad is the area of the beam bottom reinforcement;$ (5.23)



V_c is the column shear force, from the analysis in the seismic design situation;

 γ_{Rd} is a factor to account for overstrength due to steel strain-hardening and should be not less than 1.2.

(3) The shear forces acting on the joints shall correspond to the most adverse direction of the seismic action influencing the values As1, As2, and Vc to be used in expressions (5.22) and (5.23).

Beam-column joints (§5.5.3.3)

(1)P The diagonal compression included in the joint by the diagonal strut mechanism shall not exceed the compressive strength of concrete in the presence of transverse tensile strains.

(2) In the absence of a more precise model, the requirement of (1)P of this subclause may be satisfied using the subsequent rules.

(5.33)

a) At interior beam – column joints the following expression should be satisfied:

$$V_{_{jhd}} \leq \eta \cdot f_{_{cd}} \cdot \sqrt{1 - rac{v_d}{\eta}} \cdot b_{_j} \cdot h_c$$

Where

 $\eta = 0.6 \cdot (1 - f_{ck}/250);$

 v_d is the normalized axial force in the column above the joint; and

f_{ck} is given in MPa

b) At exterior beam – column joints:

V_{jhd} should be less than 80% of the value given by the right-hand-side of expression (5.33) where:

V_{jhd} is given by expressions (5.22) and (5.23) respectively;

And the effective joint width b_j is:

a) if
$$b_c > b_w$$
 then $b_c > b_w : b_j = \min\{b_c; (b_w + 0.5 \cdot h_c)\}$ (5.34a)

b) if $b_c < b_w$ then $b_c > b_w : b_i = \min\{b_w; (b_c + 0.5 \cdot h_c)\}$ (5.34b)



1.3.5 Footings

In this field, the relative checks for the footings verification are located

Structural Co	omponent Pa	rameters	
Capacity Design	Steel	Т	imber structures
Combinations Slabs Beams	Columns	Footings	Steel Reinforcement
Concrete : C20/25		Steel (N	1ain) :B500C
Checks			
Soil type			
	σal.(kN/M2) 250	
	ofr.(kN/M2)	300	
Soil failure	ta		
Bending-Shear-Punching Shear			
MV > ? 1.5 x ME	qx 3.5	qz	3.5
Max H of Footing (cm)	0		
Serviceability			
	0		
Structural failure ocr/l <= 1/	0		

In the "Soil Type" field you can activate the checkbox \mathbb{P} next to the relative stresses and type the value for the nominal stress σ_{al} (KN/m2) and stress fracture σ_{fr} (KN/m2).

In the "Soil failure" field activate the checkbox "Check" and then press the button "Data", to enter the soil data, in case you have a geotechnical study. Select a calculation method and define an internal fiction angle ϕ , the soil cohesion and the shear strength Su. The program will calculate the stresses. Otherwise, activate the checkboxes next to the stresses and enter a value.

Soil Failure calculation for Beams OEF, according to the selected calculation method, is based on the EC7 provisions.

Soil Bearing Capacity	(EC7) ×
Calculation Method	II 🗸
Internal friction angle ϕ	0
Soil cohesion C (kN/m2)	0
Without Water Pres.	
Shear Stength Su (kN/m2)	0
OK Cano	el



х

"Bending-Shear-Punching shear": In this field activate the checkbox "Max H of Footing" and type a value. The activated checkbox means that the program will perform the design check against punching shear. If the original height does not satisfy the punching shear design check, the program will calculate the height that satisfies the check. If this is higher than the limit you have set, a message is displayed that informs you that a higher footing is necessary.

In the following fields "qx" and "qz" a

In the "Serviceability" field the design checks that correspond to the serviceability limit state are included. Activate the checkbox which allows the user to define the limit value of the considered ratio δ_{cr}/l .

1.3.6 Steel Reinforcement

On the first field "Available Bars", which is common for all structural elements, specify the diameters of the reinforcement bars.

From the diameters list 6.8.10.12.14.16.18.20.22.25.28.32.35. existing diameter, respectively. Type a value in the following field and press the button

to add the new diameter to the list. To remove an existing diameter, select the value from

the list and press the following button 🛄.

In the Lmax field, you type the maximum bar length to be used in the reinforcement of the structural elements.

Structural Component Parameters

Combinations	Slabs	Beams	Column	s Footings	
Steel Reinforcement	Capacit	y Design	Steel	Timber structures	;
Available Rebars Φ(mm) 0 +	6,8,10,12,1	14, 16, 18, 20, 22, 2	5,28,32,35,	Lmax(m) 12	

1.3.6.1 Slabs

In "Slabs" the rebar of the slabs is defined:

In the field "Bar Cover" type the rebar cover value according to the climatic conditions (mm).

In the "Bar spacing" field Bars spacing (cm) type the maximum and minimum distance (cm) between the reinforcement bars inside the slab.

In the field "Solid Slabs Reinforcement" specify the minimum reinforcement (main, secondary, additional) and the minimum relative distances.

In the "Reinforcement of Zoellner-Sandwich" field specify the upper and lower, minimum main reinforcement for the solid part. For the beams inside the slab, define the bars' quantity and the upper and lower maximum and minimum diameter.



In the field "Stirrups" specify the minimum distance between the stirrups (cm), diameter/distance in support and span.

abs Columns - Wa	alls Beams	Footi	ng Connection B	eams Strip	Footings	Footings
Concrete Cover (mm) 20		Bars spacing	(cm) max	20	min 5
Solid Slabs Reinforc	ement					
	Φ	/ (cm)			Φ	/ (cm)
Main Reinforcement	8 🗸	20	Additional	Support	8 🗸	20
Secondary	8 🗸	25	Secondary		8 🗸	25
Main Reinforcement	top Slab	v 8	/ 15	τo	P ♥ 8	/ 15
	Main		Φmax	Φmax		Φmax
Main Reinforcement	1 Φ	12	✓ 20 ✓	2 0	10	✓ 20 ✓
			Stirrups		Span	1
	-				· · ·	

1.3.6.2 Columns-Walls

In "Columns-Walls" the reinforcement of the columns-walls is defined:

Type the bar cover value according to the climatic conditions (mm).

In Bars spacing (cm) type the maximum and minimum limits related to the distance (cm) between the bars.

Slabs Columns - V	Valls Beams	Footing Con	nection Beams	Strip Footings	Footings
Concrete Cover (m	m) 25		max Bars Spa	acing (cm)	20
Columns - Walls					
	Φmin	Φmax		No.of E)iameters
Columns	14 🗸	20 ~			2
Column	14 🗸	20 ~			2
Walls interior					
	Фmin Ф	max /mir	n(cm)	/min(cm) / (cm)	
Horizontal	10 ~ 1	2 ~ 5		10 ~ 15	
Vertical	10 ~ 1	2 ~ 5		10 ~ 15]
Shear (Stirrups)					
min Distance (cm	1) 5			Φ /max (c	:m)
Φmin Φ	max		Support	8 ~ 10	
8 ~ 12	\sim		Span	8 ~ 10	



In the "Columns-Walls" field specify the maximum and minimum limits of the diameter of the main steel reinforcement used for columns ("Columns" field) and columns inside the shear walls ("Column" field).

In the field "No. of Diameters" define how many differed diameters could be used in columns and shear walls, respectively.

EXAMPLE:

If you type 2, the program will consider two more diameters, i.e. 3 in total (i.e. Φ 16- Φ 18- Φ 20). If you type 0, it will consider only one.

In the "Walls interior" field define the parameters of the steel reinforcement for the body of the shear walls. Type the maximum and minimum diameter for horizontal and vertical rebar and the minimum distance between them. For distances less than the minimum defined, the program will increase the diameter.

In the field "Shear (Stirrups)" specify maximum and minimum limits for the stirrups' diameter and the minimum distance between them (for less than then minimum distance, the program increases the diameter).

		Φ	/max (cm)
	Support	8	· 10
s	Span	8	• 10

In the following field:

specify the steel reinforcement in the support and the span. If the defined values satisfy the design checks, they will be placed, otherwise, the program will take into account the limit values so as different steel reinforcement is placed.

Beams/Footing Beams 1.3.6.3

In "Beams" and "Footing Beams" define reinforcement:

Type bar cover value according to the climatic conditions (mm).

In Bar distance (cm) type the maximum and minimum limits for distance (cm) between the bars.



labs	Colum	ns - Wa	alls B	eams	Footir	ng Co	nnection	Beam	ns	Strip Fo	otings	Foot	ngs
Concre	ete Cov	er (mm)	25	Ba	ar dis	tance (cr	n) Ma	x	2	20	min	5
Web Up	Reinfo	rcemen tend	t bo	ttor 🗌 i	Exten	d		l	Upda	te All			
2	Φ	14 ~	4	Φ	14	\sim	Φmax	20	\sim	Crack	ing Φ	8	~
Side l	oars			Φmin	12	\sim	Φmax	20	\sim				
											e 101 - Z	、 Г	100
Supp	ort bars	;		Φmin	14	~	Φmax	20	~	max W	/idth (c	m) [120
Suppo	me Reir Iti-Spar r (Stirru	s nforcem n Reinfo ips) —	ent in prceme	Фтіп Span-Su nt	Jppor	t	Φmax	20	~	max W	lidth (c	m) [120

In the field "Web Reinforcement", the checkboxes "Up" and "Bottom" next to "Extend" means that the beam supports' steel reinforcement will pass through the span and will be added on this. Specify the maximum and minimum limits for the diameter of the main reinforcement in extending (upper and lower), for bars in the span and the support.

Web Reinforcement	bottor 🗌 Extend	Update All	
2 Φ 14 ×	4 Φ 14 ×	Φ max 20 \sim Cracking Φ	8 ~
Side bars	Фтіп 12 V	Фтах 20 V	
Support bars	Φmin 14 ~	Φmax 20 ∨ max Width (cm) 120

First, specify the minimum main reinforcement of the upper side. Type the number and select the diameter. Repeat for the lower side. Then define the maximum diameter.

Then specify the minimum and maximum diameter for the main reinforcement on side and supports. Finally, in the "Cracking Φ " field specify the minimum diameter of the reinforcement considered for crack control.

In the "max Width" field, determine the maximum distance to place a common bar. In case of bigger distance, the program will place two bars in the support.

Activate the following checkbox Same Reinforcement in Span-Support and the steel reinforcement will be the same for span and support.



IMPORTANT NOTE: MULTI-SPAN REINFORCEMENT

Activate the following checkbox Multi-Span Reinforcement so that the reinforcement in the spans is common to the entire beam.

In the implementation process of single beams reinforcement with common rebars of a specific length, the program places the additional rebars in the supports based on certain criteria. There are two ways to place the additional supports reinforcement.

- The first way is additional rebars to come from each span on both sides and be positioned on the respective side of the span.



- The second way is to place a common support rebar.





A. The first criterion is the <u>width of the support</u>, as determined by the parameters of beams reinforcement.

Structural Component Parar	meters				:	×
Combinations Steel Reinforcement Available Rebars	Slabs Capacit	Beams y Design	Steel	Columns	Footings mber structures	
Slabs Columns - Walls Concrete Cover (mm) Web Reinforcement Top ∠Extend E 2 Φ 14 ✓	Beams Foot 25 Rr 3ottom Ext 4 Φ 4 Φ	ing Connection ebar spacing (c end $4 \lor \Phi$ max	n Beams m) Max Upda	Strip Footing : 20 Ite All Cracking Φ	In S Footings	
Support rebars	Φmin 1 Φmin 1	2 V Φmax 4 V Φmax	20 V 20 V	max Width	(cm) 120	
Same Reinforcement i	n Span-Suppo nent	rt		-		
Shear (Stirrups) Min Spacing (cm) Preference Stirrups (90	10 () ~ 8	Φmin Φm i V 12	ax Supp	mir bort 8 n 8	nΦ / (cm) ∨ 10 ∨ 10	
				ОК	Cancel	

If it exceeds the max support width (see Figure 1), then support reinforcement per side is placed separately.

If the width of the support is less than max width, then the same rebar is placed all along the support (see Figure 2).



🔺 NOTE:

Changing this parameter post creation of the alignments of the beams requires deletion and recreation.

B. The second criterion has to do with the <u>width of the beams</u> in the support. If this width is different for the two beams, then additional support rebars are placed separately. Otherwise, a common rebar is placed.

CONCLUSION:

Common rebar is placed only if both criteria mentioned above are satisfied:

- A. Support width <max width in parameters
- B. Same beams width

In the next section "Shear (Stirrups)" the parameters for the shear reinforcement of the beams are defined.

Min Spacing (cm)	10				minΦ	/ (cm)
· ···· opcoring (crii)	10	Φmin	Φmax	Support	8 ~	10
Preference Stirr	rups (90) 🛛 🗸	8 ~	12 🗸	Span	8 ~	10

Specify,

the minimum distance between the stirrups

the angle (if they are placed perpendicularly or obliquely (45°))

the diameter limit values



In the following fields ________ specify the diameter and the distance of the reinforcement bars in the support and the span. If these values satisfy the design checks, the rebar is placed as appropriate, otherwise, the program places a different reinforcement by taking into account the limit values of the diameter as well as the distance.

NOTE:

Update All : After completing the parameter selections and before closing the window, press the Update All button to update the parameters.

1.3.6.4 Strip Footing

In "Strip Footing" the parameters of the steel reinforcement in strip footings are defined:

For the common parameters mentioned in the previous tabs, follow the previous described procedure.



Also, in the "Range Reinforcement" field define the limits for longitudinal and transversal flange

	Flange Reinforcement / (cm)							
	Longitudinal	Φ	12 🗸 / 15	Transverse	Φ	12 🖌 / 15		
reinforcement.								

1.3.6.5 Footings

In the "Footings" tab, the parameters of the steel reinforcement in footings are defined:

Type the bar cover value by the climatic conditions (mm).

In the field "max bar distance" type the maximum and minimum limit values for the distance (cm) between the reinforcement bars.

Slabs	Columns - Walls	Beams	Footing Connection Beams	Strip Footings	Footings
Concr	ete Cover (mm)	40	max bars distan	ce (cm)	5
Flang	je				
	Φmin	Φmax	/min(cm minΦ	/	
	12 🗸	20 🗸	10 12 🗸	15	

Specify the minimum and maximum diameter of the bars and the minimum distance. For lower distance, the program changes the diameter.

minΦ	/ (cm)
12 💌	15

In the following fields determine the diameter and the distance of the reinforcing bars, you wish to place. If it is sufficient, then they are placed. Otherwise, the reinforcement, resulting from the program's design verification, is placed.

1.3.7 Capacity Design

The last tab "Capacity Design" concerns concrete structures capacity design checks:

Structural Component Parameters							
Combinations Capacity I	Slabs Design	Beams	Columns Steel	Footings	Steel Reinforcement Timber structures		
Direction x Edge Middle Fixed	= acd <= 3.5 3.5 2.5 1.35 3.5 3.5			Direction z Edge Middle Fixed Eree	 acd <= 3.5 3.5 3.5 ↓ 1.35 ↓ 3.5 		



Specify the upper bound of the factor " a_{cd} " for capacity design, in each direction.

Generally, the value of α_{cd} should be less or equal to the seismic behavior factor q. For fixed columns, α_{cd} = 1.35 is taken.

Activate the corresponding checkbox and type the value you want.

Without ticks, the program takes into consideration the calculated values.

The designation of a node as free or fixed is performed by using the command "Designation".



Nodes without Designation are considered free in both directions except for the fixed ones.

Finally, considering the exploration of the column, there are the results of the capacity design check (initial and incremented moments) for each combination. Of course, in the direction in which the capacity design check has not been taken into account, the initial moments are the same as the incremented ones.

1.3.8 Steel

The next section regards the "Steel cross sections" check parameters and displays the following dialog box:



Combinations Slabs	Beams	Columns	Footings	Steel Reinforcement
Capacity Design		Steer		I imber structures
Name		^	Select All	Deselect All
Lines circles				
Concrete Columns			Сору	Paste
Concrete Jackets		-Par	ameters	
Concrete Beams				
Concrete Foundation Beams				GENERAL
Footing Connection Beams				TENSION
Foolings Stool Columns				TENSION
Steel Beams				SHEAR
Surface Mesh				
Mathematical Model				TORSION
Surface Elements				
Mesh 3D			CO	MPRESSION
Mesh 2D				
Slabs-Strips				BEINDING
			BENDIN	IG & AXIAL FORCE
Steel Columns				
Steel Beams			BENDIN	G & SHEAR FORCE
Main Beams Durling				
Girders			BENDING	SHEAR AXIAL FORCE
Secondary Columns				Default
Hor Wind bracings		~		Deiduit

First, select a layer. Click one from the list, or more using "ctrl", or all using "Select All". (By pressing the button "Deselect All" cancel the previous layers' selection.) Then activate one or more design checks by clicking on the corresponding checkbox and press the corresponding button to specify the parameters.

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

EXAMPLE:

Suppose you have set all parameters for the layer Steel Columns and you want to pass these parameters to Steel Beams. Activate the check box next to "Default" and all parameters are selected automatically.



Paran	neters
✓	GENERAL
✓	TENSION
✓	SHEAR
✓	TORSION
✓	COMPRESSION
✓	BENDING
✓	BENDING & AXIAL FORCE
✓	BENDING & SHEAR FORCE
✓	BENDING SHEAR AXIAL FORCE
✓	Default

Then press "Copy", select layer Steel Beams and press "Paste" (now activated).

^	Select All	Deselect All
	Сору	Pa
	Parameters	1
	raianeters	
	✓	GENERAL
	✓	TENSION
	✓	SHEAR
	v	TOPSION
		TORSION
	J	COMPRESSION
	•	
		BENDING
	SEN BEN	NDING & AXIAL FORCE
	BEN	DING & SHEAR FORCE
	BENDI	NG SHEAR AXIAL FORCE
	1	Default
~	•	Default
		 Select All Copy Parameters ✓ ✓ ✓ ✓ ✓ ✓ ✓ Ø Ø<

Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams. An alternative method to set the same parameters to all layer including steel sections is to select all layers by pressing "Select all" button and set the parameters once for each check category. Note that at least one (or more) layers should be selected to set the parameters.


Parar	neters	
	GENERAL	
	TENSION	
	SHEAR	
	TORSION	
	COMPRESSION	
	BENDING	
	BENDING & AXIAL FORCE	
	BENDING & SHEAR FORCE	
	BENDING SHEAR AXIAL FORCE	
	Default	
ress	the button "GENERAL"	to set the γ_{Mi} safety factors:
Ge Ge	the button "GENERAL"	to set the γ_{Mi} safety factors: γ_{M0} : partial factor for cross-sections' resistance whatever the class is γ_{M1} : partial factor for members' resistance to buckling bases on tests γ_{M2} : partial factor for resistance of cross-sections in tension to fracture
Press Ge Lim	the button "GENERAL"	GENERALto set the γ_{Mi} safety factors: γ_{M0} : partial factor for cross-sections' resistance whatever the class is γ_{M1} : partial factor for members' resistance to buckling bases on tests γ_{M2} : partial factor for resistance of cross-sections in tension to fractureIn the "Limit of Internal" field define an upper limit. Below this value, the program will not consider the corresponding stress resultants.



Holes	<u></u>	\sim	Safety Facto	r 1		Ж
	O web and Flange	ΟL			Ca	ncel
olt Holes Geometry						
Web		Flange	0			
Holes Diameter (mm)		Holes Dia	meter (mm)	- 45 -		
the force pic.1)	0	force pic.1)	Boit Rows (perp. to	otne 0		
		e2 p2		e2 p2	••••	
		ie ₁	₽ <u>1</u>	e	+ • • 1'P1	
	e2 p2 e1 p1	e2 p2 e1		e2 p2 e		0
		e2 p2 e1			•	
Distan	ce between the holes (mm)		D	istance betw	een the holes (m	nm)
e1 p1	e2 p2	e1	р1	e2	p2	
0 0 0	0	0	0	0	0	
Section L						
Hole	es Diameter (mm)	Number of Bolt	Rows (parallel to	the force)	0	
e ₁ p ₁		e1(mm) 0	p1(mm)	0	e2(mm) 0	

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

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

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

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

Press the button "SHEAR"		SHEAR		to define if the elements of the
selected layer contain stiffen	ers a	nd if so which type; web	st	iffeners or intermediate stiffeners.
Also define the spacing betwo	een t	he stiffeners and the type	e o	f the connection (rigid or not rigid).



Shear Parameters ×	
Safety Factor 1	
Stiffeners No In Support Between Dist. between Stiffeners (cm) 30	
Support	
Rigid	
Non Rigid	
OK Cancel	
	TORSION

Click the button "TORSION" to define whether the structural elements of the selected layer are loaded by a distributed or concentrated torsional moment, or not. In the following dialog box, you may as well define the support conditions based on the corresponding figures. Select the type of moment and set (i) the relative distances from the start and the end, (ii) the value of the moment and (iii) the length of the element in the corresponding fields. Also, set the support condition by typing in the "Type" field the values 0, 1, 2 or 3.



Torsion Parameters ×	
Safety Factor 1 Torsional Moment	
No Distributed Concentrated	
Distance from Start (cm)	
Distance from End (cm) 0 Value (KNm) 0	
Element's Length (cm) 300	
Support Conditions	
COMPRESSION BENDING BENDING & AXIAL FORCE BENDING & SHEAR FORCE	For all design checks presented in the figure on the left, define the safety factor in the dialog box that appears when you click one of the five buttons. The safety factor is the ratio of the resistance value versus the corresponding design value, which is set 1.0 by default.
BENDING SHEAR AXIAL FORCE	Parameters ×
	Safety Factor 1
	OK Cancel



1.3.9 Timber structures

The next section regards the **Timber cross sections** check parameters and displays the following dialog box:

Structural Component Parameters ×						
Combinations	Slabs	Beams	0	Columns	Footings	Steel Reinforcement
Capacity (Design		Ste	eel		Timber structures
					Selectall	Deselect all
Name					-	Beselectuit
Lines circles					Сору	Paste
Concrete Columns				-ym	13	Olular 125
Concrete Jackets				50110	1.5	Giulam
Concrete Beams	_			Sonvice	lass	Class 1
Concrete Foundatio	n Beams			Service c	1055	VIG501 +
Footing Connection	Beams			Load dura	ation class	Permanent V
Footings Stool Columns				✓ kh com	putation (& 3.2	- 3.3) 1
Steel Beams						·
Surface Mesh				ksys (&6.	6.) 0	kcr (&6.1.7.) 0.67
Mathematical Mode	el			🖌 kshape	e computation ((& 6.1.8) 1
Surface Elements						1.0
Mesh 3D				Km cor	nputation (& 6.	1.6)
Mesh 2D				Configura	tion (Anet)	
Slabs-Strips						· · · · ·
Steel Columns					P1	P1
Steel Beams						
Main Beams				Hole dia	meter (mm)	U
Cirdors				Number	of connector ro	ows 0 p1(mm)
Secondary Column	e			(parallel	to grain)	
Hor.Wind bracings				Stage	ered configura	ation
			~			
						OK Cancel

The definition of design parameters for timber sections is made per layer. Select the layer for setting the parameters (ex. Timber Columns). Then the program enables you to Copy And Paste to other layers.

Сору	Paste

EXAMPLE:

Suppose you set all the parameters for the Timber Columns layer and you want to copy them to the Timber Beams layer.

Left click on Timber Columns layer in the list and press "Copy", then select Timber Beams layer and press "Paste" which is now activated.

