

# User's Manual 6.LOADS







# CONTENTS

I. TH	E NEW UPGRADED INTERFAFE OF SCADA PRO	3
II. DE	TAILED DESCRIPTION OF THE NEW INTERFACE	4
	<i>c</i>	4
1	Ωεξινιτίον	4
1.1	LOAD CASES	4
1.2	LOAD GROUPS	6
2.	SLAB LOADS	7
2.1	INSERT	7
2.1.1	Overall	7
2.1.2	Ву ріск	9
2.1.3	Edit	10
2.1.4	YIELD LINES	11
2.1.5	SLAB REACTIONS	11
2.1.6	Overall:	12
2.1.7	Вү Ріск:	12
2.1.8	EQUIVALENT	12
3.	MEMBER LOAD	12
3.1	INSERT	12
2.2		/ L
2.2.1		/ 1
2.2.2		19 20
2.2.3		20 23
2.2.4		23
2.2.5		
4	WIND AND SNOW LOADS	
4.1	Parameters	
4.1.1	CODE	33
4.1.2	WIND	33
4.1.3	SNOW :	38
4.2	Edit	38
4.2.1	Walls :	38
4.2.2	ROOFS	41
4.3	VIEW	42
4.3.1	WIND	42
4.3.2	SNOW	43
4.4	Member Correspondence	43
4.5	Post-Processor	47





#### THE NEW UPGRADED INTERFAFE OF SCADA Pro Ι.



# II. DETAILED DESCRIPTION OF THE NEW INTERFACE

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

Basic	Modeling	g Vie	w T	ools	Slabs I	oads.	Analysi	is I	Post-Proc	essor	Member	rs Design	Drawing	s-Detailin	g Addons	Optimization	n
									LOA	DS							
		_	~														
		🗢 🧶 I	2-7	700.00	* 14		) =										
	Basic	Model	ina	View	Tools	Slabe	Loos	de la	Applyrig	Po	st Proces	sor Mor	abore De	ncian	Drawings Dr	tailing Add	long
	Dasic	Woue	ing	view	TOOIS	Sidus	LUau	15 1	Analysis	FU	st-Froces	sor men	inders De	sign	Drawings-De	alling Aud	IONS
	IIC	ath	ath	-	-		,			CH.	(Dort	011		011	_	_0	
		-	10		- <b>*</b>		ш.	Ш.		ALC: NO			- <b>Q</b> -		· <b></b>	E.,	
Load	Load	Incort	Edit	Viold	Slab		Incort	Edit	View	Conv	Tools	Paramotore	Edit	View	Mombor	Post Proces	scor
Casas	Groups	The contract of the contract o	Luit	Lines	Deaction		maert	- Cuit	VIEW	сору				view -	Corresponder	1031-110063	5501
Cases	Groups		CI.	Lines	Reaction					a da				14/1	Corresponder	ice	
Defi	nition		Sla	ab Loads				Mer	mper Lo	ads				wind -	Show Loads		

The 6<sup>th</sup> Unit is called "Loads" and includes the following four groups of commands:

- 1. Definition
- 2. Slab Loads
- 3. Member Loads
- 4. Wind and Snow Loads
- 1. **Definition**



"Definition" command group allows the definition of the loads and their corresponding groups.

The basic condition for loads application is the definition of the respective load cases. Each load will belong to one of those cases.

#### **1.1 Load Cases**

# LIC

Load

Cases To define the Load cases use the "Load Cases" command. In the dialog box:



L	oad Cas	e Defini	tion	×
	Self-	weight	Dead Load V	Insert
	LC	s.w.	Description	)el From Lis <sup>i</sup>
	1	Yes	Dead Load	
	2	No	Live Load	Delete Loads
				Delete All Loads
				ОК

There are two default load cases:

- 1. Dead Loads (L.C.=1)
- 2. Live Loads (L.C.=2)

Δ

The S.W. column indicates the participation of the self-weight in the specific load case.

In addition to dead and live default loads, you can also define other loads. You can choose loads from the list, or you can set your load by typing a name and then click the "Insert" button.

The imported load, takes a serial number, in LC (Load Case) column and a "Yes" or "No" indication, depending on, whether or not, the self-weight is including.

L	oad Cas	e Defini	tion	×
	Self-	weight	Dead Load V	Insert
	LC	s.w.	Description	)el From List
	1	Yes	Dead Load	
	2	No	Live Load	Delete Loads
				Delete All Loads
				ОК



Press OK to save and close the window.

▲ To include the self-weight of the structure in a load case, check "Self Weight" selection ✓ Self Weight

• To delete a load case from the list, first select it and then press Pel From List The program will ask you to confirm your choice. Click "Yes", to delete.



A You can delete a load case only if it does not include loads.



Delete

In order too delete the loads from a case first select the load and then click
 Loads

• To delete all the loads from all cases click

# 1.2 Load Groups

# LG

Groups Use "Load Group" command to create load groups that belong to an existing load case as an optional procedure. Dead and Live Loads contain as default a predefined group "Group1".

- EXAMPLE:

In case you want to define a new group, for example, a SNOW group in Dead Load and a MAN group in Live Load,

select at first Dead Load from the list, then on "Description" type "SNOW", select a color (press on **F**) and click "Insert". So, SNOW Load becomes (LG) 2.

Do the same to define MAN Group in Live Load as (LG) 2, while Group1 refers to the standard live loads.

Load Gr	oup Definition	Х	Load Group Definition X
Load C Descrip LG 1	ase Dead Load bition SNOW Description Group 1 Inse	ert	Load Case Live Load  Description MAN LG Description 1 Group 1 Insert
2	SNOW Dele Loa Ol	ete ete ids ≺	2 MAN Delete Loads OK

- Press "Delete" to delete an existing group. Prerequisite for deleting a group is that it does not include loads.
- To delete the loads included in a group (loads already entered), choose from the list of the

Delete	
Loads	

groups this group (to turn blue) and then press 📖



To display loads of each group of the same load case use the "View" command View



# 2. Slab Loads

Edit

Insert

"Slab Loads" command group contains the commands for inserting, editing and assigning the slab loads on members and nodes.

#### 2.1 Insert

The command contains two kinds of loads input:

Lines \* Reactions \*

Slab



Yield

Slab Loads

#### 2.1.1 Overall

Overall: Select the command and in the dialog box:

Unif	ormly	Distribut	ted Loads	×			
Load Case	Load Case Dead Load						
Group	Group	1		۷			
Loads				_			
0	G	eneral	Predefined				
Slab Type		Solid	Zoellner				
Cantilev	er	0	0				
Two Way In	nclined	0	0				
Two Way	Slab	0	0				
Three Way	Slab	0	0				
Four W	ay	0	0				
Triangu	lar	0	0				
	¥	Insert	Add to List				
Apply		Delete					
Replaceme	ent		Exit				

First, define the Load Case and Group and then type the load value for each type of slab.

Loads

а

• You can proceed by typing in

value in kN/m<sup>2</sup> and then press General to fill in all fields with the same value.

