Test for GreenYellow Color

Example í: Steel

Test for IndianRed Color

Test for Indigo Color Inis example will explain step by step the creation of a basic 3D steel structure. This example will be most effective if the user practice the illustrated skills as they are presented.

The structure to be entered in this example is shown below:



Text on each page

In order to simplify data entry, frame members are grouped as follows:



Test forheight Smaroeth Coloride to the scriptions shown here will be illustrated in this example.

1) Starting a new structure



Select New from the RAM Elements button menu.

In the event that there is an existing model open, RAM Elements will ask to save it.

Texanipis Light/SkyBlue Color

Test fo	r LightSlateGray Color
	Metric
Tes	r _{EhightSteelBlue} Color
Test f	o <mark>T LightYellow Co</mark> lor
	Show units
Ū	est for Lime Color
Tese	For LimeGreen Color

Press the button on the status bar, a menu will be displayed. Then, select the option Units configuration.

	50	
	Units	? 🛛
	Test for Maroon Color Units system:	English 🗸
Test	or MediumAquamarine Co	ler 🗸
т	estationdMediumBlue Color	in 🗸
Те	sReiffermeandhearchid Color	in2 🗸
То	Material Strengths:	Kip/in2
Ie	Unit Weights:	Kip/ft3 🗸 🗸
Tes	for MediumSeaGreen Colo Temperatures:	F 🗸
Tes	for MediumSlateBlue Colo Translational Springs:	Kip/in 🗸
Test f	୦୫୦MoclisymSpringGreen Co	🥠 🗸
Test	Pointweet Ennesurquoise Col	₩ip 🖌
Tes		Kip/ft 🗸
	Surface/Area Loads:	Kip/ft2 🗸
10	Stresses:	Lb/in2
	est for MintCream Color Soil stresses:	Lb/in2
	Moments:	Kip*ft 🗸
	Deflections:	in 💌
	Others:	Kip-ft 🗸 🗸
	Default units for system:]
	English SI	Metric
	ſ	OK Cancel

Text on each page

Select the English default unit system in the window displayed.

Test for Moccasin Color 2) Entering node coordinates

Test In the coordinates spreadsheet enter the coordinates shown below:



Go to the Spreadsheet Nodes/Coordinates and enter the coordinates shown above.

Test for RelevioletRed Color	(24, 16.5, 0)
Test for Mapa ya Whip Color	(24, 12, 0)
Test for PeachPuff Color	
Test for Peru Color	(24, 0, 0)
Test for Plum Color	

The entered nodes are shown on the screen.

Test for PowderBlue Color

3) Generation of frame members

Teststerc Revalative of the frame members. Select the nodes in the sequence shown below, and then

Test for RosyBrown Color



connect the selected nodes by pressing with memb

• 3	• 4	• 5
• 2		• 6
• 1		• 7

Select the nodes in the order shown. To select several nodes remember to press the SHIFT key while clicking with the mouse.

Test for SaddleBrown Color

Test for Salmon Color	
Nodes Members Shells Areas	Conn Gen
i 🖪 🎽 🖓 🚽 🖡	= 🐚 🛨 🛱 🌷
Connectivity and description	
Testible Description ell Color	NK Tribut, width Brace
Test for Sienna Color	

Go to the Spreadsheet Members/Nodes and Description Test for SkyBlue Color

As can be seen the frame members were generate.

Test for Teal Color NOTE. - Remember that it is possible to undo the last operation by pressing

Test for Tan Color

Test for Thistle Color 4) Assigning a description

It **Is steeded** and **the second second and a selection** of the second se elements, optimization, and others. To assign the same description to every member of a group rest for Transparent Color proceed as follows:

Text on each page



Select columns

4

Test for Violet Color

Test-for-Wheat-Colo Test f or YellowGreen Color COL 1 COL 1 Test for Black Font Test for₊Brown Font DESCR Test for Navy Font (additive) TestsfordBluetGolor st oseBinderodor **Test for Purple Color** Column (additive) COL +1 Test for Red Color

Example 1: Steel

Text on each page

Then assign the description to the selected members selecting the Column (additive) option.

Note. – To view the member descriptions graphically (on the screen) go to *View tab*, *Model group*, press the properties button and select the option *Description by element* from the menu displayed. Test for Antique White Color Repeat the steps explained previously to assign a Description to the other members:



Select members



Assign the description to the selected members selecting the Beam (additive) option.

Generate the beam as shown in the figure below. Assign BEAM2 description to this newly created member:

Example 1: Steelsque Color

	t for Blanched Almond Col
	Test for BlueViolet Color
1	O [est for BurlyWood Color
	Test for CadetBlue Color
_	
	est for Chartreuse Color

Test for Chocolate Color

Test for Coral Color

Connect node: To create the horizontal beam, select the nodes shown in this figure and press with members Test for Cornflower Blue Color

Test for Cornsilk Color Test for Crimson Color
Test for Cyan Color
est for Dark Cyan Color

Test fog Darkdewhip in Rochester and the members selecting the Beam (additive) option.

5) Segmenting Members

To segment frame members, follow these steps: Test for DarkGreen Color

```
Test for DarkKhaki Color
BEAM 1
```

Test for DarkMagenta Color

```
Test for DarkOliveGreen Color
```

```
Test for DarkOrange Color
```

Select members to be segmented.

Segment members		×
Number of segments per member:	3	
Options		
 Segment members 		
O Create nodes only (maintain physical m	embers)	
ОК	Canc	el

Test for DarkRed Color



Press And enter the desired number of segments. In this case, enter six segments. Then press OK or the ENTER key.



NOTE. - Remember that it is possible to undo the last operation by pressing

Notice that the segmented members have the same description as the original member and that each member is treated as one physical member.

6) Generation of vertical members

To enter the vertical truss elements, follow these steps:



Text on each page

Select the nodes shown in this figure. Notice that it is not necessary to select the exterior nodes.



Press button Y to generate vertical members (plus y generates members in the vertical up direction).

7) Generation of diagonal members

At first, generate the diagonal truss web members of the left side of the structure, and then the right side.

Diagonal members on the left side:



Select the nodes in the order shown in this figure.



Text on each page

```
11
```

Alternately connect

Press nodes with members from the ribbon (the button is visible when the Members tab is the current page in the spreadsheet and connectivity button is pressed).

To enter diagonals on the right side proceed in the same way.

NOTE. - Remember that it is possible to undo the last operation by pressing **1**.

The differences between the two buttons are as follows:



This button connects the selected nodes in a continuous line.

This button connects alternate pairs of nodes with a fragmented line. That is, the first member is generated between the first pair of selected nodes, the second member between the second pair of selected nodes, etc.

8) Assigning a Description to members

Follow these steps to assign a Description to the internal web members:

a) Select diagonal and vertical (internal) elements using the button [] (*Home tab, Selection group*)



To select the elements select one member of each group and then press. Remember that this button selects elements with a common description. In this case all internal elements belong to the group that does not have a description yet. That is to say they all have the same empty description.

b) Internal elements will be assigned a DIAG1 description. Since there is no button available to assign this description (as opposed to COL1 and BEAM1 buttons), it is necessary to enter it manually:



Enter DIAG1 description and then press located at Spreadsheet tab, Spreadsheet group, to fill the column with the value. Another way to do this would the access to command from the popup menu displayed after right click on the spreadsheet area, having selected the desired rows to fill previously.

Important - Descriptions are very important to select groups of frame members. It is also important to have entered the descriptions correctly. If this has not been done correctly the user may experience some difficulty following the next steps in this example.

9) Copying the structure

It is advisable to enter all the descriptions of a structure before copying it, because when a structure is copied the Descriptions are also copied.

To copy a structure, follow these steps:





Select all the elements that should be copied. In this case, press (Home tab, Data tab) to select the entire structure.

Execute the **Copy** command (*Home tab, Modeling group*).

e 🖻 🖬 🤊 A	R Q (U)		RAM	v
Home Sprea	dsheet View	Process Out	put Modules	
Explorer Spreadsheet	All	mts +	Delete	
Data	Selection		Modeling	
Copy elements	? 🔀			
Number of copies: 2				
Distance			Text	on each page
DeltaX: 0	[ft]			
DeltaY: 0	[ft]			
DeltaZ: 18	[ft]			
ОК	Cancel			

Enter the number of copies and the distances in X, Y, and Z between each copy. In this case, enter the values shown in this figure. Then press OK.



10) Generation of the roof beams (purlins)

To generate the roof beams, follow these steps:



Text on each page

Select the initial nodes or end nodes of the roof beams.



Then press **Z** (*Press button* **-Z** *if nothing occurs*). *Note that the* +/- *refers to the direction that the members are projected.*



Note. - Notice that the middle portal is not connected to the roof beams. The model can be left without making any changes and the program will interpret the roof beams as continuous physical members. However, if the roof beams are going to be modeled as simply supported beams (as they normally are), it is necessary to segment the beams and connect one end to the middle portal. The



Text on each page

Notice that roof beams do not connect with the middle frame



With the roof beams selected, press selection to split roof beams and connect them with the middle portal. It is necessary to select the member and the node of the middle portal that will become the point of break.



Text on each page

Choose the options shown in the figure. The members will be divided into two physical members.

11) Assigning a Description to roof beams

To assign a Description to roof beams, proceed as follows:

a) Select roof beams by description.



Select a member of the group and then press. Since the selected element does not have a description, all members with empty description will be selected.

b) ROOF1 description will be assigned to roof beams. There is no button available to automatically assign the description (as opposed to COL1 and BEAM1 descriptions). Therefore, the Description has to be entered manually:

Nodes	Members	Shells	Areas	Conn	Gen			Nodes	Members Shells	Areas	Conn	Gen		
$^{\circ}$ N	👍 🍃	ų ¹²	∳ ⊮	= ``g	<u>+</u> A	² I		ĸ	1 in 12 .	∳ ⊮	= ``g	<u>+</u> A	² I	
Conne	ctivity and de	escription	n					Connect	tivity and descriptior	٦				
Membe	r Description	-	NJ	NK	Tribut. width	Brace	^	Member	Description	NJ	NK	Tribut. width	Brace	^
82	Roof1		5	54	0	No		82	Roof1	5	54	0	No	
83			8	23	0	No		83	Roof1	8	23	0	No	
84			7	25	0	No		84	Roof1	7	25	0	No	
86			6	52	0	No		86	Roof1	6	52	0	No	
87			9	35	0	No		87	Roof1	9	35	0	No	
88			11	37	0	No		88	Roof1	11	37	0	No	
91			21	56	0	No		91	Roof1	21	56	0	No	
92			23	24	0	No		92	Roof1	23	24	0	No	
93			25	27	0	No		93	Roof1	25	27	0	No	
94			10	26	0	No		94	Roof1	10	26	0	No	
95			29	50	l n	No		95	Roof1	29	50	0	No	
								96	Roof1	35	36	0	No	
								97	Roof1	37	38	0	No	
								101	Roof1	70	~¥	on d	All h	nago
								106	Roof1	52		011 6		paye
								111	Roof1	54	21	0	No	
								116	Roof1	56	22	0	No	
								120	Roof1	26	28	0	No	

Enter ROOF 1 under description and then press \checkmark to fill the column with the entered value.

Generating DIAG2 and BEAM3 members

Now proceed to enter the DIAG2 and BEAM3 members that are shown in the figure below. Generate these elements as explained before.