Those parameters are explained in detail below :



Partial Factors γM

Partial factors' value γ_M for the materials will be used in cases of ordinary or accidental loads (the accidental loads do not include the case of an earthquake, on which the paragraph 2.9.4) is applied.

Suggested values (EC5 – Table 2.3) are presented in the table below:

Recommended partial factors	YM		
Solid timber	1.3	Members	
Glued laminated timber – Glulam	1.25		

Service classes – influence of moisture content

Humidity is associated with the environmental conditions on the structure (or the single member), i.e. the temperature and relative humidity. The regulation defines three service classes (EC5 - 2.3.1.3):

Service Classes	Temperature and relative humidity	%	Examples
1	temperature of 20°C and the relative humidity of the surrounding air only exceeding 65 % for a few weeks per year	(9±3) %	Closed constructions heated (hot roofs, interior floors, and interior walls)
2	temperature of 20°C and the relative humidity of the surrounding air only exceeding 85 % for a few weeks per year	(12±3) % (15±3) %	Closed construction unheated or heated periodically (ex. holiday homes) Open roofed construction, cold roofs, exterior walls and general structures that are not directly exposed to the elements
3	climatic conditions leading to higher moisture contents than in service class 2	> 19 %	Construction in damp areas or structures exposed to the elements (ex. direct wetting)



Load Duration Class

Since the strength of a member decreases during loading, the regulation divides load types into categories according to their duration (EC5 - 2.3.1.2):

Load-duration class	Period of time	Examples of loading	
Permanent	> 10 years	self weight	
Long-term	Six months to 10 years	storage	
Medium-term	One week to 6 months	imposed floor load	
Short-term	to 1 week	snow, wind	
Instantaneous		wind and accidental load	

Kh, Kshape, Km factors are calculated automatically by the program by the respective chapters of EC5. The values can be defined by deactivating the checkbox and typing the respective value.

kh computa	1
ksys (&6.6.)	0.67
kshape cor	1
km comput	1

- K_h is a growth factor and depends on the type of wood, the size of the member and the type of load.
- K_{sys} is a growth factor that refers to continuous loads of sectional systems.
- K_{cr} is a reduction factor with a fixed value of 0.67 and concerns the shear
- K_{shape} is a factor depending on the shape of the cross section
- K_m is a factor considering redistribution of bending stresses in a cross-section

-Configuration (Anet)

		P1
Hole diameter (mm)	0	
Number of connector rows (parallel to grain)	0	p1(mm)
Staggered configuration	ı	0

In "Configuration" field set an initial approximate order of the holes in the wooden member which will be used for members' predesigning and then it will be applied to the connections' design.

You set the diameter of the holes and the number of connector rows and the distance p1 according to the two figures.

To set a Staggered configuration activate the corresponding Checkbox.



1.4 Merge Elements

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

IMPORTANT NOTES:

Merge Elements *

A By using this command, it is now possible to define correctly, the initial length of the member

per direction to be taken into account in the buckling checks.

Buckling Members Input

Until now, this condition was considered by defining the length coefficients (see

Buckling Members Input)
Lateral Buckling	
Direction Y Member's Length	Direction Z Member's Length
○ Real	 Real Coefficient
N	

- Now, be using merging per direction, there is no need for the coefficient process, and merging will be achieved, in most cases, automatically.
- Also, note that through the merging process, the buckling length, is calculated correctly, and in the printouts of the results a merged element is printed once with the annotation of the individual members that contains.

BASIC CONCEPTS OF BUCKLING ALONG MAJOR AND MINOR AXIES. WHAT IS LY AND LZ RESPECTIVELY.

Generally, in the double T cross sections, the local axis

- y-y is the major, and
- z-z is the minor,

as in the figure below:





EXAMPLE:

For example, let's see the buckling length of this column below, which is connected laterally with griders. First, check the initial buckling lengths Ly and Lz for the column.



The local axes direction of the column and the griders are as shown in the figure below:



The columns buckling along its major axis y-y (green) means:

- Buckling because of **My** (rotation around the y-y axis), that is, buckling out of the plane, which in the specific case, the merged length should be the buckling lengh, that is, <u>the total length of the column</u>.

In the other direction, buckling along its **minor** axis **z-z** (blu) means:

- Buckling because of **Mz** (rotation around the z-z axis), that is buckling in the plane. The column is considered to be supported laterally by the griders, so, the buckling length Lz should be the length of each member.



NOTE:

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

In the same example regarding the griders:



The supports from the columns are parallel to the local z-z axis (blue, out of plane) of the griders. So, <u>merging will be in Lz</u> (total length). While in the y-y direction (green, in the plate), <u>Ly is the length of each member.</u>

Respectively, for the **inclined beam** of the figure below:



The local axis of the beam that is parallel to the purlins is the y-y. So, <u>Ly wil be the merged length</u> of the total beam, while <u>Lz will be the single members</u>.



Merge group command, contains the list of commands below:



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

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

Also, in the presentation of the results, for these merged elements, the most unfavorable results are displayed once and not for each one, as it was happening so far.

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

▲ NOTE

It is better to work in the 3D mathematical model, displaying the local axes, whenever you use these commands.



1.4.1 Auto merge

By using this command the following dialog box is displayed:

🕱 🔀	Auto			
Automa	tic Merge		6	\times
Layer	Steel Colu	mns		\sim
Colum	าร	~	Calculation	
Disco	ntinuity Level	s		_
				-
				-
	New	View	Delete	1
Pi	ck // XY	Pick // ZY	Pick	i
				-
		Cancel		

First, choose the layer of the elements to merge.

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

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

Merged elements are displayed with colors:

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





The next section is about defining and processing the **discontinuity levels**.

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

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

Predefined limits:

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

EXAMPLE:

A three floor building with 0.00, 300.00 and 600.00 height levels, in **discontinuity levels** list of the columns, only the level 300.00 will be specified by default (that is, only the intermediate level without the limits)

Automati	c Merge		×
Layer	Steel Column	ns	~
Columns		~	Calculation
Discont	inuity Levels		
1 30	0.00		
N	lew	View	Delete
Pick	: // XY	Pick // ZY	Pick
		Cancel	

Considering that, the columns merging will be interrupted at 300.00 cm. The column will merge from 0.00 to 300.00 cm and the next floor column from 300.00 to 600.00 cm.

• To set your **discontinuity levels** for **COLUMNS**:



press "**NEW**" and next "**Pick**" and point one point. The horizontal level that defines the altitude of this point is a **discontinuity level**. Select level from the list and press "**View**" to display it.



To delete a **discontinuity level**, select it from the list and press "Delete".

For the BEAMS:

The definition of vertical **discontinuity levels**, but now through "**Pick**" you define two points, that is, a line that defines a vertical **discontinuity level**.



For example, in the figure below



discontinuity level of the front and back griders is the limit of the two buildings.

Especially for the beams, and when the **discontinuity level** you want to set, is parallel to the Global XY or ZY, press the corresponding command and point only one point.

Pick // XY Pick // ZY

For Beam's and Column's discontinuity levels, *editing* can be achieved in two ways:

- 1. Either by deleting and defining a new one,
- 2. or by selecting the corresponding level and re-defining by pointing a point or points.

1.4.2 Users merging

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

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

User
Merge X
Buckling
Deformations Direction Y Direction Z
Columns
View
ОК
Cancel

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

1.4.2.1 Merge concrete columns

This command is mainly used in masonry buildings with vertical reinforced concrete elements which connect the nodes of the surface elements and which, to be designed, must be merged.





You select the command, and then you point the starting point and the end point of the members you want to merge.

1.4.3 View

Using View command, you can see the merged elements colored, accordi tong merge direction. Also, the following dialog box appears:

Siew	
Merge 🛛 🗙	<
Buckling Direction Y Direction Z	
Deformations Direction Y Direction Z	
Columns ~	•
View	
OK	
Cancel	

- Select element's type from the list, then
- check in Buckling / Deformations the direction of merging to see the corresponding merged elements.





1.4.4 Correction

"Correction" command offers the opportunity to correct elements which are already merged.

Select the command and then a merged element to display the following dialog box:





in which checks shows the merging direction.

Here you can modify the selections of the directions in Buckling and Deformations. Press View to see the member with the corresponding merged color.

ATTENTION

This command works only for the merged element, otherwise, the dialog box does not appear.





Select this command to delete a single merging. Select the command and click a merged element. Right click to delete merging.

1.4.6 Overall Delete



Select this command to delete all merging. Select to delete all merging from all the merged elements.





The "Beams" command group contains the following commands

- "Continuity of Beams"
- "Check Reinforcement"

- "Results" related to the design checks and the checks of the steel reinforcement



In case that beams include surface elements there is a need to break the members of the beams in order the necessary connections between the linear and surface elements to be ensured.





Consequently, by breaking the beam in small parts, the need for merging is born, for the beam to be designed as a single member. This is accomplished by using the command Merge Beams.

Merge Beams

Select the command, and then point one by one in succession, the parts of the beam. Continue with "Continuity of Beams" and "Check Reinforcement".

If for some reason you wish to delete a preliminary merging, select the command

Merge Beams Delete

and then the first element of the merging beam. Right click to complete.

2.1.2 Single Continuity/ Overall Continuity of Beams

Single Continuity of Beams: This command is used to (selectively) define the beams that participate in a beam continuity.

- ✓ Left click to select the beams.
- ✓ Right click to end the selection.
- ✓ The red rectangles that appear on the screen indicate the beams' continuity



Overall Continuity of Beams: This command is used to determine the beams' continuities of the current level or the entire building automatically.
Select the command and in the dialog box check Level or Total:
Beams Alignment-Total
OK Cancel
Red rectangles that describe the beams' continuity are displayed in the screen automatically.
2.1.3 Single/Overall Deletion of Continuity
Single Deletion of Continuity: This command is used to (selectively) remove already created beams' continuities.
Select the command Left click to select the beams
Right click to end the selection
Overall Deletion of Continuity: This command is used for the removal of all existing beams' continuities, of the current level or the entire building.
Beams Alignment-Total Delele X In the dialog box that appears, make the appropriate choice.
OK Cancel



2.1.4 Preferences of Beams Reinforcement

This command is used to:

- o insert one common bar or two different bars on the support of the continuous beams,
- o take into account both of them,
- change the anchorage length.

EXAMPLES:

Select the command and then select a beam. The following dialog box "Preferences of Beam Reinforcement" is displayed:

Preferences of Beam Reinforcement			×
· · · · · · · · · · · · · · · · · · ·			
0.60m	63 (220)	0.46m	
OK Cancel	Support Width (m)	0	<< >>



On the common support, the positioned bars consist of two different bars. By clicking on the two bars, they are converted into one common bar.



In the figure on the left, the positioned bars to the corresponding spans, pass through the spans on both sides, on the top and bottom of the beam.

This means that during the calculation, the program takes into account the bars from both spans (top and bottom).

Editor Results

Exploration

Diagrams Detailing







2.2 Chec	k Reinforce	nent
謿	Single	
Check - Reinforcement *	Overall	

2.2.1 Single

Single: This command is used for the design of a beam or a continuity of beams. Select the command and use left click to select the beam continuity.

The program makes the design checks and displays the results using colors and symbols indicating, in this way, the type of the failure.

The <u>colors</u> indicating the beam's failure are the following:

Red. Failure in Bending. It has exceeded the maximum steel reinforcement ratio ρ_{max} .

Pink. Failure in Shear / Torsion.

Cyan. Passed the design checks.

The symbol on the beam indicates:

Failure in Bending	М
Failure in Shear	V
Failure in Torsion	Т
Dense positioned Stirrups	Asw
It has exceeded the maximum steel reinforcement ratio	ρ
It has exceeded the maximum anchorage length	ldb
Capacity Design	αcd
It has exceeded the maximum crack width	Wk
Deflection failure	L/d

2.2.2 Overall

Overall: This command is used to apply the design checks of the beams according to their final design. Select the command and in the dialog box activate either "Level" or "Total".

Design of Beams	Alignment-T 🗴
Level	C Total
ОК	Cancel





This command includes a sub-group of commands related to the calculation process and the display of the designing as a result of beam continuity.

2.2.3.1 Editor

This is a command which, however, in the later versions of SCADA Pro is completely replaced by the command **Detailing**.

Use **Detailing** for editing a beam continuity in an integrated environment of calculation and design. You can also apply Retrofitting methods and calculate the new moment diagrams.

Detailed instructions on how to use this command are mentioned in the relative user's manual (chapter B. Beam's Detailing)

2.2.3.2 Results

This command is used to display a TXT file that contains the main design checks and calculation results of a beam continuity.

Select the command and then left click on the centroid axis of the beam to open the TXT file and read the results.

The TXT files that are displayed are the same files generated by the program for the printout.



≝	B00028 - WordPad – 🗖	х
File Edit View Insert Format Help		
	B	
B5		^
SEISMIC	C DESIGN	
BEAM : B5 - MEMB.: 158 - Co	onnection (Nodes) Start:143 End :132	
KIND : T Width bw= 0.25 H	Heig.h= 0.55 Slab thick. hf= 0.15 Leng. L= 4.15	
CONC	CRETE : C12/15	
$ fck_{(Mpa)} = 1.60 \text{ trd}(Mpa) = 0.18$	1.0 maxec(N,M)=0.0055 maxec(N)=0.002	
REINE	FORCEMENTCover c(mm) = 25	
MAIN : B500C Es(Gpa)=200.	.00 fyk(Mpa)=400 ysu/yss=1.00/1.00 max gs=0.02	
STIRRUPS : B500C Es(Gpa)=200.	.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02	
BENDING WITH AXIAL VERIFIC.	START SUPPORT SPAN END SUPPORT	
	+TopBot+TopBot+TopBot	
Effective Length beff (m)		
Applied Axiai force NSD(KN)	-21.39 /.70 /.70 /.70 /.70 -21.39 406 94 -320 67 291 55 -257 11 428 16 -308 38	
Critical Load Combinations	24(F) 21(F) 21(F) 21(F) 21(F) 21(F) 24(F)	
REQUIRED REINFORC. As (cm2)	22.91 16.42 17.14 13.24 23.97 15.80	
per Face/Critical Combin(cm2)		
	+	
SHEAR VERIFICATION	(WITHOUT SHEAR CAPACITY DESIGN)	
Seis.Shear force (KN) Start	minVSd= -8.0 / maxVSd= 242.8 = ζ=-0.03	
End	minVSd= -14.1 / maxVSd= -253.9 = ζ= 0.06	
	+-Start(Cr.Region)SpanEnd(Cr.Region)	
Beam Lengths I(m)	0.55 3.05 0.55	
Applied Shear force VEd (KN)	NO1ES+NO1ES+NO1ES	
Applied Tors, Moment TEd (KNM)		
Resist.without reinf.VRd.c(KN)		
[Resist.comp.struts VRdmax(KN)]	245.5 240.8 257.1	
Resist.tors.moment TRdmax(KNM)	66.1 64.9 69.3	
TEd/TRdmax + VEd/VRdmax <= 1.0		Υ.
For Help, press F1	NU	M //

A NEW WARNING SYMBOL FOR INEFFICIENCY OF THE ANCHORING LENGTH IN THE BEAMS.

The way of calculating the lbd anchorage lengths for different regulations is summarized below: The total lbd is calculated and divided into l1 and l2. L1 is the linear anchorage length, and l2 is the one rotated into the node.

NOTES:

- EC2 does not provide a minimum linear anchorage length but provides a minimum TOTAL anchorage length (l1 + l2) which is also called lb, min.
- EC8 in paragraph 5.6.2, among others, provides ONLY for DCH the anchorage length to be linear (exaggerating).

Based on the above, resulting:

1. For EC2 w/o EC8 scenario as well as for all EC with DCL and DCM ductility classes, it does not obey a minimum linear anchorage length lb, min, but checks the total length lbd regarding lb, min according to 8.4.4 Of the EC2.



So, there will never be an error message because if the anchorage length is bigger than the width of the support, the bar will reach the alignment and then return to the node.

- 2. For EC with DCH high ductility class, the use of the minimum linear anchorage length is allowed according to 5.6.2 of EC8. The error message will be displayed.
- Also, the affinity regions are now taken into account for calculating lb. The upper reinforcement is region II while the lower I.

2.2.3.3 Exploration

This command is used to display the TXT file that contains all design checks and calculation results of a beam. Select the command and left click on the centroid axis of the beam to open the TXT file and read the results.



8.4 Reinforcement detailing

- <u>____</u>
- Use this command for editing a beam continuity in an integrated environment of claculation and design. You're also able to apply Retrofitting methods and calculate the new moment diagrams.
- Detailed instructions on how to use this command are mentioned to the relative user's manual (chapter A. Beam's Detailing)



3. Capacity design



The command group "Capacity Design" contains commands for performing capacity design.

¹ Capacity Design always precedes columns and walls design if necessary.

3.1 Node Releases

This command is used to determine the node releases of the columns and walls.



3.1.1 Column

-

Specify the <u>node releases of the column</u> for the capacity design application.

Select the command and the column's nodes for changing their releases. Right click to end selection and the following dialog box opens:

-	
Node Designation	×
Direction y-z	<
Free	~
OK Cancel	

Direction y Direction z Direction y-z

The first list contains the directions for the capacity design application Direction y-z

The second list contains Fixed No Capacity Design the node releases for the capacity design application.

ATTENTION:

A Note that this command is used <u>only</u> for **columns'** nodes.

End Middle



3.1.2 Walls

Specify the node support releases of the <u>wall</u> for the capacity design application. Select the command and the column's nodes for changing the support conditions. Right click to end selection and the dialog box opens:

1	T	:						
	No	de Desig	gnation	×				
l	Dire	ection y-z		*				
l	Fre	e		*				
l		ОК	Cancel					
T	ne fi	irst list c	ontains t	he d	irections for capacit	ty design application	Direction y Direction z Direction y-z	
•••			oncome in the second	Fre En Mic	ddle			
Tł	ne s	econd li	st contai		ed Capacity Design	the node condition	for the designation.	
A	TTE	NTION: Note tha	at this cor	nma	nd is used <u>only</u> for	walls' nodes.		
N	OTE I i	f the us n both c	er skips t directions	he do s exco	efinition of the nod ept for the fixed on	e releases, the progr es.	am will consider them as fr	ee

3.2 Design



This command is used to apply the capacity design and display the results.

NOTES:

- 1 The beams design precedes the application of the capacity design
- The capacity design, always precedes the columns and the design of the wall, if it is necessary to be applied.
- A necessary precondition for both selective and total Design control is that beams have been designed and "Amplification" has been activated in Beams and Columns fields in the Structural Component Parameters window



3.2.1 Single

This command is used to apply the capacity design on a single node. Select the command and left click on the node.

3.2.2 Overall

This command is used to apply the capacity design on every node of the current level. Select the command and repeat the procedure for the rest of the levels.

3.2.3 Results

This command is used for the display of the TXT file that contains the results of the main design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results.

3.2.4 Exploration

This command is used for the display of the TXT file that contains all the results of the design checks of the capacity design. Select the command and left click on the node to open the TXT file and read the results.

4. Columns



The command group "Columns" contains commands about buckling check, design checks, steel reinforcement checks and results for columns and shear walls.

NOTE:



Columns' and walls' design always follows the capacity design; if it is necessary to be applied. First, the capacity design is applied on every required level, and then the design checks for columns and walls are performed.

4.1 Buckling

This command will be fully available in the next version.

4.2 Check Reinforcement

This is a command list related to the design checks for columns and walls resulting in their final design.

4.2.1 Single:



Select the command and left click to select one or more columns or walls. The program performs the appropriate design checks and displays the results by colors and symbols indicating the type of failure.

A colored dot is displayed in the center of the element. The <u>color</u> changes according to the type of failure as follows:

- Red: Failure caused by biaxial bending. The steel reinforcement exceeded the maximum ratio of 4%. Dense stirrups. No results are displayed.
- Pink: Failure by Shear / Torsion or exceedance of the ductility level. The results show the reason for failure.
- **Cyan:** All design checks are verified.

The initially indicated type of failure appears above the element as well:

Failure by biaxial bending	M-N
Failure by Shear	V
Confinement failure	ωwd
Buckling failure	λ
Failure by Torsion	Т
Dense Stirrups	Asw
Exceedance of 4% steel reinforcement ratio	ρ
Exceedance of the ductility index	ν

4.2.2 Overall

This command is used for the calculation of the final columns and/or walls design in total.



Select the command and the following dialog box is displayed:

Desi	gn Total	х
 Level 	○ Total	
Columns	✓ Walls	
ОК	Cancel	

Select whether to design all columns/walls of the current level or the building in total.

4.2.3 Single Delete

This command is used to delete the previous calculations from one or more columns or walls.

Press the command and left click to select the columns and the walls.

4.2.4 Overall Delete

This command is used to delete the previous calculations from columns or walls of the current level or the building in total according to the selection in the dialog box.

Design Delete-Total		
 Level 	◯ Total	
Columns	✓ Walls	
ОК	Cancel	

4.3 Results







This command contains a list of commands related to the elaboration and the display of the results derived from the design checks of beams continuity.

4.3.1 Editor

Thi

This is a command which, however, in the later versions of SCADA Pro is completely replaced by the command **Detailing**.

Use **Detailing** for editing a column in an integrated environment of verification and design. You can also apply Retrofitting methods and calculate the new moment diagrams.

 Detailed instructions on how to use this command refer to the related user manual (chapter B. Column's Detailing)

4.3.2 Results

Select to display the .txt file containing the main checks and design results of columns and walls. Select the command and left click on beam's axis to open the .txt file and read the results. The *txt files displayed are those generated by the program for the printout.