• You can also use predefined values by clicking on the Predefined button that opens the next dialog box:



Import from	ALTESIT AND COMPACTED 1cm AND DRY 1cm AND DRY 1cm					
Description	ARBLE CE 1cm					
Load (kN/m2) 0 Add to Library	CHIST EMENT 1cm					
Height (m) 0 OK	EMENT BULK 1cm					
Final Load (kN/m) 0 Cancel	ITUMEN 1cm					
	LANTED ROOF DRY 1cm LANTED ROOF WET 1cm					
Select from the "Import from" list the predefined load to define loads directly, or define your load, by typing a "Description", a value "Load (kN/m <sup>2</sup> )" and in case you want to save the load in the library click the button Add to Library.	LASTER 1cm LASTERBOARD 1cm LAY DRY 1cm OLYSTYRENE 1cm OODEN OOF TILES MARSEILLE OOF TILES TRADITIONAL ORTAR 1cm OSAIC					
Define the loads and press Insert						
The load case and group are displayed in the list (Lc=1: Lo 1) automatically.	oad Case 1/Lg=1: Group					
Follow the same procedure for the other load cases (ex. Live Load) Lc=2/Lg=1 to display the Load case and group (Lc=2: Load Case 2/Lg=1)	and press <mark>Insert</mark> , .: Group 1).					
By selecting Apply, all defined loads are applied to all current lev	el slabs.					
The loads' assignment for the first time means that the loads in the list applied to all current level slabs.	Lc=1/Lg=1 💽 will be					
But, in case of slabs containing existing loads, by clicking on the existing loads will be replaced.	Apply button, the					
EXAMPLE 1:						
Suppose that you have already assigned loads in all current level slabs with dead and live loads.						
- If you define a new value for the dead loads and press Apply, the program will apply						
the new dead load value and 0 live loads (the list contains only dead load load no live load).						
- But, if you want to replace dead loads and keep the existing live loads, the	en press Replacement					
- Press Delete to cancel an inserted load from the list.						





#### EXAMPLE 2:

Suppose that a dead load of 1  $kN/m^2$  is already applied on a slab and you want to add another dead load 2  $KN/m^2.$ 

Define the load and press Add to List							
I	Туре	LC	LG	Value	Status		
I	Uniform	1	1	1.00			
I	Uniform	1	1	2.00			





# Select Edit command to confirm.

# EXAMPLE 3

You can also replace only the value of a specific Slab Type. Type the value in the corresponding

snace	Four Way	0	0	and then click the ty	Four W	<b>y</b> This
space				and then ellek the ty	pc	
value v	vill replace the firs	st one to all slat	os with th	ne same type.		
Press [	Exit to clo	ose the dialog l	box with	out saving, or press	Apply	to save the
change	25.					

#### 2.1.2 By pick

select the command and then left click inside a slab. In the dialog box:

Insert Slab Loads							
Load Case	Dead Load		•				
Load Group	Group 1		•				
Load Type	Uniform 💌	Predefined					
Load(kN/m2)	0	Load					
Select Cancel							

Define "Load Case", "Load Group", and type the value in KN/m2. Then select "Load Type".

There are 3 "Load Types":

#### Uniform

Insert uniform loads over the entire surface of the slab. Define the load and left click inside the slab.

• Partial



Insert partial load in a specific area of the slab. Define the load and left click inside the slab. Then select a side to identify the direction and then left click to indicate a vertex and movthe e mouse to describe the load area.

#### ٠ Linear

Insert linear load over the slab and follow the same procedure as described in partial loads. To define the position of the load, left click to identify the two ends of the line (start and end point).

Partial and Linear loads, regarding the attribution to the slab members, will be replaced by an equivalent uniform load on the entire slab

Define "Load" in  $kN/m^2$ . 

> Predefined Load

You can also use predefined values using button, as previous.

Select to close the dialog box and click inside one or more slabs' area to apply the Press load.

#### 2.1.3 Edit

To edit and modify slab loads use/command.

Select the command and click inside a slab. In the dialog box:

E	dit Slab	Loads Slai ie []	о П1 - So Dead Loa	lid - Two-Wa d	y T	<ul> <li>Select Load Case and Group, then from the list select the load for editing.</li> <li>I</li> </ul>
	Value	0				
	Туре	LC	LG	Value	Status	
	Uniform	2	1	2.00		
		ar List Apply	Clear	by Select	Undo ät	Activate Clear List and all the loads will be deleted from the list. Otherwise, press Clear by Select to delete only the selected load from the list.
1	Loads means	are no s "read	t delet ly to de	ed immed elete".	liately, but	first "Delete" is displayed on "Status" column, which
1	уре	LC	LG	Value	Status	
l	Iniform	2	1	2.00	Delete	



command invalidates the previous action (cancels "Delete" designation in Status column)

Press Exit to close the dialog box without saving, or Apply to save the changes.

#### 2.1.4 Yield Lines

Load areas' calculation resulting from the geometric partitioning of the slab, and then used to calculate the design forces for beams (slab loads which will be imposed on beams), The calculation is automatically made by the program according to the support conditions, either Overall or By Pick.





Overall: select the command (Load areas calculation for all current level slabs) By Pick: select the command and then left click inside one or more slabs (Load areas' calculation for the selected slabs)

- Yield Lines calculation concerns rectangular slabs or slabs with equivalent rectangular modeling.
- 1 Yield Lines calculation is made according to the support conditions
- **A** Yield Lines calculation is applied to conventional slabs and <u>not to slabs with surface</u> <u>elements.</u>

#### 2.1.5 Slab Reactions

By selecting Slab Reactions, slab loads are assigned on beam members as reactions. More specifically, loads are distributed from slabs to beams and columns, based on the geometric partitioning done previously (Yield lines).





#### 2.1.6 Overall:



select the command (Load distribution from all the current level slabs).

#### 2.1.7 By Pick:

select the command and then left click inside one or more slabs (Load distribution from the selected slabs)

#### 2.1.8 Equivalent

means: to assign (Overall for By Pick for By Pick respectively) the slab loads on the connected members, without considering the yield lines evaluation (rectangular and triangular areas). Instead, the assignment is implemented by the convertion of the entire area corresponding to the member, in an equivalent rectangle.

▲ Slabs' load distribution as a reaction of the connected members, for slabs without any required simulation, is implemented without the generation of equivalent models (uniform, triangular, trapezoidal loads, etc.).

▲ In slabs identified by mathematical members entered by the user (e.g. beams with high rigidity for the simulation of basement walls), loads are assigned to the mathematical members, regardless of whether linear members (either selecting the mathematical member or the physical cross section) are chosen for the modeling and matching of the sides of the slab.

#### 3. Member Load



"Member Loads" commands' group contains the commands for inserting, editing, viewing and copying loads of members, nodes and surfaces finite elements.

#### 3.1 Insert

#### щ

loads on members, nodes and surface finite elements.



Insert Loads			×
Load Case	Dead Load	Group Group 1	<b></b>
Load Property	Land Mad		
Element	Uniformely Distributed Loads		
Description		dis.i	⊷ ⊸dis.J⊶ ́
Value i (kN/m)	0 Value j (kN/m)	-    `	
Dist.i (cm)	0 Dist.j (cm) 0	—    _	X
Angle			
Apply To	Local xy   Predefin Local xy		X
	Description		
	Description		
			Clear by Select
The lists on t	on contain all the nessik	la laad tupos d	anonding on the element type
	op, contain an the possib	ne load types u	epending on the element type
Load Case	ead Load 💌	Group Group 1	
"Load Prope	rty":		
⊢ Load Property			
Load Type	Load Kind		
Element 💌	Uniformely Distributed Loads	<b>_</b>	
Description			
Value i (kN/m)	0 Value j (kN/m	0	
Dist i (cm)	Dist i (cm)	0	
Dist.i (Cili)	Dist.j (Sill)		
Angle	0	Predefined	
Apply To	Local xy	Load	
	Elem	ent	
Select the "T	ype" of the element	and the	"Type" of the load
Uniformely Distrib	uted Loads		
Uniformely Distrib Torsional Momen	uted Loads		
Trapezoid Forces			
Concentrated For	ces		
Concentrated For Transverse tempo Slab Reactions	ces erature		

According to the element "Type" and the load "Type", the "Load Property" is modified. Fill in the fields according to the drawing, type a description, the values, and the corresponding distances.