12) Supports

To enter supports proceed as follows:



Select support nodes

e 🖻 🖬 🤊 A R Q	9					
Home Spreadsheet	View Process Outpu	it Mod	lules			
	Value	A.		x 📩		L.
Clear 📜 🗱 👪	$+ - \times - = \cdots$	Pinieu	in X	in Z	X and Z	Fixeu
Spreadsh	eet			Active spr	eadsheet t	ools
Nodes Members Shells Areas (x,y) ▲ ♣ ♣ ▲ Restraints ♣ ▲ ▲ ▲ Node TX TY TZ RX RY RZ 1 ✔ ✔ ✔ ✔ ✔ ✔ 2 ✔ ✔ ✔ ✔ ✔ ✔ 17 ✔ ✔ ✔ ✔ ✔ ✔ 19 ✔ ✔ ✔ ✔ ✔ ✔ ↓ 14 Ø Ø Ø Ø Ø Ø Ø	Conn Gen			Text	' on ea	ach page

Go to the Spreadsheet Nodes/Restraints and click on the corresponding support. In this case click on





The Supports have been entered

13) Assigning sections to frame members.

To assign a section to some member, and this section is available in the section database, proceed as follows:

Select the members to which a section will be assigned. In this case, select all the columns.



Nodes	Members	Shells	Areas	Conn	Gen			
ik R	<u>ң</u>	a∕ ^{†2}	🚽 🕨	· ``p	<u>+</u>	a ² ¹	I	» •
Sections	;							
Member	Section		lg factor	r d0	dL			^
1	W10x12		(0 0	0			
6	W10x12		(0 0	0		_	
28	W10x12		(0 0	0			
29	W10x12		(0 0	0			
38	W10x12		(0 0	0			
39	W10x12		(0 0	0			
Available	sections						•	
Group:	🛄 United	d States					~	
	_		10	ems				
	5		<u>^</u>	- We	X58		1	
	IBE		-1 d	- W8	X67		_	
				W 1	UXIZ			
	21			W 1	0/15			
	2L			W 1	0/1/			
					0722			
			*		0422		-	-

To do this, first select one column and press.

Then go to the Spreadsheet **Members/Sections**. Select W10x12 profile and press \bigtriangleup (double click on the profile will assign the selected item).

Assign sections to all members of the structure in a similar manner.



To select all the elements of the truss, select one element of each group and press.

Nodes	Members	Shells	Areas 🛛	Conn	Ge	en			
ĸ	1	v ¹² - (₽ ⊩	9)	+	a ² ¹	I	» •
Sections	5								
Member	Section		lg factor	d0	- (dL			^
2	T2L 2-1_	2x2-1_2x1	()	0	0			
3	T2L 2-1_	2x2-1_2x1	()	0	0			
4	T2L 2-1_	2x2-1_2x1	()	0	0			
5	T2L 2-1_	2x2-1_2x1	()	0	0			
7	T2L 2-1_	2x2-1_2x1	()	0	0			
8	T2L 2-1_	2x2-1_2x1	()	0	0			
9	T2L 2-1_	2x2-1_2x1	()	0	0			
10	T2L 2-1_	2x2-1_2x1	()	0	0			
11	T2L 2-1_	2x2-1_2x1	()	0	0			
12	T2L 2-1_	2x2-1_2x1	()	0	0			
13	T2L 2-1_	2x2-1_2x1	()	0	0			
	TOL 0.4	0.0 X 0.4			0	0			<u> </u>
Available	sections							▲	â
Group:	🛄 Unite	ed States						•	~
Tables			It	ems					
	Tube		~	—] т	21.2	X2X5 1	6X3 4		~
-16 1	2L			- 1	2L 2	-1 2X2	-1 2X	14	
╞╼┲╴╤	2LU			T:	2L 2	-1 2X2	-1 2X	1 4X	
- O T	s			— Т	2L 2	-1 2X2	-1 2X	1_4X	
−Ō ⊺	UBE			-🗋 т;	2L 2	-1_2X2	-1_2X	1 2	
⊢Ī v	v			-🛅 т;	2L 2	-1_2X2	-1_2X	1_2X	
∣⊢∓ v	VT			-🗋 т;	2L 2	-1_2X2	-1_2X	1 2X	
ب لحف ا	2		×	<u>т</u> т	01.0	1 202	1 22	5 12	~

Text on each page

Assign section $T2L 2 \cdot 1 \cdot 2x^2 \cdot 1 \cdot 2x^2$

Now assign sections to the DIAG2 and BEAM3 elements



Select the elements DIAG2 and BEAM3

Nodes	Members	Shells	Areas C	onn	Gen				
K	1	v ^{†2} →	🗗 🕨	9	<u>+</u>	ana international distribution of the second secon	>> •		
Sections									
Member	Section		la fector	40	d				
89	T2L 2x2x	14	0	0	0				
90	T2L 2x2x	14	0	0	0				
98	T2L 2x2x	14	0	0	0				
99	T2L 2x2x*	14	0	0	0				
100	T2L 2x2x*	1_4	0	0	0				
102	T2L 2x2x*	1_4	0	0	0				
103	T2L 2x2x*	1_4	0	0	0				
104	T2L 2x2x*	1_4	0	0	0				
105	T2L 2x2x*	1_4	0	0	0				
107	T2L 2x2x*	1_4	0	0	0				
108	T2L 2x2x*	1_4	0	0	0				
400	TOL 0.00		0	0	0				
Available	sections					-			
Group:	🛄 Unite	d States					*		
Tables			Jte	ms					
	Tube			1 T2L	2X2X1_4				
	2L			T2L	2X2X1 4	X3 8			
	2LU		-	T2L	2X2X1 4	X3 4			
- O T	5			T2L	2X2X1 8	_			
-Ō T	UBE			T2L	2X2X1 8	X3_8			
I-Ī w	/			🚺 T2L	2X2X1_8	X3_4			
⊢Ŧ w	/т			🚺 T2L	2X2X3_1	6			
L v	0		~	TOU		2V0 0	$\mathbf{\mathbf{v}}$		

Text on each page

Assign section T2L 2x2x1_4

14) Adding sections to the database.

In this example, a cold-formed C-section will be assigned to the Roof beams. This cold-formed C (with lips) profile is not available in the section database. Therefore, a new section should be added. Proceed as follows:



Go to the Home tab, Databases group and execute the Sections button

Sections		? 🔀	
Group: 🔤 United St	ates		
Tables	Items	<u>R</u>	
– C aisiCS	aisiCS 4x2x059		
_ ∩_ aisiHU	aisiCS 4x2x065		
- JC aisiILip	aisiCS 4x2x070		
– 💪 aisiLS	aisiCS 4x2x085		
- L aisiLU	aisiCS 4x2x105		
$-\mathbf{C}$ aisiSSMA_S	aisiCS 4x2.5x059		
- C aisiSSMA_T	aisiCS 4x2.5x065		
-J aisiZS	aisiCS 4x2.5x070	부	
- JaisiZU	aisiCS 4x2.5x085	Fext O	n eacn page
	aisiCS 4x2.5x105		
Cost	aisiCS 4x4x059		
- 😽 G1-1_2L	aisiCS 4x4x065		
- 🔛 G1-3_8L			
		855 M=	
		AL	
		z +	
		~	
aisiClip.leo			
		Close	

Press the button to add a New group to the database. After that, a name for the new group is required in the displayed window:

New group	×
Name:	
MyGroup	
OK Cancel	

Then, add a new Table by pressing the 📕 button. A new dialog will be displayed to enter the name

for the new table. It is also required to select the type of table, to perform this action press the button and the following dialog will be shown:

Select section	type			[?×	
Available section	types					
Y 2B2L Y 2C_1 Y 2C_0 T 2L&PL Y 2L&VPL II 2Ws II 2Ws	AS_CHS AS_CHS AS_FC AS_PFC AS_RHS AS_SHS AS_TEE AS_TFB AS_UA AS_UA AS_UA AS_UA AS_UB AS_UC AS_WB AS_WC Box2C Box2L Box2C Box2L Box2L Box2C Box4L BuiltUpl BUPGirder BUPGirder C C C C C C C C C C C C C C	 CHS ColdFormHat1 ColdFormHat2 Constants Cross3W Cross7P CrossW&2T CrossW&2TSt EA EN_IPE EN_IPN EN_IPN EN_L EN_L EN_L EN_L EN_L EN_L EN_L EN_U EN_U EN_UL Gen GL HP HP HPT HSS_Rect HSS_Rect Joist JoistsSJI L 	L LU M MC MT PFC PIPE POST RCBeamL RcBeamR RcBeamT RcColC RcColC RcColC RcColL RcColL RcColL RcColL RectBar RHS&2L RHS&2Leq RHS&2Leq SHS SS4S SHS SHS SHS	SPCb SqrBar T ST STube T STube T 12EA T 12EA T 12LU T 12LU T 12LU T 12LU T 12LU T 12LA T 12EA T	eacl	າ page
				OK Can	cel	
You can c help	reate customized sect	ion shapes and then i	insert them into this	s list. See the manual o	or the	

In the dialog window, select the desired type of profile and press OK. In this case, select the **aisiClip** profile.

New tab	le		
Name:	Roof]
Туре:	aisiClip.leo		
		ок са	ancel

Once the type of table is selected, a LEO file for the definition of the type of sections is assigned to the table.

Press the button to create a new item (section) for the current table.



Select the units system (English) and enter the values of the profile. In this case, enter the values shown in the figure. Do not forget to enter the name.

Note. - The name of a section should have the following format:

Type<space>description

For example, W 10x45, where W is the type and 10x45 the description.

The space character should be placed after the type name. A description of the section should be entered. For example 10x25, 10x15x2 (the "/" character is not accepted. It should be replaced by "_" (underscore) character)

Note - A section "Type" is determined by the characters entered before the space, e.g. W, C etc

Tip. - The Description of the profile should be self-explanatory containing the dimensions of the profile or other pertinent data.

Example of valid names:

ROOF 10X15X25

W 10X25

2L 15x2 unequal

Example of non-valid names:

W10x25 (space between Type and Description is missing)

W15/22 ("/" character is not accepted. Replace it with "_" (underscore) character)

15x22 (Type is missing)

Press OK and notice that a new section "Roof 3x6x1" has been created and saved into the sections database. A new "ROOF" group, which will contain all profiles of type "ROOF", has been created.

Important. - The Type of a profile determines the group in which this profile will be saved. Thus a "W 10x22" profile will be saved in a "W" group or type. In the same way a "TUBE 15x22" profile will be saved in a "TUBE" group. If the group does not exist, RAM Elements automatically creates a new group.

Remark. Note that the program already has a section with the name "Roof 3x6x1". The procedure described previously explains adequately the manner to perform this operation. It is recommendable that the user practices the creation of new sections for the structure in order to acquire proficiency in this task.