C00015 - WordPad – 🗆 🗙	H
	File Edit View Insert Format Help
SN====================================	==WITH MOMENT CAPACITY DESIGN===== COLUM: K8 - MEMB.: 15 - Connec KIND : T by=110 dz=25 dy=25 bz CONCRETH fck (Mpa)=25.00 ycu/ycs =1.50/1.0 r fctm (Mpa)= 2.60 trd (Mpa)=0.30 REINFORCH
200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02 200.00 fyk(Mpa)=500 γsu/γss=1.15/1.00 max εs=0.02 ITH AXIAL FORCE Critical combination 16 BOTTOM TOP	MAIN : B500C Es(Gpa)=200.00 f STIRRUPS : B500C Es(Gpa)=200.00 f BIAXIAL BENDING WITH AXIA POSITION
vd y: vd= 0.11 comb. 73 z: vd= 0.09 comb. 73 N) 927.82 893.25 M) y= 165.28 z=-1260.79 y= -104.08 z= -190.29 FORMATIONS ENVELOPE (0/00)	Max normalised axial force vd y: Applied Axial force NSd(KN) App.bend.moment MSd(KNM) y= CONCRETE DEFORMAT:
b. Deform. Apex Comb. Deform.	Apex Comb. Deform. Apex Comb. Defo: Column Bottom 1 72 -0.3160 2 0 -0.220 3 56 -0.1487 4 10 -0.352
-2.4261 5 0 -0.1987 6 13 -0.3460 -0.6098 7 4 -0.2907 8 20 -0.1503 C E VERIFICATIONWITH CAPACITY DESIGN ct minVsd= -191.75 / maxVsd= 191.95 = ζ= -1.00	5 8 -0.4919 6 15 -2.420 7 9 -2.4774 8 40 -0.609 SHEAR FORCE VI Seismic shear Y (KN) Start min
minVsd= -191.75 / maxVsd= 191.95 = ζ= -1.00 rt minVsd= 46.45 / maxVsd= 142.00 = ζ= 0.00 minVsd= 46.45 / maxVsd= 142.00 = ζ= 0.00 +-Bottom(crit.)SpanTop(critical)	End mir Seismic shear Z (KN) Start mir End mir
XN) 1738.5 1415.0 1738.5 1415.0 1738.5 1415.0 XN) 661.9 167.2 191.9 142.0 661.9 167.2	Seismic direction Design Shear resist. Vrd2(KN) 173 Applied Shear force VSd(KN) 6
XN) 240.5 162.6 291.4 268.7 212.0 178.1 XN) 421.4 449.8 (1 /24) (1 /66) (1 /9) (1 /41) (1 /24) (1 /66) Max Applied Torsional Mom. (KNM) Tsd : 0.20	Shear Kesistance vcd (KN) 24 Shear for stirrups Vwd (KN) 42 Shear critical combinations (1 / TORSION WITH SHEAR VERIFIC. Max
IM) without reinforcement Trd1: -0.06 [IM) 1088.4 885.9 750.6 611.0 1088.4 885.9 IM) 135.6 107.5 93.5 74.1 135.6 107.5 V	Tors.moment resistance (KNM) 0 Stirr.tors.mom.res. Trd2 (KNM) 10 Strut tors.mom.res. Trd3 (KNM) 13
<pre>httivsd= -191.37 / max/sd= 191.35 - (1.05) t minVsd= 46.45 / max/sd= 142.00 = (- 0.00) minVsd= 46.45 / max/sd= 142.00 = (- 0.00)+-Bottom(crit.)SpanTop(critical) +YY</pre>	Seismic shear Z (KN) Start mir End mir End mir Seismic directionBot Design Shear resist. Vrd2(KN) 173 Applied Shear force VSd (KN) 64 Shear Resistance Vcd (KN) 24 Shear for stirrups Vwd (KN) 42 Shear critical combinations (1 / TORSION WITH SHEAR VERIFIC. Max Tors.moment resistance (KNM) 43 Stirr.tors.mom.res. Trd2 (KNM) 10 Strut tors.mom.res. Trd3 (KNM) 13

4.3.3 Exploration

Select this command to display the TXT file containing the results of all design checks of columns and walls. Select the command and left click on the beam's axis to open the TXT file and read the relative results.



₿	C00015 - WordPad	- 🗆 🗙
File Edit View Insert Format Help		
	n 🖬	
COMP N M	M- 17- M-	<u> </u>
COMB N MY	-25.97 0.47 155.97 -0.02	
End 1 $1676 25 - 164 95$	-27.44 0.47 155.97 -0.02	
Start 2 1068.16 208.07	-16.11 0.16 99.48 -0.01	
End 2 1033.60 -106.29	-16.62 0.16 99.48 -0.01	
Start 3 1080.55 239.73	519.65 178.24 107.71 -0.04	
End 3 1045.99 -89.34	45.14 178.24 107.71 -0.17	
Start 4 964.30 234.82	518.40 177.60 105.81 -0.04	
End 4 929.73 -90.49	44.28 177.60 105.81 -0.17	
Start 5 1085.75 249.29	539.29 184.57 110.17 0.05	
End 5 1051.19 -87.58	44.79 184.57 110.17 -0.08	
Start 6 969.50 244.37	538.04 183.93 108.28 0.05	
End 6 934.93 -88.73	43.93 183.93 108.28 -0.08	
Start 7 1086.68 250.92	542.63 185.62 110.58 0.06	
End 7 1052.12 -87.23	44.79 185.62 110.58 -0.07	
Start 8 970.43 246.00	541.38 184.98 108.68 0.06	
End 8 935.87 -88.38	43.93 184.98 108.68 -0.07	
Start 9 1091.88 260.47	562.27 191.95 113.05 0.15	
End 9 1057.32 -85.47	44.44 191.95 113.05 0.02	
Start 10 975.63 255.56	561.02 191.31 111.15 0.15	
End 10 941.07 -86.61	43.58 191.31 111.15 0.02	
Start 11 1037.94 159.01	531.27 181.82 85.30 0.04	
End 11 1003.38 -105.04	42.61 181.82 85.30 -0.08	
Start 12 921.69 154.10	530.03 181.18 83.40 0.04	
End 12 887.12 -106.19	41.74 181.18 83.40 -0.08	
Start 13 1032.74 149.46	511.63 175.50 82.84 -0.05	
End 13 998.18 -106.80	42.96 175.50 82.84 -0.17	
Start 14 916.49 144.54	510.39 174.86 80.94 -0.05	
End 14 881.92 -107.95	42.10 174.86 80.94 -0.17	
Start 15 1044.07 170.20	554.25 189.21 88.17 0.14	
End 15 1009.51 -102.93	42.26 189.21 88.17 0.02	
Start 16 92/.82 165.28	353.01 188.57 86.27 0.14	
End 10 093.23 -104.08	TI.37 100.37 00.27 U.UZ	
End 17 1004 31 -104 60	42 61 192 99 95 71 0.00	
Start 18 022 62 155 72	12.01 102.00 00./1 -U.U0	
Start 10 522.02 133.73	555.57 152.24 65.61 0.06	*
		>

4.3.4 Detailing

- Use this command for editing a beam continuity in an integrated environment of verification and design. You can also apply Retrofitting methods and calculate the new moment diagrams.
- Detailed instructions on how to use this command refer to the related user's manual (chapter B. Column's Detailing)



4.3.5 Strength Calculation (Pushover)

This command refers to the nonlinear static analyses (Pushover analyses). First, the existing structure is designed by the design provisions that were active the time of the construction of the structure. This means that the dimensions of the structural elements as well as the existing steel reinforcement are defined. Next step is the performance of the Pushover analysis. Nevertheless, before the Pushover analysis, the "**Strength Calculation**" procedure precedes, by selecting the corresponding command.

Through this command, the program calculates the interaction diagrams between bending moment (M) and axial force (N) for all columns of the structure in all levels.

Desi	gn Total 🛛 🗙
 Level 	◯ Total
Columns	✓ Walls
ОК	Cancel

Press the command and in the dialog box choose the columns and/or the walls of the current level or the total building, to calculate the new interaction diagrams M-N.

Wait until the program completes the calculation of all selected items.



▲ The calculation of the interaction diagram M-N can also be performed for a single column or wall, using the command "Detailing" (see the corresponding **chapter B. Column's Detailing**).


4.3.6 Check-Nodes

-

In the new version of SCADA Pro, the shear strength check of the node was added according to par. 7.2.5. Of KAN.EPE.

The two checks to execute are:

- Diagonal tension cracking
- Failure in diagonal ccompression



NOTE:

- 1 This check concerns only the Greek KAN.EPE norm and is no valid in other countries.
- 1 It works only in the Greek version.



5. Footing

The "Footing" command group contains commands for footing design check, design calculation, editing and the respective results.



5.1 Check Reinforcement



This command contains a list of commands related to the calculation of the footing designing.

5.1.1 Single



Select this command and then left click to select one or more footings. The program performs the design checks and the corresponding results are displayed by colors and symbols that indicate the type of failure.

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

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

The type of failure is mentioned with a symbol as well:

Failure in Bending	М
Failure in Shear	V
Punching failure	vp
Soil failure	σuls
Damage bearing elements	∆/I
Eccentricity	E

A necessary precondition for the footing design is columns design in level 1.

5.1.2 Overall

Select the command to check all footings on the current level (foundation).



5.1.3 Single Deletion

×**∏1**

This command is used to delete the results of the previous design from one or more footings. Select the command and then left click to select the footings.

5.1.4 Overall Deletion

This command is used to delete the results of the previous design from all footings in the foundation level.

5.2 Results



This command contains a list of commands about editing the designed footings.

5.2.1 Editor

It's an editing tool used to modify the results of the footing design process. Select the command and left click on an already designed footing. Then, the following dialog box is displayed:

		Footings Editor		×
Footing	1	▲ · · · · · · · · · · · · · · · · · · ·		
Ly (cm)	320			
Lz (cm)	320			
H (cm)	100			
u (cm)	0			
αy (cm)	65	×		
<		>		
Bars S	oil Stresses			
Direction Φ	per (cm) Num	nber		
Y-Y 16	v 10 28			
Z-Z 18	v 12 24			
Load Combinations				
Comb:1(14) +1.35Lc	1+1.50Lc2	▲ · · · · · · · · · · · · · · · · · · ·		
Comb:2(1) +1.00Lc1	+0.50Lc2	2/		
Comb:3(2) +1.00Lc1	+0.30LC2+1.00LC3+0.	or v OK	Cancel	lax σso,(kN/m2)
				_
		the fellowing	Bars	Soil Stree
ne two mair	i choices are	the following		

If you choose the button "Bars", the steel reinforcement section is activated:





Here you can view and modify the reinforcing bars of this footing.

Select a new diameter and type the spacing. Colors correspond to the respective bars shown in the figure.

The number of the bars presented in the "Number" column, changes automatically according to the new spacing value.

In the upper left section, a table is displayed with the title and the values of the geometric data of the footing, and other data related to soil.

Footing	1	\sim	dz (cm)	205		\land
Ly (cm)	320		hsoil (cm)	0		
Lz (cm)	320		Footing Weight (KN)	256.00		
H (cm)	100		Soil Weight(KN)	0.00		
u (cm)	0		σal.(kN/m2) (102)	121.712		
αy (cm)	65	\sim	ofr.(kN/m2) (36)	385.601		\mathbf{v}
<		>	<		>	

If you click the "Soil Stresses" button the section of the load combinations is activated:

Comb:1(14) +1.35Lc1+1.50Lc2 Comb:2(1) +1.00Lc1+0.50Lc2 Comb:3(2) +1.00Lc1+0.30Lc2+1.00Lc3+0.3(Comb:4(2) +1.00Lc1+0.20Lc2+1.00Lc3+0.3(

In the figure on the right, the soil stresses on the four vertices of the footing are displayed, indicating the load area of the stress resultants of the footing, according to the sketch below.



Comb:1(14) +1.35Lc1+1.50Lc2 Comb:2(1) +1.00Lc1+0.50Lc2 Comb:3(2) +1.00Lc1+0.30Lc2+1.00Lc3+0.3(Comb:4/2) +1.00Lc1+0.20Lc2+1.00Lc3+0.3(Comb:4/2) +1.00Lc1+0.20Lc2+1.00Lc3+0.3(Choose a combination and scroll the list to read the respective stresses σ_1 , σ_2 , σ_3 , σ_4 and the critical stresses σ_{al} and σ_{fr} :



ofr.(kN/m2) (36)	385.601	^	σal.(kN/m2) (102)	121.712		^
Stresses (kN/m2)	0		ofr.(kN/m2) (36)	385.601		
σ1	134.442		Stresses (kN/m2)	0		
σ2	156.188	- 64	σ1	134.442	- 1	
σ3	167.607		σ2	156.188	- 1	
σ4	145.862	~	σ3	167.607	•	4
<		>	<		>	

Finally, press the button "Max σ so (kN/m2)"

Max oso.(kN/m2) to display the maximum stress

Comb:34(5) +1.00Lc1+0.30Lc2--1.00Lc3--0.: Comb:35(6) +1.00Lc1+0.30Lc2+0.30Lc3+1.(

developed, and the load combination of the origin. Combisition of the origin.

5.2.2 Results



Select this command to display the TXT file that contains the main design checks of footings and the corresponding results. Select the command and left click on a footing to open the TXT file and read the results.

The displayed TXT files are those generated by the program for the printout.

5.2.3 Exploration



Select this command to display the TXT file that contains all design checks for footings and the corresponding results. Select the command and left click on the beam's centroid axis to open the TXT file and read the results.

6. Slabs-Mesh



The "Slabs-Mesh" command group includes commands related to the analysis of slabs with the strip method and the corresponding results and commands to insert, delete, edit and generate a mesh.

6.1 Slab Strip Calculation



Slab Strip Calculation This command list is related to the analysis of slabs with the strip method.



6.1.1 Single

Select the command and left click on the strip.

The slab strips are analyzed, the stress resultants are calculated and the designing of the slab is performed. The program calculates the tension (Fe) and compression (Fe') and the steel reinforcement in cm². Also the reinforcing bars in span, additional and secondary reinforcement and stirrups, for solid and Zoellner slabs, are calculated.

6.1.2 Overall

To calculate all slab strip of the current level.

6.1.3 Single (Load Pattern)

Perform analysis of the selected strips by the load pattern as well.

6.1.4 Overall (Load Pattern)



Perform analysis of all the current level strips by the load pattern as well.



6.2 Flat	Slabs
6.2.1	lat Slabs
Slab Strip Calculation •	Image: Steel Timber abs * Steel Timber Design * Masonry Member Design *
Slabs	Flat slabs
A 🗹 🔗 🛛	Punching shear check 🔸 📚 Calculation of design strips
	Mesh Display X
	Composite Slab Display Z
	Diagrams X
	Diagrams Z
	Results

The new version of SCADA Pro offers the possibility of creating flat slabs (slabs without the presence of beams) with the finite element method.

The procedure for the modeling of flat slabs requires:

- the 3D Mesh definition,
- the External Boundary creation,
- the Holes automatically creation in place of the Columns,
- the Mesh calculation and the mathematical model calculation.





The command "Flat slabs" includes the commands:

6.2.1.1 Parameters

Flat Slab Design Pa	×	
Layer		
Flat	Flat Slab	\sim
Drop Panel	Drop Panel	\sim
Support Line xx	Support Line xx	\sim
Support Line zz	Support Line zz	\sim
	OK Cancel	
	Flat Slab Design Pa Layer Flat Drop Panel Support Line xx Support Line zz	Flat Slab Design Parameters Layer Flat Flat Slab Drop Panel Drop Panel Support Line xx Support Line xx Support Line zz OK

In the dialog box, you define the correlation between Layers.



Current	s circles				Levels XZ - St	toreys
New Flat S	lab				Update]
Number	Visibl	e Editable	Colour	^	Select All	
Timber Secondary Colu	mns 🖸	∎°	4			
Timber Hor.Wind bracin	ngs 🖸	∎°	7		Deselect All	
Timber Vert.Wind braci	ngs 🔯	∎°	8		N Code La	
_	Ø	∎°	8		VISIDIE	
Flat Slab	a	e	8		Non Visible	
Drop Panel	a	₽	8			
Support Line xx	Ø	∎°	8		Editable	
Support Line zz	Ø	∎°	8			
				\sim	Non Editable	
Delete Data						
All Model	By Level XZ	By Lay	er 🗌 M	odel Onlv	ОК	Cance

The default Layer's list of SCADA includes the layers related to the flat slabs.

- In "Flat Slab" layer transfer the outline of the slab and correspond it to the Layer "Flat".
- In "Drop Panel" layer transfer the Lines that define the area around the columns, where you will increase the thickness of the slab locally. The "Drop Panels" are inserted optionally around the columns of the slab relieving the fatigue from drilling.
- In "Support Lines xx" and "Support Lines zz" transfer the Lines that define the Support Lines. These lines are inserted in both X and Z directions between successive points of the slab. They usually connect column's nodes and end on the outline of the slab.





A Based on the designed Support Lines, the corresponding Design Strips will be generated



§ Instructions for inserting the support lines in flat slabs

- 1. The support lines should start from a column and end to a column (or free end). In any case, they must contain at least one column.
- 2. The support line must reach the outline of the slab as long as it is a free end. Otherwise, it may stop on the outline or the node of the column.
- 3. If the boundary conditions (that is, right and left of the support line) change along the line, the line must break at these points

For example, in the following case



On the supported Line 27, the red right line has not formed correctly (it is the red inclined line). This happened because a support line was inserted which:

At the right top, it is bordered by beams, that is, by the slab boundary, and so the program has delineated the red out area at this slab boundary (point 1)



At the bottom, in which the right border is another support line (19) the program calculated another length of red area (point 2)





For that reason, the inclined boundary of the red area emerged.

But, in case we had inserted two support lines, one for the upper part(with the slab boundary) and one for the bottom part (boundary with the support lines 17,18,19) the correct result is the following:



In general, the above algorithm, based on the latest improvements, is suitable for slabs with normal rectangular shapes and a cannulated array of columns. In slabs with strange shapes, overlapping loading strips may occur as well as gaps between them.

Two support lines are now formed, that is, the 28 and 32.



6.2.1.2 Calculation of design strips



According to Annex I of EC2 flat slab is divided into design strips. These are the areas that are automatically created by the program on both sides of the Support Line, according to Figure I.1 of EC2.

Select the command Calculation of design strips and the program automatically creates them.

Each Design Strip is divided into sections along its length perpendicular to the Support Line. In each section, SCADA integrates the internal forces of finite surface elements that intersect. By completing that, the bending moment around the axis of the section occurs. This intensive value is used to calculate the armature in each section.

6.2.1.3 Display X, Z



Design Strips along the X and Z axes







-						Page : 1			
Strip Calculations									
Value	Units	Code	Description	Value	Units	Code			
1			Starting point	corner	column	9.4.1&2			
1			Drop panel	Y	es				
X-X			Thickness	182.88	(cm)				
815.48	(cm)		Width		(cm)				
C20/25			Finishing point	internal	column	9.4.1&2			
20	(MPa)	Table 3.1	Drop panel	Y	es				
2.20	(MPa)	Table 3.1	Thickness	182.88	(cm)				
S400s			Width		(cm)				
400	(MPa)		Minimum reinforcement						
20	(mm)		Tension reinf.	0.00143	(cm ² /m)	9.2.1.1(1)			
0.25	(cm)		Compression reinf. (% of span reinf.)	25	%	9.3.1.2			
	Value 1 1 x-x 815.48 C20/25 20 2.20 S400s 400 20 0.25	Value Units 1 1 x-x 815.48 C20/25 (cm) 20 (MPa) 2.20 (MPa) S400s (MPa) 20 (MPa) S400s (cm) 0.25 (cm)	Value Units Code 1	ValueUnitsCodeDescription1Starting pointDrop panel1Drop panelThicknessx-xThicknessWidthC20/25Finishing point20(MPa)Table 3.12.20(MPa)Table 3.1S400sWidth400(MPa)Minimum reinforcement20(mm)Tension reinf.0.25(cm)Compression reinf.(% of span reinf.)(% of span reinf.)	ValueUnitsCodeDescriptionValue1Starting pointcorner1Drop panelYx-xThickness182.88815.48(cm)WidthC20/25Finishing pointinternal20(MPa)Table 3.1Drop panelY2.20(MPa)Table 3.1Thickness182.88S400sWidth182.8820(MPa)Table 3.1Thickness182.88S400sOMinimum reinforcementV.UU 14.320(mm)Tension reinf.V.UU 14.320(cm)Compression reinf.250.25(cm)Compression reinf.25	ValueUnitsCodeDescriptionValueUnits1Starting pointcorner column1Drop panelYesx-xThickness182.88(cm)815.48(cm)Width(cm)C20/25Finishing pointinternal column20(MPa)Table 3.1Drop panelYes2.20(MPa)Table 3.1Thickness182.88(cm)S400sMinimum reinforcement(cm)(cm)(cm)20(mm)Compression reinf.0.25(cm)25%Compression reinf.%540%			

Then the reinforcement results are displayed above and below in detail for each zone, dividing them into sub-zones.

- Left-Right -> red zone
- L-C R-C-> blue zone
- Center-> light blue zone





Analysis Results and Reinf					orcement				Тор	
		203.8	7 cm (L _{start})			407.74 cm (L _{centre})				
Zone	M (kNm)	Width (cm)	A _{srqd} (cm²/m)	A _{e,grvd} (cm ² /m)	Ф/s	M (kNm)	Width (cm)	A _{s.rqd} (cm ² /m)	A _{s.grvd} (cm ² /m)	Φ/s
Left		400.0		3.246	8/15		401.1		0.812	8/20
L-C		400.0		3.246	8/15		85.3		1.763	8/20
Center	-80.283	46.0	27.271	27.271	14/5		170.5		6.818	8/7
R-C							85.3		2.043	8/20
Right							103.8		1.471	8/20
		203.8	7 cm (L _{and})							
Zone	M (kNm)	Width (cm)	A _{erqd} (cm ² /m)	A _{e,prvd} (cm ² /m)	Φ/s					
Left	-88.070	401.1	2.873	3.246	8/15					
L-C	-44.824	85.3	7.054	7.054	8/7					
Center	-152.524	170.5	12.422	12.422	10/6					
R-C	-51.588	85.3	8.172	8.172	8/6					
Right	-45.848	103.8	5.886	5.886	8/8					
	•		Analysis I	Results ar	nd Reinf	orcement			Bot	tom
		203.8	7 cm (L _{start})				407.74	cm (L _{centre})		
			•	A		м	Midth	٨	Δ.	
Zone	M (kNm)	Width (cm)	(cm ² /m)	(cm ² /m)	Φ/s	(kNm)	(cm)	(cm ² /m)	(cm ² /m)	Φ/s
Zone Left	M (kNm) 9.207	Width (cm) 400.0	A _{εrqd} (cm ² /m) 0.294	(cm ² /m) 0.812	Φ/s 8/20	(kNm) 70.543	(cm) 401.1	(cm ² /m) 2.293	(cm ² /m) 3.246	Φ/s 8/15
Zone Left L-C	M (kNm) 9.207 9.207	Width (cm) 400.0 400.0	(cm ² /m) 0.294 0.294	(cm ² /m) 0.812 0.844	Φ/s 8/20 8/20	(kNm) 70.543 21.929	(cm) 401.1 85.3	(cm ² /m) 2.293 3.377	(cm ² /m) 3.246 3.377	Φ/s 8/15 8/14
Zone Left L-C Center	M (kNm) 9.207 9.207 80.591	Width (cm) 400.0 400.0 46.0	Asrqd (cm ² /m) 0.294 0.294 27.408	(cm ² /m) 0.812 0.844 27.408	Φ/s 8/20 8/20 14/5	(kNm) 70.543 21.929 43.857	(cm) 401.1 85.3 170.5	(cm ² /m) 2.293 3.377 3.377	(cm ² /m) 3.246 3.377 3.377	Φ/s 8/15 8/14 8/14
Zone Left L-C Center R-C	M (kNm) 9.207 9.207 80.591	Width (cm) 400.0 400.0 46.0	Asred (cm ² /m) 0.294 0.294 27.408	(cm ² /m) 0.812 0.844 27.408	Φ/s 8/20 8/20 14/5	(kNm) 70.543 21.929 43.857 21.929	(cm) 401.1 85.3 170.5 85.3	(cm ² /m) 2.293 3.377 3.377 3.377	(cm ² /m) 3.246 3.377 3.377 3.377	Φ/s 8/15 8/14 8/14 8/14
Zone Left L-C Center R-C Right	M (kNm) 9.207 9.207 80.591	Width (cm) 400.0 400.0 46.0	Asred (cm²/m) 0.294 0.294 27.408	(cm ² /m) 0.812 0.844 27.408	Φ/s 8/20 8/20 14/5	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ² /m) 2.293 3.377 3.377 3.377 3.284	(cm ² /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right	M (kNm) 9.207 9.207 80.591	Width (cm) 400.0 400.0 46.0 203.8	Asred (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end})	(cm²/m) 0.812 0.844 27.408	Φ/s 8/20 8/20 14/5	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ² /m) 2.293 3.377 3.377 3.377 3.284	(cm ² /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone	M (kNm) 9.207 9.207 80.591	Width (cm) 400.0 46.0 203.8 Width (cm)	Asrad (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end}) Asrad (cm ² /m)	(cm ² /m) 0.812 0.844 27.408 A _{s.gnd} (cm ² /m)	Φ/s 8/20 8/20 14/5 Φ/s	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ⁷ /m) 2.293 3.377 3.377 3.377 3.284	(cm ⁷ /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone Left	M (kNm) 9.207 9.207 80.591 80.591 M (kNm) 17.335	Width (cm) 400.0 46.0 203.8 Width (cm) 401.1	Astad (cm²/m) 0.294 0.294 27.408 7 Ccm (Lend) Astad (cm²/m) 0.555	(cm ² /m) 0.812 0.844 27.408 A _{s.grvd} (cm ² /m) 0.812	Φ/s 8/20 8/20 14/5 Φ/s 8/20	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ⁷ /m) 2.293 3.377 3.377 3.377 3.284	(cm /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone Left L-C	M (kNm) 9.207 9.207 80.591 80.591 M (kNm) 17.335 6.505	Width (cm) 400.0 46.0 203.8 Width (cm) 401.1 85.3	A _{kted} (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end}) A _{kted} (cm ² /m) 0.555 0.984	(cm ² /m) 0.812 0.844 27.408 A _{e.grvd} (cm ² /m) 0.812 0.984	Φ/s 8/20 8/20 14/5 Φ/s 8/20 8/20 8/20 8/20	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ⁷ /m) 2.293 3.377 3.377 3.377 3.284	(cm /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone Left L-C Center	M (kNm) 9.207 9.207 80.591 80.591 M (kNm) 17.335 6.505 23.135	Width (cm) 400.0 46.0 203.8 Width (cm) 401.1 85.3 170.5	Asred (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end}) Asred (cm ² /m) 0.555 0.984 1.762	(cm ² /m) 0.812 0.844 27.408 A _{e.grvd} (cm ² /m) 0.812 0.984 1.762	Φ/s 8/20 8/20 14/5 Φ/s 8/20 8/20 8/20 8/20 8/20 8/20	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ⁷ /m) 2.293 3.377 3.377 3.377 3.284	(cm /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone Left L-C Center R-C	M (kNm) 9.207 9.207 80.591 80.591 M (kNm) 17.335 6.505 23.135 3.583	Width (cm) 400.0 46.0 203.8 Width (cm) 401.1 85.3 170.5 85.3	Asred (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end}) Asred (cm ² /m) 0.555 0.984 1.762 0.539	(cm ² /m) 0.812 0.844 27.408 A _{e.grvd} (cm ² /m) 0.812 0.984 1.762 0.844	Φ/s 8/20 8/20 14/5 Φ/s 8/20 8/20 8/20 8/20 8/20 8/20 8/20 8/20	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm ⁷ /m) 2.293 3.377 3.377 3.377 3.284	(cm /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15
Zone Left L-C Center R-C Right Zone Left L-C Center R-C Right	M (kNm) 9.207 9.207 80.591 80.591 M (kNm) 17.335 6.505 23.135 3.583 3.311	Width (cm) 400.0 46.0 203.8 Width (cm) 401.1 85.3 170.5 85.3 103.8	Asred (cm ² /m) 0.294 0.294 27.408 7 cm (L _{end}) Asred (cm ² /m) 0.555 0.984 1.762 0.539 0.409	(cm /m) 0.812 0.844 27.408 A.grvd (cm /m) 0.812 0.984 1.762 0.844 0.821	Φ/s 8/20 8/20 14/5 Φ/s 8/20 8/20 8/20 8/20 8/20 8/20 8/20 8/20	(kNm) 70.543 21.929 43.857 21.929 25.982	(cm) 401.1 85.3 170.5 85.3 103.8	(cm /m) 2.293 3.377 3.377 3.377 3.284	(cm /m) 3.246 3.377 3.377 3.377 3.284	Φ/s 8/15 8/14 8/14 8/14 8/15

