

User's Manual 9B.MEMBERS DESIGN SBC306_17_STEEL







CONTENTS

THE	NEW UPGRADED INTERFACE OF SCADA PRO	3
DETA	AILED DESCRIPTION OF THE NEW INTERFACE	4
Мемвер	rs Design	4
1. Sc	ENARIOS	4
1.1	New	5
1.2	LIST	5
1.3	Parameters	6
1.3.1	COMBINATIONS	7
1.3.2	STEEL CROSS SECTIONS	14
1.4	Merge Elements	20
1.4.1	AUTO MERGE	24
1.4.2	USERS MERGING	27
1.4.2.1	Merge concrete columns	27
1.4.3	VIEW	28
1.4.4	CORRECTION	29
1.4.5	Single Delete	30
1.4.6	OVERALL DELETE	30
2. St	EEL	31
2.1.1	CROSS SECTION DESIGN	31
2.1.2	BUCKLING MEMBERS INPUT	39
2.1.3	COLD FORMED SECTIONS	49
7.1.4	CONNECTIONS	50
	THE DET/ MEMBER 1. So 1.1 1.2 1.3 1.3.1 1.3.2 1.4 1.4.1 1.4.2 1.4.2 1.4.2 1.4.3 1.4.4 1.4.5 1.4.6 2. ST 2.1.1 2.1.2 2.1.3 7.1.4	THE NEW UPGRADED INTERFACE OF SCADA PRO DETAILED DESCRIPTION OF THE NEW INTERFACE MEMBERS DESIGN 1. SCENARIOS 1.1 NEW 1.2 LIST 1.3 PARAMETERS 1.3.1 COMBINATIONS 1.3.2 STEEL CROSS SECTIONS 1.4 MERGE ELEMENTS 1.4.1 AUTO MERGE 1.4.2 USERS MERGING 1.4.3 VIEW 1.4.4 CORRECTION 1.4.5 SINGLE DELETE 1.4.6 OVERALL DELETE 2.1.1 CROSS SECTION DESIGN 2.1.2 BUCKLING MEMBERS INPUT 2.1.3 COLD FORMED SECTIONS





3



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	Drawings-Detailing	Addons	Optimization
	Mei	mber	s Des	ign								
	ivici											
	Basic Modeling	View To	ols Slabs	Loads	Analysis	Post-Processor	Members Design	Drawings-De	tailing Addons	Optimization		
Z,	EKOS 2000-EAK 1 (C 🔻	7, 🦻		1		7 7	1 4		L V	s 🗞 🛸 🤋	2 🗩 🥖	
New	Active Scenario	Para- Merge meters Elements	Continuity of Beams	Check - Reinforcemer	Results	Node Design Releases *	Buckling Check - Reinforceme	Results	Check Rest Reinforcement * *	Its Slab Strip Flat Res Calculation * slabs *	ults Steel Timbe Design Design	r Masonry Member Design* Diagrams*
	Scenarios			Beams		Capacity Design	Columns		Footings	Slabs - Mesh	Steel - Timber	

The 9th Unit entitled "Members Design" contains the following 8 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

1. 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 This command is us corresponding design regula	sed in order to create a new sation and then press the button	scenario. Type a name, select the
Scenario 1 Exit	X Name Type SBC304-306 New Update Design Delete Concrete Steel Apply	EKOS 2000-EAK EC2-EC3 NTC_2008 EC2_Italia EC2_Cyprus Greek old 1959-84 Greek old 1984-93 Austria <u>SBC304-306</u> 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.

In order to modify an existing scenario press the button "Update"/.

Design Delete	
Concrete	Connections
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:

Para- meters							
Structura	I Component Para	meters					Х
Stee	el Reinforcement mbinations	Capacit Slabs	ty Design Beams	Steel Colum	Tir	mber structures 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.

uctural Compon	ent Param	neters						
Steel Reinforce	ment	Сарас	ity kesign		Steel	Timber s	tructu	res
Combinations		Slabs	Bea	ms	Columns	Footings		
Combinations of L	oad Sets	(101)	(101) Ult. Serv. +X		+XX	+Z	Z	No
Combinations						ULS/SLS	Dir.	^
1(14) +1.35Lc1-	ULS							
2(1) +1.00Lc1+	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+0.3	0Lc4+1.00	Lc5+0.3	0Lc70.30Lc9	ULS	+X	
5(2) +1.00Lc1+	0.30Lc2+1	.00Lc30.3	0Lc4+1.00	Lc50.3	0Lc7+0.30Lc9	ULS	+X	
6(2) +1.00Lc1+	0.30Lc2+1	.00Lc30.3	0Lc4+1.00	Lc50.3	0Lc70.30Lc9	ULS	+X	
7(2) +1.00Lc1+	0.30Lc21	.00Lc3+0.3	0Lc41.00	Lc5+0.3	0Lc7+0.30Lc9	ULS	X	
8(2) +1.00Lc1+	0.30Lc21	.00Lc3+0.3	0Lc41.00	Lc5+0.3	0Lc70.30Lc9	ULS	X	
9(2) +1.00Lc1+	0.30Lc21	.00Lc30.3	0Lc41.00	Lc50.3	0Lc7+0.30Lc9	ULS	X	
10(2) +1.00Lc1·	+0.30Lc2	1.00Lc30.	.30Lc41.0	0Lc50	30Lc70.30	ULS	X	~
Level Multipliers		1/	(1-θ)	1			_	~
Level	Х	Y	Z		Insert Co	mbinations		
0 - 0.00	1.000	1.000	1.000		Combinatio	ns Calculatio	on	
1 - 400.00	1.000	1.000	1.000					
2 - 700.00	1.000	1.000	1.000		Combination C		101	1
3 - 1000.00	1.000	1.000	1.000		Combination G	τψ <i>2</i> · <u>γ</u>		
					Au	tomatic Desi	ign	
0		أمعما				OK	-	