To assign the new section to the roof beams proceed as follows:



Text on each page

Select roof beams

92	RUUI SXOXI	U	U	U	
93	Roof 3x6x1	0	0	0	
94	Roof 3x6x1	0	0	0	
07	D40-04	-	0	0	¥
Available	sections				*
Group:	🧱 United States				*
Tables		Item	c		
- 🗐 R	Ream		Roof 3:	x6x1	
	(Dealli				
- 🖪 R	cCol				
	acCol IndBar				
	ccol IndBar Loof				
	icCol IndBar Icoof				
	ccCol IndBar Loof 45				
	icCol IndBar Icoof I45 PCa				
	cCol IndBar Coof 45 PCa				

Assign the section by pressing

15) Assigning materials

In this example, all material elements are of steel grade A36. To assign the material, proceed with these steps:

Select elements to which a material will be assigned. In this case, select all the elements of the



structure by pressing All

Nodes	Members	Shells	Areas	Conn	Gen			
18 1	A 🍒	$\sqrt{\frac{2}{4}}$		⊨ 1	a <u>+</u>		I	»» •
Materia	ıls							
Member	Material							^
1	A36							
2	A36							
3	A36							
4	A36							
5	A36							
6	A36							
7	A36							
8	A36							
9	A35 A36							
10	A36							
40	A00							~
Available	materials						A .	
Group:	🔤 United	d States					1	-
Tables]	Items				
- M	1SR		^	Δ3	6 (weight	less)		^
- 🕒 N	Jon-NA Dime	ension L	ur	- A3	6	Ċ.		
- 📰 B	lC			- A5	00 GrA pi	pes	_	
-I s	iteel			— <u> </u>	00 GrB re	ctangula	ar	
— EX T	'imber beam	1S		- <u> </u>	00 GrB ro	unded		
- (C) T	'imber colum	nns		- <u> </u>	00 GrC re	ctangul	ar	
			~	A5	00 GrC ro	unded		~

Go to the Spreadsheet Members/Materials. Double click on the desired material, or select it and press

Material "A36" from folder Steel has been assigned to all elements.

Note. - To show/hide the section and material names on the screen, press the Sections button and the Materials button from the *View tab*, *Model group*.

16) Articulated joints (pinned joints)

By default, all frame members are rigidly connected (fixed) to the nodes. This condition is appropriate to model a fully welded joint.

For joints that cannot resist flexural moments it is necessary to release the respective moments so the model adequately represents the real structure. An element is pinned when both ends of the members are released to both bending moments. To pin a member proceed as follows:



Select the members to be pinned. In this case, select DIAG1 and DIAG2 elements. To do this, select one DIAG1 element and one DIAG2 element. Then press



Hinge

Go to the Spreadsheet Members/Hinges (Releases) and press button both ends .





Elements have been pinned

17) Rotating columns



Pressing (in the *View tab*, *Model group*) the elements with three-dimensional sections are displayed. This allows the user to see whether the elements are orientated correctly in space or need to be rotated. If necessary, sections can be rotated as required. Tool buttons are available to rotate a member 90 and 180 degrees, or as required. In this case, the middle columns will be rotated by 90 degrees.



Text on each page





Columns in the middle will be rotated 90 degrees. To rotate 90 degrees, proceed as follows:

Text on each page



Select columns to be rotated

ę	눧 🤊	A D	Q (U))								
	Home	Spreads	sheet Vi	ew Proc	ess O	utpul	: Modu	les				
Clear		₹	() Help	/alue	÷ = *	-	Local	E ³ Local axis 3 ▼	Rotate	Rotate	Default	Rotate -90°
		Sp	readsheet					Act	ive spread	isheet too	ls	
Nodes	Nodes Members Shells Areas Conn Gen iv Image: Shells Areas Conn Gen											
Orientat	tion of local a	axes			0	Deg]						
Member	Rotation	Local	NX	NY	NZ	^						
28	90	0	0	0	0							
38	90	U	U	U	L	,						
								Ŧ	490.			
								Ro	tate			

Go to the Spreadsheet Members/Local axes and press button 90°.

18) Rotating beams 180 degrees

In this example, the elements shown below need to be rotated 180 degrees.



BEAM2, BEAM3 elements need to be rotated 180 degrees To do this, follow the next steps:



Select BEAM2 and BEAM3 elements (select one BEAM2 and BEAM3 elements and press button).



Go to the Spreadsheet Members/Local axes, and press button 180° to rotate 180 degrees.



Elements have been rotated 180 degrees.

Note. - Notice that it is possible to rotate the members by entering the required angle in the

spreadsheet and pressing \checkmark to fill the column with the entered value.

19) Entering loads

In this example, a 300 Lb/ft distributed force acting downward in the "Dead Load" case will be introduced. Concentrated forces of 1200 Lb, which are acting downward on the nodes, will be added as well.

Notice that RAM Elements automatically creates a load case named "Dead Load". Therefore, it isn't necessary to create it. Later the user will see how to create a new load case and a load combination.

Before entering a load, determine if it is a:

1) Load on node

Text on each page

- 2) Load on frame members, or
- 3) Load on shell elements.

Load on frame members

To enter loads on frame members, proceed as follows:



Select frame members where the load is acting. In this case, select beams on top of the truss.



Go to the Spreadsheet Members/Loads on members, select the adequate command tools by pressing

the <i>button and then press the</i>	button.	
Distributed load	? 🛛	
Direction ● Global axes ○ X ○ X ○ Y ○ Y ○ Z ○ Z	○ Local axes 1 -1 2 -2 3 -3	
Load type: Iniferm		lext on each page
Load magnitude (V): 0.3 [Kip/ft]		
Options:	Distribute over	
Replace existing loads	 Projected length 	
	OK Cancel	

Enter the value of the distributed load (do not enter the minus sign). Then press OK.



Text on each page

The load has been entered.

Load on nodes

To enter load forces on nodes, follow the next steps:



Select the nodes on which the force is acting.

Nodes Members Shells Areas Conn Gen									
(xy) 🏤 🐂 ዿ 🗣 🤧 🛃									
Force	Forces and moments [Kip]								
Node	FX	FY	Z	MX	MY	^			
7	C	-1.2	0	0	0				
25	C	-1.2	0	0	0	_			
27	C	-1.2	0	0	0				

Go to the Spreadsheet Nodes/Forces and moments, *enter a force value (enter the* -1.2 *value) and press* \checkmark *to fill-in the column.*



Notice that the force should include its sign. Forces on nodes have been entered each page

20) Creating Wind in X load case

The Second load case acting on the structure is due to the wind force in the X direction. These steps show how to create a new load case:



Execute the shown	button located	l in the Hom	ne tab, Load	l conditions	group to	enter a new	load	case.

I Load conditions 🛛 🖓 🔀								
r	Cases:							
	Num	ID	Description	Category	Duration 🔼 📑			
	1	dl	Dead load	DL	permanen 📃 🚆			
	2	WX	Wind in X	VMND	10 minute: 🛄			
					LC.			
					्य			
	<u> </u>							
					5			
					′			
					►			
	<				>			

Then enter a load condition identifier consisting of 2-4 characters (first character should not be a number), then enter a load description and the category. In this case, enter what is shown in the figure. Press OK and the new load case in the drop-down list will be shown.



Drop-down list for load cases at the status bar.

Note that it is necessary to select a category. This feature is very useful to generate load combinations based on their categories. The user can create a template file for the local building code from which load combinations can be generated (based on the load case category, DL for dead loads, LL for live loads, etc.). The program is capable to generate load combinations according to the design standards it handles and it has example files (located at main RAM Elements directory/*combos*) which have the basic load combinations to consider for the different codes. For more details see the chapter of Other Advanced Subjects in the program manual.

21) Entering wind loads

In this case, wind loads are applied perpendicular to the roof. There is a pressure of 150 Lb/ft on the left side of the roof, and a suction of 200 Lb/ft on the right side of the roof. Wind coad entry is similar to the entry made before for the dead load condition. Notice that the distributed forces act perpendicular to the elements, not parallel to Y-axis. To enter these loads, proceed as follows:



Select the elements on which the load acts. In this case, select one member of each portal and press to select the aligned elements.

Home Spreadsheet View Process Output Modules									
▼ ▼	Image: Distributed load towards Image: Distributed load towards Image: Distributed load towards Image: Distributed load towards Image: Distributed load perpendicular to the element in Image: Trapezoidal distributed l Image: Distributed load towards Image: Distributed load perpendicular to the element in Image: Trapezoidal distributed l Image: Distributed load towards								
Spreadsheet	Active spreadsheet tools								
Nodes Members Shells Areas Conn Gen Distributed load perpendicular to the element in -2 i I Image: Conn Image: Conn									
Member Distributed 1									
1-Dir. Cat Value1 Value2 % Dist1 % Dist2 2	2-Dir. Cat								
2 0 0 0 No 0 No 0									

Go to the Spreadsheet Members/Loads on members and press the witton.

Distributed load		? 🗙
Direction Global axes	 Local axes 1 -1 2 -2 3 -3 	
Load type: Uniform		Text on each page
Options: Add to existing loads Replace existing loads 	Distribute over ● Real length ● Projected length 	Cancel

Enter the value of the distributed force (do not enter the minus sign), and press OK.



The distributed forces of the left side of the structure have been entered.



To enter the forces on the right side of the structure proceed as before. The load should be seen as illustrated in the figure.

Notice that it is necessary to press button \checkmark instead of button \checkmark to enter suction.

22) Creating load combinations

In this example, the following load combination will be created:

1.1dl + 1.2wx (1.1 times dead load plus 1.2 times wind in X)

To create it proceed as follows:



Execute the shown button located in the Home tab, Load conditions group to enter a new load case. Fill the combination equation factors in the second spreadsheets in the dialog window that appears.

I	Load	d condi	tions				?	×	
	Num	s: ID	Descript	ion		Category	Duration \Lambda 🗖		
	1	dl	Dead los	ad		DL	permanen	2	
	2	wx	Wind in 3	x		WIND	10 minute:	3	
								٩.	
								7	
								Ľ	
							2	₽	
								4	
								<u> </u>	
							Tai		
							l ex		i each page
		 a)							
		9							
ſ	<mark>Somb</mark> Formu	binations: _{nula:} c1 = 1.1dl+1.2wx				ר	۵ 🕫		
	Num	ID	dl	WY	Type			-	
	1	c1	1.1	1.2	Not assigned			<u>e</u>	
N	_					J	- 3	e	
								π.	
								×	
							~		
						ОК	Cancel		

In the dialog window, enter the information shown in the figure.

a) Enter a load condition identifier of two to four characters (the first character should not be a number).

b) Enter the formula factors for the load combination (1.1 for dl and 1.2 for wx).



Then press OK and the new load combination in the drop-down list for load cases at the status bar will be shown.
Notice that the formula factors can contain the minus sign. For example, a -1.2 factor for wx load case would define "1.1dl -1.2wx"

Note. - It is not possible to enter or edit loads data while a load combination is selected as the current load condition. Notice that the spreadsheets are locked for edition to enter loads.

23) Analyzing the structure

After the structure has been defined, the model is ready to be analyzed, designed, optimized and the results can be viewed.

To analyze the structure proceed as follows:



Text on each page

Execute Analyze model by pressing the shown button in the Process tab, Process group.

For this example a Second Order Analysis (P-Delta) will be performed. This analysis takes longer to analyze a structure as it involves iteration, but it is more accurate. In addition, buckling instability is detected in certain cases when P-Delta analysis is performed. For more about P-Delta analysis, see the Chapter of Analysis in the Manual.

Analysis	? 🔀		
Analysis Condition Finite element model Model	DOF		
-Dynamic and Response Spectra			
Number of modal shapes to calculate:	10		
Method for modal superposition			
● CQC ○ SR5S ○ AE	is 👘		
Seismic analysis results with sign			
Disable shear deformations			
Perform second order analysis (P-Delta) Ignore members with loads along their span			
Incremental/Iterative Analysis Configuration			
 Method Standard Newton-Raphson (NR) 			
O Modified Newton-Raphson (MNR)			
Number of increments:	1		
Number of iterations per increment:	10		
Convergence tolerance:	1E-3		
Cancel analysis if any of the load cases fails to converge			
Current values as default			
Analyze	Cancel		

Text on each page

Select the same options as shown in the figure above. Then press the Analyze button.