6.2.2 Punching shear checks





The punching checks are also added in the new version by EC2. The check is part of the check and design process of the flat slabs which runs automatically, but can also run as an individual check for any column. All data can be set automatically or even manually.



By Selecting "By Pick" command with the left mouse button point the node of a column and rightclick to open the dialog box where you specify all the necessary parameters.

Here are the explanations for the other positions of the columns on the perimeter of the slab:

unching Shear Check	
Control node	combinations ΔN(kN) 626.50 ΔMy(kNm) 34.283 ΔMz(kNm) 76.143
	Distributed Load. (kN/m2) 0
Materials (MPa)	Outlines of floors
Automatic	fck 20 fyk 400 Flat Slab ▼
Loaded surface	Position of loaded surface
Automatic	▼ c1(cm) 46 c2(cm) 46.00 Automatic ▼ ax 0 ay 0
Slab's elements Thickne Automatic Cover Automatic	▼ t(cm) 40.64 X ▼ ↓ upper(cr 2 Abov Φ 10 / 15 Φ 10 / 15 bottom(cr 2 Below Φ 10 / 15 Φ 10 / 15
Coefficient β	Reinforcement Calculation Type Radial Number of radii per quartile Calculation OK Results Deletion of the Report Cancel
Control node	The number of the selected node is filled

automatically and is not editable.

Combinations							
Combinations	N(kN)	626.50	∆My(kNm)	34.283	∆Mz(kNm)	76.143	
			Distributed I	.oad.(kN	/m2)	0	

In Combinations field:

40

- By choosing Combinations, the program finds automatically the combination that gives the worst Axial ΔN resulting and displays its value along with the corresponding moments.
- Choosing User, allows user values to be set for axial force and moments, in their respective fields, as well as the definition of a Distributed Load

50 Distributed Load.(kN/m2)

that works "relieving" the slab at that point, so the shear calculation be impaired relative to the original.



Shear force (V _{EdJrit.})	626.5	(kN)	
Distributed load (p)	50.0	(kN/m ²)	
Reduced shear force (V _{Ed.fn.})	478.8	(kN)	
Materials (MPa)			In the field Material, coefficients fck and
Automatic N	fck 20	fyk 400	fyk are filled in automatically with the
Automatic			Automatic option or defined by the user
_ L User			with the User option.
Outlines of floors	In O	utlines of Floors	s, choose the Layer containing the contour
Flat Slab	🧹 of th	e slab including	the columns located in the outline.
Mathematical Model	📩 Ther	n select the layer	Flat Slab that includes exactly the lines that
Surface Elements	defir	ne the overall co	ontour of the slab.
Mesh 2D			
Slabs-Strips			
Stool Columns			
Steel Beams			
Main Beams			
Purlins			
Secondary Columns			
Hor.Wind bracings			
Vert.Wind bracings			
– Timber Columns			
Timber Beams			
Timber top main beams			
Timber Purlins Timber Cirders			
Timber Secondary Columns			
Timber Hor.Wind bracings			
Timber Vert.Wind bracings			
Flat Slab			
Drop Panel			
Support Line xx			
perioramma			
perigrammaOpis	\sim		
Loaded surface		As the	loaded surface is defined the equivalent
Automatic c1(cm	n) 46 c2(cm)	46.000 surface	of the selected column.
gRectangular		Steel Rei	
Circular	05 A		
Duchessing			

By choosing:

- Automatic, the program calculates the surface of any column-shaped by a reduction in the equivalent rectangular with the corresponding dimensions c1 and c2.
- Rectangular, the user defines his dimensions c1 and c2 for calculating the loaded rectangular surface
- Circular, the user defines his diameter c1 for calculating the loaded circular surface





The Position of the loaded surface may be determined either automatically or selectively. It depends on the contour of the slab and the position of the selected column in it.

The proposed positions are:

- Interior
- Side: 4 directions
- Salient: 4 directions
- Reentrant: 4 directions

Select the position of the selected column and set the distances from the perimeter, ax, and ay, (except indoor) according to the following figures:



INTERIOR













Coefficient β Automatic appro: Automatic approxima Automatic theoretical User The Coefficient $\boldsymbol{\beta}$ for punching calculation can be calculated automatically in two ways:

- Automatic approximation or
- Automatic theoretical.

The Approximation way is a function of the position of the loaded area and the ax, ay.
 The Theoretical way is a function of the Moments My, Mz.

User selection allows the introduction of any value for the coefficient β .

Reinforcem	ient	
Туре	Radial	ĸ
	Radial	15
Number of	Cruciform	
quartile		

Finally, in Reinforcement set the layout type of punching reinforcement choosing between Radial and Cruciform.

For the Radial layout set the number of radii per quartile.

In the figure below the black box contains one quartile of the radial layout. The first perimeter has three radii of reinforcement, while there is an increase on the 3rd perimeter to 5 radii due to distance limitations set by the code. SCADA Pro automatically checks whether the distance limitations are satisfied on the first perimeter, and increases the number of radii when needed (even when the user's choice is not enough for the first perimeter).





Calculation The Calculation command performs all the necessary checks for punching, taking into account all the above parameters.

Results 🔀 This command displays the results file:

	Input Data										
Description	Value	Units	Description	Value	Units						
Level - Storey	1		Eccntr. factor (1.150							
# ofnode	40		Slab depth	40.6	(cm)						
Combination	1		Cover of reinforcement	2.0	(cm)						
Shear force (V _{Ed.Init.})	626.5	(kN)	Par aiza (autor lavar)	10	(mm)						
Distributed load (p)	0.0	(kN/m ²)	Bar size (outer layer)	10	(iiim)						
Reduced shear force (VEd,fn.)	626.5	(kN)	Spacing of bars (outer layer)	15.0	(cm)						
Bending moment (M _x)	34.3	(kNm)	Per size (assend layer)	10	(mm)						
Bending moment (M _r)	76.1	(kNm)	Dar size (second layer)	10	(mm)						
Shape of loaded area	Rectangular		Spacing of bars (second	15.0	(0m)						
ci length (along x axis)	46.0	(cm)	layer)	15.0	(ciii)						
c ₂ length (along y axis)	46.0	(cm)	Concrete (f _{sk})	20.0	(MPa)						
c diameter		(cm)	Steel (fyk)	400.0	(MPa)						
Position of loaded area	Interior		Reinforcement pattern	Radial							
Dist. of slab perim. along x (a _x)		(cm)	# of radii of reinforcement in a	2							
Dist. of slab perim. along y (a _y)		(cm)	quadrant (circular pattern)	2							

Input Data: list of all elements identified in the previous window and required to punching check.

Punching Reinforcement Layout: according to predetermined parameters and if, there is reinforcement requirement.





In Check results there are two different checks:

					С	heck re	esults				-	aye
Descriptio	on	1	Value	Units		EC2	Descript	ion	Valu	e Unit	8	EC2
Effective	depth of sla	b (d)	37.6	(cm)	(ed	q6.32)	Basic co	ntrol perimete	r 657.	0 (cm		fig6.
Perimeter area (u ₀)	of the loade	be	184.0	(cm)	(ed	q6.53)	(u1) Design v	value of the	0.29	1 (MPa	a) (eq6.3
Design va stress at u Maximum	IUE OF THE SF I ₀ (V _{Ed.0})	hear .	1.040	(MPa)	(e	q6.38)	Punch. s without s	Punch. shear resistance without shear		6 (MPa	a) (ea6.4
resistance	(V _{Rd,max})	lear	3.680	(MPa)	(e	q6.53)	reinforce	reinforcement (V _{Rdc})		6 (MP:		(eq6.)
l st check:	VEd,0 ≤ VRd,ma	x	Sufficie	ncy			2 nd chec	k: v _{Ed,1} ≤ v _{Rd,c}	No re requi	inforcem red	ent is	(040.
							•		•			Page
					C	heck r	esults					
Descripti	on depth of als	ab (d)	Value	Units	(0	EC2	Descrip	tion	Valu	le Unit	s	EC2
Perimeter	of the load	ad (D) di	37.0	(cm)	(e	q6.32)	Basic co	ontrol perimete	er 657	.0 (cm)	(fig6.
area (u ₀) Design va	alue of the s	hear	184.0	(cm)	(e	q6.53)	Design shear st	value ofthe ressatu ₁ (v _{ed}	1) 0.74	4 (MP	a) (eq6.3
stress at	u ₀ (V _{Ed,0})	ileal	2.657	(MPa)	(e	q6.38)	Punch.	shear resistan	ce			
Maximum resistance	punching s (V _{Rd,max})	hear	3.680	(MPa)	(e	q6.53)	without reinforce	shear ement (v _{Rd.c})	0.35	6 (MP	a) (eq6.4
							Constan	it (v _{min})	0.35	6 (MP	a)	(eq6.
1 st check:	VEd,0 ≤ VRd,ma	Sufficie	ency			2 nd chec	$k: v_{Ed,1} \leq v_{Rd,c}$	Reint - add reinfo - inco	forcement d punching prcement rease slat	t nece g she o long	essary ar jitudir	
		I							Tenn	Jicement		Page
					0	heck r	esults					-9°
Descripti	on		Value	Units	T	EC2	Descrip	tion	Valu	le Unit	s	EC2
Effective	depth of sla	ab (d) ed	37.6	(cm)	(e	q6.32)	Basic co	Basic control perimeter (u1)		657.0 (cm		(fig6.
area (u _o) Design va	alue of the s	hear	184.0	(cm)	(e	(eq6.53) Design value of the shear stress at u ₁ (v _{Ed,1})		1) 1.86	1.860 (MP		eq6.3	
stress at Maximum	u ₀ (V _{Ed,0})	hear	6.642	(MPa)	(e	qb. 38) Punch. shear resistance without shear 0.356		Punch. shear resistance without shear		6 (MP	a) (eq6.4
resistance	e (V _{Rd,max})	incui	3.680	(MPa)	(e	q6.53)	reinforce	ement (v _{Rd,c})				
1 st check:	I st check: v _{Ed0} ≤ v _{Rdmax} - - c			ise size ise slab f concre	ofthe l depth te of a	loaded higher	2 nd chec	$k: v_{Ed,1} \leq v_{Rd,c}$				
					Det	ailing r	esults					_
Descript	ion		Val	ue U	nits	EC2	Περιγραφ	ή	Τιμή	Μονάδες	E	2
Perimete	r u _{out,er}		483	1.7 (c	:m) (eq6.54)	Distance (d₃)		(cm)		
(a) - Dist of reinfor	ance of 1 [®] p cement∛∯on rea	erimete 1 the	r 11	.2 (0	:m)		Distance (Angle (φ)	(d ₄)	90.0	(cm)		
Limit: 0.3·d ≤ a	≤ 0.5·d		6.7	<=a<=1	1.2	(9.4.3)	(s _{tlast}) - Ta distance b	ngential etween link e last	54.4	(cm)		
(f) - Dista perimeter	ance of last r of reinforce	ment	29	.7 (0	:m)		perimeter	6 1431				_
from u _{oute}	r I= 1.5·d		33	6 (0	m)	(6 4 5)	Limit: 2.0-	d Kfalativa	44.8	(cm)		
(s _r) - Rac	lial distance	ofthe	16	.5 (c	:m)	(0.4.0)	(Tywd.er) - Et design str punching	ength of shear reinf.	306.0	(MPa)	(0.01	50)
Limit: 0.7	'5∙d		16	.8 (c	:m)	(9.4.3)	(A _{sw,1}) - Ne area ofa	ecessary link leg	0.535	(cm ²)	(ede	
(S _{L1}) - Tai between perimeter	igential dist link legs on r	ance the u ₁	54	.4 (c	:m)		(A _{sw,min}) - N area of a	/linimum link lea	0.535	(cm ²)	(eq9	.11)
Limit: 1.5 Distance	i∙d (α)		33	.6 (c	:m)	(9.4.3)	Diameter of chosen	of link leg	10	(mm)		
Distance	(d ₁)		24	.5 (c	:m)		Area of lin	k leg	0 785	(cm ²)		
Distance	(d ₂)		24	.5 (c	:m)		chosen		0.100	(em)		
		1	Gro	ouping	of pun	ching s	hear reinf	orcement				
Group	Number of lines	Φ (mm	i) N	lumber legs or	of link 1 line	He li	eight of ink leg (cm)	Perimeter wh 1 st link leg o line stan	ere the of the ds	Distance link leg f loaded	of the from ti i area	1" he
1	8	10		2			21.4	1		11.3	20	
	4			1			21.4			21.	10	

If the 1st check presents *Sufficiency*, then the 2nd check specifies the requirement or not of punching reinforcement.

If the 1^{st} check presents *Insufficiency*, then the 2^{nd} check is not performed and some interventions for the slab are proposed.

In **Detailing results** the results from the two checks by their respective formulas and chapters of EC2 are listed in detail.

In **Grouping of punching shear reinforcement** the posted punching reinforcement resulting from the above controls, and the characteristics of the layout are listed.





The 3rd page shows the Legend containing characteristics by the position of the Loaded Surface.

6.2.2.2 Overall



6.2.2.3 Edit



The Edit command allows you to change the parameters set by the "By Pick" check or "Overall" check.

Select the command and left-click on the node of the column to edit. Automatically the previous parameters window opens with the original settings for the selected column. You can make any changes you want and repeat the check using the command Calculate.

6.2.2.4 User check



The user has the opportunity to test different data on different nodes for a supervisory result image. This is a "draft" that is not saved in the printout but allows the user to make tests to reach the desired solution.





6.3 Results

This command contains a command list about editing and viewing the design checks' results.



6.3.1 Editor



It's an editing tool used to modify the results of the slabs' design. Select the command and left click on an already designed strip. A dialog box is displayed:



	Slab Re	einforce	ement Ed	litor		×	
Geometry Slab ∶∏5 Solid			Π	5		П3	
Thick.: 150 L1 : 4.21	Δ		20	Δ	Δ		
L2 : 3.50			J.,	70		4.70 << >>	
	SUPPOR Up b	r ottom	SI Up	PAN bottom	suppo Up	RT bottom	
Bending Moments		0.39		2.57	-19.93		
Required (cm2)	0.00	0.07	0.00	1.64	3.89	0.00	
Placed (cm2)	1.26	1.26	0.00	2.51	5.45	2.93	
Shears (kN)	-3.21				13.4	8	
Required (cm2)	0.00				0.00)	
Placed (cm2)	0.00				0.00)	
		REIN	FORCE	MENT BAR	t S		
Span			_Φ_	Φ8/20			
Support	_Φ_	_Φ_			Φ8/20	_ Φ _	
Stimups				Φ			
Secondary / Shear		Φ8/	/25	/ Φ8/	25		
0 Φ 🗸	Φ	~ /	0	OK		Cancel	

At the top left, you see the slab data such as name, size, etc. The dimensions are measured from axis to beam axis. Then the bending moments calculated for the supports and the corresponding span and the square centimeters of reinforcement calculated at the corresponding points are listed.

	SUPP Up	SUPPORT		PAN bottom	SUPPORT Up bottom		
Bending Moments		0.39		2.57	-19.93		
Required (cm2)	0.00	0.07	0.00	1.64	3.89	0.00	
Placed (cm2)	1.26	1.26	0.00	2.51	5.45	2.93	

The next section includes shear calculation and the corresponding required square centimeters.

Shears (kN)	-3.21	13.48
Required (cm2)	0.00	0.00
Placed (cm2)	0.00	0.00

The last section includes the corresponding reinforcing bars.

		REINFORCEMENT BARS							
Span		_Φ_ Φ8/20							
Support	_Φ_	_ Φ _		Φ8/20	_Φ_				
Stimups			_¢_						
Secondary / Shear		Φ8/25	/	Φ8/25					

When arming, the minimum reinforcement designated parameters are also taken into account.



Listed in order: span and supports reinforcements, stirrups for Zoellner slabs, as well as the secondary and shear reinforcement.

SCADA Pro always calculates the main reinforcement parallel to the strip direction and in the other direction calculates the secondary and shear reinforcement, regardless of how bending (in one or both directions).

To modify the calculated bars, click on the corresponding size, $\frac{\Phi^{10/7}}{2}$ and automatically activate the field at the bottom of the dialog box where you can select the new diameter and enter the

```
new distance \Phi 10 \checkmark / 7
```

6.3.2 Slab Strips



Select to display the TXT file that contains the design checks for slabs and the corresponding results.

Select the command and left click on a strip to open the TXT file and read the results. The TXT files presented, are generated by the program for the printout.

BENDING VERIFICATION	-F5 (EDGE)	F5 (SPAN)	F5-(EDGE)
	+TopBot	+TopBot	+TopBot
Bending Moment MSd(KNM)	1.19	17.20	1.19
REQ. REINFORCEMENT As (CM2)	0.00 0.16	0.00 2.34	0.00 0.16
-SHEAR VERIFICATION	+	+	++
Applied Shear force VEd (KN)	11.78 1	I	11.78 1
Resist.without reinf VRd,c(KN)	82.94		82.94
Compressed.trut res.VRdmax(KN)	575.56	1	575.56
REQ.ADDIT.SUPPORT BARS (CM2)	0.00	1	0.00
FINAL REINFORCEMENT As (CM2)	1.26 1.26	0.00 2.51	1.26 1.26
+	+	+	++
FINAL REINFORCEM. BARS		Φ 8 / 20	
+	+	+	+*

NOTES:

Respect the older versions, changes have also been made in the presentation of the reinforcement results.

Titles above "Tension" – "Compression" has changed to "Up" - "Down" and determine the position of the reinforcements on the slab.

Now only one moment value is written, the sign of which determines whether the armature will be inserted up or down

- For a positive moment value, the tension strength is below and the reinforcement is entered in the bottom.
- For a negative moment value, the tension strength is up and the reinforcement is entered on the top.

It is possible to have a positive value, mainly on the support, and a requirement for a compressive reinforcement, in which case the required reinforcements goes both up and down.



Especially for the Polish scenario, the slabs reinforcements are considered to be straight. This means that half of the reinforcement on the support is not considered, and where required, more support reinforcement is inserted.

NEW FEATURES:

In the newest SCADA Pro version, the slabs Deflection Control is also included. Deflection control based on 7.4.2 and 7.4.3 of EC2 is presented at the end of the results of each slab. The results of the two checks are displayed separately.

+				-DEFLECT	TION CON	TROL	(EC2 7	7.4.2	٤7.	4.:	3)			+
1	1/d	1/d	Suf.	Suggest	ted.min	1	Max. M	d	ul	I.	а	1/a	(perm.)	Suf.
1	- E	perm.	I	thick.	hs(mm)	- I	(kNm)	(m	m)	1		1	(mm)	1 1
1	+-		+	+	+	- +		+		-+-		+		+
j	34.59	80.10	YES	I 7	7 I		-7.64	1 0	.42	T	250	1	18.40	YES

From the first check a minimum recommended thickness results, but that can not be proposed for the initial slab recognition, because its reinforcement is required to calculate it.

The calculation of the sizes of the first check does not involve intensive forces, while the second check considers the serviceability combination (s).

A Red sign marks on slab symbol when the thickness is less than permitted by regulation

6.3.3 M Diagrams







6.3.4 Q Diagrams

Select the command to display a qualitative representation of shear diagrams over the slab strip, selected with a left click. The shear diagrams result from the load combination 1.35G+1.50Q, after the multiplication by qx and qz, for strips parallel to x or z axes, respectively.



6.3.5 M Diagrams (Load Pattern)

Select the command to display a qualitative representation of the moment diagrams over the slab strip, selected with left click, resulting from a load pattern.