#### **Member Load Sign**:

Loads' sign convention is made versus the local coordinate system of each member, which is based on the rule of the "right hand".

#### Specifically:

#### 1. BEAMS :

x-x is the local axis directed from the start to the end point (red vectors),

**y-y** is the vertical axis (perpendicular to the local axis x-x) parallel to the height of the slab (green vector). It is always directed like Y Absolut axis (bottom-up).

**z-z** is the third vertical axis, perpendicular to the plane defined by the xx and yy local axes (blue vector).



#### 2. COLUMNS :

**x-x** is the local axis directed from the start to the end point meaning bottom-up direction (red vectors),

**y-y** is the vertical axis (perpendicular to the local x-x) directed like X Absolut axis (green vector).

**z-z** is the third vertical axis, perpendicular to the plane defined by the xx and yy local axis (blue vector).



node (j) X (+)

A Beams and columns local axes can also be defined using the rule of the right hand with your thumb along the positive axis xx, the index finger along the positive yy and the middle one along the positive zz.

#### Nodes Load Sign :

Nodes' loads are always directed according to the Absolute X, Y, Z axes.





The next dialog box section is the load list.

LC	LG	Description	Insert
			Clear List
			Clear by Select
			ОК
•		Þ	Cancel

The list is filled in by defining the loads and selecting the "Insert" command.

LC	LG	Description	Insert
1	1	U.D.F. Wall 2.00/2.00/0.00/0.00/0.00	Clear List
			Clear by Select
			ОК
•			Cancel



Insert a uniform distributed load (U.D.F. Uniformly Distributed Force) that belongs to Load Case (LC) 1 (Dead Loads) and Load Group (LG) 1. The numbers after the description (Wall) are: start load value, end load value, the distance of the load from the beginning, the distance of load from the end and the angle.

By activating/, all the loads on the list will be deleted. Otherwise, press	Clear by Select	to
delete only the selected load from the list.		





Load Case	Dead Load 🗸 Group	Group 1
Load Property Load Type	Load Kind	i - (+) (-) -
Plate ~	Pressure ~	
Description	Pressure Uniform Temperature Variation Linear Temperature Variation	
Value (kN/m2)	0 Value j (kN/m) 0	x
Dist.i (cm)	0 Dist.j (cm) 0	
Angle	0 Dradafaad	Z X
Apply To	Local z v Load	I <sup>d</sup> y∕ y d k
LC LG De	scription	Insert
		Clear Li
		Clear by Sele
		OK

Allows you to define a **Pressure**, and also the possibility to enter **Temperature Variations** load for finite surface elements is added.

More specifically, for Plate (shell) elements **Uniform Temperature Variations** and **Linear Temperature Variations** loads are added.

- **Uniform Temperature Variations** causes membrane deformation in the plane of the element, while
- Linear Temperature Variations causes deflection.

#### NOTE:

- We have to note that the two loads of the plate element can be integrated either on the same loading or in two different loadings.
  - Integrating both loads at the same analysis scenario, you will get aggregated results in one load (the first).
  - Considering two different loadings, to obtain individual results, each load MUST go to a different analysis scenario.

The procedure to follow is:



oad C	ase De	finition	×			
Self	f-weigh	t Uniform Temperature Var 🗸 🗸	Insert			
10	<b>C</b> 111	Description	el From Lis			
1	S.W.	Description				
2	No	Live Load	Dalata			
3	No	Uniform Temperature Var	Loads			
4	No	Linear Temperature Var				
			Delete All Loads			
			ОК			
enario	)	X	Scenario			>
Renum	bering		Renumberin	ng		
Nodes	Cuth	ill-McKee(II) V Advanced Multi-Threaded Solver	Nodes	Cuthill-McKee(II)	~	Advanced Multi-Threaded Solver
Disabl	e	Name Un. Temp. Var	Disable		Name	Lin. Temp. Var
C8_Gen	ieral Stat	ic (0)	EC8_General	Static (0)	Analysis	Static
tatic Un	. Temp.	Var (2) Load Case participation	Static Un. Ter	np. Var (2)	Load Cas	e participation
tatic Lin	n. Temp.	Var (3) Static Un. Temp. VarStatic	Static Lin. Ter	mp. Var (3)	Ctatia Li	n Tomn VorStatia
		Scenario g(m/sec2)	9.8		Load Ca	n. Temp. varstatic
			=		Scenario	p g(m/sec2)
					1	A 10 161
		3 LC1 0.00			2	
		4 LC2 0.00			3	
					5	1.03 0.00
		7			6	
		8			7	
		9			8	
OTE: For pos	r <b>Plar</b> ssible	e elements (Stress, Strain, Axisy	mmetric) o	only <b>Unifor</b>	m Temj	perature Variat
2.2	2 Edi	t:				
Edit	for	editing the existing loads' propert	ies.			
2.2	2.1	Overall:				



ad Properties		2
Load Case	Dead Load Group	Group 1
Load Property Load Type	Load Kind	<i>₹</i> /77
Element 💌	Uniformely Distributed Loads	i o dia i dia i
Description		Y↑
Value i (kN/m)	0 Value į (kN/m)	y y
Dist.i (cm)	0 Dist.j (cm) 0	y (+)
Angle	0 Predefined	
Apply To	Local xy  Load	V Z Z
ld Status	Description	<b></b>
1	U.D.F. wall 2.00/2.00/0.00/0.00/0.00	Delete
2	S.R. П1(1) -11.69/-0.94/230.00/0.00/0.00	
4	S.R. П1(1) -0.93/-11.69/0.00/230.00/0.00	Clear by Select
8	S.R. П1(1) -1.40/-11.69/0.00/220.00/0.00	
10	S.R. П1(1) -11.69/-1.40/220.00/0.00/0.00	Apply
12	S.R. П1(1) -0.94/-11.69/0.00/230.00/0.00	

In the load list, you can see all the imported loads according to the selection. For example, select Dead Load/Group 1/Element/ Uniformly Distributed Loads. The list describes all the existing loads according to the selection. (U.D.F. Uniformly Distributed Force, S.R. Slab reactions)

When you choose a load, the values appear on the top of the window where you can modify them.

Press the Apply command to save the changes.

By activating Delete, all the loads on the list will be deleted. Otherwise, press

to delete only the selected loads.

Loads are not deleted immediately but	ld	Status	Description	^
first, the "Delete" indication is displayed	9	Delete	U.D.F. 1.00/1.00/0.00/0.00/0.00	
on "Status" column. which means "ready	11	Delete	S.R. П1(3) -0.32/-5.93/0.00/109.27/0.00	
to delete"	13	Delete	S.R. П1(3) -5.93/-1.01/215.73/0.00/0.00	
	16	Delete	S.R. П1(3) -1.01/-5.93/0.00/305.73/0.00	
To delete them permanently, press	18	Delete	S.R. П1(3) -5.93/-5.93/109.27/215.73/0.00	
Apply	<	D-I-t-	C D D1/0) E 00 / 0 00 /100 07 /0 00 /0 00	Ť
Press Exit to close the dialog box w	/ithc	out savi	ng, or Apply to save the changes	5.