24) Designing the structure



Select the command Design all in the Process tab, Process group.

After that, a dialog window will be shown to specify the design standard to be used in the design of members. For this example select AISC 360-05, AISI-01 (ASD) for steel members. The other material options may use the code defined by default.

Design			? 🗙
Design code	Seismic	design	
-Steel members-			
Hot	olled	AISC-05 ASD	-
Cold	formed	AISI-01 ASD	-
Use MBMA special considerations			

Text on each page

Select the design code shown above and press the Design button.

Once the elements are designed, the user has the option to optimize the sections with the following command.



Select the command Optimize model in the Process tab, Process group.

For this example the optimization is not performed.

25) View results graphically

As can be seen, several buttons (in the *Analysis* and *Design* groups of the *View tab*) are enabled once the structure has been analyzed and designed. These newly enabled buttons allows the user to select what results to display.



Result buttons from the Analysis and Design groups are enabled when the structure has been analyzed/designed.

In order to see results graphically, press the button corresponding to those desired items, and then select the elements to see the results.

Notice that the selected display options will only be seen on the selected elements.

	- dl=Dead load	
	 wx=Wind in X 	
English - Conditions:	🔹 dl=Dead load 🔤 🔤	a^ ar 🔒 🔍 🔍 🛟

Select the load condition.

26) Deformed shape

One of the first display options that should be viewed is the deformed shape of the structure.



In this view the elements are drawn as lines. To see the deformed shape with the original shape select the option with original shape.

27) 3D Sections Deformed shape

It is possible to see the deformed shape with the extruded sections.



Activate the deformed shape accessing the option from the Rendering button menu (*View tab, Model group*). Notice that this view may take a longer time to draw.



The graphic shown corresponds to the Dead load case.

Text on each page

28) Stress

Another important view option is the information related to the element stress contour. This is of particular importance in light gage structures where stress concentrations are significant to the design.



Press the stresses button to see frame member stresses. Note that the button has a menu where there are some options to see the stresses in the deformed shape or in shrunken members.

To select only those elements that are stressed within a certain range, mark a block of stresses with the mouse and press

Stresses Members and shells [Kip/in2]		
7.80 6.75 5.71 4.66 3.62 2.57 1.53 0.48 -0.56 -1.61 -2.65 -3.70 -4.74 -5.79 -6.83		
-7.88 Smooth Env		
Ang (')		
Von Mises 💙		

Text on each page

To see element stresses within a certain range, mark the range and press



RAM Elements selects those elements whose maximum stress is within the marked range of stresses. Note that the remaining members are recalibrated (color changes).

Note. – To see only the axial stress (without bending moments, press \checkmark Axial stress).

29) Stress and deformation



Text on each page

💳 Stresses 💌		
Deformed shape		
	Shrunken members	
Ø	Transparent	
€	Lines only	

To view stress and deformation of the elements, activate these buttons



30) Forces diagrams

The buttons shown above (View tab, Analysis group, Member forces displayed menu) allow the user to see the forces diagrams of the frame members:

Bending moment around element axis 3 (Typically strong axis bending)



Bending moment around element axis 2 (Typically weak axis bending) Text on each page



Shear forces in element axis 2 (typically weak axis) (Dead load case)



Shear forces in element axis 3 (typically strong axis) (Dead load case)



Text on each page





Axial forces (Dead load case)



Select the Show values option (Member forces menu) to simultaneously display the magnitude of the forces.



Select Show units option (menu displayed for units at the status bar) to display the units.



31) Displacements of nodes

To see the nodal displacement values, press choose the degree of freedom to be viewed:

View tab, Analysis group) and

Tx
Ту
Tz
Rx
Ry
Rz

The relation between a degree of freedom and its respective displacement in the global coordinate system is as follows:

1: Tx: X translation

- 2: Ty: Y translation
- 3: Tz: Z translation
- 4: Rx: Rotation about X
- 5: Ry: Rotation about Y
- 6: Rz: Rotation about Z

Note. – Notice that X, Y, and Z represents the global coordinate system.

Each element has its own system of coordinates, named local axes. These axes are designated with the numbers 1, 2 and 3, which are equivalent to X, Y, and Z-axis. Local axes are Cartesian and follow the right hand rule. To see the local axes, press $\frac{1}{2} \frac{1}{4} \frac{1}{2} \frac{1$



Press Nodal displacement and the degree of freedom corresponding to the displacement.

To see the displacement units press the Show units option in the Units menu at the status bar.



32) Reactions

To see reactions, press $\overset{\text{Leactions}}{\longrightarrow}$ (*View tab, Analysis group*) and the degree of freedom corresponding to the desired action.

Τx
Ту
Tz
Rx
Ry
Rz

This is the relation between degree of freedom and force:

- 1: Tx: X force
- 2: Ty: Y force
- 3: Tz: Z force
- 4: Rx: Moment about X
- 5: Ry: Moment about Y
- 6: Rz: Moment about Z



Press $\stackrel{\text{Reactions}}{\longrightarrow}$ and the degree of freedom that corresponds to the desired reaction. (*Case: Wind in X*)

33) Deflections

One of the most important results of an analysis is the ratio between deflection and length of the element. To view this ratio press \checkmark Deflections \checkmark (*View tab, Analysis group*).

Defl/L	
Defl. in axis 2	
Defl. in axis 3	
Defl=f(L) in axis 3	
Defl=f(L) in axis 2	

Options displayed in the Deflections button menu.

This ratio may vary across an element. RAM Elements displays the maximum ratio found within an element.

Note. - The Defl/L ratio should never exceed a value suggested by the design code and judgement.



Text on each page

Press \smile Deflections \checkmark *to see the element colored Defl/L ratios.*

Maximum Defl/L Deformation/L in 2 and 3		
	4.683E-04 4.234E-04 3.785E-04 3.336E-04 2.887E-04 2.438E-04 1.989E-04 1.540E-04	
	1.091E-04 6.41/E-05 Not designe	d

In this panel mark a range of Defl/L ratios and press to select the elements that have slopes within the marked range.

34) Deflection values

To see the Deflection values (in function of L) in local axis 2 direction, selection

```
option Defl=f(L) in axis 3
```

To see the Deflection values (in function of L) in local axis 3 direction, press

```
button Defl=f(L) in axis 2
```



Deflection in function of L for the Load combination C1.

35) Design: Colored Interaction Values To view interaction values graphically, by color, press Stress ratio (View tab, Design group). Important! To view the interaction colors scaled from 0 to 1.0, press Stress ratio . To view the controlling interaction value for all Load Combinations (not load cases) press Stress ratio Weximum stress ratio Stress ratio For the controling combination Text on each page

Press

to see interaction values.

To select the elements with stress ratio within a certain range, mark a range of stress ratios and press

Stres: AISC/	s ratio AISI/NDS	
	0.69	
	0.61	
	0.54	
_	0.46	
	0.38	
	0.31	
	0.23	
	0.15	
	0.08	
	5.777E-04	

Mark a block with the mouse and press button to select elements with stress ratio within the range.

Note that most of the results displayed to this point are for the selected load condition.

36) Design: Interaction Values

To see interaction values for the currently selected load condition, press



Choose the Stress ratio option to view interaction values for the current load condition. The last option of the menu should be enabled to see the ratios for the governing load combination.

37) Design: OK and NG (No Good) elements



To view the elements that failed code check (for the current load condition), press *Design group*):



×



Text on each page

Press button to see elements that pass code check.



Press button Market to quickly select all elements that failed code check.

The user can print the results of the steel design in a report. To print them, go to the *Output tab*, *Reports group*. For more information about reports see the Printing Graphics and Reports Chapter in the manual.

The user can also use the optimization feature that is valid only for steel and wood members. This option allows the user to change the existing sections with sections that are recommended (based on explicit criteria) from a collection of sections. In other words, the original section can be replaced with another that resists the imposed loads with an allowable deflection and that is located above the original section in the list of sections specified for the optimization. To use the optimization feature go to the *Process tab, Process group, Optimize model command*. For more details see *Chapter 11: Steel and Wood Structure Optimization and Code Check* of the Manual.

YOUR PACKAGE CONFIRMATION

Test for GreenYellow Color

<u>Test for Honeydew Color</u>





Test for Ivory Color

NAME: Test for Khaki Color

Test for Lavender Colo

EMAIL ADDRESS: der Blush Color

Test for LawnGreen Color CUSTOMER NUMBER:

Test for LightBlue Colo

PACKAGE ID: LightCoral Color

Test for LightCyan Color ISSUE DATE: Coldenrod Yellow Color

EXPIRES: Test for LightGray Color Test for LightGreen Color

PACKAGE INFORMATION: Test for LightSalmon Color

Test for LightSeaGreen Color

Important Information

ext on each page

We ask that you confirm your travel dates with us at least 45 days prior to arrival. You may reschedule your reservation once without charge as long as the request is made at least 45 days prior to the date of your arrival. For special events, peak seasons and major holidays allow 60 days.

All resort destination accommodations are provided at nationally recognized hotels such as The Radisson Hotel or other comparable properties. While in Florida you and (spouse/traveling companion) will be required to attend tours about ownership at our partner timeshare resorts. It takes part of a morning or afternoon and it's fun and informative.

*Travelers must meet the requirements within 18 months from the date of purchase.

*If married or cohabiting quest must have matching identification (matching IDs, passport, bill) with the same address.

*Full time students and those who are unemployed do not qualify for this offer.

INITIAL

ADDRESS: Test for LightSkyBlue Color Test for LightSlateGray Color

CITYest for LightSteelBlue Color

est for LightYellow Color

STATE/PROVINCEIme Color

Test for LimeGreen Color POSTAL CODE: Linen Color

Test for Magenta Color COUNTRY: Test for Maroon Color

Test for MediumAquamarine Color TIME ZONE: Test for MediumBlue Color

PHONE I. Test for MediumOrchid Color Test for MediumPurple Color

PHONE 2-MediumSeaGreen Color Test for MediumSlateBlue Color

Test for MediumSpringGreen Color PURCHASE INFORMATION:

You have authorized the purchase of the above Trest of oill Allet dither Vierdie treed lister bbelow.

AMOUNT:

MERICAN

EXPRES

PURCHASE DATE:

DISCOVER

AUTI49RIZE MEREghtBlue Color

Fest for MintCream Color

Important Information

*These promotional holidays are provided for Married or cohabiting couples 25-75 and single females 28+ unless otherwise stated. These vacations are fully transferable to anyone who meets the promotional requirements within 18 months.

ext on each page

If you have trouble dialing the number please contact your operator and have the operator dial the number.

Additional numbers are available on our Website.

Our Goal is to provide you a VIP Hospitality Travel Experience so enhanced, and so wonderful you will always reach out to us plan & book all of your future holidays.

Seller of Travel Registration No.

This document is powered by

is not operated by, or affiliated with, Walt Disney World® Resort, Universal Orlando® or SeaWorld®. As to Disney artwork/properties: ©Disney. Universal elements and all related indicia ™ & ©Universal Studios. All rights reserved. All SeaWorld© images are a copyright of SeaWorld Parks & Entertainment.

Page 10

TOTAL:

EXP DATE:

VISA

CARD NUMBER:

Test for GreenYellow Color

Example í: Steel

Test for IndianRed Color

Test for Indigo Color Inis example will explain step by step the creation of a basic 3D steel structure. This example will be most effective if the user practice the illustrated skills as they are presented.