6.3.6 Q Diagrams (Load Pattern)



Select the command to display a qualitative representation of shear diagrams over the slab strip, selected by left click, resulting from a load pattern.



7.	Steel			
Steel Design	Timber Design •	Masonry Design *	Mer Diagi	"Steel Design" command group contains commands for the cross-sections design, the buckling resistance, and the steel connections design.
	Cross-Sectio	on Design		
/	Buckling M	embers Inp	ut	
ſ	Cold Forme	d Sections		
+	Connection	s		
+	IDEA StatiCa	a Connectio	on	Always remember to calculate the corresponding load combinations in the parameters dialog box.

7.1.1 Cross Section Design

This command is used to check the adequacy of the steel cross-section.

eel Design (Layer)						
Name	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Secti
Steel Columns	IPE 450					
Steel Beams	IPE 360					
Main Beams	HEA 180					
Purlins	IPE 100					
Girders	IPE 100					
Secondary Columns						
Hor.Wind bracings						
Vert.Wind bracings	CHS 219,1X6,3					
-						
Timber Columns						
Timber Beams						
Timber top main beams						
Timber Purlins						
Timber Girders						
Timber Secondary Columns						
Timber Hor.Wind bracings						

102



The first column contains the layers of the current project and the other columns the cross-sections that belong to each layer.

- Select the button "Calculate All" for the calculation of all sections.
- Alternatively, select the layers one by one and then click the button "Calculate".

		Stee	el Design (La	ayer)				×
Name Steel Columns	Cross Sectio IPE 450	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	^
Steel Beams	IPE 330							
Surface Mesh								
Mathematical Model								
Surface Elements								
Mesh 3D								
Mesh 2D								
Slabs-Strips								
Steel Columns								
Steel Beams								
Main Beams	IPE 330							
Purlins	IPE 100							
Girders	IPE 100							
Secondary Columns								
Hor.Wind bracings	CHS 114.3							
Vert.Wind bracings	CHS 219.1							~
<							>	
Edit	Calculate		Calculate	All		Cancel	ОК	

Name	Cross Sectio						
Steel Columns	IPE 450						
Steel Beams	IPE 330						
Surface Mesh							
Nathematical Model							
Surface Elements							
Nesh 3D							
lesh 2D							
Blabs-Strips							
Steel Columns							
Steel Beams							
lain Beams	IPE 330						
Purlins	IPE 100						
Girders	IPE 100						
Secondary Columns							
lor.Wind bracings	CHS 114.3						
/ert.Wind bracings	CHS 219.1						
							>



Green color indicates that all sections of this layer satisfy the design criteria (stress/resistance \leq 1) red color that they don't.

To locate the inadequate members or just see the check results, select the layer and click "Edit".

					S	teel Des	ign - La	ayer Da	ta								×
Layer: Mai	in Beams		VERIFI	CATION OK	:					Capacity D	esign	Ampli	ificatio	'n			
Different Cr	oss	IPE	IPE 330 V										CK SEI	ECTIO	DN		
Description	Membe	Membi Comb. N Vy Vz Mx My Mz									N	м	v	Мх	M-N	M-V	M-V-N
Max N	161	1	27.30	0.26	-3.38	-0.00	-4.35	0.91		V							
Min N	152	1	-13.76	-0.18	-2.74	-0.00	-4.74	-0.49		V							
Max QY	157	37	-1.19	0.44	-1.34	0.00	-1.11	1.50		V							
Min QY	156	62	-3.10	-0.44	-2.00	-0.00	-3.38	-1.48									
Max QZ	160	1	27.30	-0.26	3.38	0.00	-4.35	0.91		V							
Min QZ	161	1	27.30	0.26	-3.38	-0.00	-4.35	0.91		V							
Max MX	153	5	-6.29	0.15	-1.30	0.00	-1.02	0.61		V							
Min MX	152	7	-6.29	0.18	1.34	-0.00	-1.02	0.61		V							
Max MY	159	1	12.82	0.01	0.00	0.00	7.02	-0.02		V							
Min MY	161	96	11.93	0.05	-2.83	-0.00	-5.40	0.17		X							
Max MZ	157	7	-0.69	0.44	-1.35	0.00	-1.15	1.51		X							
Min MZ	157	64	-3.10	0.09	1.94	0.00	-3.38	-1.48									
User			0	0	0	0	0	0	V								
	For a	all memb	ers that b	elong to thi	is GROUP				V								
ОК		Can	cel		Layer De	esign		La	yer Exp	lorer				Calc	ulatior	Printo	out

					S	teel Des	sign - La	ayer Da	ta								×
Layer: Pur	lins	,	/ERIFICAT	тои иот с	ж				Ca	apacity De	esign	Ampli	ficatio	n			
Different Cr	OSS	IPE	100	_		-	CHE	CK SEL	ECTIC	N							
Description	Membi Comb. N Vy Vz Mx My Mz								NO	Auto	N	м	v	Mx	M-N	M-V	M-V-N
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44		V							
Min N	187	1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82		V							
Max QY	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84		V							
Min QY	173	1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84		V							
Max QZ	197	1	0.06	1.25	2.18	-0.00	-2.60	-1.42		V							
Min QZ	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42		V							
Max MX	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42		V							
Min MX	197	1	0.06	-1.25	-1.41	-0.00	-0.00	-1.41		V							
Max MY	167	1	0.01	-0.33	0.00	0.00	1.93	0.80		V							
Min MY	196	1	-0.03	1.25	2.17	-0.00	-2.61	-1.42		V							
Max MZ	179	1	0.55	-0.00	0.07	0.00	0.92	0.91		V							
Min MZ	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84		V							
User			0	0	0	0	0	0	V								
	For all members that belong to this GROUP								V								
		_													1.11		
OK		Can	cel		Layer De	esign		La	yer Explo	orer				Calcu	ulation	Printe	out

Getting the mouse indicator over a green cell, a value lower than 1.0 is displayed (adequacy), while on a red cell a value greater than 1.0 (failure) is displayed.



All design checks' results are displayed, in the dialog box above, for all cross-sections of the current layer.

Apart from the automatic process, the user can follow his check process. Select the combination of the design checks by clicking on the corresponding button from "CHECK SELECTION"

and then the button "Layer Design".

Checking one by one you could notice that for some members, for example, the M & M-V check fail (red color). This happens because in this case the program uses only the values of MY, VZ and ignores N value (worst case).

You can also type your values in "User" line and do your own checks.

To read the main results (automatic procedure, manually or "user") click on the button "Calculation Printout" or "Layer Explorer" for all results. The displayed TXT files are those generated by the program for the printout.

MORE DETAILS:

For each section of each layer, the program calculates, for each load combination, the maximum and minimum value of all stress resultants (N, Mx, My, Mz, Qx, Qy, Qz). The load combination that gives, for example, the maximum value of the axial force N and the corresponding structural member stressed with the N, is identified. The other cells of the same line are filled in with the corresponding values obtained for the same member and the same load combination.

In this way a table is created with 12 lines (maximum and minimum value) and six columns (6 stress resultants).

-Max N ... and the relative values for Mx, My, Mz, Qx, Qy -Min N ... and the relative values for Mx, My, Mz, Qx, Qy -Max Mx... and the relative values for N, My, Mz, Qx, Qy -Min Mx... and the relative values for N, My, Mz, Qx, Qy -Max My... and the relative values for N, Mx, Mz, Qx, Qy -Min My... and the relative values for N, Mx, Mz, Qx, Qy -Max Mz ... and the relative values for N, Mx, My, Qx, Qy -Max Mz ... and the relative values for N, Mx, My, Qx, Qy -Max Qy ... and the relative values for N, Mx, My, Qx, Qy -Max Qy ... and the relative values for N, Mx, My, Mz, Qx -Min Qy ... and the relative values for N, Mx, My, Mz, Qy -Max Qz ... and the relative values for N, Mx, My, Mz, Qy -Max Qz ... and the relative values for N, Mx, My, Mz, Qy

The "Member" column contains the number of the structural member with the maximum or minimum value of the resultant stress.

The "Comb." column contains the number of the load combination that corresponds to the maximum and minimum values.



- The sign convention used by the program: Axial force with NEGATIVE sign => TENSION Axial force with POSITIVE sign => COMPRESSION But in TXT files the condition is the opposite: (+) TENSION, (-) COMPRESSION.
- 2. The column "NO" allows excluding one or more maximum or minimum obtained values. To exclude, for example, max Mz and min Mz, activate the checkboxes "NO" in the relative lines. So, for these checks, Mz max and Mz min will be excluded.

Layer: Pur	lins		VERIFICAT	ю пот с	ж					apacity D
Different Cr	OSS]	IPE 100					\sim		
Description	Memb	Com	b. N	۷у	Vz	Мх	Му	Mz	NO	Auto
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44		V
Min N	187	1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82		V
Max QY	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84		V
Min QY	173	1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84		V
Max QZ	197	1	0.06	1.25	2.18	-0.00	-2.60	-1.42		V
Min QZ	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42		V
Max MX	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42		V
Min MX	197	1	0.06	-1.25	-1.41	-0.00	-0.00	-1.41		V
Max MY	167	1	0.01	-0.33	0.00	0.00	1.93	0.80		~
Min MY	196	1	-0,03	1,25	2,17	-0,00	-2,61	-1.42	V	
Max MZ	179	1	0.55	-0,00	0.07	0,00	0,92	0.91	V	
Min MZ	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84		V

3. How to exclude one or more intensive forces from the layer's design

If for some reason you want to exclude one or more intensive forces from the layer's design,

press the corresponding Intensive Force column (for example N) and again "Layer Design" to receive the new results without considering the axial forces.



Different Cro	DSS	IPE 100					~	
Description	Membi Con	nb. N	Vy	Vz	Mx	Му	Mz	NO
Max N	218 1	4.07	1.25	1.86	-0.00	-2.31	-1.44	
Min N	187 1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82	
Max QY	181 1	0.05	1.61	1.90	0.00	-2.54	-1.84	
Min QY	173 1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84	
Max QZ	197 1	0.06	1.25	2.18	-0.00	-2.60	-1.42	
Min QZ	221 1	0.06	-1.25	-2.18	0.00	-2.60	-1.42	
Max MX	221 1	0.06	-1.25	-2.18	0.00	-2.60	-1.42	
Min MX	197 1	0.06	-1.25	-1.41	-0.00	-0.00	-1.41	
Max MY	167 1	0.01	-0.33	0.00	0.00	1.93	0.80	
Min MY	196 1	-0,03	1.25	2.17	-0.00	-2.61	-1.42	
Max MZ	179 1	0.55	-0.00	0.07	0.00	0.92	0.91	
Min MZ	181 1	0.05	1.61	1.90	0.00	-2.54	-1.84	
User		0	0	0	0	0	0	$\overline{\mathbf{v}}$
	For all m	embers that b	elong to th	is GROUP				V

- 4. The "AUTO" column offers an automatic process through which the program calculates for each line of internal forces whose check should be done based on the values corresponding to each intensive force. This means that in case of considering N, My, Mz and Mx=Qy=Qz=0 the program performs Bending, Bending with Axial, Compression & Tension checks only and doesn't perform Torsion and Shear checks.
- **5.** Choosing the **manual process** the user is free to check which checks to perform and then click "Layers Design" to see the results:

					S	teel Des	sign - La	ayer Da	ta								×
Layer: Pur	lins		VERIFICAT	ION NOT C	ж				✓ C	Capacity Design Amplification							
Different Cr	ss IPE 100 V											CHE	CK SEI	ECTIO	N		
Description	Membe	Comb.	Ν	Vy	Vz	Mx	Му	Mz	NO	Auto	*	M	v	Mx	M-N	M-V	M-V-N
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44			V	V			V		
Min N	187	1	-3.62	-1.61	-2.08	-0.00	-2.49	-1.82			V	V			V		
Max QY	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84			V	V			V		
Min QY	173	1	0.05	-1.61	-1.90	-0.00	-2.54	-1.84			V	×			V		
Max QZ	197	1	0.06	1.25	2.18	-0.00	-2.60	-1.42			V	V			V		
Min QZ	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42			V	V			×		
Max MX	221	1	0.06	-1.25	-2.18	0.00	-2.60	-1.42			V	V			×		
Min MX	197	1	0.06	-1.25	-1.41	-0.00	-0.00	-1.41			V	V			×		
Max MY	167	1	0.01	-0.33	0.00	0.00	1.93	0.80			V	V			V		
Min MY	196	1	-0.03	1.25	2.17	-0.00	-2.61	-1.42			V	V			×		
Max MZ	179	1	0.55	-0.00	0.07	0.00	0.92	0.91			V	V			V		
Min MZ	181	1	0.05	1.61	1.90	0.00	-2.54	-1.84			V	V			V		
User			0	0	0	0	0	0			√	V			•		
	For a	all mem	bers that be	elong to th	is GROUP						V	V			√		
ОК		Car	ncel		Layer D	esign		La	iyer Expl	orer				Calc	ulatior	n Print	out



- a green check means stress/resistance ≤1
- red check means: stress / resistance >1
- yellow check means: not required
- ▲ Getting closer the mouse indicator over a green cell, a value lower than 1.0 is displayed (adequacy), while on a red cell a value greater than 1.0 (failure) is displayed.
- **6.** Activating "**User**" the user can type his values of the intensive forces to check the sections. In the next dialog box:

Layer: Mai	in Beam	s	VERIFIC	CATION OK					Ca
Different Cro	OSS	IP	E 330					~	
Description	Memb	Comb.	Ν	Vy	Vz	Mx	Му	Mz	NO
Max N	161	1	27,30	0,26	-3,38	-0,00	-4.35	0,91	V
Min N	152	1	-13.76	-0,18	-2,74	-0,00	-4,74	-0,49	V
Max QY	157	37	-1.19	0,44	-1.34	0,00	-1,11	1,50	V
Min QY	156	62	-3,10	-0,44	-2,00	-0,00	-3,38	-1,48	V
Max QZ	160	1	27,30	-0,26	3,38	0,00	-4,35	0,91	4
Min QZ	161	1	27,30	0,26	-3,38	-0,00	-4,35	0.91	v
Max MX	153	5	-6,29	0,15	-1,30	0,00	-1,02	0,61	V
Min MX	152	7	-6,29	0,18	1,34	-0,00	-1,02	0,61	V
Max MY	159	1	12,82	0.01	0,00	0,00	7,02	-0,02	V
Min MY	161	96	11.93	0,05	-2,83	-0,00	-5,40	0,17	4
Max MZ	157	7	-0,69	0,44	-1,35	0.00	-1,15	1,51	V
Min MZ	157	64	-3,10	0,09	1,94	0,00	-3,38	-1,48	V
User			-15.23	0.52	-1.23	0	-3.51	3.61	
	For	all mem	pers that be	elong to thi	is GROUP				V

intensive forces are given by the user and those estimated by the program analysis are disabled.

- **Attention to the Convention on the sign of the axial force!!!!**
- 7. "Different Cross Sections" contains the different sections included in the "Steel Beams" layer.

Layer: Ma	in Beams		VERIFIC	ATION OK				
Different Cr	OSS		IPE 330					\sim
Description	Memb	Con	HEA 300 IPE 220 IPE 270					Mz
Max N	161	1	HEA 550					0.91
Min N	152	1	-13.76	-0.18	-2.74	-0.00	-4.74	-0.49


Follow the same procedures described above to design manually the other sections or to see the results:





7.1.2 Buckling Members Input

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

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

By selecting the command the following window opens:

Member D	esign		~	×
			2	
Layer	Main Beams		N	~
Member	129 HEA 180	\sim	Param	eters
Group	Докоі			~
A	Apply to all members of the Layer			
Checkir	ng with Min, Max	of all co	mbinatio	ons
Check in	Layer			
Explo	Exploration of Member Buckling			
Exploration of Member Servicability				
Memb	Member Results Layer Results			ults
	ОК		Cancel	

Checking is performed by layer. So first select the layer from the drop down list and the "Member" list loads all members of this layer and its cross sections.

	Member I	Desi	gn ×
Layer	Steel Columns		~
Member	1 IPE 450	~	Parameters
Group	1 IPE 450 3 IPE 450 4 IPE 450	^	~
1	5 IPE 450 6 IPE 450		ne Layer
Check in	7 IPE 450 9 IPE 450 10 IPE 450		
Exp	11 IPE 450 12 IPE 450		g
Explo	13 IPE 450 15 IPE 450		ility
Buckling F	16 IPE 450		ricability Results
	18 IPE 450 19 IPE 450 21 IPE 450		Cancel

Timber Vert.Wind bracings



EXAMPLE:

	Member Design	×	Select from the drop-down list the layer "Steel
Layer Member Group Check ir Expl Buckling	Steel Columns Concrete Beams Concrete Foundation Beams Footing Connection Beams Footings Steel Columns Steel Beams A Surface Mesh Mathematical Model Surface Elements Mesh 3D Slabs-Strips Steel Columns Steel Beams Main Beams Purlins Girdore	~	Columns". In the "Members" list all the structural members that belong in the selected layer are displayed. If you want to define different parameters for some of them, you can create different "Groups" in the same layer. The program has two default Groups: "Beams" and "Columns".
	Secondary Columns Hor.Wind bracings Vert.Wind bracings Timber Columns Timber Beams Timber Beams Timber Purlins Timber Purlins Timber Girders Timber Secondary Columns Timber Hor.Wind bracings Timber Vert.Wind bracings	~	If you want to apply the same parameters to all members of the layer, then set the parameters once, keep the default name "Columns" and press the "Apply to all members of the layer". Calculations will consider the same parameters for all members of the layer. Otherwise to set different parameters for some of the

Otherwise, to set different parameters for some of the

members of the layer, the procedure that should be followed is explained below. But first, let's see how to set the parameters.

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

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

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



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

Δ NOTE

1 In older SCADA Pro versions as well as before the command was created



Merge

Elements, the user was asked to specify the length of the member and the buckling length along both directions Y and Z respectively, following the procedure:

In "Member's Length":

- By choosing "Real", you have to fill in the real length of the member (in m)
- By choosing "*Coefficient*", you have to type in a coefficient by which the different lengths of the members which belong to the particular parameters' group will be multiplied.

In case you want the program to take into account the real members' lengths, during buckling check, choose "Coefficient" with one value.

In case you have some members with different or equal lengths that are laterally secured at the same distance (eg 1/3), then you define the value of 0.33 and of course, you create a separate group of parameters to which these members will belong.

	1410
-	new
	11-12

In the new version of SCADA Pro, the buckling length is defined by using the command "Merge Elements " and so no action is required in this field. Having followed the procedure of Merge Elements, in the Parameters field and specifically in the Member's Length, you leave it as it is and proceed with the definition of the remaining parameters.

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

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

Lateral Buckling resistance check: Activate the corresponding checkbox. Set the length of the structural member and the buckling lengths for both Y and Z directions. On the field

"Member's Length" activate the label "Real" and type the real length in m, or activate the label "Coefficient" Coefficient and type a factor ("1" means the real length).

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

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



Buckling Lengths Y	The icons are di The first group depending on th	vi in ne	ded clue me	into two des icons ember su	grou with oport	ps: a s coi	pecific factor nditions
				Buckl	ing Ler	ngth	is Y X
					*		
By choosing you can define the pos	itions of lateral		_				
blocks if there are any to calculate the corresponding				Length(m)	Coef.	^	fixed support
reduced buckling lengths.			1	0.00	1.00		
			2	0.00	1.00		
			3	0.00	1.00		0.5
The second group			4	0.00	1.00		
			5	0.00	1.00		
			6	0.00	1.00		2.0
includes cases of members in multi-storey	steel structures		7	0.00	1.00		ОК
and allows setting the concurrent to the node members.			8	0.00	1.00	~	Cancel



(the most complex case) the user sets for the vertical member the 6 Members By choosing (2 vertical and four horizontal) that offers succor to it (3 on the top and three at the end).

By selecting the icon the following dialog box is displayed:



Frame Members Parameters					×	
Jo	oint Membe	ers		IH	1	Type of Load
top Column	52	IPE 450	1.63	\odot \bigcirc		
B top left	53	IPE 450	1.63	\odot \bigcirc	$\odot \bigcirc \bigcirc$	Concrete Slabs v
B top Right	158	IPE 330	6.80	\odot \bigcirc	$\odot \bigcirc \bigcirc$	Concrete Slabs 🗸
						Concrete Slabs
B lower left	0			• •	$\odot \circ \circ$	Direct Loads
B lower right	0			• •	$\odot \bigcirc \bigcirc$	Concrete Slabs 🗸
Lower Column	0			\odot \bigcirc		
2. 158			ОК		Cano	cel

where for the respective fields the respective members that offer succor to the top and the bottom nodes of the member that specifies the buckling length are indicated graphically.

Every time you click a member, in the corresponding field, the number of the cross-section and the length is automatically filled in. To select the members to follow the indications on the left (Top column, Beam top Left, ecc).

Then indicate their orientation an especially for beams the type of support on the other end, and the type of load imposed on them.

Pressing "OK", on buckling length the corresponding icon and the factor -1 appears, which generally means that the program based on the data you entered automatically calculates the buckling length for this member.

Finally, choosing/user can type his buckling length.

Sway Frame set if the frame to which the member belongs is transposable or irremovable.

Flexural Buckling resistance check:



✓ Flexural Buckling		
Ends Constraint		
Member Loading		
Loading Level		
Activate the checkbox and press	ionstraint	constraints onons
The End Constraints window, contain	ing the valious types of	constraints opens.
Press one of the first four buttons to au	tomatically calculate	Lateral Buckling
the flexural buckling factor:		4
End Constraints		
The last one gives the user the op	portunity to consider	Length(m) Coef intermediate
different constrains along the same me	mber.	1 0.00 1.00
		2 0.00 1.00
		6 0.00 1.00
		7 0.00 1.00 OK
		8 0.00 1.00 Cancel
Member Loading	-	Member Loadi ×
The next parameter	refers to the	e load
selecting the corresponding icon the fo	s y, and z respectivel llowing ontions annear	y. By
sciecting the corresponding icon, the to		
In which you choose the type of Member Finally, the last parameter/concerns the	er Loading. e	
	Determination of th	e Loading level of the member.
	the following five o	prions are displayed by selecting
	Loading levels for ea	ch icon:
	1 st icon: on the uppe	r flange of the element
2 nd icon: near and upward from the axis	of symmetry of the ele	ement

3rd icon: on the axis of symmetry of the element



4th icon: near and below the axis of symmetry of the element 5th icon: on the lower flange element.

- > For Lateral Torsion Buckling resistance check: activate the corresponding checkbox.
- **NOTE**: For the lateral buckling and the lateral torsion buckling resistance check, the parameters are the same.
- For Serviceability checks: activate the checkbox "Serviceability Check" and the checkboxes "Member Deflection Limits" and "Node Displacement Limits".

<	 Serviceability Check 				
V N	✓ Member Deformation Limits				
Y	Y 200 Z 200				
 ✓ Node Displacement Limits × 150 Z 150 					

Then type the corresponding values in each direction, X, and Z. For example in the figure on the left, the limits are defined as I/200 and I/150, where I is the member's length.

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

Member Design X				
Layer	Steel Columns	~		
Member	21 IPE 450	✓ Parameters		
Group	Στύλοι	~		
A	pply to all memb	ers of the Layer		
Checki	ng with Min, Ma	x of all combinations		
Check in	Check in Layer			
Explo	Exploration of Member Buckling			
Exploration of Member Servicability				
Memb	er Results	Layer Results		
	ОК	Cancel		

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

Click the button "Check in Layer" to check every member of the current layer, for every load combination. The results of the design checks are displayed in the black window that becomes green if it the checks are satisfied with all members of the active layer and red, if not.