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

Press "Save" and type a name





Jave AS							>
→ ~ ↑	> This PC > BOOTCAMP (C:) > me	eletes > 1dokos > scaanal >		~ ō	Search scaanal		P
rganize - N	ew folder					855 •	0
OneDrive This PC Desktop Documents Downloads Music Pictures Videos Stores New Volume NINTENSO (F;	 Name Scen000 Scen02 ▲ sbc.sdp 	Date modified 10/5/2017.118 μμ 10/5/2017.1148 μμ 10/5/2017.1353 μμ	Type File folder File folder VLC media file (.s	Size 9	6 KB		
INTENSO (F:)	test.sdp						

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:



		SBC301 Sta	itic (0).cmb)			\sim							
rom the drond	own list	lefault.cmb EC8_Genera SBC301 Sta	al Nonlinea tic (0).cmb	ar (1)).cmb		wi	th auto	mati	c c	alcu	lation	or	
			Insert C	omb	ination	s			inac					
nrough the co	mmand							that	oper	ns t	the	folder	r witl	1 the
gistered .cmb	files. Sel	ect the fil	le and p	ress	5	Co	ombina	ations Ca	lculati	on				
			•											
tructural Compo	nent Para	meters										×		
Steel Reinford	ement	Сара	acity Desig	n		Steel		Timb	er stn	uctu	res			
Combinations	•	Slabs	В	eam	s	С	olumns	3	Fo	otin	gs			
Combinations of	Load Sets	(101	I) Ult.	Se	erv.	+X	-X	+Z	Z		No			
Combinations								ULS/SI	S	Dir.	~			
1(14) +1.40Lc1	+1.70Lc2							ULS						
2(1) +1.20Lc1+	1.00Lc2+1	.00Lc3+1.0	0Lc4+1.00	Lc5	+0.30L	c7+0.3	OLc9	ULS		۰X				
3(2) +1.20Lc1+	1.20Lc2+1	.00Lc3+1.0	0Lc4+1.00	Lc5	+0.30L	c70.3	OLc9	ULS		۰X				
4(2) +1.20Lc1+	1.20Lc2+1	.00Lc3+1.0	0Lc4+1.00	Lc5	0.30L	c7+0.3	OLc9	ULS		+X				
5(2) +1.20Lc1+	1.20Lc2+1	.00Lc3+1.0	0Lc4+1.00	Lc5	0.30L	c70.3	OLc9	ULS		+X				
6(2) +1.20Lc1+	1.20Lc21	.00Lc3+1.0	0Lc41.00)Lc5	+0.30L	c7+0.3	OLc9	ULS		-X				
7(2) +1.20Lc1+	1.20Lc21	.00Lc3+1.0	0Lc41.00)Lc5	+0.30L	c70.3	OLc9	ULS		-X				
8(2) +1.20Lc1+	1.20Lc21	.00Lc3+1.0	0Lc41.00)Lc5	0.30L	c7+0.3	OLc9	ULS	· ·	-X	_			
9(2) +1.20Lc1+	1.20Lc21	.00Lc3+1.0	0Lc41.00)Lc5	0.30L	c70.3	OLc9	ULS	· ·	-X				
10(2) +1.20Lc1	+1.20Lc2+	1.00Lc3+1.	00Lc4+1.()0Lc	5+0.30	Lc8+0.	30Lc9	ULS		+X ≥	~			
Level Multipliers		1	/ (1.A)	_	SBC3	01 Stat	ic (0) c	mb		_	~			
			7 (10)	_	defau	lt.cmb	.0 (0).0							
Level	Х	γ	Z	^	EC8_	Genera	Nonlir	near (1).c	mb					
0 - 0.00	1.000	1.000	1.000		SBC3	01 Stat	ic (U).c IDINALIO	mb iris caicu	auon					
1 - 300.00	1.000	1.000	1.000				En	d Calc						
2 - 600.00	1.000	1.000	1.000		,	Combin	ation (6+m20	99		7			
3 - 900.00	1.000	1.000	1.000				Διιστή ο	tomatic [Deelar		1			
4 - 1200.00	1.000	1.000	1.000	¥			Au	itomatic L	Jeargi					
								OK		0				
								UK		Ca	incel			

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.