#### 2.2.2 Pick

n P

for editing the load properties of the selected element. Select the command and left click on a member, node or surface finite element and the dialog box appears:

ad Properties		3
Load Case	Dead Load Group	Group 1
Load Property Load Type	Load Kind	<i>¥ 11</i> 7
Element 💌	Uniformely Distributed Loads	
Description		YA
Value i (kN/m)	0 Value j (kN/m)	
Dist.i (cm)	0 Dist.j (cm) 0	<b>y</b> (+)
Angle	0 Predefined	, i inj
Apply To	Local xy  Load	Vz - x
ld Status	Description	
1	U.D.F. wall 2.00/2.00/0.00/0.00/0.00	Dalata
2	S.R. П1(1) -0.94/-11.69/0.00/230.00/0.00	
4	S.R. П1(1) -11.69/-0.93/230.00/0.00/0.00	Clear by Select
		Apply
•		► Exit

In the load list, you can see all the imported loads of the selected element.

For example, select Dead Load/Group 1/Element/ Uniformly Distributed Loads. In the list, all the existing loads according to the selection will be descripted.

(U.D.F. Uniformly Distributed Force, S.R. Slab reactions)

When you choose a load the values appear on the top of the window where you can modify them.

Press Apply command to save the changes.

In the list, loads of the specific member are displayed. For example, the Uniform Distributed Force and the Slab Reactions of the selected member. By choosing a load the values appear on the top of the window where you can modify them. Then press "Apply" to save.

Loads are not deleted immediately but first, the "Delete" is displayed on "Status" column, that

Apply

	means '	"ready to	delete"	.For the	deletion	to be	applied,	press	
--	---------	-----------	---------	----------	----------	-------	----------	-------	--

Press Exit to close the dialog box without saving, or Apply to save the changes.



#### 2.2.3 View



for the display of the loads for all elements, in 3D view as vectors, with or without values, or in 2D view as a number.

Select the command and in the dialog box:

the defined Load Cases and the Load Groups contained are displayed. Each load group contains a switch ON or OFF (display or not display), that changes by clicking on it.

Disp	Display Loads													
Loa	Load Case Live Load Level XZ													
L	C	LG1	LG2	LG3	LG4	LG5	LG6	LG7	LG8	LG9	LG10		A/A	
L	C1	ON	ON										0	ON
L	C2	ON	ON										1	ON
I B-3d I Truss I B-3def I Node I Plate I Slabs														
Scale (1 Load Unit ) = 10 cm Display as Vector Value														
Filter : From 0 To 0 OK Cancel														

In the picture above there are two loads LC1 (Dead) and LC2 (Live). Each load contains a default group LG1 and a created group LG2, that are all "ON", which means that all loads will be displayed.

On "Levels XZ" select ON or OFF, to display or not, loads of the corresponding level.



🔽 B-3d Truss B-3def Node

The following options are related to the elements' loads that will be displayed.

Plate Slabs

Activate Value to display the value of the loads.

2



	Scale (1 Load Unit) = 10 cm to set the visualization scale of the vectors. Type in value in cm.	the
<b>Сору</b>	Display as Vector Value in 3D view select "Vector" and activate Value to dis the value of the loads. In 2D view select "Number" ("Value" activation doesn't change anyth Load values are visible only in 3D view).	play 1ing.
	Finally, using the filter From To you can specify a range of load be displayed.	ls to
	Also, by choosing isplayed inside the slabs, in the 2D display. Image: the values of the slab loads displayed inside the slabs, in the 2D display. Image: the value of the slab loads displayed inside the slabs, in the 2D display. Image: the value of the slab loads displayed inside the slabs, in the 2D display. Image: the value of the slab loads displayed inside the slabs, in the 2D display. Image: the value of the slab loads displayed inside the slabs, in the 2D display. Image: the value of the slab loads displayed inside the slabs, in the 2D displayed. Image: the value of the slab loads displayed inside the slabs, in the 2D displayed. Image: the value of the slab loads displayed inside the slabs. Image: the value of the slab loads displayed inside the slabs. Image: the value of the slab loads displayed inside the slabs. Image: the value of the slab loads displayed inside the slabs. Image: the value of the slab loads displayed inside the slabs. Image: the value of the slabs of the slab loads displayed inside the slabs. Image: the value of the slabs. Image: the value of the value of the slabs. Image: the value of the value	are
	■B-3d Truss ■B-3doef ■Node ■Plate ■Slabs	
	Scale (1 Load Unit) = 10 cm Display as Number Value	
	the presence of loads in letters and numbers is displayed on the member.	





And the number indicating how many loads of that type exist



	Display as	Vector ~	
Finally, in the option Filter From O for the loads you wish to appear.	То 0	you can d	efine a value range
<ul> <li>2.2.4 Copy</li> <li>To copy slabs and loads from one level</li> <li>Use the command <u>only in case you have a state</u></li> </ul>	to another. t <u>ypical floor</u> , i.	.e. the floor is the	e same.
Select the command and a 2 part dialog box ap Slab - Loads Copy Slabs SLABS Current Level Copy to levels From 1-300.00 Up to 1-300.00 Slab loads copy	opears		
Loads ✓ LOADS ✓ Replacement Total Yes Live Load LC LG1 LG2 LG3 LG4 LG5 LG6 LC1 OFF OFF LC2 OFF OFF			
Apply Exit			



1. "Slabs" part:
<ul> <li>- activate "SLABS", select current level (copy level) and "paste levels" from/to.</li> <li>- Activate Stab loads copy if you want to copy the slab loads as well.</li> </ul>
2. "Loads" part:
- activate "LOADS", press
-activate "LOADS", press to switch OFF all load groups and then select by clicking on individually.
$\boxed{ Replacement } and \boxed{ Apply } \rightarrow to replace loads of the selected load groups }$
$\square \text{ Replacement} and \square Apply \rightarrow to apply loads of the selected load groups on the existing}$
loads.
<b>2.2.5</b> Tools Command used for the automatic distribution of loads on a mesh area
Tools
Load distribution on the surface

The new version of SCADA Pro includes a new tool for the automatic distribution and application of the loads on the mesh area.

Select the command, and on the dialog box that opens, define:



Autor	matic distri	ibutio	n of loa	ads on th	e surface	×	
Lo	Load Dead Load						
Gr	Group Group 1					~	Tł
Di	stribution D	ata				~	ar
Su	rface						In
Me	esh 3D	$\sim$	PLATE			~	м
Se	lect graphic	ally	S1/6/2			~	M
Lo	ad						Se
Na	ame						su
	Vertices	Valu	le		Coordinate (cm)		
1.	Pick	0			Not Pick		
2.	Pick	0			Not Pick		
3.	Pick	0			Not Pick		
4.	Pick	0			Not Pick		
4		4			Single surface defined by the points Distribution Esc	e nree	

The type of load from the "Load" field and the group from the "Group" field.

In the "Surface" field,

Mesh 3D 🛛 🗸 Mesh 2D Mesh 3D

Select the type of the surface and the surface group that you wish to load.

In case that the selected mesh group has more than one surfaces you must select from the list, the preferred surface as well.

Mesh 3D $\sim$	PLATE ~
Mesh 2D Mesh 3D	S1/6/2 ~
	S1/1/2 S1/2/2 S1/3/2 S1/4/2 S1/5/2 S1/6/2 S1/7/2 S1/8/2





The load application can be performed graphically as well, by clicking the button Select graphically.

The dialog box closes automatically so that you can identify the surface by clicking on one of its shell element.

Then the dialog box reopens with the pointed out surface identified.



On the "Load" field give a name for the load. Afterwards, define the way of the load distribution on the selected surface.

The definition can be performed graphically:

- By pointing the four corners of the surface and setting the load value.
- By pointing 3 points the first two of which define a straight line that the first load value will be applied, and the third which defines the height that the second load value will be applied.

It is not necessary for these points to belong to the same level, while the outline of the surface can contain lines and arcs.

More specifically:



Point the four corners that define the surface by clicking successively the buttons each corner as shown in the image below.





In this way, the coordinates for the four corners are automatically recognized and filled in.

V	ertices	Value	Coordinate (cm)	
1.	Pick	10	948.3 , 1094.3 , 300.0	
2.	Pick	10	947.7 , 634.6 , 300.0	
3.	Pick	50	948.3 , 1094.3 , 0.0	
4.	Pick	50	947.7 , 634.6 , 0.0	

Next, you set the pressure values (in kN/m<sup>2</sup>) for each corner point

The load distribution on the selected surface is completed and is graphically represented by the elements of the current surface.





A similar process is followed by the below differences being the only ones:



utom	atic distri	butio	n of loa	ds on the surface	×
Loa	d	Dea	ad Load		~
Gro	up	Gro	up 1		~
Dist	ribution Da	ata			~
Surf	ace				
Mes	h 3D	$\sim$	PLATE		~
Sele	ct graphica	ally	S1/6/2		×
Nan	Vertices	Valu	e	Coordinate (cm) Not Pick	
2.	Pick	0	_	Not Pick	_
3.	Pick	0		Not Pick	
4.	Pick	0		Not Pick	
4	K SA	5		Single surfa defined by points Distributi	ce three on

• Using the "Select graphically" command you only select one element form one of the surfaces that are going to be loaded.

• Check the command "Single surface defined by three points" and the 4<sup>th</sup> option is automatically disabled.

As previously described, using the Pick button, you point the 3 points that define the combined area.

Then specify the pressure values in kN / m2 for 3 points.

Automatic distri	bution of loa	ds on the surface X
Load Group Distribution D Surface Mesh 3D Select graphic	Dead Load Group 1 ata PLATE ally \$1/6/2	~ ~ ~ ~
Load Name PR Vertices 1. Pick 2. Pick 3. Pick 4. Pick	ESSURE Value 10 10 50 0	Coordinate (cm) Not Pick Not Pick Not Pick Not Pick
		Single surface defined by three points Distribution Esc

and the second
· 《新聞》中,如果真是出来的「「如果」」「「可以做什么做」」「「如果」」「「」









Follow the same procedure:

Graphical selection with one click.

utomatic distribution of loads on the surface $X$						
Load	Dead Load	~				
Group	Group 1	~				
Distribution	Data	~				
Surface						
Mesh 3D	✓ PLATE	~				
Select graph	ically S1/7/2	~				
Name Vertices	Value	Coordinate (cm)				
1. PICK		040.2, 1004.2, 200.0				
Z- Pick	10	948.3 , 1094.3 , 300.0				
3. Pick	50	492.7 , 1255.0 , 0.0				
4. Pick	0	Not Pick				
		Single surface ocints				

Check the option "Single surface defined by three points" and the  $4^{th}$  option is automatically disabled. Define the surface by pointing to the three points that define the

surface using the Pick	buttons.	Fill in	the pressur	e
values (in $kN/m^2$ ) and click	Distribution	and	Esc	



# 4. Wind and Snow Loads



"Wind-Snow Loads" commands group contains tools for the automatic calculation of wind and snow loads and the distribution to members by Eurocode 1.

Also Greek, Italian, Germany, and Poland Eurocode 1 appendices, as well as the Italian NTC08 Regulation, are included.

It is an extraordinary tool that includes:

- Automatic calculation of characteristic values of snow load on the ground and the roofs determined by EN 1990 for all types of the roof: flat, single, double, quadruple, vaulted, with proximity roof tallest building drift in protrusions and obstacles of the Roof shape coefficients automatic calculation.
- 2D and 3D display of snow load distribution.
- Basic wind velocity automatic calculation.
- Automatic calculation of average wind speed VM (z) at height z (according to soil roughness and orography)
- Categories and soil parameters
- Wind turbulence
- Max velocity
- Wind pressure distribution on surfaces
- Wind forces
- Pressure coefficients for buildings (vertical walls or roofs)

The procedure for calculating wind and snow loads and their distribution to members includes five groups of commands:

- 1. Parameters: Code selection, Wind-snow general parameters
- 2. Edit: wall-roof
- 3. View: wind-snow
- 4. Member correspondence
- 5. Post-Processor



4.1 Parameters	
Code	
Wind	
Parameters Snow	
<b>4.1.1 Code</b> In the dialog box that appears	
Tachnical Standard	
EC1	alian
Technical EC1 VIC_2	008
EC1_PC	olish ustrian
EC1_G	erman
Saudi B	Building Code (301)
011	
Wind         Define wind parameters by Eurocode 1 in the original statements         EC1 WIND PARAMETERS	dialog box:
Wind Define wind parameters by Eurocode 1 in the of EC1 WIND PARAMETERS	dialog box:
Wind Define wind parameters by Eurocode 1 in the o EC1 WIND PARAMETERS Regulation EC1 Zone Rest of Greece	dialog box:
Wind Define wind parameters by Eurocode 1 in the of EC1 WIND PARAMETERS Regulation EC1 Zone Rest of Greece Altitude from sea level (m) Mean Wind Velocity (m/sec) Vb,0 27	dialog box:
Wind Define wind parameters by Eurocode 1 in the of ECI WIND PARAMETERS Regulation EC1 Zone Rest of Greece Altitude from sea level (m) Mean Wind Velocity (m/sec) Vb,0 Snow Density (Kg/m3) p 1.25	dialog box:
Wind Define wind parameters by Eurocode 1 in the of ECI WIND PARAMETERS Regulation EC1 Zone Rest of Greece Altitude from sea level (m) A 500 Mean Wind Velocity (m/sec) Vb,0 27 Snow Density (Kg/m3) $\rho$ 1.25 Directional Factor Cdir 1 Season Factor Cseason 1	dialog box:
Wind Define wind parameters by Eurocode 1 in the of CI WIND PARAMETERS Regulation EC1 Zone Rest of Greece Altitude from sea level (m) A 500 Mean Wind Velocity (m/sec) Vb,0 27 Snow Density (Kg/m3) p 1.25 Directional Factor Cdir 1 Season Factor Cseason 1 Soll Type	dialog box:
Wind Define wind parameters by Eurocode 1 in the of Regulation EC1 Zone Rest of Greece Altitude from sea level (m) A 500 Mean Wind Velocity (m/sec) Vb,0 27 Snow Density (Kg/m3) $\rho$ 1.25 Directional Factor Cdir 1 Season Factor Cseason 1 Soli Type 0 Sea or coastal area exposed to the open sea Distance from More than 40 Km	dialog box:
Wind Define wind parameters by Eurocode 1 in the of Regulation EC1 Zone Rest of Greece Altitude from sea level (m) A 500 Mean Wind Velocity (m/sec) Vb,0 27 Snow Density (Kg/m3) p 1.25 Directional Factor Cdir 1 Season Factor Cseason 1 Soil Type Distance from More than 40 Km V Z0(m) 0.003 Zmin(m) 1 Distance from Zmin(m) 1	dialog box:
Wind         Define wind parameters by Eurocode 1 in the operation of the second	dialog box:
Wind         Define wind parameters by Eurocode 1 in the operation of the second	dialog box:
Wind         Define wind parameters by Eurocode 1 in the openation of the second	dialog box:
Wind         Define wind parameters by Eurocode 1 in the openation of the second	dialog box:
Wind         Define wind parameters by Eurocode 1 in the openation of the second	dialog box:
Wind         Define wind parameters by Eurocode 1 in the of         Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2">Colspan="2"         Colspan="2">Colspan="2"         Colspan="2">Colspan="2"         Altitude from sea level (m)         A 500         Mean Wind Velocity (m/sec)       Vb,0         One Rest of Greece         Colspan="2"         Mean Wind Velocity (m/sec)       Vb,0         Directional Factor       Cdir         Soid Type         Oistence from       More than 40 Km         Oistence from       More than 40 Km         Z0(m)       0.003       Zmin(m)       1         Corthography Factor         Upwind       Lu(m)       500         More than 40 Km       Ymin         Oistence from       More than 40 Km         Upwind       Lu(m)       500       Xmin       150       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12       1.0.12	dialog box:
Wind         Define wind parameters by Eurocode 1 in the openation of the search of the	dialog box:



Select from the list: "Regulation" and "Zone" and the respect fields are automatically updated. In "Soil Type": select type from the list, category, and distance from the coast. In "Orthography Factor": define topography and wind direction. The other fields are updated automatically based on the previous selections. In "Roughness Factor": when Automatic Calculation is activated, the program automatically

calculates the Cr(z) value, otherwise type a value reaction manually.

Press "OK" to save the parameters.

**A** The user can modify the calculated values. By typing different values in the fields, data is updated automatically.

.....



The latest version of SCADA Pro integrates the **Saudi Arabia code (SBC 301)** for wind loads as well.

In the following, a detailed description of the parameters choosing SBC 301 is described:

Technical Star	×	
Technical	Saudi Building Code (301)	•
Wind	Wind :	

By selecting "Wind" parameters the following dialog box appears:



Wind Parameters (SBC)			
Wind Design Procedure     Method 2 - Analytical Procedure       Classification of Bulding     I			
Basic Wind Speed (km/h) V= 0 ? Exposure Category B ?			
Structure Type (Kd) Main Wind Force Resisting System			
Topographic Factor, Kzt			
2-dimensional ridges Upwind Lh(m) -0 H(m) 0 H(m) 0 x(m) -0 H(m) 0 x(m) -0 kzt 0 kzt 0			
Bulding Type Rigid ? Flexible $OK$ Damping ratio $\beta = 0$ Cancel			

SBC 301 provides three methods for calculating wind loads (par. 6.1.2)

- 1. Simplified Procedure (Section 7.1)
- 2. Analytical Procedure (Section 7.2)
- 3. Wind Tunnel Procedure (Section 7.3)

SCADA Pro incorporates the first two methods (The third method is based on experimental measurements).

First, choose one of the two methods for the calculation of the wind loads Wind Design Procedure The first method is applied only to buildings that meet specific criteria (par. 7.1.1).

The second parameter

Classification of Bulding

regards the choice of the class of the building

based on the Table 1.6-1. Press next to the parameter to open the corresponding table. Then define the Base Wind Speed/parameter based on the values of the map (FIGURE 6.4-1)

that appears when pressing





The parameter Exposure Category regards the choice of the class exposure of the building by the paragraph 6.4.2.2 & 6.4.2.3.

Structural Type selection Structure Type (Kd) (TABLE 6.4-1) regards the choice of the Kd (Directionality Factor).

The next parameter section



identifies the influence of the topographic configuration of the region where the structure is situated (FIGURE 6.4-2). According to these parameters, calculate the topography coefficient (Topographic Factor) Kzt.

The first selection regards the soil topology There are five choices:



- 2-Dimensional Ridges
- 2-Dimensional Escarpments



- 3-Dimensional Axisymmetric Hill
- Flat, unobstructed areas and water surfaces
- User

Leeward,

The 3<sup>rd</sup> choice sets Kzt=1.

On the 4<sup>th</sup> choice, the user defines a value to Kzt.

Select the position of the structure relative to the wind direction (Windward, Upwind or

Downwind)	Upwind -

The next three fields

500
100
200

Are related to the topography of the area

Lh: Upwind distance of crest to where the difference in ground elevation is half the height of the hill or escarpment, in meters.

H: Height of hill or escarpment relative to the upwind terrain, in meters.

x: distance upwind or downwind of the crest as in Figure 6.4-2, in meters.

On the next field choose the type of the building based on the stiffness of the diaphragms. There are three choices:

- Rigid
- Flexible
- Parapets

If the type of the building is Flexible you must also set the following two parameters:

Flexible	
Frequency (Hz) n1 =	0
Damping ratio $\beta =$	0



#### 4.1.3 Snow :

Define snow parameters according to **Eurocode 1** in the dialog box:

Snow	
EC1 SNOW PARAMETERS	
Regulation EC1	Select from the list the parameters: "Regulation". "Topography" and "Zone" and the related fields are automatically updated. In the "Accidental Snow Load": select a condition. Press "OK" to save the parameters.
Snow Load (at sea level) Sk,0 kN/m2     0.8       Altitude (from sea level) A m     500       Snow Load (at Altitude A) Sk kN/m2     1.04       Accidental Snow Load     1.04       Design State     Case A (No exceptional falls/No exceptional drift       Exceptional Loads Factor Cesl     1       OK     Cancel	
4.2 Edit 4.2.1 Walls : Edit Edit Walls	×.
Umail       Umail <td< td=""><td></td></td<>	



Automatic Calculation

The take advantage of the "Templates" command, in which all the geometric characteristics of the walls are filled in automatically by the program, and save a lot of time and work!

#### Without using "TEMPLATES"

Select from the list the wall according to the wind direction (clockwise direction).

Press next to b(m). To define the length of the selected wall, left click on start and end points.

Press next to how. To define the width of the selected wall, left click on start and end points.

"Partial Walls" list is filled in automatically.

Then type/openings percentage (for doors and windows) and press

The program calculates automatically the "Equivalent Wall." Press "OK" command to save the parameters. Repeat the procedure for all four directions of the walls.

#### NOTE:

- ▲ The height of the lower wall is always defined starting from level 0 even if the steel structure begins at a higher level.
- ▲ If the front view consists of several walls at one or more levels, press the button "New" and repeat the above procedure to set the whole face.



In above table fill in the geometrical characteristics of the "Sub-walls".



Finally type the percentage of Automatic Calculation The program calculates autom The red rectangle should c	openings for each direction and press, every time,
Edit Wall	Image: A state of the walls.

#### Using "TEMPLATES"

By using "TEMPLATES" tool, the user saves a lot of time and work because the geometric characteristics of the walls are updated automatically by the program.

Select from the list the wall according to the wind direction. "Partial Walls" list is filled in automatically, without using "Pick" as mentioned previously.

The user needs only to type in the openings percentage and press and press The program calculates automatically the "Equivalent Wall." Press "OK" to save the parameters. Repeat the procedure for all four directions of the walls.



Edit	
4.2.2 Roofs	
Edit Roof	×
	Roof No. 1 Duopitch
	Orientation 90           Peaks - Sides (m)           L0         5           L1         5           L3         5
	Sharp Edges hp(m)= 0
	Number of 1
	h1 3 h2 0
	h         0         a1         o         a2         o         a3         o           L         0         b1         0         b2         0         b3         0
	Roof Proximity (Snow Drift) (m) Side 0 V No Proximity V h 0 b2 0

### Without using "TEMPLATES"

Select from the lists the roof number and the form.

Press next to "Peaks-Sides". To define the geometry of the roof, left click on the four peaks of the floor plan of the roof and the cells will be filled in automatically.

Sharp Edges	•
Sharp Edges	
Parapets	
Curved Edges	

Select from the lists Indined Edges and type in the height of the barrier in m.

In "Geometrical Data" type in the number of frames and the other geometrical data in m.

**Roof Proximity** 

If the structure is close to another taller building, in the "Roof Proximity" select the side which



The field changes according to the proximity and the side. Type in the geometry data and





Press "OK" to save the parameters.

Repeat the procedure for all four directions of the roof (clockwise direction).

#### Using "TEMPLATES"

Select from the lists the roof number and the form. "Geometrical Data" fields are filled in automatically.