By activating the option Checking with Min, Max of all combinations, in checks,

only the maximum and minimum values of the intensive forces resulting from all combinations, excluding the intermediate values, will be taken into account so that the process will be completed at noticeably shorter times.





To define another set of parameters for some of the members of the layer follow these steps:

1st step: Press the button "Parameters" and open again the parameters dialog box.
 Type the "Group Name" for the new set of parameters that will be created and press the button "New Group Creation".

	Member Design	×
Group Name Column	ns_1	New Group Create
Safety Factor 1 Lateral Buckling Direction Y Member's Length Real Coefficient Buckling Lengths	Limit of Direction Z Member's Le Real Coefficien Buckling Ler	0.1
Flexural Buckling Ends Constraint	Serviceabi	1 ility Check eformation Limits Z 200
Loading Level	X 150	z 150

Then set the parameters and press the button "OK".

2nd step: define the members of the layer that will belong in "Columns_1" group.

Returning to the original dialog, the only member that gets the parameters automatically, is the current member in the list of members.

All other members have the parameters of the group "Columns"

To move the members from a group to another, select each one from the Members list and change the Group.

Layer	Steel Columns		~
Member	1 IPE 450	~	Parameters
Group	Columns_1		~



Member	1 IPE 450	~	Group	Columns_1		\sim		
As soon	1 IPE 450 3 IPE 450 4 IPE 450 5 IPE 450 6 IPE 450 9 IPE 450 10 IPE 450 11 IPE 450 12 IPE 450 13 IPE 450 13 IPE 450 15 IPE 450 16 IPE 450 17 IPE 450 18 IPE 450 19 IPE 450 21 IPE 450 22 IPE 450 23 IPE 450 23 IPE 450 24 IPE 450 as the layee	r check is	comple	Columns Beams Columns_1	he window	on the right	gives t	he
informat	ion:		eep.e	-			8.000 0	
Member D	esign		\times	Red: if there is a fa	ailure			
Layer Member Group	Steel Columns 21 IPE 450 Στύλοι pply to all member ng with Min, Max o	 Parar rs of the Layer of all combinat 	v neters v	Green. In there is i	io failure.			
Check in	Layer							
Explo Explora	ration of Member	r Buckling Servicability						
Memb	er Results	Layer Re	sults					
	ОК	Cancel						
By doubl results of	e clicking on pens:	the colore	ed windo	ow, the dialog box c	ontaining m	embers checl	< summa	iry

				Men	nber Ch	eck Resul	ts
Member	Cross Section	Lateral	Side	Lat.Torsional	Serv.Def	Serv.Displ	
151	IPE 330	4/0.00	25/0.08	25/0.08			
152	IPE 330	4/0.00	25/0.05	25/0.05			
153	IPE 330	Not Req	25/0.05	25/0.05			
154	IPE 330	Not Req	25/0.08	25/0.08			
155	IPE 330	4/0.00	25/0.08	25/0.08			
156	IPE 330	4/0.00	25/0.05	25/0.05			
157	IPE 330	18/0.00	25/0.05	25/0.05			
158	IPE 330	18/0.00	25/0.08	25/0.08			
159	IPE 330	Not Req	Not Req	Not Req			
160	IPE 330	18/0.00	Not Req	53/0.01			
161	IPE 330	4/0.00	Not Req	53/0.01			
162	IPE 330	4/0.00	Not Req	Not Req			
ТХТ	Г				ОК		

118



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

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

"Not Required" means that there is no corresponding size or that the intensive axial force is tensile and not compressive.

NOTES:

▲ The check for the three types of buckling is performed for each member and all combinations. For each group of (N, My and Mz) the checks are made four times based on the following combinations:

N with min My and min Mz N with min My and max Mz N with max My and min Mz N with max My and max Mz

▲ That's why in the output results and in the exploration text the number of the combination has two numbers: The first is the number of the combination and the second refers to the number for each of the four previous cases.

Member Design		×
Layer Steel	Columns	~
Member 21 IP	E 450 V	Parameters
Group Στύλ	01	~
Apply to	all members of t	ne Layer
Checking with	n Min, Max of all o	ombinations
Check in Layer	Mem:54/5	54 Comb:4/4
Exploration	of Member Buckl	ing
Exploration o	f Member Servica	bility
Member Res	sults L	ayer Results
ОК		Cancel

Selecting the Exploration of Member (Buckling /Serviceability) opens the files containing the analytical results of all checks for all combinations for the active member.

By selecting Results the files that include the summary results of the checks on the active member Member Results and all members of the active layer/opens.



7.1.3 Cold Formed Sections

Cold Formed Sections

This command concerns the checks of cold formed sections.

The design of cold formed sections concerns the:

- Resistance check in cross sections
- Resistance check in members
- Serviceability check

The procedure of selecting the members and the checks that are going to follow is similar to the warm formed sections' buckling.

The main difference between the warm formed and the cold forms sections elements are that cross-sections' and members' checks are now done with <u>a common command</u> (see figure) rather than separately. An important feature is that all members and their cross sections are checked <u>for all combinations</u>.

		44	
Steel Design •	Timber Design *	Masonry Design *	Mer Diagr
	Cross-Sectio	on Design	
<i>]</i>] =	Buckling M	embers Inp	ut
	Cold Forme	d Sections	
+	Connection	s	
*	DEA StatiCa	a Connecti	on

Member D	esign	×						
Layer	Purlins	~						
Member	103 ELASTRON	103 ELASTRON Z 🗸 Parameters						
Group	∆окоі	∆окоі						
A	Apply to all members of the Layer							
Checki	ing with Min, Max	x of all combinations						
Check in	Layer							
Explo	Exploration of Member Buckling							
Exploration of Member Servicability								
Memb	per Results	Layer Results						
	ОК	Cancel						

For the rest, the steps to design are the same as those for warm formed elements (per layer, members' merge, buckling parameters, etc.).

DESIGN PRINTOUT

The design results are displayed either per member or per layer. In the second and more general case, the printout's form is the following:

- 1. Page 1: General cross-section 1 data Information about dimensions and properties of the initial and virtual cross-section
- Page 2: Active cross-section 1 (A part)
 Information about N, My, and Mz ratios of active cross section dimensions
- 3. Page 3: Active cross-section 1 (B part)



Information about N, My and Mz ratios of active cross-section properties

- 4. **Page 4: Cross-Section check for the 1st member with cross-section 1** Resistance checks based on §6.1
- Page 5: Member check for the 1st member with cross-section1 Resistance checks based on §6.2 & 6.3 and serviceability check based on §7
- 6. **Repeat steps 4 & 5 :** If multiple members have the same cross section within the layer.
- 7. Repeat steps 1 to 6:

In case of multiple cross sections within the layer.

The printout per layer can also be extracted while creating the <u>Study Printout</u>. (See "Cold Formed sections" in the User's manual)



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

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





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

Steel connections				×
Beam to Beam	Beam to Beam Type	Beam to Column (Flange) F	Beam to Column (Web) F	Next Connection Group Connection Name dok_styl_asthenis
			,	Member Group Definition (Node) Edit Connection (Geometry/Checks)
				Cancel

EXAMPLE:

Left click to select member 30 (column) and member 116 (beam) and right click to open the library with the four possible types of connection. Select the last one "Beam to Column (Web) Γ ".

Fist, type a name (e.g. dok_styl_asthenis).

No space between words.

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

To add groups of members, click into the field "Lower Column" and pick the column 19. Then click into the field "Right Beam" and pick the beam 115 (or just enter the numbers in the corresponding fields) and then click the button "Add".



Group I	Members				×
		N(kN)	Ν	1(kNm)	V(kN)
Lower Column	0	0	0		0
Right Beam	0	0	0		0
	0	0	0		0
	0	0	0		0
	0	0	0		0
30: 116.	30.		1		Add
60: 118, 40: 117,	60. 40.			Up	odate
				D	elete
					Exit

Use this dialog box for the design of steel connections with the same type and the same crosssections in total (i.e. column IPE 450 - beam IPE 330).

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

In the end, click "Exit" and select the command "Edit Connections (Geometry/Checks)". In the new dialog box, you can define the type and the geometry of the specific connection. Select the type and enter the geometrical parameters of the cross-section. Then select the material and define the bolts' parameters. In each type of connection, the relative parameter fields are active. Then for the design calculations by using the analysis' combinations select the command "Calculation (Combinations)".

First, the program performs the geometrical checks of the connection (e.g. if the bolts are located too close to the edge of the plate). If there is a problem, the corresponding error message appears in the field on the right. In the specific connection, change the distance e1 from 14 to 15 cm and then click again the button "Calculation (Combinations)".



H - I Beam C	Column Connection (Secondary Axis of	^c Column)	×
Type Type Στυλος ΝΠΛ	Beam - Column (Γ) With End Plate	End-plate in bending verification is ok Column many time bending verification is ok Beam web in tension verification is ok Beam web in tension verification is ok Column flange in compression verification is ok Beam web in bending verification is ok Column web in bending verification is ok Column web in bending verification is ok Fillet weld is required Wield's verification end-plate - right beam is ok Calculation (Combinations) Calculations Simplified Results	ve kit
End-plate (F Extension h 150 Weld Thick Bolts Bolt (Rows (e1 (mm) (Gap (g)	(mm) ded Materia S235 b 100 t 12 hup 20 kness 6 Angle 90 412 Materi 4.6 General Parameters 4 Same Distances 10 e2 (mm) 20 ?		1 2 3 Σ/Κ
Cross Sect	ion M12 Materi 4.6 Y Y Column: 2 Beam : Y Lin(Y) e1 10 p1 20 e2 ev1 pv1 20 10 m) Welded Quality \$235 Y 6 STIFFENERS \$1000000000000000000000000000000000000		30

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

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



vne	Beam - Column (Г)	Node	Bolts	Weld	Section	End-Plate/	Max
		1) 60	S11 = 0.05	S5 = 0.03	S5 = 0.08	S5 = 0.08	S5 = 0.08
pe	With End Plate	Max	S11 = 0.05 (1)	S5 = 0.03 (1)	S5 = 0.08 (1)	S5 = 0.08 (1)	S5 = 0.08 (1)
ιλος hπλ	Докос	Calculation	s) Calculations	Simplifi	ed Ex	(ploration	Save
nd-plate (Extend 500 Weld Thick iolts outs 1 (mm)	(mm) ded Materik S235 ▼ b 200 t 12 hup 20 kness 6 Angle 90 412 ▼ Materi 4.6 ▼ General Parameters 4 ▼ Same Distances 15 e2 (mm) 20 ?	Combinations				<u>Resolus</u>	
Ingles (L) Sap (g) Cross Sect EQ 100×10	Bolts (mm) ion General Parameters M12 Materi 4.6 Column:2 Beam : Lin: Image: State 1 p1 20 e2 80 ev1 pv1 20 10				Ļ		
Velded (m	m) Welded S235 V	1					

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

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

Connections' drawings are located in the folder of the project "sxedia": C:\SCADApro\ "project name" \scades_Synd\sxedia

You can import them using:





А	nd in the	dialog bo	x:		
	🔺 in File	s of Type	select SCADA Con	nection	
	🔺 press	Find			
	🔵 Open		N		×
			2		
	Look in:	1.dwg	~	g Ø 🕫 🖻 🛄 🕇	
	-	Name	~	Date modified	Туре
	Quick access		No items match you	ır search.	
	Desktop				
	Libraries				
	This PC				
	.	<			>
	Network	File name:		~	Open
		Files of type:	Scada connection(*.con)	~	Cancel
	Συντελ	εστής 1.0	Οροφος 1 Fin	d N	
				NS	

In the Search File in the window that opens, select the connection to import the designs, views and section, and the detailed table of the link elements.







"Timber Design" command group contains commands for the cross-sections design, the buckling resistance, and the connections design.

NOTES:

Always remember to calculate the corresponding load combinations in the parameters dialog box.

The design process of the timber sections is similar to that of steel sections.

7.2.1 Cross Section Design

This command is used to check the adequacy of the timber cross-section.

Cross-Section Design

Select this command to open the following dialog box:

		Timb	ber Design (l	Layer)			
Name	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio	Cross Sectio
Purlins							
Girders							
Secondary Columns							
Hor.Wind bracings							
Vert.Wind bracings							
_							
Timber Columns	200x200						
Timber Beams	100x100	100x100					
Timber top main beams	140x140						
Timber Purlins	100x100						
Timber Girders	100x100						
Timber Secondary Columns	150x150						
Timber Hor.Wind bracings							
Timber Vert.Wind bracings							
tegides	100x100						
columns	200x200						
							•
<							>

The first column contains the layers of the current project and the other columns the cross-sections that belong to each layer.



Select the button "Calculate All" for the calculation of all sections.

Alternatively, select the layers one by one and click the button "Calculate".

Green color indicates that all sections of this layer satisfy the design criteria (stress/resistance \leq 1), while red color indicates that they don't.

To locate the inadequate members or just see the check results, select the layer and click "Edit".

						Timber	Design	(Layer)	r) ×
Layer: Tin	nber Bea	ms	VERIF	ICATION N	от ок				Capacity Design Amplification
Different Cr	OSS	10	0x100					\checkmark	CHECK SELECTION
Description	Membe	Comb.	Ν	Vy	Vz	Mx	Му	Mz	NO Auto N M V Mx M-N M-V M-V-N
Max N	705	1	34.40	-3.25	0.08	-0.00	-0.01	0.90	
Min N	937	1	-25.39	0.17	-0.00	0.00	0.00	-0.01	
Max QY	495	23	23.87	3.01	-0.41	-0.01	-0.08	1.50	
Min QY	628	21	25.47	-3.37	-0.84	0.01	0.25	1.57	
Max QZ	726	59	21.55	-2.32	2.44	0.00	0.22	0.05	
Min QZ	628	55	26.53	-2.81	-2.40	-0.00	-0.20	0.10	
Max MX	746	23	-0.63	1.12	0.04	0.06	0.04	0.19	
Min MX	613	59	-2.28	-0.93	-0.19	-0.06	0.13	-0.13	
Max MY	735	59	12.00	0.03	1.68	-0.00	1.04	-0.00	
Min MY	734	59	13.12	-0.01	1.94	-0.01	-1.02	-0.01	
Max MZ	628	21	25.47	-3.37	-0.84	0.01	0.25	1.57	
Min MZ	806	24	-10.80	0.79	-0.01	0.01	0.03	-0.75	
User			0	0	0	0	0	0	
	For a	all memb	pers that b	elong to th	is GROUP				
ОК	OK Cancel Layer Design Layer Explorer Calculation Printout								

1 The whole procedure for the layer design is described in detail in the respective section of steel sections (See Steel> Steel Design> Cross Sections Design)

7.2.2 Buckling Members Input

Buckling Members Input

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

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

Always remember to calculate first the corresponding load combinations in the parameters dialog box

By aelecting the command the following window opens:



	Member Design									
Layer	Timber Columns	3		~						
Member	1 200x200 V Parameters									
Group	Group									
Apply to all members of the Layer										
Check i	Check in Layer									
Ex	ploration of Layer	Buckling								
Expl	oration of Layer S	ervicabili	ity							
Buckling	Results (Layer)	Servi	cability Results	6						
	ОК		Cancel							

A check is performed by layer. So first select the layer from the drop down list and the "Member" list shows all members of this layer and its cross sections.

	Member Design	×	Member Design ×
Layer	Timber Columns	~	Layer Timber Columns 🗸
Member	Concrete Beams Concrete Foundation Beams	^	Member 1200x200 V Parameters
Group	Footing Connection Beams Footings Steel Columns		Group 3 200x200 4 200x200
	Steel_Beams Surface Mesh		A 6 200x200 7 200x200
Check in	Mathematical Model Surface Elements Mash 3D		Check in 8 200x200 9 200x200
Exp	Mesh 2D Slabs-Strips		Exp 10 200x200 g 11 200x200 liter
Explo	Steel Columns		Buckling F 15 200x200 ricability Results
Buckling F	Steel Beams Main Beams Purlins		16 200x200 17 200x200 18 200x200 Cancel
	Girders Secondary Columns		19 200x200 20 200x200
	Hor.Wind bracings Vert.Wind bracings	E	21 200x200 23 200x200 24 200x200
	Timber Columns Timber Beams		25 200x200 26 200x200
	Timber top main beams Timber Purlins		27 200×200 28 200×200 29 200×200
	Timber Girders Timber Secondary Columns		- 30 200x200 31 200x200
	Timber Hor.Wind bracings	~	33 200x200 34 200x200

EXAMPLE: select from the drop-down list the "Timber Columns" layer. In the "Members" the list displays all the structural members that belong in the selected layer. If you want to define different parameters for some of them, you can create different "Groups" in the same layer. The program has two default Groups: "Beams" and "Columns".



Group	Columns	~
	Columns	
	/ Beams	

In case you want to apply the same parameters to all members of the layer, then set the parameters once, keep the default name "Columns" and press the "Apply to all members of the layer".

Calculations will consider the same parameters for all members of the layer.

Otherwise, to set different parameters for some members of the layer, the procedure that should be followed is explained below. But first, let's see how to set the parameters.

×
New Group Creation 0.1 Igth 1 gths 1
ty Check formation Limits Z 200 cement Limits Z 150
f

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

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

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

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

The rest of the form is divided into four parts, one for each check:Lateral BucklingLateral Torsion BucklingFlexural BucklingServiceability checks

1 The whole procedure for the layer buckling resistance check is described in detail in the respective section for steel sections (See Steel> Steel Design> Buckling Members Input)



7.2.3 Connections

Connections

The last command of the "Timber Members Design" group command is the "Connections", used for the timber connections' design. Press the command and select the connected members sequentially.



Right click to complete the selection and open the following form:

		Timber Joint	t	×
Name				
Parameters	Members connectivity			
Characteristics of m	ember's connection			
781 🗸	Edit 2D/3D			
Check	Save			
Search	Results			
ОК	Cancel			



In the right part of the window, the connected members with b and h casually are displayed. Through Members connectivity command, the user defines the real dimensions of the members.

Name the connection and select the command Members connectivity. Node and timber members (Connectivity) Seq./no. Name Type Direction Prior. Επιπ. xz 💌 Connected 💌 781 0 1 xy 💌 Continuous 💌 2 1 733 xz 💌 Connected 3 782 2 • xy 💌 Main 3 3 • xy 💌 5 0 Inactive vv 🗸 🗸 • 6 Inactive ٥ Grade Name 370 C20 \sim b(mm) 100 h(mm) 200 Angle 269.9 Member Eccentricity (mm) ,×i ⁽ⁱ⁾ 0 х (i) 0 7 Xi=0 Zi=0 ОК Cancel View

In the first field, you specify the Type of the member.

Select graphically with a left click on the member to define the *Main Member* (each connection has <u>only one</u> Main member). In the left list, the selected member is automatically marked.



 $\mathbf{\widehat{V}}$ In this example, the Main member is 370.





If there is a collinear member (ex. 733) this can be defined either as Connected or Continuous.

All the other members of the connection are Connected.

- Main Member: may be any member of the connection
- **Continuous:** is the member continued to the Main member without interruption. It is a single member and cannot have dimensions different than the main member.
- Connected: is the member connected with other members and can have different • dimensions than the connected.

Therefore, set, in the same way, the Type of all members.

The next step is to define the **dimensions** of each member.

Choose from each list the values of b and h. **b**= the thickness of the member (dimension perpendicular to the screen) **h**= the height of the section (dimension in the screen level)

			Ν	lode	and tin
Seq./no.	Name	Туре	Direction	Prior.	Επιπ.
1	781	Connected	0		xz 💌
	733	Continuous 💌	1	1	xy 💌
3	782	Connected 💌	2		xz 💌
4	370	Main 💻	3	3	xy 💌
5		Inactive 💌	0		xy 💌
6		Inactive	n		vv v
lame 73	33 Gra	ade C20		~	
(mm) 10	00 h((mm) 200	Angle 90		1
Member E	ccentrici	ty (mm)			
x 0		(i) xi (i)	(1)		
z 0		Xi-0			
		Zi=0			
ОК		Cancel	Viev		2

Angle: Is the angle of the member as to the connection. The angles are defined counterclockwise with 0 in + x.



	C14
	C16
	C18
	C20
	C22
	C24
	C27
	C30
	C35
	C40
	C45
	C50
(right to the connection)	D18
	D24
	D30
Grade: to set the quality of each member, select the member and its quality	D35
	D40
	D50
	D60
A ATTENTION:	D70
The Main and the Continuous Manshare annual have different dimensions	GL24h
The Main and the Continuous Members cannot have different dimensions.	GL28h
They are the same element!	GL32h
	GL36h
	GL24c
	GL28c
View command displays the connection in total with the lengths of the	GL32c
	GL36c
members.	



Direction: The direction of each member is defined according to the icon.

Therefore, start by selecting the left-hand member to define the 0 directions and continue defining the direction of the remaining members of the connection.







In the priority column set a number only for the Connected members.

				Ν	lode a	and ti	mb
Seq./no.	Name	Туре		Direction	Prior.	Lev.	^
1	781	Connected	•	2	1	xz 💌	
2	733	Continuous	•	3		ху 💌	
3	782	Connected	•	0	2	xz 💌	
4	370	Main	•	1		ху 💌	
5		Inactive	•	0		ху 💌	
6		Inactive	•	0		vv 🔻	4

Levels:





The connection A is in XY level i.e. the steel connecting plate will be introduced at this level (vertically).

The connection B is in XZ level i.e. members 1 and two will be connected with a horizontal steel plate.

Only members that belong to the same level can be connected. Therefore column members can't be connected in B connection.

NOTES:

- ▲ The connection level defines the bending level of the members that will be considered according to their local axes, as well.
- Therefore from the six intensive forces (N, Mz, Vy, My, Vz, Mx) of each member, in the connection node, 3 of them will be taken into account, N, Mz, Vy in the xy level, and N, My, Vz in xz level.



Choosing the right level of each member based on local axes is defined in the **Level** column.

Member Eccentricity:

Through **Member Eccentricity**, the end of a connected member can be moved from the connection node according to the eccentricity.

In this way, structure eccentricities are covered.



Select the member and according to the icon set the eccentricities in X and Z.