Decentention	cement	Cap	acity Desi	gn	Steel	Timber	structu	res		
Combinations	s	Slabs	1	Beams	Column	s	Footin	gs		
Combinations of	f Load Sets	; (10	01) Ult	. Serv.	+XX	+Z ·	Z	No		
Combinations						ULS/SLS	Dir.	^		
1(14) +1.35Lc	1+1.50Lc2	2				ULS				
2(1) +1.00Lc1	ULS									
3(2) +1.00Lc1+0.30Lc2+1.00Lc3+0.30Lc4+1.00Lc5+0.30Lc7+0.30Lc9 ULS										
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	301 - 41	0.01 5 0						
			J.JULC41	.00LC50	.30Lc7+0.30Lc9	ULS	X			
10(2) +1.00Lc	1+0.30Lc2	21.00Lc3-	-0.30Lc4	-1.00Lc50	.30Lc7+0.30Lc9 0.30Lc70.30	ULS	X X	~		
10(2) +1.00Lc <	1+0.30Lc2	21.00Lc3-	-0.30Lc4	-1.00Lc50	.30Lc7+0.30Lc9 0.30Lc70.30	ULS	X X	~		
10(2) +1.00Lc «	1+0.30Lc2	21.00Lc3-	-0.30Lc4 1 / (1-θ)	-1.00Lc50	.30Lc7+0.30Lc9	ULS	X X	*		
10(2) +1.00Lc evel Multipliers 	1+0.30Lc2	21.00Lc3-	-0.30Lc4 1 / (1-θ) Z	-1.00Lc50	.30Lc7+0.30Lc9 0.30Lc70.30 Insert C	ULS ULS	X X >	*		
10(2) +1.00Lc Level Multipliers Level 0 - 0.00 	1+0.30Lc2 X 1.000	21.00Lc3 Y 1.000	-0.30Lc4 1 / (1-θ) Z 1.000	-1.00Lc50	.30Lc7+0.30Lc5 0.30Lc70.30 Insert C Combinati	OULS ULS Combinations	X X >	~		
10(2) +1.00Lc < evel Multipliers Level 0 - 0.00 1 - 300.00	X 1.000 1.000	21.00Lc3 Y 1.000 1.000	-0.30Lc4 1 / (1-θ) Z 1.000 1.000	-1.00Lc50	.30Lc7+0.30Lc5 0.30Lc70.30 Insert C Combinatio	OLS ULS Combinations	X X >	~		
10(2) +1.00Lc < Level Multipliers 0 - 0.00 1 - 300.00 2 - 600.00	x 1.000 1.000	Y 1.000 1.000 1.000	-0.30Lc4 1 / (1-θ) Z 1.000 1.000 1.000	-1.00Lc50	.30Lc7+0.30Lc5 0.30Lc70.30 Insert C Combination C	ombinations	X X X s ion	•		
10(2) +1.00Lc evel Multipliers Level 0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 	X 1.000 1.000 1.000 1.000	Y 1.000 1.000 1.000 1.000	J. JOLC4-1 1 / (1-θ) Z 1.000 1.000 1.000		.30Lc7+0.30Lc5 0.30Lc70.30 Insert C Combination Combination C	ombinations combinations cons Calculat G+ψ2Q [atic Design	X X >	*		
10(2) +1.00Lc evel Multipliers Level 0 - 0.00 1 - 300.00 2 - 600.00 3 - 900.00 4 - 1200.00	1+0.30Lc2 X 1.000 1.000 1.000 1.000 1.000	Y 1.000 1.000 1.000 1.000 1.000 1.000	-0.30Lc4 1 / (1-0) Z 1.000 1.000 1.000 1.000 1.000		.30Lc7+0.30Lc5 0.30Lc70.30 Insert C Combination C Autom Recalculate	OULS ULS combinations ons Calculat G+ψ2Q [atic Design KAN.EPE. v:	X X X >	~		

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 in order 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.

Indicatevely, in the following printout:



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	4.00
9	144.800	15.876	11.857	4.000	144.800	2.907	64.759	4.00
10	144.800	15.876	11.857	4.000	144.800	2.907	64.759	4.00
11	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.00
12	134.000	15.569	11.189	4.000	134.000	3.416	50.993	4.00
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	4.00
15	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.00
16	144.800	15.876	11.857	4.000	134.000	5.114	34.063	4.00
17	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.00
18	144.800	15.569	12.091	4.000	144.800	3.416	55.103	4.00
19	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.00
20	134.000	14.853	11.728	4.000	134.000	4.605	37.831	4.00
21	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.00
22	134.000	14.547	11.975	4.000	144.800	2.907	64.759	4.00
23	144.800	14.547	12.941	4.000	134.000	2.907	59.929	4.00
24	144.800	14.547	12.941	4.000	134.000	2.907	59.929	4.00
25	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.00
26	144.800	14.853	12.673	4.000	144.800	4.605	40.880	4.00
27	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.00
28	134.000	14.547	11.975	4.000	134.000	5.114	34.063	4.00
29	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.00
30	134.000	14.853	11.728	4.000	144.800	3.416	55.103	4.00
31	144.800	14.853	12,673	4.000	134,000	3,416	50,993	4.00
32	144.800	14.853	12,673	4.000	134,000	3,416	50,993	4.00
33	144.800	14.547	12,941	4.000	144.800	5,114	36.808	4.00
34	144,800	14.547	12,941	4.000	144.800	5,114	36,808	4.00
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	0.00
39	134.000	4.205	41,422	0.000	144.800	12.774	14.736	0.00
40	134.000	4.205	41,422	0.000	144.800	12.774	14,736	0.00
41	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.00
42	144.800	5.228	36.006	0.000	144.800	12.264	15.349	0.000
13	134.000	4.921	35, 397	0.000	134.000	12.774	13.637	0.00
44	134 000	4 621	35, 367	0.000	134 000	12.774	13.637	0.00
45	144 900	3 900	48 282	0.000	134.000	12 264	14 204	0.00
16	144.800	3.033	49 292	0.000	134.000	12.201	14 204	0.000
10	124 000	3.039	10.203	0.000	144 800	12.204	15 240	0.000
10	134.000	3.039	44 692	0.000	144.000	12.204	15.349	0.000
10	134.000	3.099	30.052	0.000	144.800	12.204	10.349	0.000
19	144.800	4.921	38.250	0.000	144.800	12.774	14.736	0.000