The user only needs to select Indined Edges

define the "Roof Proximity" as mentioned previously. Press "OK" to save the parameters.

Repeat the procedure for all four directions of the roof (clockwise direction).

#### **4.3 View**

#### 4.3.1 Wind

Select the command to see the wind pressure distribution on the walls and the roofs of the building. In the dialog box, select the wind direction, the wall or the roof and the type of pressure. The distribution is automatically displayed with colors. The zones with different pressure have a different color.





#### View

#### 4.3.2 Snow

Select the command to see the snow distribution on the roofs.



In the dialog box, select from the list the number of "roof" of the "opening" i.e. the number of the frame, (if there are more than one), and "Case"



for the load distribution of snow.

Activate "Load" checkbox to display the values and "3DView" checkbox to receive snow distribution as is displayed in the following pictures.



#### **4.4 Member Correspondence**

Member Correspondence to assign the calculated loads to the members, through the influence zones. Select the command and in the dialog box: select a wall, or a roof and define the dimension of the influence zones.

In the new version of SCADA Pro, the automatic calculation of influence zones for linear members to make the distribution of wind and snow loads is completed and integrated as well.



We remind you that until now the automatic distribution was possible only for structures derived from Templates. Now you have the opportunity to make this distribution on any surface.

Let's see in detail the manual, semiautomatic and automatic distribution:

By selecting the command the following dialog box now opens

Member Co	rrespondenc	e			×
Wall left (p	erpendicular v	wind o	dir. 0)		~
Add Memb Influence	oers Zones (m)		Vertices	Coordinat	te (cm)
1.0		1.	Pick	0.0,500.	0,0.0
Lett	0	2.	Pick	0.0,0.0	, 0.0
Right	U	3.	Pick	0.0,0.0,	300.0
Pi	ick			Distribution	View
	Initialisation of all members (Walls-Roofs)				
Membe	ers Initialisatio	n		[	Cancel

Concerning the old definition of the influence zones nothing changed, while a new part to define the area with three points was added to the right.

 $\sim$ 

The definition always concerns the active area

Wall left (perpendicular wind dir. 0)

It is better to start either the **manual** or **semi-automatic** procedure by pressing the "Members Initialization" button.

#### Attention:

▲ In the **automatic** procedure coming from the TEMPLATES, DO NOT press "Members Initialization" button, because it will delete the automatic load distribution to members!!!

• Manual Procedure - Without using "TEMPLATES"

Influence Zones (m)			
Left	0		
Right	0		

In define the influence zones of a member by typing the corresponding widths in m, on the left and right of that, "Pick" and left click on the member (or parts of the member). The "Influence Zone" is displayed as in the figure below.

"Left" and "Right" are determined based on the local axis x (red).





#### • Semi-automatic Procedure - Without using "TEMPLATES"

A new part to define the area with three points is added to the right. The definition always concerns the active area:

 $\sim$ 

Wall left (perpendicular wind dir. 0)

It is better to start the procedure by pressing the "Members Initialization" button.

Indicate the point graphically along with the following particularity:

- The first two points define the direction by which the automatic calculation of influence surfaces is made for items which are parallel to this direction.
   Note also that the distribution will be for all linear members belonging to this level and are parallel to the first direction.
- Since you define the three points, press the "Distribution" button and the program automatically makes the distribution and displays it.

Respectively the definition is made for the other walls.

Concerning the roofs, the definition can be made sequentially. First, select the roof

Roof No. 1

 $\succeq$ , then you must define the individual

areas.

#### EXAMPLE:

For example, first, define the left slope indicating graphically the three points





and then, the right ones. The overall result is the following:



- ▲ Finally, it is worth noting that if the walls are properly defined there is NO need for more definition. Just select each wall and press «Distribution». The distribution is made and is displayed simultaneously on the linear members belonging to this wall.
- A Same for the flat roofs only.

#### • Automatic Procedure - Using "TEMPLATES"

By activating "Purlins" and "Girders" in "Load Attribution" of "Templates", just select "Pick" and the program automatically calculates the influence zones distributing the pressure in all purlins and girders.



**V**1



#### 4.5 Post-Processor

The last command is "Post-Processor".



In the dialog box, in "Load Attribution" there are two labels:

one with the wind loads, 4 cases for four directions, i.e. 12 cases to each load and one with snow loads, 3 cases for typical snow.

The numbers that appear on labels correspond to the Load Cases serial numbers.

Remember:

Load Case1: Dead Load Case2: Live

structure members, or

and now there are 16 new Load Cases for the Wind (from 3 to 18) and 3 for the Snow (19, 20 and 21).

Total Load Deletion (Snow-Wind Loads)

20	a	iu
Se	e	ct

wind			100	070	Sno	w-	
Cpe_p+Cpi	3	90	180	15			Acci-\d Typical
Cpe_p-Cpi	4	8	12	16	Ca	se i	19 22
Cpe_n+Cpi	5	9	13	17	Ca	se ii	20 23
Cpe_n-Cpi	6	10	14	18	Ca	se iii	21 24
	Т	otal Load	d Deletion	n (Snow-\	Wind Loa	ds)	
	Load A	ttributior	n in Meml	bers(from	n Wind ar	id Sn	ow)
Sceneries							Results
Sceneries		Nev	w Scenar	io 💌			Results
Sceneries Wind 0 Wind 90		Nev	w Scenar w Scenar	io 💌			Results
Sceneries Wind 0 Wind 90 Wind 180	)	Nev Nev	w Scenar w Scenar w Scenar	io 💌 io 💌			Results
Sceneries Wind 0 Wind 90 Wind 180 Wind 270	)	Nev Nev Nev	w Scenar w Scenar w Scenar w Scenar	io V io V io V			Results
Sceneries Wind 0 Wind 90 Wind 180 Wind 270 Vind 270 Vind 270	) ) now	Nev Nev Nev Nev	v Scenar v Scenar v Scenar v Scenar v Scenar	io V io V io V io V			Results
Sceneries Wind 0 Wind 90 Wind 180 Wind 270 Wind 270 Ypical S Accident	) ) now al Snow	Nev Nev Nev Nev	v Scenar v Scenar v Scenar v Scenar v Scenar v Scenar	io  io io io io io io io io io io io io io			Results

 Wind Loads)
 to attribute wind and snow loads on the

 Load Attribution in Members(from Wind and Snow)
 to delete them all.

In "Scenarios" there is a list with the analysis scenarios created automatically by selecting the Scenery Analysis Creation command.



SCADA Pro not only calculates automatically the load distribution of wind and snow, but it also creates automatically all the analysis scenarios too, saving the user from hard work and much time.

					Scenario		×
					No	•	
					Disable	Name	
Load Ca	se Defin	nition		×	EC8 Static EAK Static	Analysis EC-8	•
<mark>I ∕ Sel</mark> t	f Weight	Dead Load	•	Insert	NTC_2008 Static EC8 Static	Type Static	-
LC	S.W.	Description	<b></b>		Static Wind 0	Properties	
1	Yes	Dead Load			Static Wind 50	Elements	Nodes
2	No	Live Load			Static Wind 270		
3	No	Snow			Static Typical snow	Load Cases	Masses
4	No	Wind W (+x)		Delete			
5	No	Wind W (+z)					
6	No	Wind (+x)				New	Update
7	No	Wind (+z)	-				
1		· · · ·		OK		Exi	it
					1		

Press to open the txt results file, containing in detail all data and calculations derived from all Eurocode 1 procedures1.

E d001 - Word	Pad – 🗆 🗙
File Edit View Insert Format Help	
WIND / SNOW LOADS CALC ACCORDING TO EN 1991-1-3/4:2005 N	ULATION AD GREECE
SNOW DATA	
TOPOGRAPHYV EXPOSURE COEFFICIENT Ce THERMAL COEFFICIENT Ct DESIGN STATE EXCEPTIONAL SNOW LOADS COEFFICIENT Cesl WEIGHT DENSITY OF SNOW γ(Kn/m^3) Zone III (Yπόλοιπη Χώρα) SNOW LOAD (AT SEE LEVEL) Sk,0 (Kn/m^2) ALTITUDE A(m) SNOW LOAD (AT LEVEL 500.00m)Sk (Kn/m^2) 	: Normal : 1.00 : 1.00 : Case A (No exceptional falls/No e : 1.00 : 3.00 : 0.80 : 500.0 : 1.04
TERRAIN CATEGORY Z0 (m) Zmin (m) FUNDAM. VALUE OF THE BASIC WIND VELOCITY (m/sec) AIR DENSITY ρ(Kg/m^3) VIECTIONA EACTOR Cot ~ <	: 0 Sea or coastal area exposed t : 0.003 : 1.00 : 27.0 : 1.25
For Help, press F1	NUM //

1



. .

. . . . . . . . .

Choosing the SBC 301 Regulation, the printing is as follows:

										Page: 1
		W	/IND	L	OADS	; C/	ALCULA	TION		
	AC	co	RDI	N	G ТО 3	SBO	C 301 CH	IAPTER :	7	
Design Method : M	ethod 2	-A	nalyti	ica	I Proced	dure				
				Ba	asic Win	d Pa	arameters			
DESCRIPTION	S	YMBOL U		JNI	TS		V	ALUE		REFERENCE
Classification of Buildin	9									Table 1.6-1
Importance Factor							(	0.77		Table 6.5-1
Country										Figure 6.4-1
City										Figure 6.4-1
Basic Wind Speed		V	(	(km	v/h)		1	65.00		Figure 6.4-1
Exposure Category								В		6.4.2.3
Structure Type						Mai	in Wind Ford	e Resisting	System	Table 6.4-1
Wind Directionality Fact	or	Kd					(	0.85		Table 6.4-1
	Торо	gra	phic	Fa	ctor (Kz	t) C	alculation	(Figure 6.4	-2)	
DESCRIPTION		SY	MBO	L	UNITS		VALUE			
Topography						2-d	imensional ridge	6		
Hill Height		н			(m)		-500.00	-	ŕ	Speed-up
Half Hill Length		Lh			(m)		100.00	110	x(Upwind)	A z(Desumind)
Distance from top of cre	est	x			(m)		-100.00	17	All and	NO 11
Building up/down wind							Upwind	-	4	NO T
Topographic Factor		Kzt					1.49			
Building Type :	Rigid	F	Frequence		cy (Hz)	n1 :	=	Dam	ping Ratio	β =
	(	Gus	t Effe	ect Dir	Factor	(G)	Calculatio	n (7.2.7)		
	1			ruş	giù Stiù	ctui	WALL	WALL	WALL	
DESCRIPTION	SYMB	OL	דואט	ſS	UEFT (N	L w0)	FRONT (w90)	RIGHT (w180)	BACK (w270)	REFERENCE
Mean height	h		(m)	)	4.00		4.00	4.00	4.00	
Width	В		(m)	)	6.00	)	6.00	6.00	6.00	
Equivalent height	/z		(m)	)	10.00	)	10.00	10.00	10.00	7.2.7.1
Constant	/ε				0.33		0.33	0.33	0.33	Table 7.2-1
Constant	1		(m)	)	100.0	0	100.00	100.00	100.00	Table 7.2-1
Minimum height	zmir	1	(m)	)	10.00	)	10.00	10.00	10.00	Table 7.2-1
Constant	С				0.30		0.30	0.30	0.30	Table 7.2-1
Constants	9 <b>9</b> , 9	jv 🛛			3.40/3.	40	3.40/3.40	3.40/3.40	3.40/3.40	
Integral length scale	Lz		(m)	)	100.0	0	100.00	100.00	100.00	Table 6.4-1
Intensity of turbulence	lz				0.30		0.30	0.30	0.30	Eq. 7.2-2
Background response	Q				0.93		0.93	0.93	0.93	Eq. 7.2-3
Gust Effect Factor	G				1.00		1.00	1.00	1.00	Eq. 7.2-1



														F	age :	
					W	IND	LO	ADS	CAL	CULAT	ION	_				
				AC	co	RDI	NG	TO S	SBC 3	01 CH/	APTER	17				
								WALI	S DAT	Α						
		Individual Wa					5	-		EquivalentWalls						
	Num	Sect	Leng	gtn	Heigi		Ag n2)	Open (%)	Ao (m2)	Length	Height	Ag (m2)	Open (%)	A0 (m2)	Ao>0	
		1	6.0	0	3.00	18	3.00	0.00	0.00	6.00	3.00	18.00	0.00	0.00	No	
Wall		<u> </u>	-	-		-										
Left	W1					—										
		1	6.0	0	4.00	24	1.00	0.00	0.00	6.00	4.00	24.00	0.00	0.00	No	
Wall Front	W2															
			_	_		_									_	
			6.0	<u></u>	2.00		00	0.00	0.00	6.00	2.00	40.00	0.00	0.00	No	
w11		<u> </u>	0.0	~	3.00		5.00	0.00	0.00	0.00	3.00	18.00	0.00	0.00	NO	
Right	W3															
		1	6.0	0	4.00	24	1.00	0.00	0.00	6.00	4.00	24.00	0.00	0.00	No	
Wall																
Back	VV4															
To	tals					84	1.00		0.00	1		84.00	J	0.00		
BU	ILDIN	GISC	UAL	IFIED	) A S	OPE	N		NO							
Maria	1			1.0	<u> </u>	14		ROOI		A	Demetit	-	1-2	1	T	
ber	R	oof Ty	pe	1 (m	<u>.</u>	L1 (m)		LZ L3 1 (m) (m) R0		of Edge ions		(m)	(m)	h (m)	L (m	
1	Duopit	ch		6.00	6.00		6.00	6.0	0 Sha	rp Edges	1	3.00	3.00	4.00	3.00	
							Ļ									
Num	a0 (m)	a1	a2	a3 (m)	b0	b1	b	2 b	3 A	9	1.2					
Der	(111)	(111)	(111)	(m)	(111)	(m)		·/ (	36 (	1/	LJ	4				
		_				-			00.1	~			т 📡	, P	South States	
							+			1.0		12	h1			
						-	T			-"		L2 .	1	<u>ا</u> " ا	l í	
													10	U		
										2	-11-	3		FL-	1	
								То	tal 36.	00						
					Enc	losu	re C	lassi	ficatio	n (Sectio	on 6.2)					
Num			Re	feren	ice A	rea	-	Aoi	Agi	Aoi/Agi	Ag	Ao	Ao>	Ref. A	\o>1.1	
ber	VV.	ali	(m2)	(m2	9	(m2)	(	m2)	(m2)	<=0.20	(m2)	(m2)	Are	a	Aoi	
1	Wall	Left	0.4	0.18	0	0.180	0	.000	102.000	Yes	18,000	0.000	N		No	
	Wall	Front	0.4	0.24	0	0.240	0	.000	96.000	Yes	24,000	0.000	N		No	
2		Diabt	0.4	0.18	0	0.180	0	.000	102.000	Yes	18.000	0.000	N	)	No	
2	Wall	NUGHLI														
2 3 4	Wall   Wall	Back	0.4	0.24	10	0.240	0	.000	96.000	Yes	24.000	0.000	N		No	