The structure to be entered in this example is shown below:



Text on each page

In order to simplify data entry, frame members are grouped as follows:



Test forheight Smaroeth Coloride to the scriptions shown here will be illustrated in this example.

1) Starting a new structure



Select New from the RAM Elements button menu.

In the event that there is an existing model open, RAM Elements will ask to save it.

Texanipis Light/SkyBlue Color

Test fo	r LightSlateGray Color
	Metric
Tes	r _{Ehight} SteelBlue Color
Test f	o <mark>T LightYellow Co</mark> lor
	Show units
Ū	est for Lime Color
Tese	For LimeGreen Color

Press the button on the status bar, a menu will be displayed. Then, select the option Units configuration.

	50	
	Units	? 🛛
	Test for Maroon Color Units system:	English 🗸
Test	or MediumAquamarine Co	ler 🗸
т	estationdMediumBlue Color	in 🗸
Те	sReiffermeandhearchid Color	in2 🗸
То	Material Strengths:	Kip/in2
Ie	Unit Weights:	Kip/ft3 🗸
Tes	for MediumSeaGreen Colo Temperatures:	F V
Tes	for MediumSlateBlue Colo Translational Springs:	Kip/in 🗸
Test f	୦୫୦MoclisymSpringGreen Co	🥠 🗸
Test	Pointweet Ennesurquoise Col	₩ip 🖌
Tes		Kip/ft 🗸
	Surface/Area Loads:	Kip/ft2 🗸
10	Stresses:	Lb/in2
	est for MintCream Color Soil stresses:	Lb/in2
	Moments:	Kip*ft 🗸
	Deflections:	in 💌
	Others:	Kip-ft 🗸 🗸
	Default units for system:]
	English SI	Metric
	ſ	OK Cancel

Text on each page

Select the English default unit system in the window displayed.

Test for Moccasin Color 2) Entering node coordinates

Test In the coordinates spreadsheet enter the coordinates shown below:



Go to the Spreadsheet Nodes/Coordinates and enter the coordinates shown above.

Test for RelevioletRed Color	(24, 16.5, 0)
Test for Mapa ya Whip Color	(24, 12, 0)
Test for PeachPuff Color	
Test for Peru Color	(24, 0, 0)
Test for Plum Color	

The entered nodes are shown on the screen.

Test for PowderBlue Color

3) Generation of frame members

Teststerc Revalative of the frame members. Select the nodes in the sequence shown below, and then

Test for RosyBrown Color



connect the selected nodes by pressing with memb

• 3	• 4	• 5
• 2		• 6
• 1		• 7

Select the nodes in the order shown. To select several nodes remember to press the SHIFT key while clicking with the mouse.

Test for SaddleBrown Color

Test for Salmon Color	
Nodes Members Shells Areas (Conn Gen
i 🖬 🎽 🖓 🔶 ⊨	🐚 🛨 🛱 🌷
Connectivity and description	
Teethie Descrathell Color	NK Tribut, width Brace
Test for Sienna Color	

Go to the Spreadsheet Members/Nodes and Description Test for SkyBlue Color

As can be seen the frame members were generate.

Test for Teal Color NOTE. - Remember that it is possible to undo the last operation by pressing

Test for Tan Color

Test for Thistle Color 4) Assigning a description

It **Is steeded** and **the second second and a selection** of the second se elements, optimization, and others. To assign the same description to every member of a group rest for Transparent Color proceed as follows:

Text on each page



Select columns

4

Test for Violet Color

Test-for-Wheat-Colo Test f or YellowGreen Color COL 1 COL 1 Test for Black Font Test for₊Brown Font DESCR Test for Navy Font (additive) TestsfordBluetGolor st oseBinderodor **Test for Purple Color** Column (additive) COL +1 Test for Red Color

Example 1: Steel

Text on each page

Then assign the description to the selected members selecting the Column (additive) option.

Note. – To view the member descriptions graphically (on the screen) go to *View tab*, *Model group*, press the properties button and select the option *Description by element* from the menu displayed. Test for Antique White Color Repeat the steps explained previously to assign a Description to the other members:



Select members



Assign the description to the selected members selecting the Beam (additive) option.

Generate the beam as shown in the figure below. Assign BEAM2 description to this newly created member:

Example 1: Steelsque Color

es	t fer Blanched Almend Cel
	Test for BlueViolet Color
1	O [est for BurlyWood Color
	Test for CadetBlue Color
	I date ibide color
	est for Chartreuse Color

Test for Chocolate Color

Test for Coral Color

Connect node: To create the horizontal beam, select the nodes shown in this figure and press with members Test for Cornflower Blue Color

Test for Cornsilk Color Test for Crimson Color
Test for Cyan Color
est for Dark Cyan Color

Test fog Darkdewhite in Roche order cted members selecting the Beam (additive) option.

5) Segmenting Members

To segment frame members, follow these steps: Test for DarkGreen Color

```
Test for DarkKhaki Color
BEAM 1
```

Test for DarkMagenta Color

```
Test for DarkOliveGreen Color
```

```
Test for DarkOrange Color
```

Select members to be segmented.

Segment members		×						
Number of segments per member:	3							
Options								
 Segment members 								
 Create nodes only (maintain physical members) 								
ОК	Canc	el						

Test for DarkRed Color



Press And enter the desired number of segments. In this case, enter six segments. Then press OK or the ENTER key.



NOTE. - Remember that it is possible to undo the last operation by pressing

Notice that the segmented members have the same description as the original member and that each member is treated as one physical member.

6) Generation of vertical members

To enter the vertical truss elements, follow these steps:



Text on each page

Select the nodes shown in this figure. Notice that it is not necessary to select the exterior nodes.



Press button Y to generate vertical members (plus y generates members in the vertical up direction).

7) Generation of diagonal members

At first, generate the diagonal truss web members of the left side of the structure, and then the right side.

Diagonal members on the left side:



Select the nodes in the order shown in this figure.



Text on each page

```
11
```

Alternately connect

Press nodes with members from the ribbon (the button is visible when the Members tab is the current page in the spreadsheet and connectivity button is pressed).

To enter diagonals on the right side proceed in the same way.

NOTE. - Remember that it is possible to undo the last operation by pressing **1**.

The differences between the two buttons are as follows:



This button connects the selected nodes in a continuous line.

This button connects alternate pairs of nodes with a fragmented line. That is, the first member is generated between the first pair of selected nodes, the second member between the second pair of selected nodes, etc.

8) Assigning a Description to members

Follow these steps to assign a Description to the internal web members:

a) Select diagonal and vertical (internal) elements using the button [] (*Home tab, Selection group*)



To select the elements select one member of each group and then press. Remember that this button selects elements with a common description. In this case all internal elements belong to the group that does not have a description yet. That is to say they all have the same empty description.

b) Internal elements will be assigned a DIAG1 description. Since there is no button available to assign this description (as opposed to COL1 and BEAM1 buttons), it is necessary to enter it manually:



Enter DIAG1 description and then press located at Spreadsheet tab, Spreadsheet group, to fill the column with the value. Another way to do this would the access to command from the popup menu displayed after right click on the spreadsheet area, having selected the desired rows to fill previously.

Important - Descriptions are very important to select groups of frame members. It is also important to have entered the descriptions correctly. If this has not been done correctly the user may experience some difficulty following the next steps in this example.

9) Copying the structure

It is advisable to enter all the descriptions of a structure before copying it, because when a structure is copied the Descriptions are also copied.

To copy a structure, follow these steps:





Select all the elements that should be copied. In this case, press (Home tab, Data tab) to select the entire structure.

Execute the **Copy** command (*Home tab, Modeling group*).

e 🖻 🖬 🤊 A	R Q (U)		RAM	v
Home Sprea	dsheet View	Process Out	put Modules	
Explorer Spreadsheet	All		Delete	
Data	Selection		Modeling	
Copy elements	? 🔀			
Number of copies: 2				
Distance			Text	on each page
DeltaX: 0	[ft]			
DeltaY: 0	[ft]			
DeltaZ: 18	[ft]			
ОК	Cancel			

Enter the number of copies and the distances in X, Y, and Z between each copy. In this case, enter the values shown in this figure. Then press OK.



10) Generation of the roof beams (purlins)

To generate the roof beams, follow these steps:



Text on each page

Select the initial nodes or end nodes of the roof beams.



Then press **Z** (*Press button* **-Z** *if nothing occurs*). *Note that the* +/- *refers to the direction that the members are projected.*



Note. - Notice that the middle portal is not connected to the roof beams. The model can be left without making any changes and the program will interpret the roof beams as continuous physical members. However, if the roof beams are going to be modeled as simply supported beams (as they normally are), it is necessary to segment the beams and connect one end to the middle portal. The



Text on each page

Notice that roof beams do not connect with the middle frame



With the roof beams selected, press selection to split roof beams and connect them with the middle portal. It is necessary to select the member and the node of the middle portal that will become the point of break.



Text on each page

Choose the options shown in the figure. The members will be divided into two physical members.

11) Assigning a Description to roof beams

To assign a Description to roof beams, proceed as follows:

a) Select roof beams by description.



Select a member of the group and then press. Since the selected element does not have a description, all members with empty description will be selected.

b) ROOF1 description will be assigned to roof beams. There is no button available to automatically assign the description (as opposed to COL1 and BEAM1 descriptions). Therefore, the Description has to be entered manually:

Nodes	Members	Shells	Areas	Conn	Gen			Nodes	Members Shells	Areas	Conn	Gen		
$^{\circ}$ N	👍 🍃	₽. Na	∳ ⊮	= ``g	<u>+</u> A	² I		ĸ	1 in 12 .	∳ ⊮	= ``g	<u>+</u> A	² I	
Conne	ctivity and de	escription	n					Connect	tivity and descriptior	٦				
Membe	r Description	-	NJ	NK	Tribut. width	Brace	^	Member	Description	NJ	NK	Tribut. width	Brace	^
82	Roof1		5	54	0	No		82	Roof1	5	54	0	No	
83			8	23	0	No		83	Roof1	8	23	0	No	
84			7	25	0	No		84	Roof1	7	25	0	No	
86			6	52	0	No		86	Roof1	6	52	0	No	
87			9	35	0	No		87	Roof1	9	35	0	No	
88			11	37	0	No		88	Roof1	11	37	0	No	
91			21	56	0	No		91	Roof1	21	56	0	No	
92			23	24	0	No		92	Roof1	23	24	0	No	
93			25	27	0	No		93	Roof1	25	27	0	No	
94			10	26	0	No		94	Roof1	10	26	0	No	
95			29	50	l n	No		95	Roof1	29	50	0	No	
								96	Roof1	35	36	0	No	
								97	Roof1	37	38	0	No	
								101	Roof1	70	~¥	on d	All h	nago
								106	Roof1	52		011 6		paye
								111	Roof1	54	21	0	No	
								116	Roof1	56	22	0	No	
								120	Roof1	26	28	0	No	

Enter ROOF 1 under description and then press \checkmark to fill the column with the entered value.

Generating DIAG2 and BEAM3 members

Now proceed to enter the DIAG2 and BEAM3 members that are shown in the figure below. Generate these elements as explained before.