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	ooting	gs
Combinations of Load S	Gets (101)	Ult. Serv.	+XX	+Z:	z I	Vo
Combinations			l	JLS/SLS	Dir.	^
1(14) +1.35Lc1+1.50	Lc2) i	JLS		
2(1) +1.00Lc1+0.50L	c2		l	JLS		
3(2) +1.00Lc1+0.30L	c2+1.00Lc3+0.30	Lc4+1.00Lc5+0.3	0Lc7+0.30Lc9	JLS	+X	
4(2) +1.00Lc1+0.30L	c2+1.00Lc3+0.30	Lc4+1.00Lc5+0.3	0Lc70.30Lc9 (JLS	+X	
5(2) +1.00Lc1+0.30L	c2+1.00Lc30.30	Lc4+1.00Lc50.3	0Lc7+0.30Lc9	JLS	+X	
6(2) +1.00Lc1+0.30L	c2+1.00Lc30.30	Lc4+1.00Lc50.3	0Lc70.30Lc9 (JLS	+X	
7(2) +1.00Lc1+0.30L	c21.00Lc3+0.30	Lc41.00Lc5+0.3	0Lc7+0.30Lc9	JLS	X	
8(2) +1.00Lc1+0.30L	c21.00Lc3+0.30	Lc41.00Lc5+0.3	0Lc70.30Lc9	JLS	X	
9(2) +1.00Lc1+0.30L	c21.00Lc30.30	Lc41.00Lc50.3	0Lc7+0.30Lc9	JLS	X	
10(2) +1.00Lc1+0.30	Lc21.00Lc30.3	0Lc41.00Lc50.	30Lc70.30 I	JLS	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.

ctural Component Par	ameters					>
Combinations	Slabs	Beams	Colum	ns	Footings	
Steel Reinforcement	Capaci	ty Design	Steel	Tim	ber structures	
Direction x = acc Edge 3.5 Middle 3.5 Fixed 2.1.3 Free 3.5	5		Direction z Edge Middle Fixed Free		acd <=	
			3			

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

SECTION 2.3

COMBINING FACTORED LOADS USING STRENGTH DESIGN

2.3.2. Basic Combinations. Structures, components, and foundations shall be designed so that their design strength equals or exceeds the effects of the factored loads in the following combinations: U = 1.4 (D + F) (9-1)



U = 1.4 (D + F + T) + 1.7(L + H) + 0.5 (Lr or R) (9-2) U = 1.2D + 1.6 (Lr or R) + (1.0 L or 0.8 W) (9-3) U = 1.2D + 1.6W + 1.0L + 0.5(Lr or R) (9-4) U = 1.2D + 1.0E + 1.0L (9-5) U = 0.9D + 1.6W + 1.6H (9-6)U = 0.9D + 1.0E + 1.6H (9-7)

In the preceding expressions, the following values are used: U = the design or ultimate load the structure needs to be able to resist D = dead load L = live load Lr = roof live load S = snow load R = rain load W = wind load E = seismic or earthquake load effects

"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 $1/(1-\theta)$ in order to take into account the P-Delta effect during the design check. The stress resultants will be increased automatically at the corresponding levels, where $0.1 < \theta < \theta$ max.

10.9.7.2

When the stability coefficient (ϑ) is greater than 0.10 but less than or equal to ϑ max the incremental factor related to P-delta effects (ad) shall be determined by rational analysis. To obtain the story drift for including the P-delta effect, the design story drift determined in Section 10.9.7.1 shall be multiplied by 1.0/(1 - ϑ).

ATTENTION:

For modification purposes, press the following button

Combinations Calculation

▲ The following field/concerns only the Greek EKOS.



Structural Component Parameters Slabs Combinations Beams Columns Footings Steel Reinforcement Steel Capacity Design Timber structures Select All Name ~ Deselect All Lines circles Paste Copy Concrete Columns Concrete Jackets Parameters Concrete Beams Concrete Foundation Beams GENERAL Footing Connection Beams TENSION Footings Steel Columns SHEAR Steel Beams Surface Mesh TORSION Mathematical Model Surface Elements COMPRESSION Mesh 3D Mesh 2D BENDING Slabs-Strips **BENDING & AXIAL FORCE** Steel Columns Steel Beams **BENDING & SHEAR FORCE** Main Beams Purlins BENDING SHEAR AXIAL FORCE Girders Secondary Columns Default Ý Hor Wind bracings OK Cancel

1.3.2 Steel cross sections

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.



Parameters						
✓	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).

Name	^	Select All	Deselect All
Lines circles			
Concrete Columns		Сору	Par
Concrete Jackets		Parameters	1.1
Concrete Beams		raianeteis	
Concrete Foundation Beams		✓	GENERAL
Footing Connection Beams			
Footings		✓	TENSION
Steel Columns			
Steel Beams		✓	SHEAR
Surface Mesh		J	TODSION
Mathematical Model			TORSION
Surface Elements			COMPRESSION
Mesh 3D			
Mesh 2D			BENDING
Slabs-Strips		•	
		 BEND 	ING & AXIAL FORCE
Steel Columns			
Steel Beams		BENDI	NG & SHEAR FORCE
Iviain beams			
Circlere		BENDING	SHEAR AXIAL FORCE
Girders			
Secondary Columns	~	v	Default
Hor wind bracinds			

Now all the parameters defined for Steel Columns are defined also for the layer Steel Beams. An alternative method in order 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 in order to set the parameters.



Param	neters	
	GENERAL	
	TENSION	
	SHEAR	
	TORSION	
	COMPRESSION	
	BENDING	
	BENDING & AXIAL FOR	Ε
	BENDING & SHEAR FOR	CE CONTRACTOR OF CONTRACTOR
	BENDING SHEAR AXIAL F	RCE
	Default	
Press	the button "GENERAL" eneral Parameters afety Factors yM0 1 yM1 1 yM2 1.25 it of Internal 0.01 OK Cancel values are recommende	GENERAL to set the γ _{Mi} safety factors: Y _{M0} : partial factor for cross-sections' resistance whatever the class is Y _{M1} : partial factor for members' resistance to buckling based on tests Y _{M2} : partial factor for resistance of cross-sections in tension to fracture In the "Limit of Internal" field define an upper limit. Below this value, the program will not consider the corresponding stress resultants. d by Eurocode.
Press		TENSION



Holes		\bigcirc I	Safety Facto	or 1		ОК
	U Web and I lange	Ú.			Ca	ancel
olt Holes Geometry		Flango				
Holes Diameter (mm)		Holes Dia	meter (mm)			
Number of Bolt Rows (perp. to the force pic.1)	0	Number of force pic.1	Bolt Rows (perp. 1)	to the 0		
		e2 p2 e1	Pt	e2 p2 e	T ^I P1	
	e2 p2 e1 p1	e2 p2 e1		e21 p2		0
e2 p2 e1p1		e21 p2 e ₁				
Distan	ce between the holes (mm)			Distance betw	een the holes (mm)
el pl	e2 p2	el	p1	e2	p2	
0 0 0	0	0	0	0	0	
Section L						
Hole	es Diameter (mm)	Number of Bol	t Rows (parallel to p1(mm)	the force) 0	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 in order to define if the elements of the selected layer contain stiffeners and if so which type; web stiffeners or intermediate stiffeners. Also define the spacing between the stiffeners and the type of the connection (rigid or not rigid).



Shear Parameters ×	
Safety Factor 1	
Stiffeners	
No In Support Between	
Dist. between Stiffeners (cm)	
Support	
Rigid	
O Non Rigid	
OK Cancel	
	l
	TORSION
elements of the selected laver are loaded	by a distributed or concentrated torsional moment. or
not. In the following dialog box, you ma	y as well define the support conditions based on the
corresponding figures. Select the type of r	noment and set (i) the relative distances from the start
fields. Also, set the support condition by t	yping in the "Type" field the values 0, 1, 2 or 3.



Torsion Parame	eters ×	
Safety Factor Torsional Moment No Distri	1 buted	
Distance from Start (cm) Distance from End (cm) Value (KNm) Element's Length (cm) Support Conditions University of the second seco	0 0 0 300	
	FORCE	For all design cher the safety factor i one of the five b resistance value v set 1.0 by default.
BENDING SHEAR AX	IAL FORCE	Pa Safety Factor OK

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.

	Parameters	×
Safety Factor		1
ОК		Cancel



Buckling Members Input

1.4 Merge Elements

Ruckling Manhard Input

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.

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

Bucking Members Input)
✓ Lateral Buckling	
Direction Y Member's Length	Direction Z Member's Length
○ Real● Coefficient	○ 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.

In addition, in the presentation of the results, for these merged elements, the most unfavorable results are displayed once and not for each individual 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:

Sa 🖓	uto			
Automati	c Merge		6	\times
Layer	Steel Columns			\sim
Columns	;	~	Calculation	
Discont	inuity Levels			
				_
1	lew	View	Delete	
Pid	< // XY	Pick // ZY	Pick	
		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.
- **A** 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)

Automatic	Merge		×
Layer	Steel Column	s	~
Columns		~	Calculation
Discontir	nuity Levels		
1 300	.00		
Ne	ew	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 own **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**, <u>editing</u> 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
Direction Y
Direction Z
Deformations
Direction Y
Direction Z
Columns \sim
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, in order 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:

😴 View 🔓	
Merge 🛛 🗙	
Buckling 😡 🖓	
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 in order 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:

Correction	2
Merge	Х
Buckling	
Direction Y	
Direction Z	
Deformations	
Direction Y	
Direction Z	
Columns	\sim
View	
ОК	
Cancel	



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 in order 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.



2. 9	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.
🗾 Ci	ross-Sectio	n Design		
Л в	uckling Me	mbers Inp	ut	
	old Formed	Sections		
 co	onnections			
-	EA StatiCa	Connectio	on	Always remember to calculate the corresponding load
				combinations in the parameters dialog box.

2.1.1 Cross Section Design

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

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						
<						>

Select this command to open the following dialog box:



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	

Steel Columns Steel Beams Surface Mesh	IPE 450			01000 00000	
Steel Beams Surface Mesh	IPE 330				
Surface Mesh	II E 330				
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				
<					>



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

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

					S	teel Des	sign - La	ayer Da	ata	x		
Layer: Ma	in Beams		VERIFI	CATION OK					Capacity Design Amplification			
Different Cr	nt Cross IPE 330 V							\checkmark	CHECK SELECTION			
Description	Memb	Comb.	N	Vy	Vz	Mx	Му	Mz	NO Auto N M V Mx M-N M-V M-V-	-N		
Max N	161	1	27.30	0.26	-3.38	-0.00	-4.35	0.91				
Min N	152	1	-13.76	-0.18	-2.74	-0.00	-4.74	-0.49				
Max QY	157	37	-1.19	0.44	-1.34	0.00	-1.11	1.50				
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				
Min QZ	161	1	27.30	0.26	-3.38	-0.00	-4.35	0.91				
Max MX	153	5	-6.29	0.15	-1.30	0.00	-1.02	0.61				
Min MX	152	7	-6.29	0.18	1.34	-0.00	-1.02	0.61				
Max MY	159	1	12.82	0.01	0.00	0.00	7.02	-0.02				
Min MY	161	96	11.93	0.05	-2.83	-0.00	-5.40	0.17				
Max MZ	157	7	-0.69	0.44	-1.35	0.00	-1.15	1.51				
Min MZ	157	64	-3.10	0.09	1.94	0.00	-3.38	-1.48				
User			0	0	0	0	0	0				
	For a	all memb	ers that b	elong to thi	is GROUP							
ОК		Can	cel		Layer De	esign		La	ayer Explorer Calculation Printout			

					S	teel Des	sign - La	ayer Da	ata	×			
Layer: Pur	lins	,	VERIFICAT	ТОМ МОТ С	ж				Capacity Design Amplification				
Different Cr	oss	IPE	100					~	CHECK SELECTION				
Description	Memb	Comb.	N	Vy	Vz	Mx	Му	Mz	NO Auto N 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					
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					
	For all members that belong to this GROUP												
									· · ·				
ОК		Can	cel		Layer De	esign		La	ayer Explorer Calculation Printout				



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 own check process. Select the combination of the design checks by clicking on the corresponding button from "CHECK

SELECTION" N M V Mx M-N M-V

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 own 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 6 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 -Max Mz ... and the relative values for N, Mx, Mz, 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, Qx -Min 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.