Fastener Cutting of members Type Oovels Fuk(MPa) 360 Plates In the Fastener Type Side plates Vividth of slot same as thickness of plate Image: Connected members Vividth of slot same as thickness of plate Image: Connected members Vividth of slot same as thickness of plate Image: Connected members Vividth of slot same as thickness of plate Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Vividth of slot same as thickness of plates Image: Connected members Instance between plates e1 (mm) Image: Connected members Instance between plates Image: Connected members Solts, and Nails) and the Grade and the Fuk tensile strenge i	(General Parameters	×
GradeBolts NailsAlternatively, the user can enter a value for the tensile stree be taken into account during the checks.	Fastener Type Dowels Grade \$235 Fuk(MPa) 360 Plates Type Type \$5235 Number 3 Thickness Grain angle / Panel axis ✓ Width of slot same as thickness Width of slot (mm) 20 ✓ Side plates ✓ Uniform placement of plates Distance between plates e1 (mm)	Cutting of members Main member Main member O Connected members O Connected members Connected members O Connected members Connected members Con	
	Fastener Type Dowels V	In the Fastener field define the connect Bolts, and Nails) and the Grade and th	tor Type (between Fuk tensile streng
Platoc	Fastener Type Dowels V Dowels Grade Bolts Nails Fuk(MPa) 300	In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check	tor Type (between te e Fuk tensile strengt e for the tensile stre s.
Plates Finish birch plyw Type Steel Scale Scale Number 3 Scale Scale Grain angle / Panel axis Scale Scale Scale Grain angle / Panel axis Scale	Fastener Type Dowels Grade Bolts Nails Fuk(MPa) 300 Plates Type Steel V S235 Number 3 S235 S275 S355 Grain angle / Panel axis S450	In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check Plates Type Wood Finnish bi Finnish bi Finnish bi Grain angle / Panel axis OSB/2 OSB/2	tor Type (between e Fuk tensile strengt e for the tensile stre s.
Plates Finites Type Steel Steel Number 3 Steel Steel Steel Steel Number 3 Steel Steel Steel Steel Grain angle / Panel axis Steel Steel Steel Steel Steel <td>Fastener Type Dowels Grade Bolts Nails Fuk(MPa) Plates S235 Type Steel ✓ Steel ✓ S235 Number 3 S275 S355 Grain angle / Panel axis S450 ✓ Width of slot same as thickness Width of slot (mm) 20</td> <td>In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check Plates Type Wood Finnish bi Finnish bi Finnish bi Grain angle / Panel axis OSB/2 OSB/3 OSB/4 Particlebo Width of slot same as Width of slot (mm)</td> <td>tor Type (between e Fuk tensile streng e for the tensile stre s. rch plyw v rch plywood oftwood plyw ard P4 bard P5</td>	Fastener Type Dowels Grade Bolts Nails Fuk(MPa) Plates S235 Type Steel ✓ Steel ✓ S235 Number 3 S275 S355 Grain angle / Panel axis S450 ✓ Width of slot same as thickness Width of slot (mm) 20	In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check Plates Type Wood Finnish bi Finnish bi Finnish bi Grain angle / Panel axis OSB/2 OSB/3 OSB/4 Particlebo Width of slot same as Width of slot (mm)	tor Type (between e Fuk tensile streng e for the tensile stre s. rch plyw v rch plywood oftwood plyw ard P4 bard P5
Plates Finnish birch plywi ✓ Type Steel ✓ S235 Number 3 S275 Sz75 S355 Grain angle / Panel axis S450 ✓ Width of slot same as thickness of plate ✓ Width of slot (mm) 20 ✓ Side plates ✓	Fastener Type Dowels Grade Bolts Nails Fuk(MPa) Plates S235 Type Steel ✓ Plates S235 Number 3 S275 Grain angle / Panel axis S450 ✓ Width of slot same as thickness Width of slot (mm) 20 ✓ Side plates	In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check Plates Type Wood Finnish bi Finnish bi Finnish bi Sorain angle / Panel axis OSB/2 OSB/3 OSB/4 Width of slot same as Particlebo Width of slot (mm)	tor Type (between e Fuk tensile streng e for the tensile streng s. rch plyw v rch plywood oftwood plyw ard P4 hard P5
Plates Finnish birch plyw v Type Steel v S235 Number 3 S275 S275 S355 Grain angle / Panel axis S450 V Width of slot same as thickness of plate Vidth of slot same as thickness of plate Vidth of slot (mm) Vidth of slot (mm) V Side plates Side plates Side plates Side plates V Uniform placement of plates V Uniform placement of plates V Uniform placement of plates	Fastener Type Dowels Grade Bolts Nails Fuk(MPa) Plates S235 Type Steel ✓ Plates S235 Number 3 S275 Grain angle / Panel axis S355 Grain angle / Panel axis S450 ✓ Width of slot same as thickness Width of slot (mm) 20 ✓ Side plates ✓ Uniform placement of plates	In the Fastener field define the connect Bolts, and Nails) and the Grade and the is automatically updated. Alternatively, the user can enter a value be taken into account during the check Plates Type Wood Finnish bi Finnish bi Finnish bi Finnish bi Grain angle / Panel axis OSB/2 OSB/2 OSB/4 Width of slot same as Particlebo Width of slot (mm)	tor Type (between e Fuk tensile strengt e for the tensile stre s. rch plyw v rch plywood oftwood plyw ard P4 hard P5



 In case of Wooden Flange, the parameter Grain angle / Panel axis where the user defines the angle of the grain of the wooden flange as to the axis of the Main member, is activated. When the Width of the slot is bigger than the plates, deactivate the checkbox and define the width of the slot. 	Plates Type Wood Finnish birch plyw Number 3 Thickness 0 Rumber 3 Thickness 20 Grain angle / Panel axis 0 Grain angle / Panel axis 0 Width of slot same as thickness of plate Width of slot (mm)
Side plates activate the introduction of the side plates.	
Otherwise only the intermediate plates exist.	
✓ Uniform placement of plates Plate's placement (along with its the cross section intervence) Distance between plates e1 (mm) 0 Distance between plates e1 (mm) 35	nt in the section of the timber member hickness) can be <i>Uniform</i> , i.e. divides the co equal parts (active checkbox) or not.





In the second case deactivate the check and set the distance $\ensuremath{\mbox{ei}}$ as defined in Figure.

▲ Connections with Side plates require uniform placement of plates.



In **Cutting of members**, field chooses the sectioning modes of member's profiles.

Cutting of members Main member

The first two options concern the cut of the Main member with the Connected member with Priority 1:

- 1. Main and Connected member are both trimmed
- 2. The Main member prevails to the Connected.

The other two options concerning the cutting of Connected members based on the priority:



- 1. Connected member both cut
- 2. the Connected with bigger priority prevails.

Bolt parameters	to set additio	nal parameters concerning bolts in steel plate
Bolts General Pa	arameters ×	
Staggered Rows in Web Countersunk Bolts The thread is included in the S Types of Shear Connec ks Values (Tabl	Oversized ctions A ✓ e 3.6) 1.00 ✓	
Slide factor for prestressed Bolts (T	e3(mm) 0 e4(mm) 0 d0(mm) 0	



haracteristics of member's connection						
Timber Joint						
Name 222 Parameters Member's connectivity Characteristics of member's connection 733 V Edit 20/30 Check Save Search Results						
OK Cancel						

After Member's connectivity and Parameters definition, the connection's **Editing** follows. Choose a member, starting from the *Main*, either graphically in the right figure, or from the list





Joint characteristics (mm) Diameter M12 V Washer diameter Plug length of dowel Hole	- Sel 12 dow 14 - Typ 0 - Typ 1	ect the dia el) be Washer fine plug le be the hole	ameter of the diameter (onl ength (only for e tolerance in t	bolt (or type the y for bolts) dowels) mm.	diamete	r of the
		Connection	detailing			×
Joint characteristics (mm) Diameter M12 Washer diameter 14 Plug length of dowel 0 Hole 1 Fasteners arrangement Rows 3 Distance 55 Columr 4 Distance 100 Staggere Placement angle Parallel to the grain v $\phi1$ $\phi2$ 0 Circular pattern (mm) Seq./no. Radius Seq./no. Radius Number Angle 1 0.00 0.000 2 Distance from loaded end a3t 0 2 Ecco Distance from the centre of the fastener to the pl Parallel to the grain 0	Dia Λ 1 12.00 ✓ 2 12.00 ✓ 3 12.00 ✓ 4 12.00 ✓ 5 12.00 ✓ 6 12.00 ✓ 9 12.00 ✓ 10 12.00 ✓ 9 12.00 ✓ 10 12.00 ✓ entricity (mm) 0 ate edge (mm) endicular	Membi 370	Fastener Bolts	Assign to all members	ΟΚ	Cancel

In the field Fasteners arrangement:

• Activated *Rectangle pattern*:

Type the number of the Rows (parallel to the wood fibers) and the Columns (perpendicular to the wood fibers) and the respective distances. The figure on the right is updated showing the connectors and the outline of the plate.





Through the column of the connectors, the user can modify the diameters by typing directly the diameter for the selected connector, either graphically in the figure or the column. There is also the possibility to exclude connectors by deactivating the checks.

Connection detailing x Assign to all members OK Cancel Membi 370 Fastener Bolts Joint chi M12 V 12 Diameter α3,t 14 Washer diameter Plug length of dowel Hole Fasteners arrangement Rectangle pattern (mm🖌 Dia. 3 Distance 55 Rows 12.00 🗸 1 Columr 4 Distance 100 2 12.00 Stagere 3 12.00 🔽 ent angle 4 Parallel to the grain 12.00 5 12.00 φ1 0 φ2 0 6 12.00 🔽 ular pattern (mm 7 12.00 8 12.00 ▼ 9 12.00 ▼ 10 12.00 ▼ 12.00 Seq./no. Radius Number Angle 🛆 0.00 0 0.00 0.00 0.00 0.00 0 0 ? EC Distance from loaded end a3t ntricity (mm) nce from the centre of the fastener to the plate edge (mm) Parallel to the grain 0 Perpendicular 0

The Staggered activation excludes all intermediate connectors.

The **Placement angle** of the connectors is selected from the list and is displayed in the figure.





	Connection detailing	×
boint characteristics (mm) Diameter M12 \checkmark 12 Washer diameter 14 Plug length of dowel 0 Hole 1 Fasteners arrangement Rectangle pattern (mm Rows 3 Distance 55 Columr 4 Distance 100 Staggere Placement angle Perpendicular to the loaded end \checkmark ϕ_1 5 ϕ_2 5 Circular pattern (mm] \checkmark Seq./no. Radius Number Angle 1 8 0.00 9 30.00 2 50.00 6 0.00 ϕ_1 5 ϕ_2 5 $1 22.00 \checkmark$ 6 12.00 \checkmark 8 12.00 \checkmark 9 12.00 \checkmark 1 12.00 \checkmark	Connection detailing Memb: 470 Fastener Bolts	Assign to all members OK Cancel
3 0.00 0 0.00 III 12.00 V Distance from loaded end a3t 100 ? Eccentricity (mm) 0 Distance from the centre of the fastener to the plate edge (mm) Parallel to the grain 10 Perpendicular 10		

In Fasteners arrangement field

• Activated Circular pattern:

Complete the table defining for each circle of connectors the Radius and the Number of the connectors. The angle rotates the corresponding cycle from + x in the counterclockwise direction.

At the bottom of the remaining window, the user can :

Set the distance a3t (the distance from the edge of the member of the closest to this edge connector, parallel to the fibers of the member)







By pressing? The connection center is automatically transferred to the edge of the member.



Eccentricity: transfers the connection by the eccentricity, perpendicular to the fibers of the member.




Modify the vertical and parallel to the fibers of the two ends of the plate distances from the center of the connection.



Assign to all members By selecting the command, all the connecting member details, are applied to the other members of the connection as well.

Click OK to save the choices and close the window.

You can create complex connections with a large number of members and display them in 2D/3D representation.





Name	new connection			
Paramet	ers Member	rs connectivity		
Characteri 990 Check Searc	istics of member's Control Edit Control Edit	connection 2D/3D Save Results		
Save Check Search Results	: saves cor : makes th : displays t : displays t	nection e necessary checks accordin he checks results in detail the checks results in tables	g to EC5 (for wood) EC3 (for st	eel)
Exercise 1.1 - The second	Trans Trans Specification constraints Specification constrain	Total Total <th< td=""><td>Part // Part // Part // 1000000000000000000000000000000000000</td><td>Number Number Numer Numer Numer</td></th<>	Part // Part // Part // 1000000000000000000000000000000000000	Number Numer Numer Numer



8. Masonry Design Masonry Design Assessment (EC8-3)	Command to design masonry structures. Command for the assessment of masonry structures.
 A Basic requirement either to desin, of perform first: 1. The modeling of the structure by using 	or for the assessment of masonry structure is to g either 3D surfaces finite elements or the templates
 tool (with or without using command 2. Define the parameters of the mason Type" in the analysis "Parameters" for 	Front View Identification) may and select the respective "Structura r the automatic calculation of "q" factor:
Properties of masonry Masonry Brick blocks wall - M2 25 cm Name Masonry Brick blocks wall - M2 25 cm Type Load-bearing ✓ Single-leaf wall Masonry uni Common brick 6x9x19 Thickness 25 fb=1.6733 fbc=2.0000 ε=15.00 Mortar Mortar Cement-M2 General purpose designed masonry mortar fm=2.0000 Wall ? L1 (cm) 0 t1 (cm) 0 t2 (cm) 0 Shell Bedded Wall Total width of the two mortar strips g (cm) 0	Type Existing Concrete jacket Thickness Thickness Single Sided Cocrete Steel Colorete Steel
Masonry uni Thickness Mortar Wall ? L1 (cm) 0 t1 (cm) 0 Concrete infill fck (N/mm2) Thickness C20/25 20 0	Masonry units - Modulus of elasticity (GPa) 1000 0.794381 Characteristic strength fvkm 0.1 0.10876€
Data reliability level KL1:Limited V Execution control dass 1 V	Save Flexural strength fxk1 0.1 Exit Flexural strength fxk2 0.2 (N/mm2) 0.2 0.2



	EC8 Parameters	×						
Seismic Area Seismic Areas Zone I \checkmark a 0.16 Importance Zone II \checkmark Y ⁱ 1 Spectrum Response Spectrum Design ζ 5 Hori Response Spectrum U Structural Type a	EC8 ParametersCharacteristic PeriodsSpectrum TypeHorizontal VerticalType 1S,avg1.20.90.150.05SoilTB(S)0.150.150.50.15BTC(S)0.50.51TD(S)2.51vDuctility ClassDCMvDuctility ClassDCMsontal b02.5Vertical b03Sd(T) >=0.2	Levels XZ Down 0 - 0.00 Up 1 - 680.00 ✓ Dynamic Analysis Eigenvalues 10 Accuracy 0.001 Spectrum Participation factors PFx 0 PFy 0 PFz 0 Eccentricities Sd (T) Sd (TX) 1 1 e TIX 0.05 *Lz Sd (TZ) 1						
Structural Type q 3.15 qz 3.15 Bays Setbacks S Concrete Steel X One X All the other cases S Concrete Steel X One Z All the other cases Z One Z All the other cases Z One Z Unreinforced masonry Frames type a Z Moment resisting frames type a Z One Z All the other cases Z One Z All the other cases Z One Z All the other cases Vinreinforced masonry Reinforced masonry Setbacks Z One Z All the other cases Vinreinforced masonry Setbacks Z Concrete Moment Resisting Frames Z Concrete Moment Resisting Frames V X Concrete Moment Resisting Frames Z Concrete Moment Resisting Frames V Interstorey Drift Limit 0.005 Walls KANEPE Default OK Cancel								

3. A design scenario by Eurocode is produced so all the required checks are incorporated

Open "Members Design" unit and in the "Scenarios" command group choose the "EC2" from the drop-down list as the Active Scenario.

Click on the command "Parameters" and load the combinations file that is automatically saved by the analysis. Inset Combinations, click Calculation and "OK" to close.

	Structural	Compon	ent Pa	ramet	ers			×	
Combinations Slabs Be	ams Column	Footings	Reinfor	cement	Сарас	ity Desig	n Ste	el	
Combinations of Load Sets	; (102)	Fail.	Serv.	+X	X	+Z -	Z	No	
Combinations						SE/ST	Dir.	^	
1(14) +1.35Lc1+1.50Lc2 F									
2(1) +1.00Lc1+0.50Lc2 F									
3(2) +1.00Lc1+0.30Lc2+	1.00Lc3+0.30	Lc4+1.00Lc	:5+0.30Lo	c7+0.30	Lc9	F	+X		
4(2) +1.00Lc1+0.30Lc2+	1.00Lc3+0.30	Lc4+1.00Lc	:5+0.30Lo	c70.30	Lc9	F	+X		
5(2) +1.00Lc1+0.30Lc2+	1.00Lc30.30	Lc4+1.00Lc	:50.30Lo	c7+0.30	Lc9	F	+X		
6(2) +1.00Lc1+0.30Lc2+	1.00Lc30.30	Lc4+1.00Lc	:50.30Lo	c70.30	Lc9	F	+X		
7(2) +1.00Lc1+0.30Lc21.00Lc3+0.30Lc41.00Lc5+0.30Lc7+0.30Lc9 FX									
8(2) +1.00Lc1+0.30Lc2	1.00Lc3+0.30	Lc41.00Lc	:5+0.30Lo	c70.30	Lc9	F	X		
9(2) +1.00Lc1+0.30Lc2	1.00Lc30.30	Lc41.00Lc	:50.30Lo	c7+0.30	Lc9	F	X		
10(2) +1.00Lc1+0.30Lc2	1.00Lc30.3	0Lc41.00L	.c50.30	Lc7-0.3	0Lc9	F	X	× .	
Level Multipliers	1	/ (1,4)	1	_			_		
		(10)	Insert Combinations						
Level X	Υ	Z		(`ombinat	ione Calo	ulation		
0 - 0.00 1.000	1.000	1.000			Jonibina	ions caic	ulation		
1 - 400.00 1.000	1.000	1.000							
2 - 700.00 1.000	1.000	1.000		Co	mbinatio	n G+ψ2G	101		
3 - 1000.00 1.000	1.000	1.000							
					Autor	matic Des	lign		
					0	К	Ca	incel	





Press the button "**Pick**" (the first one) to define the x starting and ending points of the part (i.e. length definition). Since the starting point is clicked, an elastic chord emerges from it, waiting to link it with the ending point (second click).







Press the corresponding button and in the dialog box, the following parameters are presented:

Au		Crite	SIMPLE Ex Criteria The Perpend Joints are: - Joints fully grouted with mortar. ✓ - Ungrouted joints. - Ungrouted joints. ✓ - Ungrouted joints. - Ungrouted joints.						xit						
Previous 1/37 Next															
Level	Γ	Lx(m)	Lz(m)	Εσα	οχές Εμβα	δόν (m2)	Μάζα(KN/g)	n	ΣL(m)	Awtot(m2) ΣL>	2m(m)	к	1
0 - 0.00	x	16.30	15.50				0.000								
	z														
1 - 400.00	x	16.40	15.50				230.19	5							
	z														
2 700 00		14.10	15 20				170 10								1
Walls Data															
Level		L(m)	h(m))	t(m)	hανοιγμ.	m)	hef(m))	fb(N/	mm2)	fm(N/	mm2)		_
						ï									

The field "Criteria" presents one by one, the 37 requirements according to EC8, for a building to be defined as "Simple". The user should see a tick next to any of the requirements that are satisfied, and move on to the next one.

NOTE:

▲ All the requirements must be satisfied, otherwise, the building cannot be defined as "Simple". As previously mentioned, only in case of a "Simple building", the design checks of EC6 are optional.

NOT SI	(MPLE	Exit
Criteria		
The Perpend Joint - Joints fully grout - Ungrouted joints - Ungrouted joints masonry units.	s are: ed with mortar. with mechanical interlock	ing between

The 37 criteria of the previous stage are the initial steps of the "simple building" characterization procedure. It must also conform to the demands in Table 9.3 of EC8, for the characterization to



be finalized. These demands concern both the building in total and each wall consecutively, and the design check process starts with the command "Automatic Data Calculation" Automatic Data Calculation

Again, a failed check means that the building cannot be characterized as "Simple".

Level	Lx(Lz(Εσοχές Εμβαδόν (Μάζα(ΚΝ	n	ΣL(Awtot(ΣL>2m(к	
0 - 0.00	: 13	9.00		0.000	7	13.02	6.51	7.02	1	NOT SIMP
	:				1.	9.19	4.60	3.00	1	NOT SIMP
1 - 300	: 13	9.00		147.850	0	0	0	0		
	:				0	0	0	0		

```
Walls Data
```

	Level	L(m)	h(m)	t(m)	hανοιγμ.(m)	hef(m)	fb(N/mm2)	fm(N/mm2)		^
1-1	0	9.02	3.00	0.50	1.00	2.70	1.68	2.00	NOT SIMPLE	
1-2	0	9.00	3.00	0.50	2.20	2.70	1.68	2.00	NOT SIMPLE	
1-3	0	9.00	3.00	0.50	2.20	2.70	1.68	2.00	NOT SIMPLE	
1-4	0	6.02	3.00	0.50	2.79	2.40	1.68	2.00	NOT SIMPLE	
1 5	0	4 71	2.00	0.00	0.00	0.00	0.00	0.00	NOT CIMPLE	×

Check

Automatic application of the seven design checks for a selected part of the wall.

	Ь	~	۲	Ŀ
 u		C	u	ĸ

	Masonry Verification ×									
1-1 ×										
Description 1-1										
l(cm)	901	.77	Pick	Check	Ratio	Strength	Load	σδ/Φ	T	^
h(cm)	300		Pick	Check 1	0.70(39)	88.54	62.25	58.14	2.50	
Support	: 4 Si	des	~	Check 2	0.49(1)	160.81	79.36	152.82	9.02	
	_			Check 3	0.00(0)	0.00	0.00	0.00	0.00	
New		U	lpdate	Check 4	2.42(37)	47.08	-114.09	1.90	2.50	\sim
Delet	e	le Masonry Buil <								
Ched	k	Total Check Results Total Results Exit								

Total Check

Total Check Automatic application of the seven checks in the structure in total.



	Masonry Verification								
1-1 ~									
Descripti	ion	1-1							
l(cm)	901.	77 Pick	Check	Ratio	Strength	Load	σδ/Φ	I	^
h(cm)	300	Pick	Check 1	0.70(39)	88.54	62.25	58.14	2.50	
Support	: 4 Sid	les 🗸	Check 2	0.49(1)	160.81	79.36	152.82	9.02	2
			Check 3	0.00(0)	0.00	0.00	0.00	0.00	
New		Update	Check 4	2.42(37)	47.08	-114.09	1.90	2.50	\sim
Delet	Delete le Masonry Buil <								
Chec	k	Total Check		Results	Tota	l Results		Exit	

The design checks are applied to sections (horizontal and vertical) concerning the EC6 design code.

- SCADA Pro scans each selected wall, at first horizontally and then vertically, the wall sections (strips of finite elements) are detected, and all the checks are applied to each section.
- During the scan, each strip of finite elements is colored according to the results of the design checks; bluegreen (all design checks of the section are satisfied) or red (one or more design checks of the sections are not satisfied).



Since the design checks' procedure has been completed, the user can elaborate on the results.

The command "Results"	Results	presents the results of all design checks for the selected
wall or part of the wall.		

			Ma	sonry Ver	ification				×
1-1									¥
Descripti	ion	1-1							
l(cm)	901.	77 Pick	Check	Ratio	Strength	Load	σδ/Φ	1	^
h(cm)	300	Pick	Check 1	0.70(39)	88.54	62.25	58.14	2.50	
Support	: 4 Sid	es 🗸	Check 2	0.49(1)	160.81	79.36	152.82	9.02	
			Check 3	0.00(0)	0.00	0.00	0.00	0.00	
New		Update	Check 4	2.42(37)	47.08	-114.09	1.90	2.50	×
Delet	e l	e Masonry Buil	<					>	
Ched	k	Total Check		Results	Tota	l Results		Exit	

The command "Total Results" Total Results presents the results for all the walls of the building.