12) Supports

To enter supports proceed as follows:



Select support nodes

e 🖻 🖬 🤊 A R Q	9					
Home Spreadsheet	View Process Outpu	it Mod	lules			
	Value	A.		x 📌		Fixed
Clear 📜 🗱 👪	$+ - \times - = \cdots$	Pinieu	in X	in Z	X and Z	Fixeu
Spreadsh	eet			Active spr	readsheet t	ools
Nodes Members Shells Areas (x,y) ☆ ☆ ☆ Restraints Node TX TY TZ RX RY RZ 1 ✔ ✔ ✔ ✔ ✔ 2 ✔ ✔ ✔ ✔ ✔ 17 ✔ ✔ ✔ ✔ 19 ✔ ✔ ✔ ✔ 24 Ø Ø Ø Ø	Conn Gen			Text	t on ea	ach page

Go to the Spreadsheet Nodes/Restraints and click on the corresponding support. In this case click on





The Supports have been entered

13) Assigning sections to frame members.

To assign a section to some member, and this section is available in the section database, proceed as follows:

Select the members to which a section will be assigned. In this case, select all the columns.



Nodes	Members	Shells	Areas 🛛 C	onn 🗍 🤇	Gen				
ik 🛛	F	Vi -	🗗 🕨	9	<u>+</u>	₽ ¹	I	» •	
Sections	;								
Member	Section		lg factor	d0	dL			^	
1	W10x12		0	0	0				
6	W10x12		0	0	0				
28	W10x12		0	0	0				
29	W10x12		0	0	0				
38	W10x12		0	0	0				
39	W10x12		0	0	0				Text on each na
ñuailable	sections					-			
Avaliable	Sections						<u>^</u> [₂	E	
Group:	📕 United	d States					~		
Tables			Ite	ms					
- O T	S		<u>~</u> –	🗋 W 8	X58		-	^	
	LIBE			1 W 8	X67		_		
	/			W 1	0X12			-	
-i v	/T			W 1	UX15				
⊢⊣⊢ ×	2L		-	🔄 W 1	0X17				
				_] ₩ 1	0X19				
			~	W 1	0X22			~	
				1 00 4	0226		_		

To do this, first select one column and press.

Then go to the Spreadsheet **Members/Sections**. Select W10x12 profile and press \bigtriangleup (double click on the profile will assign the selected item).

Assign sections to all members of the structure in a similar manner.



To select all the elements of the truss, select one element of each group and press.

Nodes	Members	Shells	Areas	Conn	Ge	en						
ĸ	1	v ¹² - (₽ ►	9)	+	a ² ¹	I	» •			
Sections												
Member	Section		lg factor	d0	- (dL			^			
2	T2L 2-1_	2x2-1_2x1	()	0	0						
3	T2L 2-1_	2x2-1_2x1	()	0	0						
4	T2L 2-1_	2x2-1_2x1	()	0	0						
5	T2L 2-1_	2x2-1_2x1	()	0	0						
7	T2L 2-1_	2x2-1_2x1	()	0	0						
8	T2L 2-1_	2x2-1_2x1	()	0	0						
9	T2L 2-1_	2x2-1_2x1	()	0	0						
10	T2L 2-1_	2x2-1_2x1	()	0	0						
11	T2L 2-1_	2x2-1_2x1	()	0	0						
12	T2L 2-1_	2x2-1_2x1	()	0	0						
13	T2L 2-1_	2x2-1_2x1	()	0	0						
	TOL 0.4	0.00 A - 0.04			0	0			<u> </u>			
Available	sections							▲	â			
Group:	🛄 Unite	ed States						•	~			
Tables			It	ems								
	Tube		~	—] т	21.2	X2X5 1	6X3 4		~			
-16 1	2L			- 1	2L 2	-1 2X2	-1 2X	14				
╞╼┲╴╤	2LU			T:	2L 2	-1 2X2	-1 2X	1 4X				
- O T	s			— Т	2L 2	-1 2X2	-1 2X	1_4X				
−Ō ⊺	UBE			-🗋 т;	2L 2	-1_2X2	-1_2X	1 2				
⊢Ī v	v			-🛅 т;	2L 2	-1_2X2	-1_2X	1_2X				
∣⊢∓ v	VT			-🗋 т;	2L 2	-1_2X2	-1_2X	1 2X				
ب لحف ا	2		×	<u>т</u> т	01.0	1 202	1 22	5 12	~			

Text on each page

Assign section $T2L 2 \cdot 1 \cdot 2x^2 \cdot 1 \cdot 2x^2$

Now assign sections to the DIAG2 and BEAM3 elements



Select the elements DIAG2 and BEAM3
Nodes	Members	Shells	Areas C	onn	Gen		
K	1	v ^{†2} →	🗗 🕨	9	<u>+</u>	ana international distribution of the second secon	>> •
Sections	;						
Member	Section		la fector	40	d		
89	T2L 2x2x	1 4	0	0	0		
90	T2L 2x2x	14	0	0	0		
98	T2L 2x2x	14	0	0	0		
99	T2L 2x2x*	14	0	0	0		
100	T2L 2x2x*	1_4	0	0	0		
102	T2L 2x2x*	1_4	0	0	0		
103	T2L 2x2x*	1_4	0	0	0		
104	T2L 2x2x*	1_4	0	0	0		
105	T2L 2x2x*	1_4	0	0	0		
107	T2L 2x2x*	1_4	0	0	0		
108	T2L 2x2x*	1_4	0	0	0		
400	TOL 0.00		0	0	0		
Available	sections					-	
Group:	🛄 Unite	d States					*
Tables			Jte	ms			
	Tube			1 T2L	2X2X1_4		
	2L			T2L	2X2X1 4	X3 8	
	2LU		-	T2L	2X2X1 4	X3 4	
- O T	5			T2L	2X2X1 8	_	
-Ō T	UBE			T2L	2X2X1 8	X3_8	
I-Ī w	/			🚺 T2L	2X2X1_8	X3_4	
⊢Ŧ w	/т			🚺 T2L	2X2X3_1	6	
L v	0		~	TOU		2V0 0	$\mathbf{\mathbf{v}}$

Assign section T2L 2x2x1_4

14) Adding sections to the database.

In this example, a cold-formed C-section will be assigned to the Roof beams. This cold-formed C (with lips) profile is not available in the section database. Therefore, a new section should be added. Proceed as follows:



Go to the Home tab, Databases group and execute the Sections button

Sections		? 🔀	
Group: 🔤 United St	ates		
Tables	Items	<u>B</u>	
– C aisiCS	aisiCS 4x2x059		
_ ∩_ aisiHU	aisiCS 4x2x065		
- JC aisiILip	aisiCS 4x2x070		
– 💪 aisiLS	aisiCS 4x2x085		
- L aisiLU	aisiCS 4x2x105		
$-\mathbf{C}$ aisiSSMA_S	aisiCS 4x2.5x059		
- C aisiSSMA_T	aisiCS 4x2.5x065		
-J aisiZS	aisiCS 4x2.5x070	부	
- JaisiZU	aisiCS 4x2.5x085	Fext O	n eacn page
	aisiCS 4x2.5x105		
Cost	aisiCS 4x4x059		
- 😽 G1-1_2L	aisiCS 4x4x065		
- 🔛 G1-3_8L			
		855 M=	
		AL	
		z +	
		~	
aisiClip.leo			
		Close	

Press the button to add a New group to the database. After that, a name for the new group is required in the displayed window:

New group	×
Name:	
MyGroup	
OK Cancel	

Then, add a new Table by pressing the 📕 button. A new dialog will be displayed to enter the name

for the new table. It is also required to select the type of table, to perform this action press the button and the following dialog will be shown:

Select section	type			[?×	
Available section	types					
Y 2B2L Y 2C_1 Y 2C_0 T 2L&PL Y 2L&VPL II 2Ws II 2Ws	AS_CHS AS_CHS AS_FC AS_PFC AS_RHS AS_SHS AS_TEE AS_TFB AS_UA AS_UA AS_UA AS_UA AS_UB AS_UC AS_WB AS_WC Box2C Box2L Box2C Box2L Box2L Box2C Box4L BuiltUpl BUPGirder BUPGirder C C C C C C C C C C C C C C	 CHS ColdFormHat1 ColdFormHat2 Constants Cross3W Cross7P CrossW&2T CrossW&2TSt EA EN_IPE EN_IPN EN_IPN EN_L EN_L EN_L EN_L EN_L EN_L EN_L EN_U EN_U EN_UL Gen GL HP HP HPT HSS_Rect HSS_Rect Joist JoistsSJI L 	L LU M MC MT PFC PIPE POST RCBeamL RcBeamR RcBeamT RcColC RcColC RcColC RcColL RcColL RcColL RcColL RectBar RHS&2L RHS&2Leq RHS&2Leq SHS SS4S SHS SHS SHS	SPCb SqrBar T ST STube T STube T 12EA T 12EA T 12LU T 12LU T 12LU T 12LU T 12LU T 12LA T 12EA T	eacl	າ page
				OK Can	cel	
You can c help	reate customized sect	ion shapes and then i	insert them into this	s list. See the manual o	or the	

In the dialog window, select the desired type of profile and press OK. In this case, select the **aisiClip** profile.

New tab	le		
Name:	Roof]
Туре:	aisiClip.leo		
		ок са	ancel

Once the type of table is selected, a LEO file for the definition of the type of sections is assigned to the table.

Press the button to create a new item (section) for the current table.



Select the units system (English) and enter the values of the profile. In this case, enter the values shown in the figure. Do not forget to enter the name.

Note. - The name of a section should have the following format:

Type<space>description

For example, W 10x45, where W is the type and 10x45 the description.

The space character should be placed after the type name. A description of the section should be entered. For example 10x25, 10x15x2 (the "/" character is not accepted. It should be replaced by "_" (underscore) character)

Note - A section "Type" is determined by the characters entered before the space, e.g. W, C etc

Tip. - The Description of the profile should be self-explanatory containing the dimensions of the profile or other pertinent data.

Example of valid names:

ROOF 10X15X25

W 10X25

2L 15x2 unequal

Example of non-valid names:

W10x25 (space between Type and Description is missing)

W15/22 ("/" character is not accepted. Replace it with "_" (underscore) character)

15x22 (Type is missing)

Press OK and notice that a new section "Roof 3x6x1" has been created and saved into the sections database. A new "ROOF" group, which will contain all profiles of type "ROOF", has been created.

Important. - The Type of a profile determines the group in which this profile will be saved. Thus a "W 10x22" profile will be saved in a "W" group or type. In the same way a "TUBE 15x22" profile will be saved in a "TUBE" group. If the group does not exist, RAM Elements automatically creates a new group.

Remark. Note that the program already has a section with the name "Roof 3x6x1". The procedure described previously explains adequately the manner to perform this operation. It is recommendable that the user practices the creation of new sections for the structure in order to acquire proficiency in this task.