IMPORTANT NOTES:

- 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.
- 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		Capacity D					
Different Cr	oss	I	PE 100					\sim		
Description	Membe	Com	D. N	Vy	Vz	Mx	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		
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		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		

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 Cr	OSS	IPE	100					~	
Description	Memb	Comb.	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	V
	For a	ll memb	ers that b	elong to th	is GROUP				V
	For a	ll memb	ers that b	elong to th	is GROUP				, . , .

- 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:

					St	teel Des	sign - La	ayer Da	ta								×
Layer: Pur	lins		VERIFICAT	тон нот с	Ж				✓ C	apacity D	esign	Ampl	lificatio	n			
Different Cr	OSS	IP	E 100					\sim				CHE	CK SEL	ECTIO	N		
Description	Membe	Comb.	Ν	Vy	Vz	Mx	Му	Mz	NO	Auto	N	M	v	Mx	M-N	м-v	M-V-N
Max N	218	1	4.07	1.25	1.86	-0.00	-2.31	-1.44			V	V		\Box	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		
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	×			V		
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			V	×			V		
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			×	×			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	V			V		
For all members that belong to this GROUP																	
ОК	OK Cancel Layer Design Layer Explorer Calculation Printout																



- 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 own values of the intensive forces in order to check the sections. In the next dialog box:

Layer: Mai Different Cro	in Beam: oss	s	VERIFIC	CATION OK				~	Ca
Description	Memb	Comb.	Ν	۷у	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	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	V
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 members that belong to this 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					~
Description	Memb	Cor	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



NUM

Follow the same procedures described above in order to design manually the other sections or to see the results: Layer Design Layer Explorer Calculation Printout In table - Analytically - As Printout _ × D019 000 - WordPad Steel Design - Layer Data File Edit View Insert Format Help Layer: Purlins Capacity Design Amplificat D 🛋 🖉 🗛 🔏 📾 📾 💁 Different Cross IPE 100 CHECK SELECTIO STEEL CROSS SECTIONS DESIGN Vz Mx My Mz Comb. N Vy M V Mx M-N M-V-N -2.31 -2.49 -2.54 -2.54 218 187 181 173 197 221 221 197 167 196 179 181 Layer : Purling Section : IFE 100 CROSS SECTION GENERAL PROPERTIES TYPE OF CROSS SECTION I h = 10.00 (cm) d = 7.46 (cm) b = 5.50 (cm) r = 0.01 (cm) Max N Min N 4.07 1.25 1.86 -0.00 -1.44 -1.61 1.61 -2.08 -0.00 -1.82 -3.62 Max QY Min QY 0.05 -1.90 -1.84 0.05 -1.61 -0.00 -0.00 -0.00 0.00 -0.00 -0.00 -0.00 0.00 -2.60 -2.60 -2.60 -0.00 1.93 Max QZ Min QZ Max MX Min MX -1.50 2.18 -2.18 -2.18 -1.41 -1.42 -1.42 -1.42 -1.41 0.80 0.06 1.25 0.06 0.06 0.06 0.06 0.01 -1.25 -1.25 -0.33 0.00
2.17
0.07 Max MY Min MY -0.03 1.25 -2.61 0.92 -1.42 Max MZ 0.55 -0.00 Min MZ DT019 000 - WordPac File Edit View Insert Format Help 3) : S235 fy=235.00 MPa fu=360.00 MPa OK Cano STEEL CROSS SECTIONS DESIGN middle of the web) = 2.54 (cm) mid.of the up.flan.) = 4.71 (cm) (cm) middle of the web) = 2.54 (cm) mid.of the end edge) = 4.71 (cm) center weight) center weight) TIA (11) 0 Sz=0.0000000 = 0.00 (cm) = 0.00 (cm) ********STEEL CROSS SECTIONS DESIGN - CROSS SECTIONS RESISTANCE****** 0 Sz=0.0000000 5 Sz=2.1553125 5 Sz=2.1553125 0 Sz=0.0000000 5 Sz=0.0000000 25 Sz=0.0000000 0 Sz=0.0000000 BENDING, SHEAR 4 AXIAL VERIF. : Member: 218 Combinet. : 1 Bending-Shear-Axial Verification (prEN 1999-1-1: 2004(E) 6.2.10) Section class. in Bending = 1 (web = 1, flanges = 1) Design value of Assistance Vp12Ad = 45.908 Design value of Assistance Vp12Ad = 45.908 Internal force : I (BM) My(IMb) Mi(IMb) Qy(IM) Qz(IM) Internal force : I (BM) My(IMb) Mi(IMb) Qy(IM) Qz(IM) Internal for : 0.50 (<-1.00 Hz) Mi(IMb) Qy(IM) (2005) Batio : 0.50 (<-1.00) Interaction formula criterion = 0.06 <- 1.00 Satisfied Interaction formula criterion = 0.06 <- 1.00 Satisfied Interaction formula criterion = 0.03 <- 1.00 Satisfied Interaction formula cr 0 Sz=0.0000000 25 Sz=-2.1553125 25 Sz=-2.1553125 0 Sz=0.0000000
 Number
 0.03 <= 1.00 Satiafied</th>

 BENDIND, SHEAR & AXIAL VERIF.
 : Member: 107 Combinat. : 1

 Bending-Shear-Akail Verification (prEN 1993-1:12 2004(E) 6.2.10)

 Section class. in Bending = 1 (web = 1, flanges = 1)

 Design value of Resistance VpIR4 = 66.9908

 Design value of Resistance VpIR4 = 85.0697

 N(KN) My(KNM) Mir(KNM) QY(KN) QZ(KN)

 Internal force :
 3.62 - 2.49 - 1.62 - 1.61 - 2.08

 Design resistance :
 2.42.60 - 9.26 2.15 - 85.07 - 68.99

 Ratio :
 1.10 (>1.00)
 BENDING, SHEAR & AXIAL VERIF. : Member: 181 Combinat. : 1 Bending-Shear-Axial Verification (prEN 1993-1-1: 2004(E) 6.2.10) Section class. in Bending = 1 (web = 1, flanges = 1) Design value of Resistance VpIrAd = 68.0697 Design value of Resistance VpIrAd = 5.0697 For Help, press F1 NUM Capacity Design Amplification Layer Design 8. Activate and press if you want to apply the capacity design in your checks. 38



2.1.2 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:

esign		~	×				
		5					
Main Beams ~							
129 HEA 180 V Parameters							
∆окоі							
Apply to all members of the Layer							
ng with Min, Max	of all co	mbinatio	ons				
Layer							
Exploration of Member Buckling							
Exploration of Member Servicability							
Member Results Layer Results							
OK		Cancel					
	esign Main Beams 129 HEA 180 Δοκοί pply to all membing with Min, Max Layer ration of Member er Results OK	esign Main Beams 129 HEA 180 ν Δοκοί pply to all members of th ng with Min, Max of all co Layer ration of Member Bucklin tion of Member Servicab ter Results La	esign Main Beams 129 HEA 180 V Param Δοκοί pply to all members of the Layer ng with Min, Max of all combination Layer rration of Member Buckling tion of Member Servicability ter Results Layer Res OK 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	Desi	gn ×
Layer	Steel Columns		~
Member	1 IPE 450	~	Parameters
Group	1 IPE 450 3 IPE 450 4 IPE 450	^	~
ļ	5 IPE 450 6 IPE 450		ne Layer
Check in	7 IPE 450 9 IPE 450		
Exp	11 IPE 450 12 IPE 450		9
Explo	13 IPE 450		ility
Buckling F	16 IPE 450		ricability Results
	18 IPE 450 19 IPE 450 21 IPE 450		Cancel



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 1 Surface Elements Mesh 3D Slabs-Strips Steel Columns Steel Beams Main Beams Purlins Girdere	× <	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, in order to set different parameters for

 \sim Otherwise, in order 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.

Group Name Columnsi New Group Cree Safety Factor 1 Limit of Internal 0.1 ✓ Lateral Buckling Direction Z Member's Length Real 1 Ø Coefficient Ø Coefficient Ø Coefficient Ø Coefficient 0 Buckling Lengths I Ø Coefficient I 0 Ø Coefficient I Ø Coefficient I I Ø Coefficient I I I I I Ø Coefficient I Ø I I I I Ø Flexural Buckling I <th>Member Design</th>	Member Design
Safety Factor 1 Limit of Internal 0.1 Image: Coefficient Image: Coefficient Image: Coefficient Image: Coefficient Buckling Lengths Image: Coefficient Image: Coefficient Image: Coefficient Buckling Lengths Image: Coefficient Image: Coefficient Image: Coefficient Image: Coefficient Image: Coefficient Image	Imnsi New Group Creation
Flexural Buckling Ends Constraint Member Loading y Member Loading y	1 Limit of Internal 0.1
Loading Level X 150 Z 150	z Member Deformation Limits Y 200 z 200 Node Displacement Limits X 150 z 150

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 own 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.

Λ ΝΟΤΕ

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 1 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.



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 4 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"
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.



intermediate

fixed support

ОК

Cancel



corresponding reduced buckling lengths.

The second group



includes cases of members in multi-storey steel structures and allows setting the concurrent to the node members.

(the most complex case) the user sets for the vertical member the 6 Members By choosing (2 vertical and 4 horizontal) that offers succor to it (3 on the top and 3 at the end).

By selecting the icon the following dialog box is displayed:



Frame Members Parameters							
Jo	oint Membe	ers		IH	*	Type of Load	
top Column	52	IPE 450	1.63	• •			
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 \bigcirc	$\odot \bigcirc \bigcirc$	Direct Loads	
B lower right	0			• •	$\odot \bigcirc \bigcirc$	Concrete Slabs 🗸	
Lower Column	0			• •			
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 own 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 The "End Constraints" window, containing the various types of	constraints opens.
	Lateral Buckling ×
Press one of the first four buttons in order to automatically calculate the flexural buckling factor:	
	1
	intermediate
The last one gives the user the opportunity to consider	Length(m) Coef.
different constrains along the same member.	
	3 0.00 1.00
	4 0.00 1.00
	5 0.00 1.00
	6 0.00 1.00 +
	7 0.00 1.00 ОК
	8 0.00 1.00 Cancel
	Member Loadi ×
Member Loading	
The next parameter refers to the	e load
type of the member at the local axis y, and z respectivel	у. Ву
selecting the corresponding icon, the following options appear	
In which you choose the type of Member Loading.	
Finally, the last parameter/concerns the	
determination of the	Loading level of the member. The
Loading Level × following five option	is are displayed by selecting the
icon.	. , ,
Loading levels for ea	ch icon:
1 st icon: on the uppe	r flange of the element
$2^{n\alpha}$ 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 								
Member Deformation Limits								
Y 200 Z 200								
 ✓ Node Displacement Limits X 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 V Parameters							
Group	Στύλοι 🗸							
Apply to all members of the Layer								
Checkir	ng with Min, Ma	x of all combinations						
Check in	Check in Layer							
Exploration of Member Buckling								
Exploration of Member Servicability								
Memb	er Results	Layer Results						
	ОК	Cancel						

In order 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 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".

Men	nber Design ×
Group Name Columns_1 Safety Factor 1	New Group Creation Limit of 0.1
Lateral Buckling Direction Y Member's Length Real Coefficient Buckling Lengths	Direction Z Member's Length Real Coefficient Buckling Lengths
Flexural Buckling Ends Constraint Member Loading Loading Level	Serviceability Check Member Deformation Limits Y 200 z 200 Node Displacement Limits X 150 z 150
Lateral Torsional Buckling	OK Cancel

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		~

Exploration of Member Buckling Exploration of Member Servicability

Layer Results

Cancel

Member Results

ОК



right gives the

Member	1 IPE 450	~	Group	Columns_1	~
As soon informat	1 IPE 450 3 IPE 450 4 IPE 450 5 IPE 450 6 IPE 450 7 IPE 450 9 IPE 450 10 IPE 450 11 IPE 450 12 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 24 IPE 450 24 IPE 450 as the laye ion:	r check is	comple	Columns Beams Columns_1	he window on the
Member D	esign		Х	Red: if there is a fa	ailure
Layer	Steel Columns		~	Green: if there is r	no failure.
Member	21 IPE 450	✓ Para	meters		
Group	Στύλοι		\sim		
A	pply to all member	rs of the Laye	r		
Checki	ng with Min, Max	of all combinat	tions		
Check in	Layer				

By double clicking on the colored window, the dialog box containing members check summary results opens:

				Men	nber Ch	eck Resul
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		



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 for all combinations.

For each group of (N, My and Mz) the checks are made 4 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

1 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 X								
Layer	yer Steel Columns ~							
Member 🤇	21 IPE 450 V Parameters							
Group	Group Στύλοι 🗸							
A	pply to all memb	ers of th	e Layer					
🗹 Checkir	ng with Min, Max	of all co	ombinations					
Check in	Layer M	em:54/5	4 Comb:4/4					
Explo	ration of Memb	er Buckli	ng					
Explora	tion of Member	Servical	bility					
Memb	Member Results Layer Results							
OK 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.



2.1.3 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>.

		÷	
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 Connectio	on

Member Design X						
Layer	Purlins		~			
Member	103 ELASTRON	103 ELASTRON Z 🗸 Paramete				
Group Докоі V						
Apply to all members of the Layer						
Check	ing with Min, Max	of all co	mbinations			
Check in	n Layer					
Exploration of Member Buckling						
Exploration of Member Servicability						
Member Results Layer Results						

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

- 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 B	Beam to Column (Flange) Г	Beam to Column (Web) Γ	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 4 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 own 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 N	1embers					×
			N(kN)	N	/(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: 116,3	30,			1	1	Add
60: 118,6 40: 117,4	50. 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	olumn Connection (Secondary Axis of	Column)	×
Type Type	Beam - Column (Г)	End-plate in bending verification is ok Column web panel in shear verification is ok Column flange in bending verification is ok Beam web in tension verification is ok Beam flange in compression verification is ok	<u> </u>
Στυλος	Δοκος	Total design connection verification in bending is ok Column web in bending verification is ok Fillet weld is required Wields verification end-olate - right beam is ok Calculation (Couldations) Calculations (Combinations) User) Simplified Exploration Sa	ve
End-plate (Extend h 150	(mm) ded Materic S235 💌 b 100 t 12 hup 20		1 2 3
Weld Thick Bolts	kness 6 Angle 90		
Rows e1 (mm)	Materij 4.6 Parameters 4 Image: Same Distances ? 10 e2 (mm) 20 ?		<u> Σ/</u> Κ
Angles (L) Gap (g) Cross Sect	10 Bolts (mm) General Parameters M12 Y Materi 4.6 Y		
100x10	Column:2 Beam : Lini P e1 10 p1 20 e2 ev1 pv1 20 10		
Welded (m	m) Welded S235 V G STIFFEIVERS		3D

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.



pe	Beam - Column (Г)	-	Node	Bolts	Weld	Section	End-Plate/	Max
	y and place		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)
ίλος hπλ	Δοκος	2	Calculation (Combinations)	Calculations (User)	Simplifie	ed Ex	ploration	Save Exit
nd-plate (i Extend	mm) led Materiz S235 b 200 t 12 hup	20		T				
Veld Thick olts	ness 6 Angle	90 eral						-
ows 4	Same Distances e2 (mm) 20	?				11		
ngles (L) - ap (g) ross Secti	10 Bolts (mm) General Parame	ters						
EQ 100×10	Column:2 Beam :	l.6 ▼				7		
b [8	e1 10 p1 20 ev1 10 pv1 20	10			r .			
/elded (mr	n) Welded S2	35 💌						

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:





A	nd in the o	dialog box			
	🔺 in File	s of Type	select SCADA Conn	ection	
	🔺 press	Find			
	Open		N		×
	Look in:	1.dwg	₩ ₩	G 🌶 🖻 🛄 -	
	Quick access	Name	^ No items match you	Date modified r search.	Туре
	Desktop				
	Libraries				
	This PC				
	Network	<			>
		File name:		<u> </u>	Open
		Files of type:	Scada connection(*.con)	~	Cancel
	Συντελ	εστής 1.0	Οροφος 1 Find	\mathbf{k}	

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.