Wall	Check 1	Check 2	Check 3	Check 4	Cher ^
1_1	0.60(1)	0.01(1)	0.02(1)	0.06(1)	1.29(1)
1_3	0.61(1)	0.01(1)	0.02(1)	0.11(1)	0.31(1)
1_4	0.61(1)	0.04(1)	0.01(1)	0.73(1)	0.13(1)
1_2	0.60(1)	0.06(1)	0.01(1)	0.51(1)	0.09(1)
•					►





8.2 Assessment (EC8-3)

The design checks are applied to the cross section of the pier/spandrel, where the dominant stress resultant is either:

- The axial force and bending moment, or
- The shear force

Consequently, the critical failure of the structural element is determined and the corresponding structural capacity is calculated for the three performance levels A, B, C.

After all the preliminary procedure select ert



In the dialog box that opens, the user must identify the parts of the walls to make the required checks as for the "New Masonry Building":

	Assessment	of mase	onry bu	uilding	(EC8-3))	×
1111						V Perf	ormance ojective
Description 1111	L					A	-DL 🗸
l(cm) 400	Pick Check	Ratio	D	Vf1	Vf2	Ved	δυ
h(cm) 300	Pick						
Support: 4 Sides	~						
New U	pdate						
Delete Reinf	orcement						
Check T	otal Check	Results		Total Re	sults		Exit

NOTE that:

The program automatically performs recognition of piers/spandrels. Therefore set the entire wall with the openings and the program checks automatically (separately) the piers and the spandrels (means wall sections above and below the openings)





Check to make all the checks for the selected wall for each pier/spandrel section.

Спеск										
			Asses	sment c	of maso	nry bi	uilding	(EC8-3)		×
2222									Perf Ot	ormance ojective
Descript	ion	222	2						A	- DL 🗸
l(cm)	200)	Pick	Check	Ratio	D	Vf1	Vf2	Ved	δυ
h(cm)	300)	Pick	Pier 1	1.172	1.00	6.78	68.99	-7.95	0.000
Cumpert		idee.		Pier 2	1.416	0.50	2.91	33.61	-4.12	0.000
Support	:45	des	*	Υπερθ. 1	5.381	0.50	1.80	25.93	9.69	0.000
New		l	Update	Υπερθ. 2	4.605	1.50	4.35	77.78	-20.01	0.000
Delete	е	Rein	forcement	<						>
Chec	k		Total Check		Results		Total Re	sults		Exit



Assessment of masonry building (EC8-3) 1111 Performance Objective A - DL I(cm) 200 Pick Check Batio D Vf1 Vf2 Ved õu I(cm) 200 Pick Check Batio D Vf1 Vf2 Ved õu V Pick Check Batio D Vf1 Vf2 Ved õu	otal Cheo Total Che	<mark>ck</mark> to make eck	all the c	checks f	or all t	the defi	ined wal	ls and	each pie	r/spandrel s
1111 V Performance Objective Description 1111 A - DL V I(cm) 200 Pick Check Batio D Vf1 Vf2 Ved õu		Asses	sment c	of maso	nry bu	uilding	(EC8-3)		×	
I(cm) 200 Pick Check Ratio D Vf1 Vf2 Ved õu	1111 Description	1111						✓ Pe	erformance Objective A - DL →	
	l(cm) 200	0 Pick	Check	Ratio	D	Vf1	Vf2	Ved	δυ	
h(cm) 300 Pick 1111 5.381 0.50 1.80 25.93 9.69 0.000	h(cm) 300	0 Pick	1111	5.381	0.50	1.80	25.93	9.69	0.000	
Support: 4 Sides 2222 5.381 0.50 1.80 25.93 9.69 0.000	Support: 4 S	Sides 🗸	2222	5.381	0.50	1.80	25.93	9.69	0.000	
New Update	New	Update								
Delete Reinforcement < >	Delete	Reinforcement	<						>	
Check Total Check Results Total Results Exit	Check	Total Check		Results		Total Re	sults		Exit	





The adequacy checks are performed per pier/spandrel section regarding forces and deformations depending on the Performance Level.

8.2.1 Masonry Reinforcement

SCADA Pro offers the possibility of reinforcing the masonry with:

- simple or double of the reinforced concrete jacket to increase the compressive, shear and flexural strength of the element
- Textile Reinforced Mortar for shear reinforcement in a plane
- Also, in cases of reinforcing with **Deep Jointing** or **Grouting**, you define the compressive strength of the of the reinforced masonry according to the corresponding formulas:

$$f_{wc} = \frac{1}{\gamma_{Rd}} \cdot \zeta \cdot f_{wc,o} \qquad \qquad f_{wc,i} = f_{wc,0} (1 + \frac{V_i}{V_w} \frac{J_{c,in}}{f_{wc,0}})$$
(Grouting)

After completing the checks through the "Masonry Assessment" printing files, you can read the characterization of fault resulting and reinforce accordingly.



Calculation's Printout													×	
Availiable Chapters	Printout			Nu	mber	of Pa	iges							
General Analysis Decim	Assessment Assessme	of \	Wall:	1111						Bu	uilding	Data		
Reinforcement	Assessme		C-14			Din	Wall	: 1111	m =7 70	(m) Heid	ht (h) =3.00	(m)	Assessm	age : 2 nent
Hasonry	Assessme	有利に行		201-1-1-1-1-1	たな話	Nar Typ Thio	me xe ckness (e	Mason : Single quivalen	Leaf Wal	wall - M2 : I = 50.00	50 cm	(
Masonry Assessment 1111 2222	133633116	2		19 A		Par Lim Kno	tial facto it State : owledge I	r for mas Level :	KL1:Li	= 2.20 A - DL mited	EC6	(82.4.3) Cl	EC8 (&9.6.(3)))
- 3333 - 4444 - 6666		N	Aasonry	properti	es :	Cha Me: Cha Me:	aract. cor an compr aract. initi an initial :	npressiv ressive s al shear shear stre	e strength trength fr strength f ength fvm	i fk (N/mn n (N/mm2 ivk0 (N/mi i0 (N/mm2	n2)) m2) ?)	-	2.62 3.12 0.10 0.15	
 Bill of Materials						Mar Shear	ximum sh Pler pro	ear stren perties a pacity	nd chara	ax (N/mm cterisatio Shea	2) n r force cap	acity	0.20	
		α/α	Height (cm)	Thick ness (cm)	(C Ho (cm)	D (cm)	N (kN)	vd (x10-3)	vf (kN)	(Capar D' (cm)	fvd (MPa)	Vf (kN)	Capacity controlled by:	Com binat ion
		2	300.0	20.0	600.0 155.6	200.0	-0.5 -0.8 -0.4	0.9	0.2	200.0	74.0 74.9 74.6	22.5 30.0 26.7	Flexure Flexure	20 68 28
			Lim	it state	of DL	Pier	strength	checks (in terms Limit s	of drift o tates of S	r force) D or NC			
		α/α	(in te Ved (kN)	Vf (kN)	Force) Ved / V	f (mn	n) (I	ui nm)	(in çj (rad)	çi (rad)	Drift)	δu (rad	j) δed / δυ	Suffi cien cy
			-0.1	0.2	0.56	9	_					-	_	Tes

8.2.1.1 Reinforcement with a concrete jacket

To reinforce a wall with a single or double jacket, in "Masonry" Library define the characteristics for the concrete jacket. Automatically change all the characteristics of the initial wall also.



Masonry	$^{\prime}$ stone wall - M5 50 cm $^{\prime}$ $^{\prime}$		Type Existing
Name	Masonry stone wall - M5 50 cm		Concrete jacket
			Thickness 10 Double-Leaf
Гуре	Load-bearing V Double-lear wall V		Cocrete Steel
Masonn	y uni Stones - stones drilled 20x20x25 V		
	Thickness 25 fb=8.0000 fbc=8.0000 ε=20.00		Φ 10 / 10 cm fRd,c (MPa)= 0.30
Mortar	Mortar Cement-M5 V		Anchorage Without any additional car V
	General purpose designed masonry mortar fm=5.0000		A BARANCE A BARANCE
Wall	? L1 0 t1 (cm) 0 t2 (cm) 0		
Shell Be	edded Wall		
Tatal			
TOLATV	vidth of the two mortar strips g (cm)		
tef=2	5.00 k=0.45 fk=3.1266		Filled vertical joints (3.6.2)
tef=2	5.00 k=0.45 fk=3.1266		Filled vertical joints (3.6.2) Red join of thickness >15 mm
tef=2	vidth of the two mortar strips g (cm)		Filled vertical joints (3.6.2) Bed join of thickness >15 mm
tef=2	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266 y uni Stones - stones drilled 20x20x25 Thickness 25 fh=8.0000 fb=8.0000 s=20.00		Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent)
tef=2	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266		Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight
Total V tef=2: Masonny Mortar	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266		Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight 21.42857 Compressive strength fk
Tef=2: Masonn Mortar Wall	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266	Masonry units - Mortars library	Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight 21.4285 Compressive strength fk 11.74376 Modulus of elasticity 1000 10.80461
Masonn Mortar Wall	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266 ? y uni Stones - stones drilled 20x20x25 ~ Thickness 25 fb=8.0000 fbc=8.0000 ɛ=20.00 Mortar Cement-M5 ~ General purpose designed masonry mortar fm=5.0000 ? 2 L1 0 t1 (cm) 0 t2 (cm) 0 5.00 k=0.45 fk=3.1266 ?	Masonry units - Mortars library	Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight 21.42855 Compressive strength fk 11.74376 Modulus of elasticity 1000 Characteristic strength fk 0.1
Masonry Mortar Wall tef=22	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266	Masonry units - Mortars library	Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight 21.4285 Compressive strength fk 11.74376 Modulus of elasticity 1000 Characteristic strength fvk0 (N/mm2) Maximum shear strength fvkmax (N/mm2)
Masonn Mortar Wall tef=2! Concret	vidth of the two mortar strips g (cm) 0 ? 5.00 k=0.45 fk=3.1266	Masonry units - Mortars library	Filled vertical joints (3.6.2) Bed join of thickness >15 mm Thickness (Equivalent) 70 Specific weight 21.42857 Compressive strength fk 11.74376 Modulus of elasticity 1000 Characteristic strength fk/k0 (N/mm2) Flexural strength fxk1 (N/mm2) Conserver

Set a new name for the reinforced element and save it for using it later, defining the reinforced wall.

Masonry	stone wall - M5.5	50 cm			\sim	Type	Existing	
						Concrete jacket	5	
lame	REINF/Masonry	stone wall	- M5 50 cm			Thickness 10	Doub	ole-Leaf
уре	Load-bearing	~ D	ouble-leaf wall	~	?	Cocrete	Steel	
						C20/25	~ S500)
Masonry	uni Stones - stor	nes drilled 2	20x20x25	~		⊕ 10 / 10 g	m 60 d a (MC	
	Thickness	25	fb=8.0000 fbc=	-8.0000 ε=20.00			IN IRU,C (MP	'a)= 0.30
Mortar	Mortar Ceme	ent-M5		×		Anchorage With	out any addit	ional car
-ior car	General nurn	iose designe	ed masonry morta	r fm=5 0000			1111111	
A/-11		obe designe	t1 (cm) 0	+2 (cm) 0				
/Vdll	: 11	0					E	
Shell Bed	ded Wall						1	
Total wi	dth of the two m	nortar strips	s g (cm)	0 ?				
tef=25.	00 k=0.45 fk=3.	1266				Filled vertical j	oints (3.6.2)	[
tef=25.	00 k=0.45 fk=3.	1266				Filled vertical j	joints (3.6.2) ckness >15 m	.m
tef=25. Masonry	00 k=0.45 fk=3. _{uni} Stones - stor	.1266 nes drilled 2	20x20x25			Filled vertical j	joints (3.6.2) ckness >15 m	im
tef=25. Masonry	00 k=0.45 fk=3. _{uni} Stones - stor	.1266 nes drilled 2	20x20x25	-9 0000		Filled vertical j Bed join of thic Thickness (Equiv	ioints (3.6.2) ckness >15 m ralent)	im 70
tef=25. Masonry	00 k=0.45 fk=3. _{uni} Stones - stor Thickness	.1266 nes drilled 2 25	20x20x25 fb=8.0000 fbc=	-8.0000 ε=20.00		Filled vertical j Bed join of thic Thickness (Equiv Specific weight	oints (3.6.2) ckness >15 m ralent)	rm 70 21.428
tef=25. Masonry Mortar	00 k=0.45 fk=3. _{uni} Stones - stor Thickness Mortar Ceme	1266 nes drilled 2 25 ent-M5	20x20x25] fb=8.0000 fbc=	-8.0000 ε=20.00		Filled vertical j Bed join of this Thickness (Equiv Specific weight	joints (3.6.2) ckness >15 m ralent)	m 70 21.428
tef=25. Masonry Mortar	00 k=0.45 fk=3. uni Stones - stor Thickness Mortar Ceme General purp	1266 nes drilled 2 25 ent-M5 ose designe	20x20x25] fb=8.0000 fbc= ed masonry mortar	≈8.0000 ε=20.00 × fm=5.0000	Masonry units	Filled vertical j Bed join of thi Thickness (Equiv Specific weight Compressive str	ioints (3.6.2) ckness >15 m ralent) ength fk	rm 70 21.428 11.743
tef=25. Masonry Mortar Wall	00 k=0.45 fk=3. Uni Stones - stor Thickness Mortar Ceme General purp ? L1	1266 nes drilled 2 25 ent-M5 ose designe	20x20x25 fb=8.0000 fbc= ed masonry mortar t1 (cm) 0	=8.0000 ε=20.00 r fm=5.0000 t2 (cm) 0	Masonry units - Mortars library	Filled vertical j Bed join of thia Thickness (Equiv Specific weight Compressive str Modulus of elast (GPa)	ioints (3.6.2) ckness >15 m ralent) ength fk icity 1000	m 70 21.428 11.743 10.804
tef=25. Masonry Mortar Wall tef=25.	00 k=0.45 fk=3. Uni Stones - stor Thickness Mortar Ceme General purp ? L1 00 k=0.45 fk=3.	1266 nes drilled 2 25 ent-M5 ose designe 0 1266	20x20x25 fb=8.0000 fbc= ed masonry mortar t1 (cm) 0	-8.0000 ε=20.00 r fm=5.0000 t2 (cm) 0	Masonry units - Mortars library	Filled vertical j Bed join of thia Thickness (Equiv Specific weight Compressive stru- Modulus of elast (GPa) Characteristic st (N/mm2)	ioints (3.6.2) ckness >15 m ralent) ength fk icity 1000 rength fvk0	m 70 21.428 11.743 10.804 0.1
tef=25. Masonry Mortar Wall tef=25. Concrete	00 k=0.45 fk=3. uni Stones - stor Thickness Mortar Ceme General purp ? L1 00 k=0.45 fk=3. infill	1266 25 ent-M5 ose designe 0 1266 fck (N/mm2	20x20x25 fb=8.0000 fbc= ed masonry mortan t1 (cm) 0 2) Thickness	=8.0000 ε=20.00 r fm=5.0000 t2 (cm) 0	Masonry units Mortars library	Filled vertical j Bed join of thia Thickness (Equiv Specific weight Compressive strr Modulus of elast (GPa) Characteristic st (N/mm2) Maximum shear fvkmax (N/mm2	ioints (3.6.2) ckness >15 m ralent) ength fk icity 1000 rength fvk0 strength	m 70 21.428 11.743 10.804 0.1 0.36
tef=25. Masonry Mortar Wall tef=25. Concrete C20/25	00 k=0.45 fk=3. Uni Stones - stor Thickness Mortar Ceme General purp ? L1 00 k=0.45 fk=3. infill	1266 res drilled 1 25 ent-M5 ose designe 0 1266 fck (N/mm2 20	20x20x25 fb=8.0000 fbc= ed masonry mortar t1 (cm) 0 2) Thickness 0	-8.0000 ε=20.00 r fm=5.0000 t2 (cm) 0	Masonry units - Mortars library	Filled vertical j Filled vertical j Bed join of thia Thickness (Equiv Specific weight Compressive stru- Modulus of elast (GPa) Characteristic st (N/mm2) Maximum shear fvkmax (N/mm2 (N/mm2)	ickness >15 m ralent) ength fk icity 1000 rength fvk0 strength) n fxk1	m 70 21.428 11.743 10.804 0.1 0.36 0.1
tef=25. Masonry Mortar Wall tef=25. Concrete C20/25 ata reliab	00 k=0.45 fk=3. Uni Stones - stor Thickness Mortar Ceme General purp ? L1 00 k=0.45 fk=3. infill pillty level	1266 nes drilled ; 25 ent-M5 ose designe 0 1266 fck (N/mm2 20	20x20x25 fb=8.0000 fbc= ed masonry mortar t1 (cm) 0 2) Thickness 0 Execution	-8.0000 ε=20.00 r fm=5.0000 t2 (cm) 0	Masonry units - Mortars library New Save	Filled vertical j Filled vertical j Bed join of thia Thickness (Equiv Specific weight Compressive stru- Modulus of elast (GPa) Characteristic st (N/mm2) Maximum shear fvkmax (N/mm2) Flexural strengtl (N/mm2)	ickness >15 m ralent) ength fk icity 1000 rength fvk0 strength) n fxk1 n fxk2	m 70 21.428 11.743 10.804 0.1 0.36 0.1 0.4



Open Mess 3D command and Calculation to identify the sub-groups that need reinforcement:



Then inside the window of the mesh group identify the sub-groups and modify their **quality** and **thickness**:

Plate Elements Creation					×
Description PLATE		Material Mason	ry ~	Туре	REINF/Masonr $ imes $
Element	Ks (Mpa/cm)	Isotropic	Orthotr	opic	Angle 0
Plate 🗸 🖓	300				
Density Width (cm)	Thickness	Exx (GPa)	10.80468436	Gxy (GPa)	4.321873747
0.05 ~ 30	70	Eyy (GPa)	10.80468436	ε (kN/m3)	21.42857142
Descriptions Mesh	ו ו	Ezz (GPa)	10.80468436	atx*10-5	1
Mesh Groups	Surface	vxy(0.1-0.3)	0	aty*10-5	1
2S PL/ 3P S1/	ATE(1) /3/3	vxz(0.1-0.3)	0	atxy*10-5	1
4P S1, 5P S1,	/4/2(2) /5/2	vyz(0.1-0.3)	0	Exx * v	z = Eyy * vxy
6P S1,	/6/2	Podofinitio			
8P S1	/8/2	Redefinitio	Steel Re	inforcement	ОК
9P S1	/9/2	Del From Li	st s220		
10P S1 11P S1	/11/3	New	3220		Exit

Remember to press Redefinition every time you modify something

Then, repeat the analysis process, updated with new data, and check again the reinforced wall to receive new adequacy ratios, until you manage to get ratios smaller than one. The process is iterative and can be done repeatedly.



§ WALLS WITH CONCRETE JACKET – NOTES:

What is affected?

The assignment of concrete jackets affects the following:

- The Equivalent Thickness
- The Specific Weight
- The Modulus of Elasticity
- The Characteristic Compressive Strength
- The Characteristic Shear Strength

Note: Since the equivalent thickness and the Modulus of Elasticity change, the tension of the elements is different than the non concrete jacket ones. As a result, you will have to change the thickness of the surface elements and run the analysis again.

What checks are being made?

The checks being made are the same as those on walls without a concrete jacket. That is to say, the provisions of Eurocode EC8-3 (annex C) which concern the following are applied:

- In-plane shear
- In-plane bending

Which parameters change?

The assignment of concrete jackets brings out the following changes:

-Equivalent Thickness
-Specific Weight
-Compressive Strenght
-Characteristic Compressive Strenght
-Modulus of Elasticity

It is obvious that some parameters do not change. The reasons why this happens is because:

- 1. Either they are not used or they are not necessary for the checks of the EC8-3.
- 2. These are parameters which do not change (eg shear strength of a masonry without loading), but they are used or needed for the EC8-3 checks.

In the evaluation printout, we see similar differences.

8.2.1.2 Textile Reinforced Mortar (TRM)





Use Textile Reinforced Mortar for shear reinforcement in the plane, defined by the corresponding window for the selected wall from the list.

Select the "Design Method".

To SCADA Pro contains two methods and you can select between

ACI 549.4R-13	~
ACI 549.4R-13	
Triantafillou & Antonopoulos (2000)	

Specify the characteristics of the mesh, based on catalogs and commercial materials.

1 In SCADA Pro company materials have been introduced

EM4C Sika

By selecting the company and the corresponding material the mesh features are automatically filled in by the program.







Then press again the "Checks" button and check the results obtained after the introduction of the grid.

You can repeat the process. The program is making checks each time to take into account the latest features you specified.







Member Diagrams

This command is used for the display of the structural members' stress resultants diagrams in 2D.

Press the command and then left click to select a structural member. In the dialog box, all the diagrams of the stress resultants of the corresponding member are displayed. Select a load combination or a load case and move the mouse in the framework of the diagrams to read the corresponding values of the stresses resultant along the member.





NOTE

Right mouse click on a member opens a command list containing also Members Diagrams.





9.3 Nodes Displacements

7 Nodal Displacements

There is now the possibility to see the nodes displacements.

Select the command and point the mouse on a node. In the dialog box that opens, all the movements of that node are displayed. Select a combination or load to read the corresponding values.

Displacements	of Node : 34	D	×
Coordinates :	1366.83,1000.00	,450.99	
Dx(mm)	0.2681		
Dy(mm)	-1.2344		
Dz(mm)	0.1889		
Rx(rad)	0.0000		
Ry(rad)	0.0000		
Rz(rad)	0.0001		
Load Case	~	1	\sim
Maximu	um Rates	Exit	

Press Maximum Rates to see the maximum value for each movement and rotation, as well as the combination from which it originates. The number of this combination is written in brackets next to the value.

splacement	s of Node : 34	
Coordinates	: 1366.83,1000.00,450.99	
Dx(mm)	0.2681(1)	
Dy(mm)	-1.2344(1)	
Dz(mm)	0.2998(5)	
Rx(rad)	0.0000(1)	
Ry(rad)	0.0000(7)	
Rz(rad)	0.0001(1)	
Load Case	✓ 1	~
Maxim	um Rates Exit	



Right mouse click on a member opens a command list containing also Nodal Displacements.



9.4 Plate Elements Internal Forces

🛞 Plate Element Internal Forces

There is now the possibility to see the Internal Forces in Plate

Elements.

Select the command and point the mouse on a plate element.

The dialog box that opens display all stresses and moments for that plate element. Select a combination or load to read the corresponding values of stresses and moments.

Plate Elements Internal Forces : 53			Х
Nodes : 997 , 1027 , 995 , 966			
σXX(kN/m2)	2476.0000		
σYY(kN/m2)	-3654.0938		
σXY(kN/m2)	212.5625		
MXX(kNm/m)	193.9150		
MYY(kNm/m)	-19.8182		
MXY(kNm/m)	-0.9999		
Load Case \checkmark 1 \checkmark		~	
Maximum Rates		Exit	

Press Maximum Rates to see the maximum value for each stress and moment, as well as the combination from which it originates. The number of this combination is written in brackets next to the value.

NOTE

Right mouse click on a node opens a command list containing also Plate Element Internal Forces.





9.5 Slab Strips Diagrams

Slab Strips Diagrams

There is now the possibility to see the Slab Strips Diagrams.

Select the command and point the mouse on a Slab Strip.

The dialog box that opens display all diagrams of the internal forces for each span. Select a combination or load and move the mouse on the diagrams to read the corresponding values along the section.

In case of Load Pattern, activate ZLoad Pattern to see the diagrams resulting.

Move to the next span using the arrows on the bottom. |>>