To assign the new section to the roof beams proceed as follows:



Text on each page

Select roof beams

92	RUUI SXOXI	U	U	U	
93	Roof 3x6x1	0	0	0	
94	Roof 3x6x1	0	0	0	
07	D40-04	-	0	0	· · · ·
Available	sections				*
Group:	🧱 United States				*
Tables		Item	c		
- 🗐 R	Ream		Roof 3:	x6x1	
	(Dealli				
- 🖪 R	cCol				
	acCol IndBar				
	toof				
	icCol IndBar Icoof				
	ccCol IndBar Loof 45				
	icCol IndBar Icoof I45 PCa				
	cCol IndBar Coof 45 PCa				

Assign the section by pressing

15) Assigning materials

In this example, all material elements are of steel grade A36. To assign the material, proceed with these steps:

Select elements to which a material will be assigned. In this case, select all the elements of the



structure by pressing All

Nodes	Members	Shells	Areas	Conn	Gen			
18 1	A 🍒	$\sqrt{\frac{2}{4}}$		⊨ 1	a <u>+</u>		I	»» •
Materia	ıls							
Member	Material							^
1	A36							
2	A36							
3	A36							
4	A36							
5	A36							
6	A36							
7	A36							
8	A36							
9	A35 A36							
10	A36							
40	A00							~
Available	materials						A .	
Group:	🔤 United	d States					1	-
Tables]	Items				
- M	1SR		^	Δ3	6 (weight	less)		^
- 🕒 N	Jon-NA Dime	ension L	ur	— A3	6	Ċ.		
- 📰 B	lC			- A5	00 GrA pi	pes	_	
-I s	iteel			— <u> </u>	00 GrB re	ctangula	ar	
— EX T	'imber beam	1S		- <u> </u>	00 GrB ro	unded		
- (C) T	'imber colum	nns		- <u> </u>	00 GrC re	ctangul	ar	
			~	A5	00 GrC ro	unded		~

Go to the Spreadsheet Members/Materials. Double click on the desired material, or select it and press

Material "A36" from folder Steel has been assigned to all elements.

Note. - To show/hide the section and material names on the screen, press the Sections button and the Materials button from the *View tab*, *Model group*.

16) Articulated joints (pinned joints)

By default, all frame members are rigidly connected (fixed) to the nodes. This condition is appropriate to model a fully welded joint.

For joints that cannot resist flexural moments it is necessary to release the respective moments so the model adequately represents the real structure. An element is pinned when both ends of the members are released to both bending moments. To pin a member proceed as follows:



Select the members to be pinned. In this case, select DIAG1 and DIAG2 elements. To do this, select one DIAG1 element and one DIAG2 element. Then press



Hinge

Go to the Spreadsheet Members/Hinges (Releases) and press button both ends .





Elements have been pinned

17) Rotating columns



Pressing (in the *View tab*, *Model group*) the elements with three-dimensional sections are displayed. This allows the user to see whether the elements are orientated correctly in space or need to be rotated. If necessary, sections can be rotated as required. Tool buttons are available to rotate a member 90 and 180 degrees, or as required. In this case, the middle columns will be rotated by 90 degrees.



Text on each page





Columns in the middle will be rotated 90 degrees. To rotate 90 degrees, proceed as follows:



Select columns to be rotated

ę	e 🗁 🔚 🤊 🗛 🗛 🕲											
	Home	Spreads	sheet Vi	ew Proc	ess O	utpul	: Modu	les				
Clear		₹	() Help	/alue	÷ = *	-	Local	E ³ Local axis 3 ▼	Rotate	Rotate	Default	Rotate -90°
		Sp	readsheet					Act	ive spread	isheet too	ls	
Nodes	Nodes Members Shells Areas Conn Gen Rotate 90° $\overset{1}{\overset{1}{\overset{1}{\overset{1}{\overset{1}{\overset{1}{\overset{1}{\overset{1}$											
Orientat	tion of local a	axes			0	Deg]						
Member	Rotation	Local	NX	NY	NZ	^						
28	90	0	0	0	0							
38	90	U	U	U	L	,						
								Ŧ	490.			
								Ro	tate			

Go to the Spreadsheet Members/Local axes and press button 90°.

18) Rotating beams 180 degrees

In this example, the elements shown below need to be rotated 180 degrees.



BEAM2, BEAM3 elements need to be rotated 180 degrees To do this, follow the next steps:



Select BEAM2 and BEAM3 elements (select one BEAM2 and BEAM3 elements and press button).



Go to the Spreadsheet Members/Local axes, and press button 180° to rotate 180 degrees.



Elements have been rotated 180 degrees.

Note. - Notice that it is possible to rotate the members by entering the required angle in the

spreadsheet and pressing \checkmark to fill the column with the entered value.

19) Entering loads

In this example, a 300 Lb/ft distributed force acting downward in the "Dead Load" case will be introduced. Concentrated forces of 1200 Lb, which are acting downward on the nodes, will be added as well.

Notice that RAM Elements automatically creates a load case named "Dead Load". Therefore, it isn't necessary to create it. Later the user will see how to create a new load case and a load combination.

Before entering a load, determine if it is a:

1) Load on node

Text on each page

- 2) Load on frame members, or
- 3) Load on shell elements.

Load on frame members

To enter loads on frame members, proceed as follows:



Select frame members where the load is acting. In this case, select beams on top of the truss.



Go to the Spreadsheet Members/Loads on members, select the adequate command tools by pressing

the <i>button and then press the</i>	button.	
Distributed load	? 🛛	
Direction ● Global axes ○ X ○ X ○ Y ○ Y ○ Z ○ Z	○ Local axes 1 -1 2 -2 3 -3	
Load type: Iniferm		lext on each page
Load magnitude (V): 0.3 [Kip/ft]		
Options:	Distribute over	
Replace existing loads	 Projected length 	
	OK Cancel	

Enter the value of the distributed load (do not enter the minus sign). Then press OK.



The load has been entered.

Load on nodes

To enter load forces on nodes, follow the next steps:



Select the nodes on which the force is acting.

Nodes Members Shells Areas Conn Gen										
(x,y)	(xa) 🏤 🐂 度 🗣 🤧 🔀									
Force	s and mome	nts			[Kij	5]				
Node	FX	FY	Z	MX	MY	^				
7	C	-1.2	0	0	0					
25	C	-1.2	0	0	0	_				
27	C	-1.2	0	0	0					

Go to the Spreadsheet Nodes/Forces and moments, *enter a force value (enter the* -1.2 *value) and press* \checkmark *to fill-in the column.*



Notice that the force should include its sign. Forces on nodes have been entered each page

20) Creating Wind in X load case

The Second load case acting on the structure is due to the wind force in the X direction. These steps show how to create a new load case:



Execute the shown	button located	l in the Hom	ne tab, Load	l conditions	group to	enter a new	load	case.

Ι	I Load conditions 🛛 🤶 🔀							
r	Cases	82						
	Num	ID	Description	Category	Duration 🔼 📑			
	1	dl	Dead load	DL	permanen 📃 🚆			
	2	WX	Wind in X	VMND	10 minute: 🛄			
					LC.			
					्य			
	<u> </u>							
					5			
					′			
					►			
	<				>			

Then enter a load condition identifier consisting of 2-4 characters (first character should not be a number), then enter a load description and the category. In this case, enter what is shown in the figure. Press OK and the new load case in the drop-down list will be shown.



Drop-down list for load cases at the status bar.

Note that it is necessary to select a category. This feature is very useful to generate load combinations based on their categories. The user can create a template file for the local building code from which load combinations can be generated (based on the load case category, DL for dead loads, LL for live loads, etc.). The program is capable to generate load combinations according to the design standards it handles and it has example files (located at main RAM Elements directory/*combos*) which have the basic load combinations to consider for the different codes. For more details see the chapter of Other Advanced Subjects in the program manual.

21) Entering wind loads

In this case, wind loads are applied perpendicular to the roof. There is a pressure of 150 Lb/ft on the left side of the roof, and a suction of 200 Lb/ft on the right side of the roof. Wind coad entry is similar to the entry made before for the dead load condition. Notice that the distributed forces act perpendicular to the elements, not parallel to Y-axis. To enter these loads, proceed as follows:



Select the elements on which the load acts. In this case, select one member of each portal and press to select the aligned elements.

Home Spreadsheet View Process Output Modules				
▼ ▼	Image: Distributed load towards Image: Distributed load towards Image: Distributed load towards Image: Distributed load towards Image: Distributed load perpendicular to the element in Triangular distributed l Image: Distributed load towards Image: Distributed load perpendicular to the element in Triangular distributed l Image: Distributed load towards			
Spreadsheet	Active spreadsheet tools			
Nodes Members Shells Areas Conn Gen i I I I I I I Loads on members I I I I Distributed loads I I I	Distributed load perpendicular to the element in -2			
Member Distributed 1				
1-Dir. Cat Value1 Value2 % Dist1 % Dist2 2	-Dir/Cat			
2 0 0 0 No 0 No 0				

Go to the Spreadsheet Members/Loads on members and press the witton.

Distributed load		
Direction Global axes	 ● Local axes ○ 1 ○ -1 ○ 2 ○ -2 ○ 3 ○ -3 	
Load type: Uniform		Text on each page
Options: Add to existing loads Replace existing loads 	Distribute over Real length Projected length	Cancel

Enter the value of the distributed force (do not enter the minus sign), and press OK.



The distributed forces of the left side of the structure have been entered.



To enter the forces on the right side of the structure proceed as before. The load should be seen as illustrated in the figure.

Notice that it is necessary to press button \checkmark instead of button \checkmark to enter suction.

22) Creating load combinations

In this example, the following load combination will be created:

1.1dl + 1.2wx (1.1 times dead load plus 1.2 times wind in X)

To create it proceed as follows:



Execute the shown button located in the Home tab, Load conditions group to enter a new load case. Fill the combination equation factors in the second spreadsheets in the dialog window that appears.

I	Load	d condi	tions				?		
	Num	s: ID	Descript	ion		Category	Duration 🔨 🔽		
	1	dl	Dead los	ad		DL	permanen	2	
	2	wx	Wind in 3	x		WIND	10 minute:	3	
								c	
								7	
								Ľ.	
							2	₽	
								4	
								<u> </u>	
							Tai		
							/ ex	t Or	i each page
		 a)							
		9							
ſ	<mark>Somb</mark> Formu	instion _{Ila:} c1 =	» 1.1dl+1.	2wx		ר	2 🕏		
	Num	ID	dl	WY	Type			-	
	1	c1	1.1	1.2	Not assigned			<u>د</u>	
N	_					J		e	
								n.	
								24	
							~		
						ОК	Cancel		

In the dialog window, enter the information shown in the figure.

a) Enter a load condition identifier of two to four characters (the first character should not be a number).

b) Enter the formula factors for the load combination (1.1 for dl and 1.2 for wx).



Then press OK and the new load combination in the drop-down list for load cases at the status bar will be shown.

Notice that the formula factors can contain the minus sign. For example, a -1.2 factor for wx load case would define "1.1dl -1.2wx"

Note. - It is not possible to enter or edit loads data while a load combination is selected as the current load condition. Notice that the spreadsheets are locked for edition to enter loads.

23) Analyzing the structure

After the structure has been defined, the model is ready to be analyzed, designed, optimized and the results can be viewed.

To analyze the structure proceed as follows:



Text on each page

Execute Analyze model by pressing the shown button in the Process tab, Process group.

For this example a Second Order Analysis (P-Delta) will be performed. This analysis takes longer to analyze a structure as it involves iteration, but it is more accurate. In addition, buckling instability is detected in certain cases when P-Delta analysis is performed. For more about P-Delta analysis, see the Chapter of Analysis in the Manual.

Analysis	? 🔀			
Analysis Condition Finite element model Model	DOF			
-Dynamic and Response Spectra				
Number of modal shapes to calculate:	10			
Method for modal superposition				
● CQC ○ SR5S ○ AE	is 👘			
Seismic analysis results with sign				
Disable shear deformations				
P-Delta Perform second order analysis (P-Delta) Ignore members with loads along their span				
Incremental/Iterative Analysis Configuration				
 Method Standard Newton-Raphson (NR) 				
O Modified Newton-Raphson (MNR)				
Number of increments:	1			
Number of iterations per increment:	10			
Convergence tolerance: 1E-3				
Cancel analysis if any of the load cases fails to converge				
Current values as default				
Analyze	Cancel			

Select the same options as shown in the figure above. Then press the Analyze button.

24) Designing the structure



Select the command Design all in the Process tab, Process group.

After that, a dialog window will be shown to specify the design standard to be used in the design of members. For this example select AISC 360-05, AISI-01 (ASD) for steel members. The other material options may use the code defined by default.

Design			? 🗙
Design code	Seismic	design	
-Steel members-			
Hot	rolled	AISC-05 ASD	-
Cold	formed	AISI-01 ASD	-
Use MBMA special considerations			

Text on each page

Select the design code shown above and press the Design button.

Once the elements are designed, the user has the option to optimize the sections with the following command.



Select the command Optimize model in the Process tab, Process group.

For this example the optimization is not performed.

25) View results graphically

As can be seen, several buttons (in the *Analysis* and *Design* groups of the *View tab*) are enabled once the structure has been analyzed and designed. These newly enabled buttons allows the user to select what results to display.



Result buttons from the Analysis and Design groups are enabled when the structure has been analyzed/designed.

In order to see results graphically, press the button corresponding to those desired items, and then select the elements to see the results.

Notice that the selected display options will only be seen on the selected elements.

	- dl=Dead load	
	 wx=Wind in X 	
English - Conditions:	🔹 dl=Dead load 🔤 🔤	a^ ar 🔒 🔍 🔍 🛟

Select the load condition.

26) Deformed shape

One of the first display options that should be viewed is the deformed shape of the structure.



In this view the elements are drawn as lines. To see the deformed shape with the original shape select the option with original shape.

27) 3D Sections Deformed shape

It is possible to see the deformed shape with the extruded sections.



Activate the deformed shape accessing the option from the Rendering button menu (*View tab, Model group*). Notice that this view may take a longer time to draw.



The graphic shown corresponds to the Dead load case.

28) Stress

Another important view option is the information related to the element stress contour. This is of particular importance in light gage structures where stress concentrations are significant to the design.



Press the stresses button to see frame member stresses. Note that the button has a menu where there are some options to see the stresses in the deformed shape or in shrunken members.

To select only those elements that are stressed within a certain range, mark a block of stresses with the mouse and press

Stresses Members and shells [Kip/in2]	
7.80 6.75 5.71 4.66 3.62 2.57 1.53 0.48 -0.56 -1.61 -2.65 -3.70 -4.74 -5.79 -6.83	
-7.88 Smooth Env	
Ang (')	
Von Mises 💙	

To see element stresses within a certain range, mark the range and press



RAM Elements selects those elements whose maximum stress is within the marked range of stresses. Note that the remaining members are recalibrated (color changes).

Note. – To see only the axial stress (without bending moments, press \checkmark Axial stress).

29) Stress and deformation



Text on each page

💳 Stresses 🔽		
	Deformed shape	
	Shrunken members	
Ø	Transparent	
€	Lines only	

To view stress and deformation of the elements, activate these buttons



30) Forces diagrams

The buttons shown above (View tab, Analysis group, Member forces displayed menu) allow the user to see the forces diagrams of the frame members:

Bending moment around element axis 3 (Typically strong axis bending)



Bending moment around element axis 2 (Typically weak axis bending) Text on each page



Shear forces in element axis 2 (typically weak axis) (Dead load case)



Shear forces in element axis 3 (typically strong axis) (Dead load case)



Text on each page





Axial forces (Dead load case)



Select the Show values option (Member forces menu) to simultaneously display the magnitude of the forces.



Select Show units option (menu displayed for units at the status bar) to display the units.



31) Displacements of nodes

To see the nodal displacement values, press choose the degree of freedom to be viewed:

View tab, Analysis group) and

Tx
Ту
Tz
Rx
Ry
Rz

The relation between a degree of freedom and its respective displacement in the global coordinate system is as follows:

1: Tx: X translation

- 2: Ty: Y translation
- 3: Tz: Z translation
- 4: Rx: Rotation about X
- 5: Ry: Rotation about Y
- 6: Rz: Rotation about Z

Note. – Notice that X, Y, and Z represents the global coordinate system.

Each element has its own system of coordinates, named local axes. These axes are designated with the numbers 1, 2 and 3, which are equivalent to X, Y, and Z-axis. Local axes are Cartesian and follow the right hand rule. To see the local axes, press $\frac{1}{2} \frac{1}{4} \frac{1}{2} \frac{1$



Press Nodal displacement and the degree of freedom corresponding to the displacement.

To see the displacement units press the Show units option in the Units menu at the status bar.



32) Reactions

To see reactions, press $\overset{\text{Leactions}}{\longrightarrow}$ (*View tab, Analysis group*) and the degree of freedom corresponding to the desired action.

Τx
Ту
Tz
Rx
Ry
Rz

This is the relation between degree of freedom and force:

- 1: Tx: X force
- 2: Ty: Y force
- 3: Tz: Z force
- 4: Rx: Moment about X
- 5: Ry: Moment about Y
- 6: Rz: Moment about Z



Press $\stackrel{\text{Reactions}}{\longrightarrow}$ and the degree of freedom that corresponds to the desired reaction. (*Case: Wind in X*)

33) Deflections

One of the most important results of an analysis is the ratio between deflection and length of the element. To view this ratio press \checkmark Deflections \checkmark (*View tab, Analysis group*).

Defl/L	
Defl. in axis 2	
Defl. in axis 3	
Defl=f(L) in axis 3	
Defl=f(L) in axis 2	

Options displayed in the Deflections button menu.

This ratio may vary across an element. RAM Elements displays the maximum ratio found within an element.

Note. - The Defl/L ratio should never exceed a value suggested by the design code and judgement.



Press \smile Deflections \checkmark *to see the element colored Defl/L ratios.*

Maxin Defor in 2 a	num Defl/L mation/L nd 3	
	4.683E-04 4.234E-04 3.785E-04 3.336E-04 2.887E-04 2.438E-04 1.989E-04 1.540E-04	
	1.091E-04 6.41/E-05 Not designe	d

In this panel mark a range of Defl/L ratios and press to select the elements that have slopes within the marked range.

34) Deflection values

To see the Deflection values (in function of L) in local axis 2 direction, selection

```
option Defl=f(L) in axis 3
```

To see the Deflection values (in function of L) in local axis 3 direction, press

```
button Defl=f(L) in axis 2
```



Deflection in function of L for the Load combination C1.

35) Design: Colored Interaction Values To view interaction values graphically, by color, press Stress ratio (View tab, Design group). Important! To view the interaction colors scaled from 0 to 1.0, press Stress ratio . To view the controlling interaction value for all Load Combinations (not load cases) press Stress ratio Weximum stress ratio Stress ratio For the controling combination Text on each page

Press

to see interaction values.

To select the elements with stress ratio within a certain range, mark a range of stress ratios and press

Stress ratio AISC/AISI/NDS 🛛 🥅		
	0.69	
	0.61	
	0.54	
_	0.46	
	0.38	
	0.31	
	0.23	
	0.15	
	0.08	
	5.777E-04	

Mark a block with the mouse and press button to select elements with stress ratio within the range.

Note that most of the results displayed to this point are for the selected load condition.

36) Design: Interaction Values

To see interaction values for the currently selected load condition, press



Choose the Stress ratio option to view interaction values for the current load condition. The last option of the menu should be enabled to see the ratios for the governing load combination.

37) Design: OK and NG (No Good) elements



To view the elements that failed code check (for the current load condition), press *Design group*):



×



Text on each page

Press button to see elements that pass code check.



Press button Market to quickly select all elements that failed code check.

The user can print the results of the steel design in a report. To print them, go to the *Output tab*, *Reports group*. For more information about reports see the Printing Graphics and Reports Chapter in the manual.

The user can also use the optimization feature that is valid only for steel and wood members. This option allows the user to change the existing sections with sections that are recommended (based on explicit criteria) from a collection of sections. In other words, the original section can be replaced with another that resists the imposed loads with an allowable deflection and that is located above the original section in the list of sections specified for the optimization. To use the optimization feature go to the *Process tab, Process group, Optimize model command*. For more details see *Chapter 11: Steel and Wood Structure Optimization and Code Check* of the Manual.

YOUR PACKAGE CONFIRMATION

Test for GreenYellow Color

<u>Test for Honeydew Color</u>





Test for Ivory Color

NAME: Test for Khaki Color

Test for Lavender Colo

EMAIL ADDRESS: der Blush Color

Test for LawnGreen Color CUSTOMER NUMBER:

Test for LightBlue Colo

PACKAGE ID: LightCoral Color

Test for LightCyan Color ISSUE DATE: Coldenrod Yellow Color

EXPIRES: Test for LightGray Color Test for LightGreen Color

PACKAGE INFORMATION: Test for LightSalmon Color

Test for LightSeaGreen Color

Important Information

ext on each page

We ask that you confirm your travel dates with us at least 45 days prior to arrival. You may reschedule your reservation once without charge as long as the request is made at least 45 days prior to the date of your arrival. For special events, peak seasons and major holidays allow 60 days.

All resort destination accommodations are provided at nationally recognized hotels such as The Radisson Hotel or other comparable properties. While in Florida you and (spouse/traveling companion) will be required to attend tours about ownership at our partner timeshare resorts. It takes part of a morning or afternoon and it's fun and informative.

*Travelers must meet the requirements within 18 months from the date of purchase.

*If married or cohabiting quest must have matching identification (matching IDs, passport, bill) with the same address.

*Full time students and those who are unemployed do not qualify for this offer.

INITIAL

ADDRESS: Test for LightSkyBlue Color Test for LightSlateGray Color

CITYest for LightSteelBlue Color

est for LightYellow Color

STATE/PROVINCEIme Color

Test for LimeGreen Color POSTAL CODE: Linen Color

Test for Magenta Color COUNTRY: Test for Maroon Color

Test for MediumAquamarine Color TIME ZONE: Test for MediumBlue Color

PHONE I. Test for MediumOrchid Color Test for MediumPurple Color

PHONE 2-MediumSeaGreen Color Test for MediumSlateBlue Color

Test for MediumSpringGreen Color PURCHASE INFORMATION:

You have authorized the purchase of the above Trester oill Allet dither Vierdie treed lister bbelow.

AMOUNT:

MERICAN

EXPRES

PURCHASE DATE:

DISCOVER

AUTI49RIZE MEREghtBlue Color

Fest for MintCream Color

Important Information

*These promotional holidays are provided for Married or cohabiting couples 25-75 and single females 28+ unless otherwise stated. These vacations are fully transferable to anyone who meets the promotional requirements within 18 months.

ext on each page

If you have trouble dialing the number please contact your operator and have the operator dial the number.

Additional numbers are available on our Website.

Our Goal is to provide you a VIP Hospitality Travel Experience so enhanced, and so wonderful you will always reach out to us plan & book all of your future holidays.

Seller of Travel Registration No.

This document is powered by

is not operated by, or affiliated with, Walt Disney World® Resort, Universal Orlando® or SeaWorld®. As to Disney artwork/properties: ©Disney. Universal elements and all related indicia ™ & ©Universal Studios. All rights reserved. All SeaWorld© images are a copyright of SeaWorld Parks & Entertainment.

Page 10

TOTAL:

EXP DATE:

VISA

CARD NUMBER: