



RAM Advanse[®]

Full Featured 3D Finite Element Analysis and Design for
Virtually Any Type of Structure

INTRODUCTORY TRAINING
Handout

Copyright

RAM Advanse 6.0 and all associated documentation are proprietary and copyrighted products. Both United States copyright law and international treaty provisions protect this software and related documentation. Any unauthorized copying or reproduction is strictly prohibited and subject to civil and criminal penalties. Please refer to the License Agreement for authorization to make a backup copy of the software and documentation. You may not sell this software or documentation or give copies of them away to anyone else.

For further information please contact:

RAM International, L.L.C.
5225 Avenida Encinas, suite E
Carlsbad, CA 92008
U.S.A.

tel. (800) 726-7789

fax (760) 431-5214

e-mail: advansesupport@ramint.com

Website: <http://www.ramint.com>

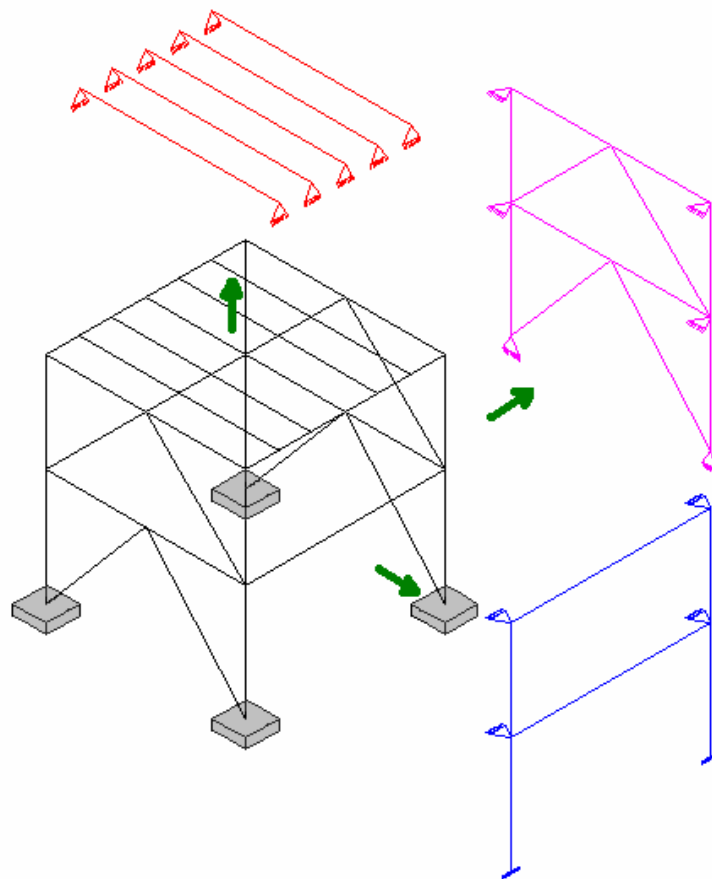
Copyright 1990-2004, RAM International, L.L.C.
All Rights Reserved

Tutorial

1) Good Modeling Practices

In general, the modeling of a three dimensional structure can be achieved in three steps:

- At first, the 3D structure can in general be considered as an assembly of members (see figure below). These components can be plane sub-assemblies such as floor structures, individual frames consisting of linear beams, columns, and braces, and even 2D or 3D elements such as slabs or walls. This idealization of the structure should also include the proper support (boundary) conditions. Your model should be such that the effects of actions on the actual structure are properly judged. In particular, stability aspects, performance of joints and foundation conditions should be correctly considered.
- Then, in general, the individual plane frames may be analyzed only against in-plane actions, using in-plane resistances. In other words, out-of-plane effects of such actions and limitations may be neglected in this kind of analysis.
- In the final step, member checks can be carried out by isolating them from the frame and applying all internal and external forces, restraints and out-of-plane deficiencies.

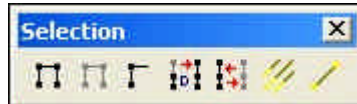


It is the engineers choice based on his judgment, if the structure can be modeled in 2D or if it should be modeled in 3D. RAM Advanse allows you to go either way.

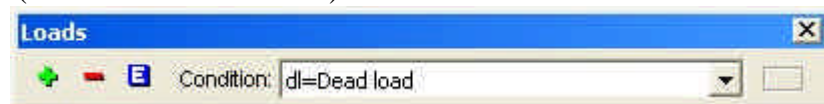
2) Screen Layout in RAM Advanse

RAM Advanse has a main window in which all the model geometric, section and load data is input, manipulated and viewed. The main window is displayed below with all of the individual work areas identified. In short the areas are:

- a) Member selection toolbar – Commands to help you rapidly select groups of members in your model.



- b) Load condition toolbar – Commands to create, edit, and select between different load conditions (cases and combinations).




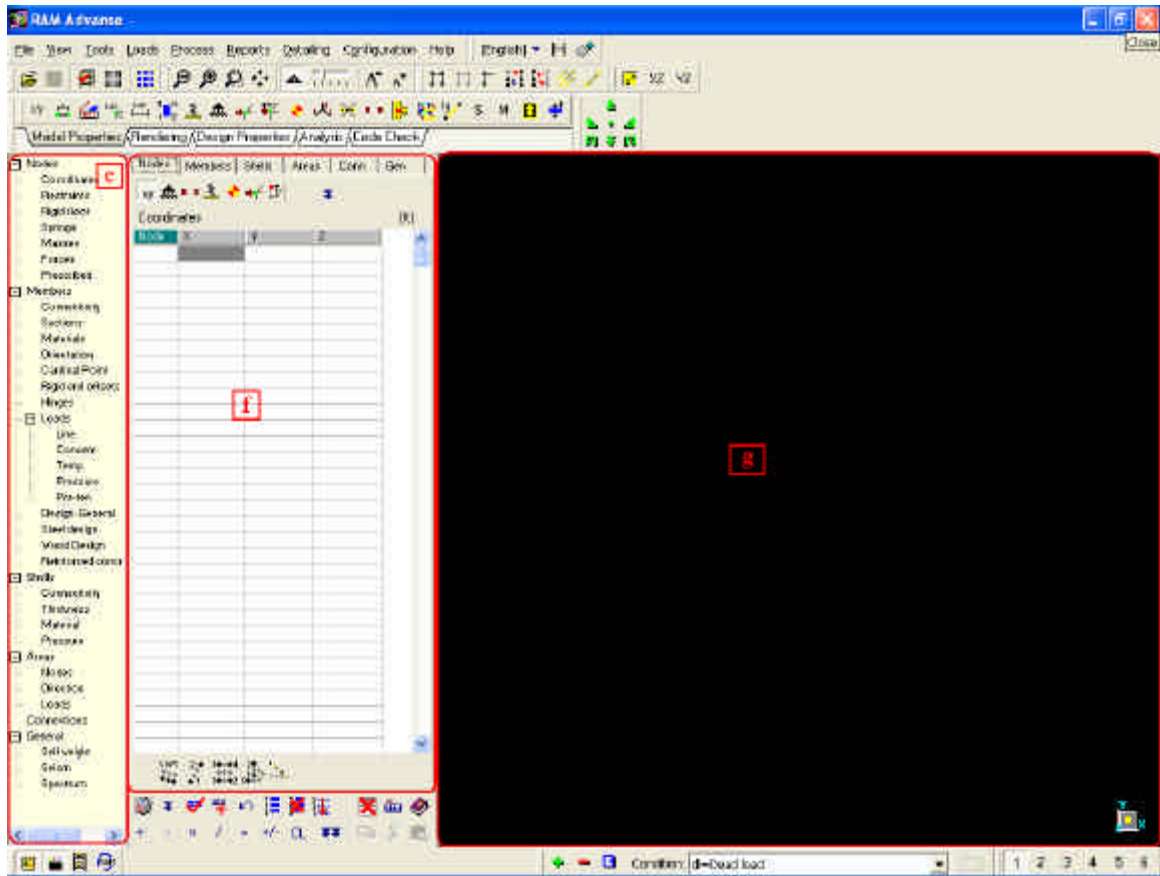
- c) Panels toolbar – Commands that influence what is displayed on the main window and how the engineer will enter data.



- d) Display options combo-toolbar – Commands to display different properties of the model on the graphic display area.



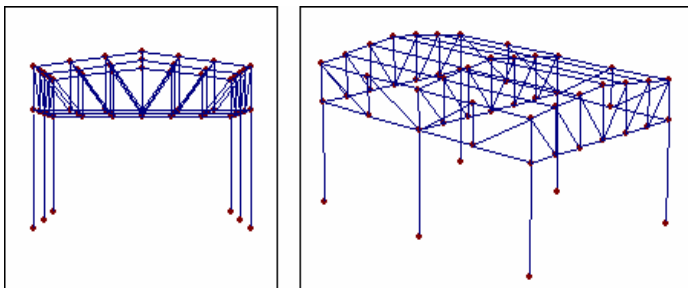
- e) Data explorer – Area to allow engineer to quickly navigate to desired input spreadsheet and to review what data has been input. Note: Initially, the data explorer is disabled (hidden). To activate it you have to press the  button in the panels toolbar (c).
- f) Data panel worksheet/spreadsheet area – Area where member properties and geometric data are input for the selected members.
- g) Graphic area – Area where model and any other selected display option are displayed.



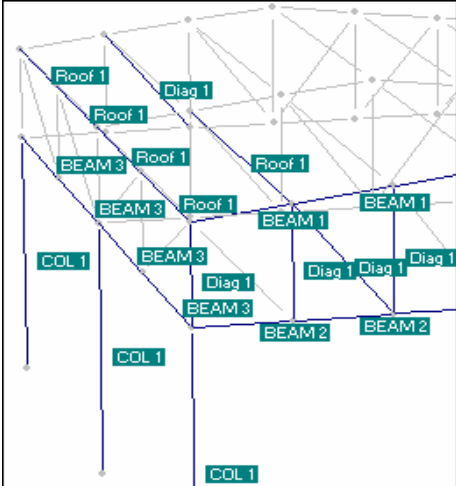
3) Starting a New Structure

This example will take you step by step through the creation of a basic 3D steel structure. This example will be most effective if you practice the illustrated skills as they are presented.

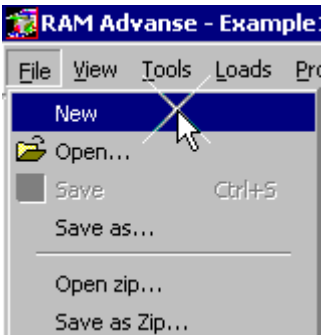
The structure to be entered in this example is shown below:



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

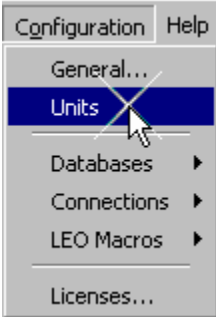


The assignment of the member descriptions shown here will be illustrated in this example.

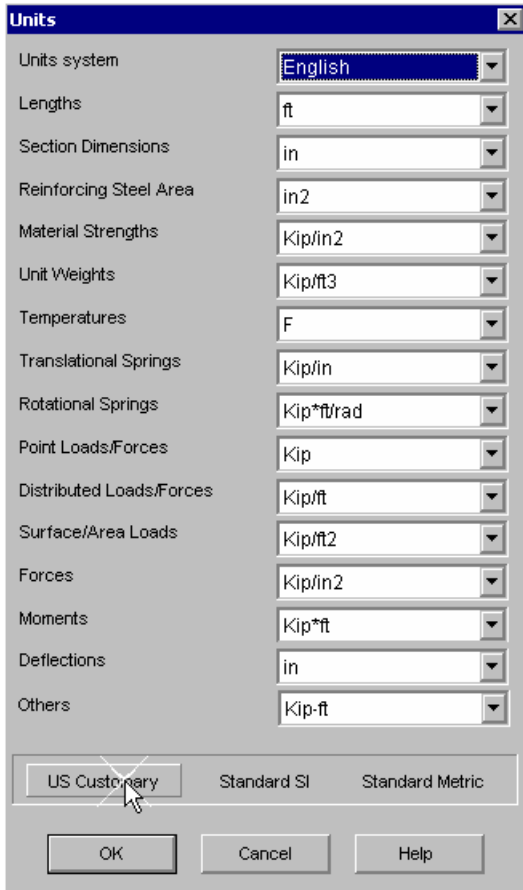


Select the File/New command.

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



Select units



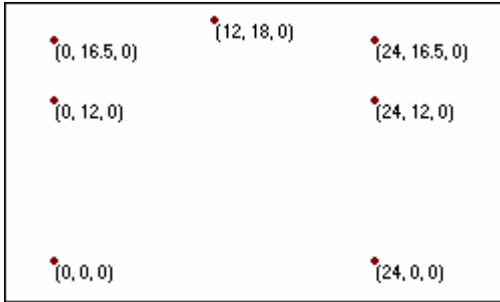
Select the US customary option for this example.

4) Entering Node Coordinates

In the coordinates spreadsheet enter the coordinates shown below:


Node	X	Y	Z
1	0	0	0
2	24	0	0
3	0	12	0
4	24	12	0
5	0	16.5	0
6	24	16.5	0
7	12	18	0

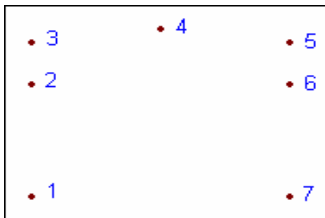
Go to the Spreadsheet Nodes/Coordinates and enter the coordinates shown above.



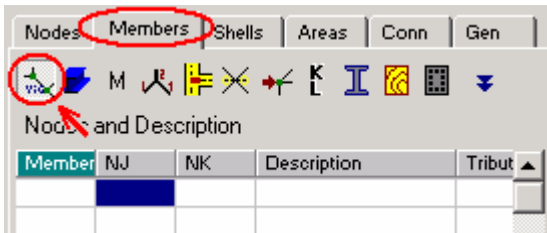
You can see the entered nodes on the screen.

5) Generation of frame members


Select the "path" of the frame members. Select the nodes in the sequence shown below, and then connect the selected nodes by pressing 

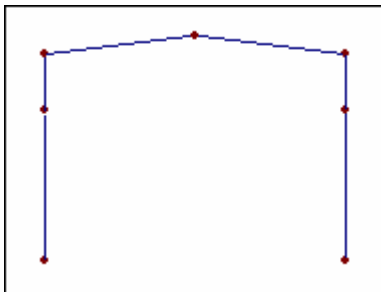


Select the nodes in the order shown. To select several nodes remember to press the SHIFT key while you click with the mouse.




Go to the Spreadsheet Members/Nodes and Description

Then press  to generate the frame members.

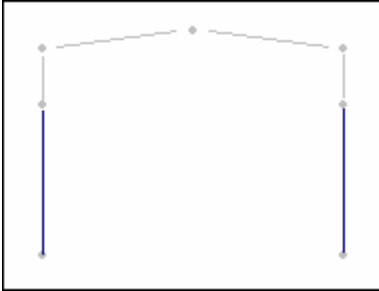


As you can see the frame members were generated

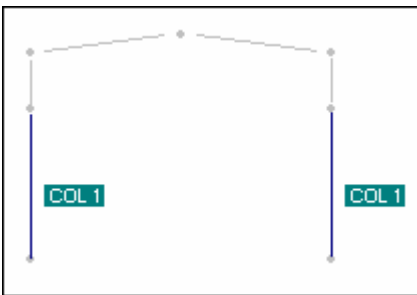
NOTE. - Remember that you can undo the last operation by pressing 


6) Assigning a description

It is necessary to group frame members in order to simplify later operations such as selection of elements, optimization, and others. To assign the same description to every member of a group proceed as follows:



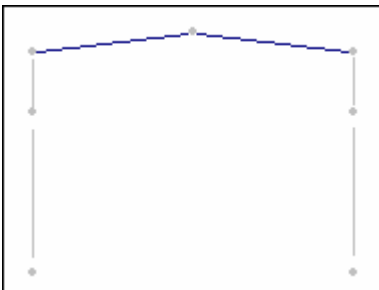
Select columns



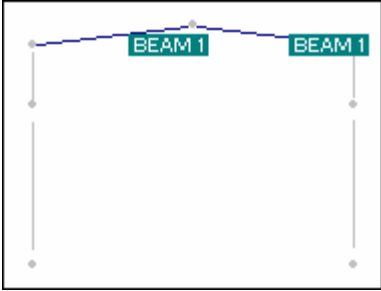
Then press  to assign COL1 description to the selected members

Note. – To view the member descriptions graphically (on the screen) press .

Repeat the steps explained previously to assign a Description to the other members:

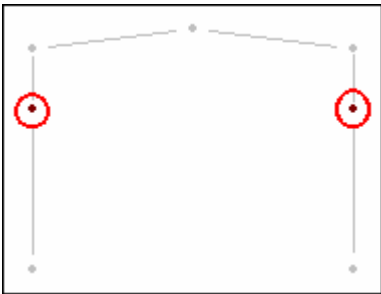



Select members

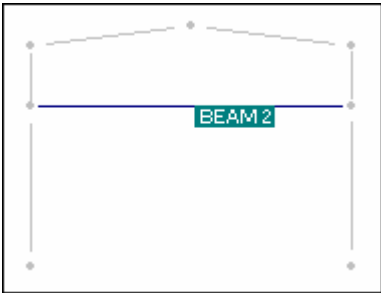


Press  to assign BEAM1 description.

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



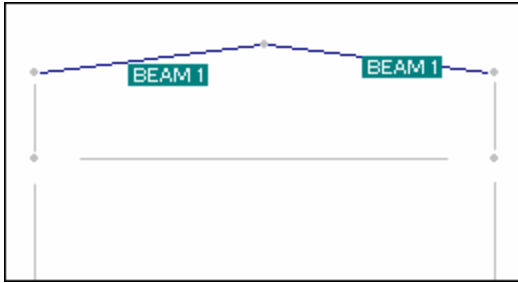
To create the horizontal beam, select the nodes shown in this figure and press 



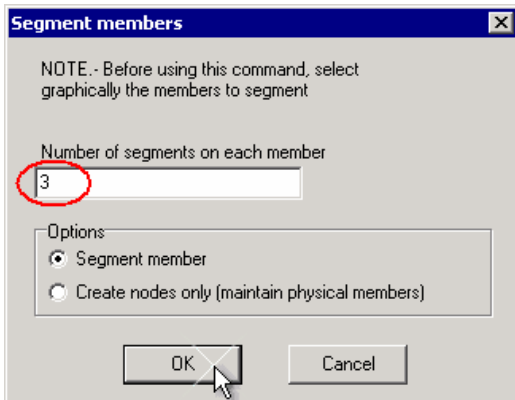
Press  to assign BEAM2 description.


7) Segmenting Members

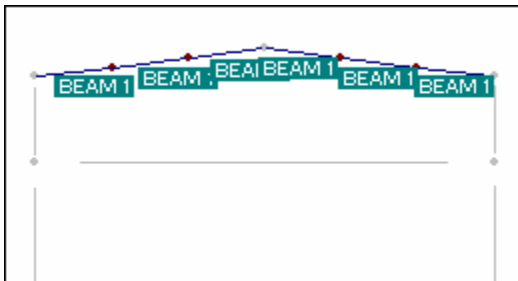
To segment frame members, follow these steps:



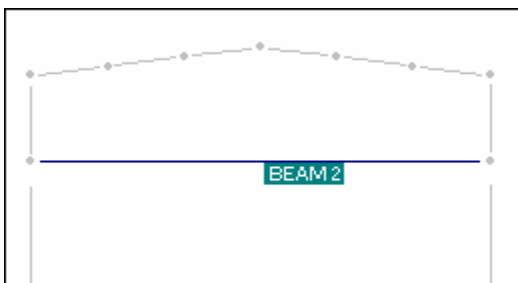
Select members to be segmented.



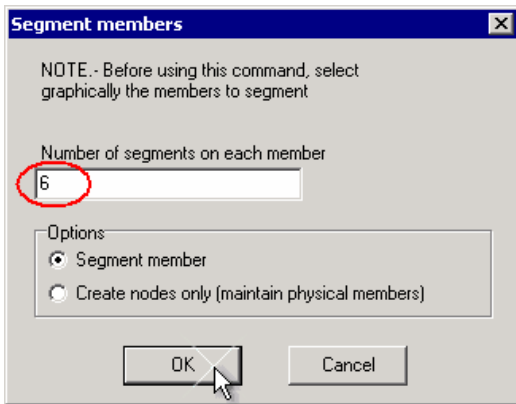
Press  and enter the desired number of segments (3 segments in this case). Then press OK or the ENTER key. Notice that in this case 3 physical element will be created.




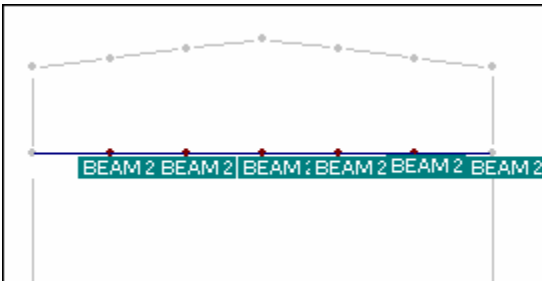
Next, segment the horizontal member BEAM2. To do this:




Select BEAM2 member.



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

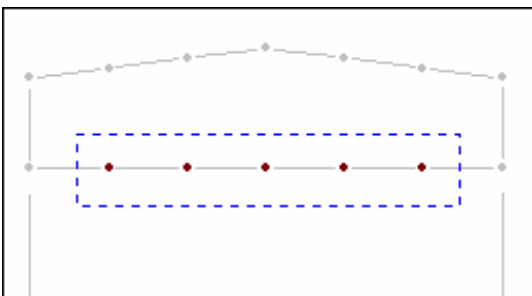


NOTE. - Remember that you can undo the last operation by pressing .

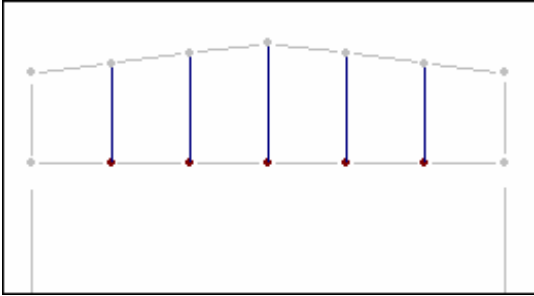
Notice that the segmented members have the same description as the original member and that each member is treated as one physical member.


8) Generation of vertical members

To enter the vertical truss elements, follow these steps:



Select the nodes shown in this figure. Notice that you should not select the exterior nodes.

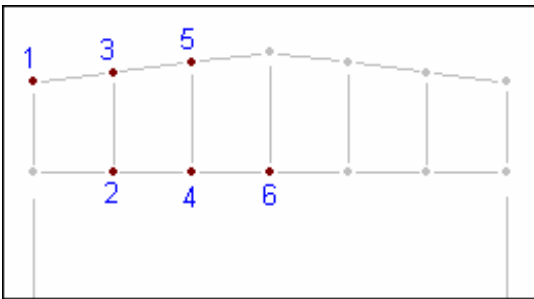


Press button  to generate vertical members (plus y generates members in the vertical up direction).

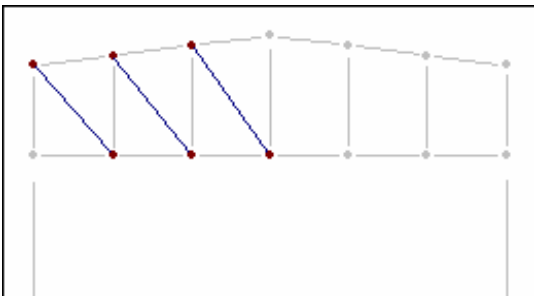
9) Generation of diagonal members

We will 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.



Press 

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

NOTE. - Remember that you can undo the last operation by pressing .

The differences between the two buttons is as follows:



This button connects the selected nodes in a continuous line.

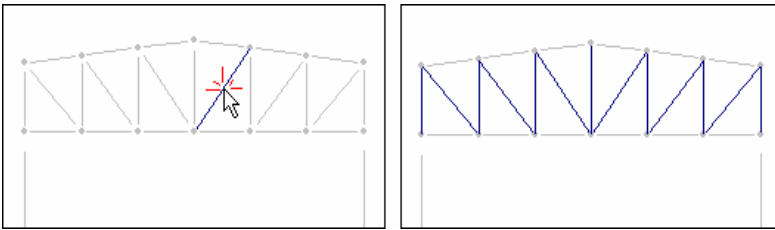


This button connects alternate pairs of nodes with a fragmented line. That is: the first member is generated between the first pair of selected nodes, the second member between the second pair of selected nodes, etc.

10) 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



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:

Member	NJ	NK	Description
2	3	4	Diag 1
5	6	5	
17	13	11	
18	14	12	
19	15	7	
20	16	9	
21	17	10	
22	4	13	
23	11	14	
24	12	15	
25	6	17	
26	10	16	
27	9	15	

Member	NJ	NK	Description
2	3	4	Diag 1
5	6	5	Diag 1
17	13	11	Diag 1
18	14	12	Diag 1
19	15	7	Diag 1
20	16	9	Diag 1
21	17	10	Diag 1
22	4	13	Diag 1
23	11	14	Diag 1
24	12	15	Diag 1
25	6	17	Diag 1
26	10	16	Diag 1
27	9	15	Diag 1

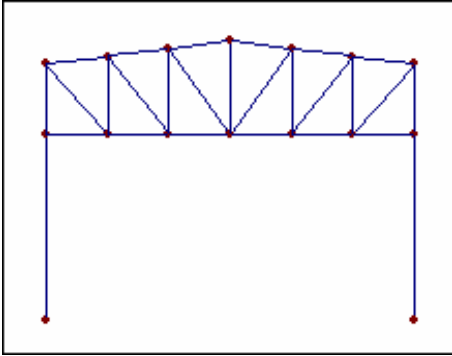
Enter DIAG1 description and then press to fill the column with the value.


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

11) Copying the structure

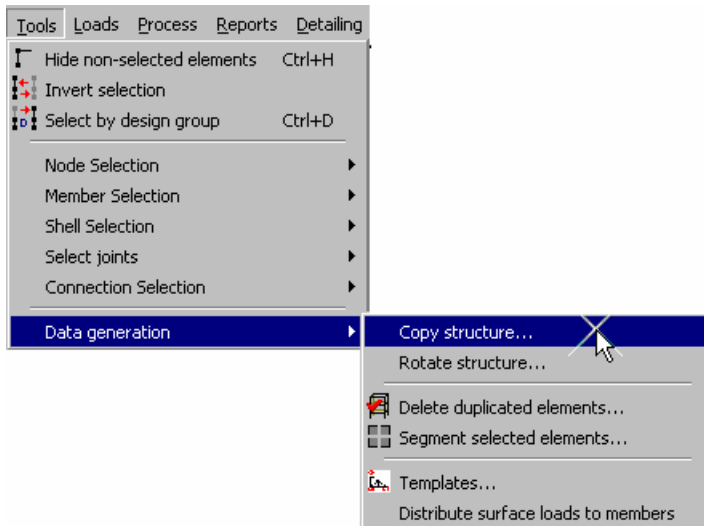
It is advisable to enter all the descriptions of a structure before copying it, because when a structure is copied the Descriptions are also copied.

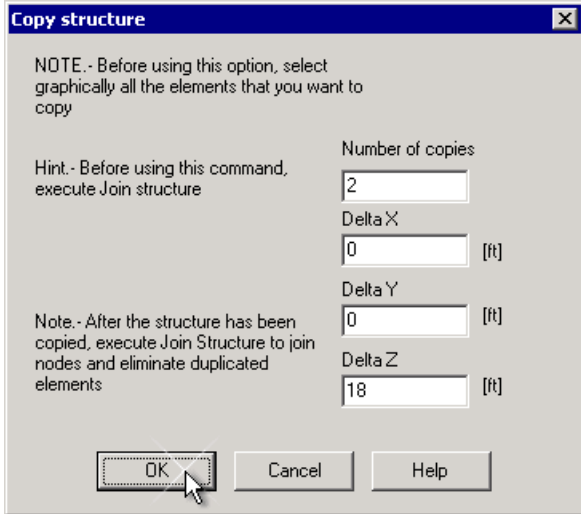
To copy a structure, follow these steps:



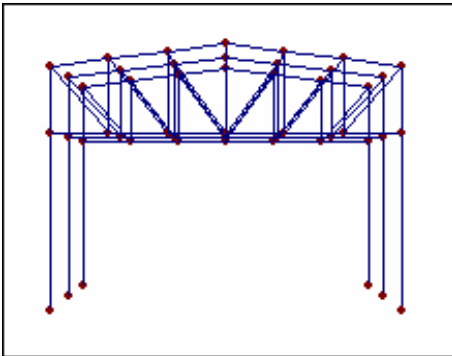
Select all the elements that should be copied. In this case, press  to select the entire structure.

Execute **Copy Structure** command



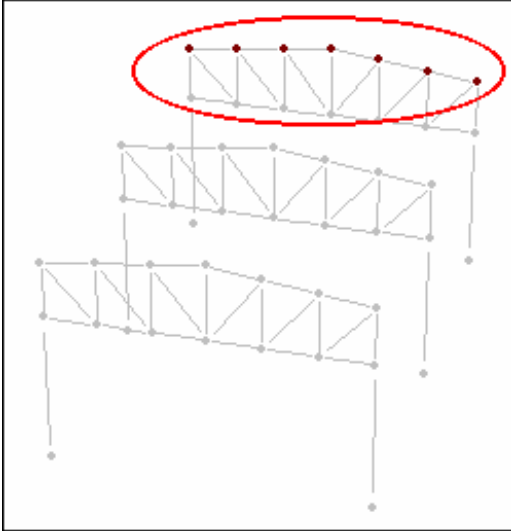


Enter the number of copies and the distances in X, Y, and Z between each copy. In this case, enter the values shown in this figure. Then press OK.

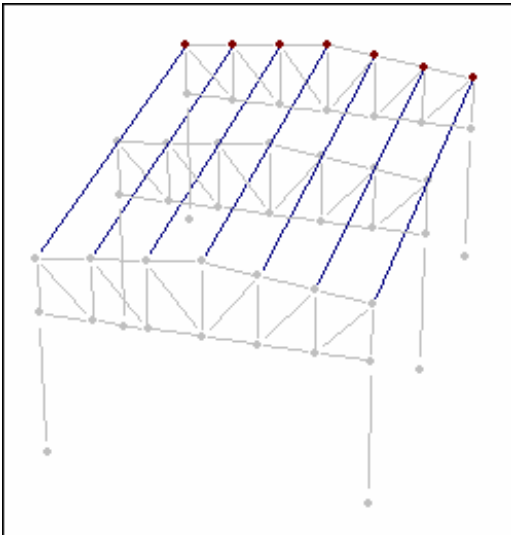


12) Generation of the roof beams

To generate the roof beams, follow these steps:




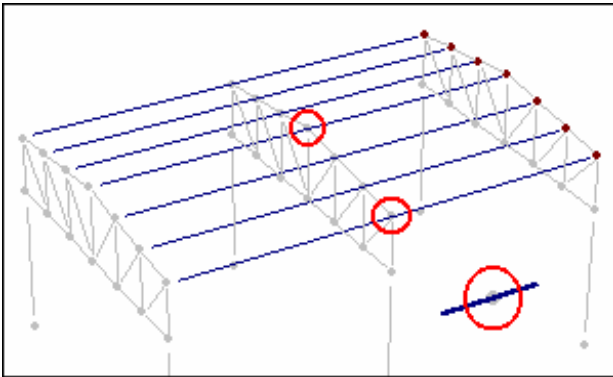
Select the initial nodes (you can also select the end nodes) of the roof beams.



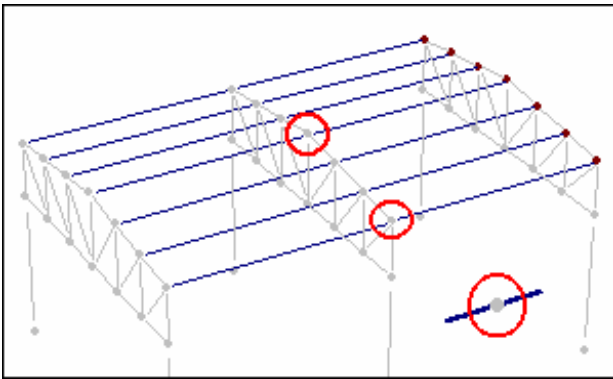
Then press **+z** (Press button **-z** if nothing occurs). Note that the +/- refers to the direction that the members are projected.

Note. -Notice that the middle portal is not connected to the roof beams. You can live as it is 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), you have to

segment the beams and connect one end to the middle portal. The command Segment Elements  may be applied in this case.



Notice that roof beams do not connect with the middle frame

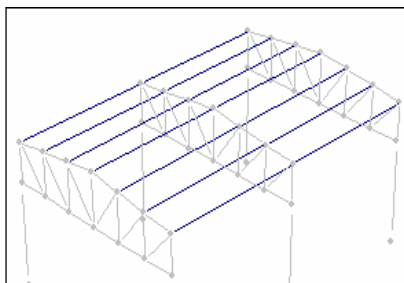
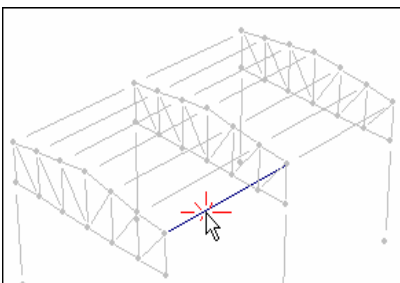


Press  (Segment Elements) to split roof beams and connect them with the middle portal.

13) 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:

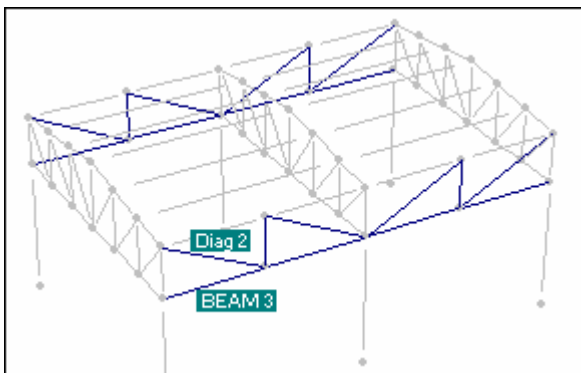
Nodes and Description			
Member	NJ	NK	Description
82	4	21	Roof 1
83	10	23	
84	11	39	
85	7	25	
86	8	26	
87	9	37	
88	6	29	
89	21	22	
90	23	24	
91	39	40	
92	25	27	
93	26	28	
94	37	38	
95	29	31	

Nodes and Description			
Member	NJ	NK	Description
82	4	21	Roof 1
83	10	23	Roof 1
84	11	39	Roof 1
85	7	25	Roof 1
86	8	26	Roof 1
87	9	37	Roof 1
88	6	29	Roof 1
89	21	22	Roof 1
90	23	24	Roof 1
91	39	40	Roof 1
92	25	27	Roof 1
93	26	28	Roof 1
94	37	38	Roof 1
95	29	31	Roof 1

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

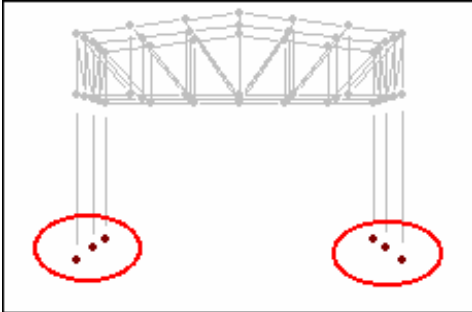
Generating DIAG2 and BEAM3 members

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

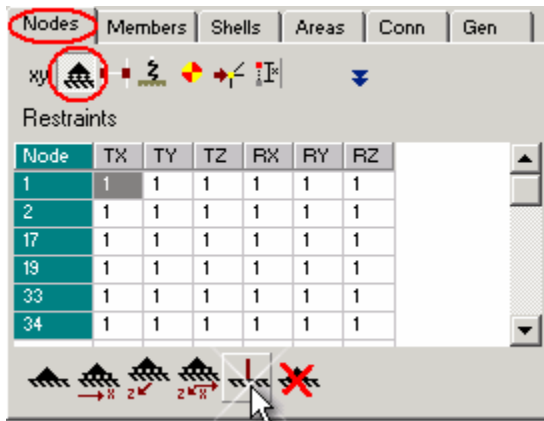


14) Supports

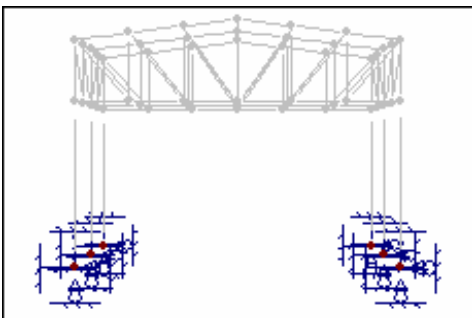
To enter supports proceed as follows:



Select support nodes



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

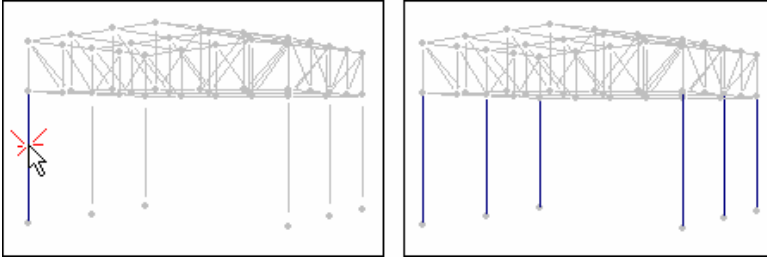



The Supports have been entered

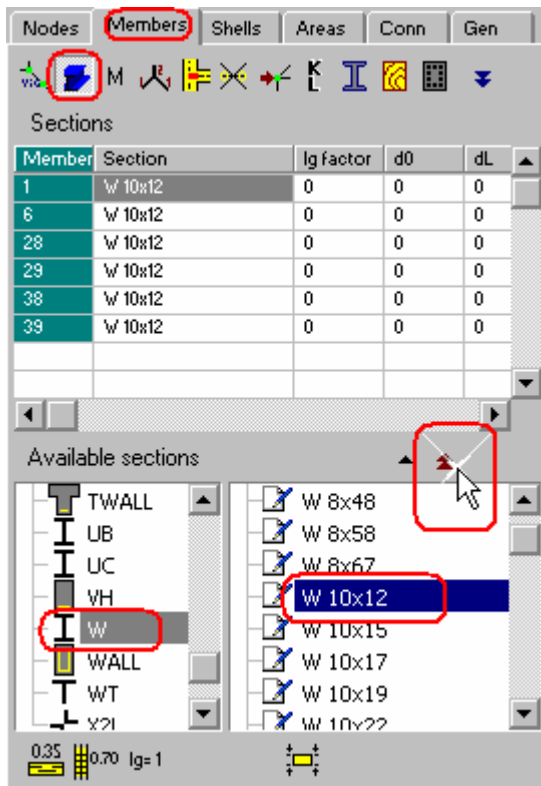
15) Assigning sections to frame members.


When you want 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.

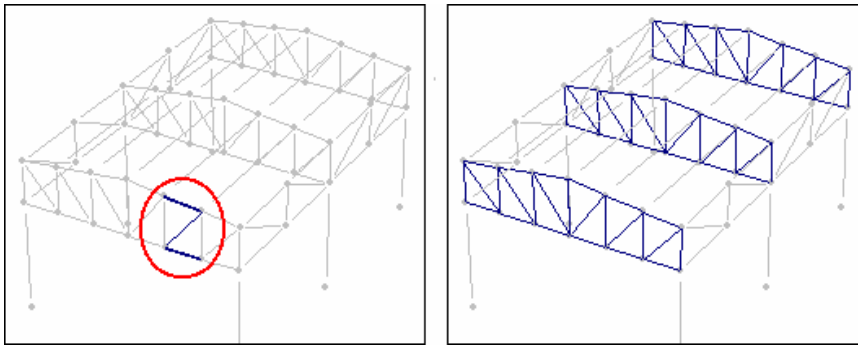



To do this, first select one column and press 

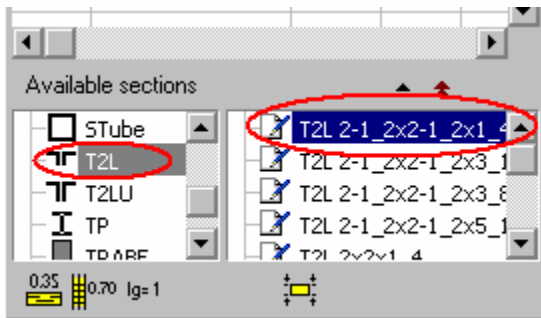


Then go to the Spreadsheet Members/Sections. Select W10x12.sec profile and press  (you can also double click on the profile).

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

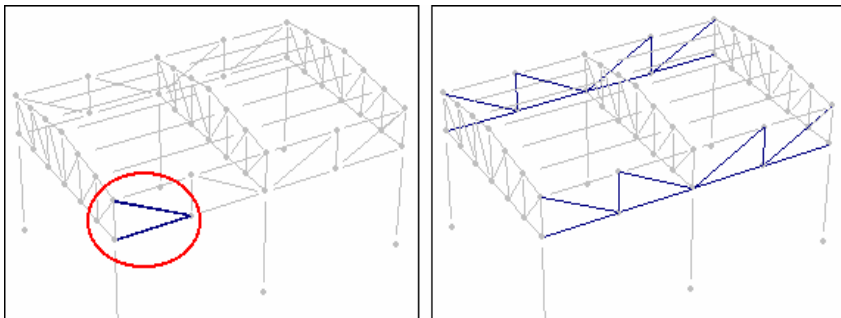


To select all the elements of the truss, select one element of each group and press 

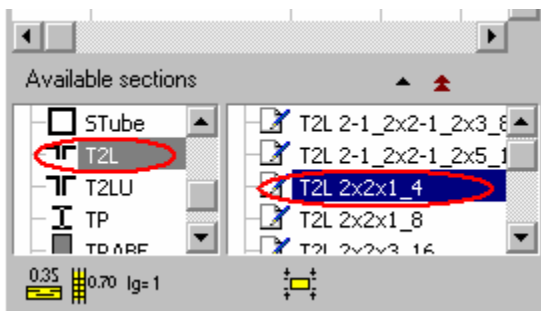


Assign section T2L 2-1_2x2-1_2x1_4 to the truss elements.

Now assign sections to the DIAG2 and BEAM3 elements



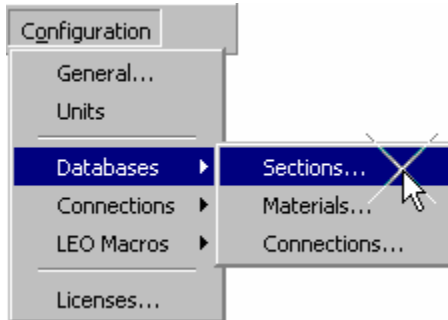
Select the elements DIAG2 and BEAM3



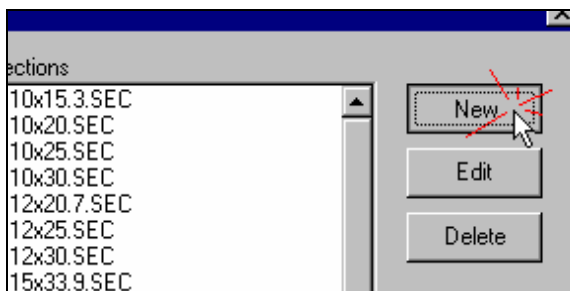
Assign section T2L 2x2x1_4

16) Adding sections to the database.

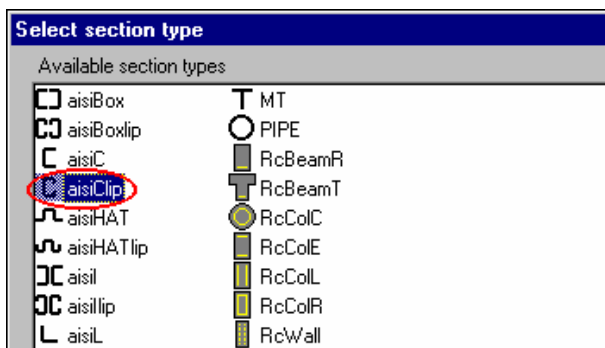
In this example, a cold-formed C-section will be assigned to the Roof beams. This cold-formed C (with lips) profile is not available in the section database. Therefore, a new section should be added. Proceed as follows:



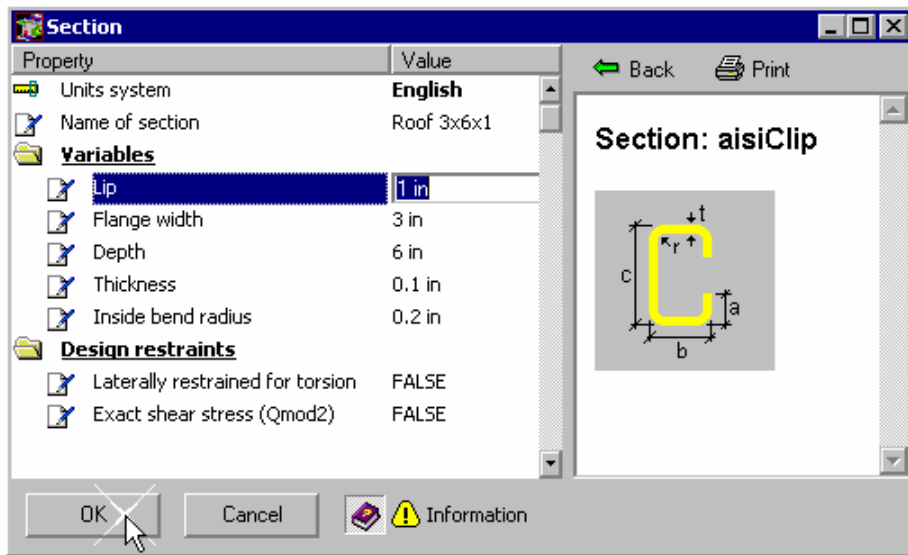
Go to the Menu Configuration/Databases/Sections



Select button New



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



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

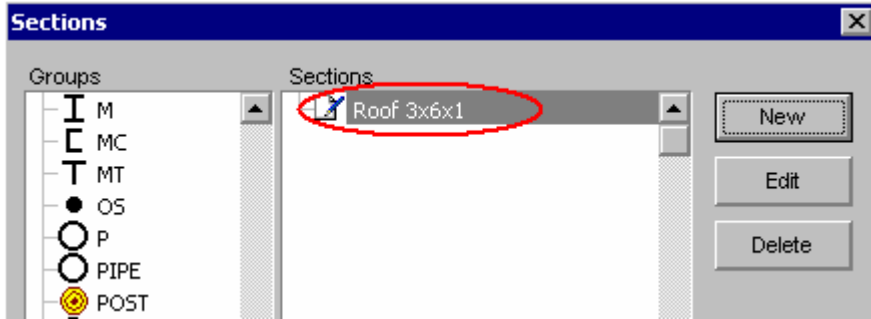
Tip. - The Description of the profile should be self-explanatory containing the dimensions of the profile or other pertinent data.

Example of valid names:

ROOF 10X15X25
W 10X25
2L 15x2 unequal

Example of non-valid names:

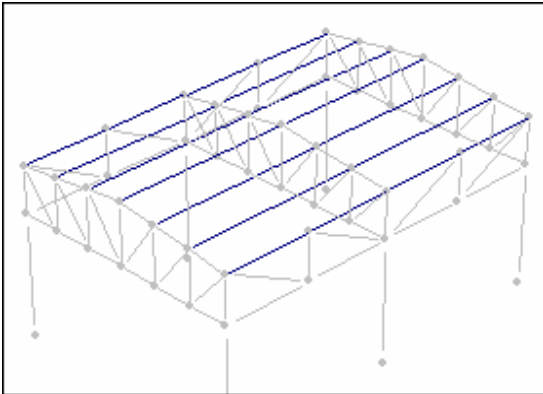
W10x25 (space between Type and Description is missing)
W15/22 ("/" character is not accepted. Replace it with "_" (underscore) character)
15x22 (Type is missing)



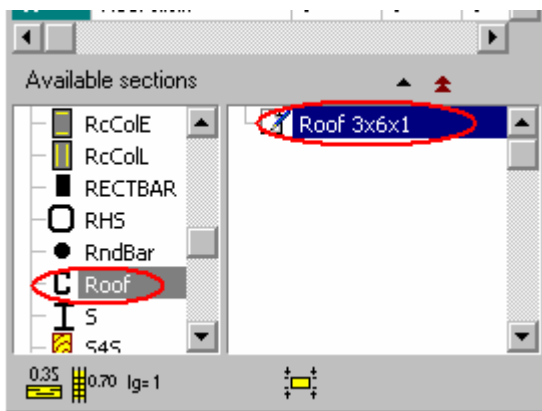
Press OK and notice that a new section "ROOF 3x6x1" has been created and saved into the sections database. A new "ROOF" group, which will contain all profiles of type "ROOF", has been created.

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 Advanse automatically creates a new group.

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




Select roof beams

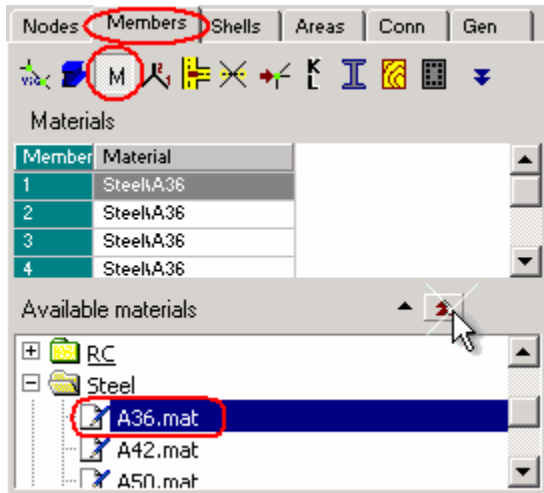



Assign the new profile by pressing 

17) Assigning materials




In this case, all material elements are of steel grade A36. To assign 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 



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

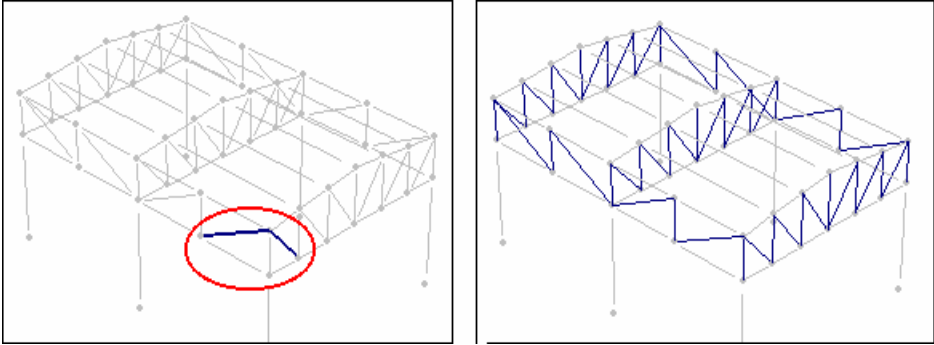
Material “A36.Mat” from folder Steel has been assigned to all elements.

Note. - To hide the section and material names on the screen, toggle the  and  buttons up or press .

18) 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 

Nodes **Members** Shells Areas Conn Gen


        

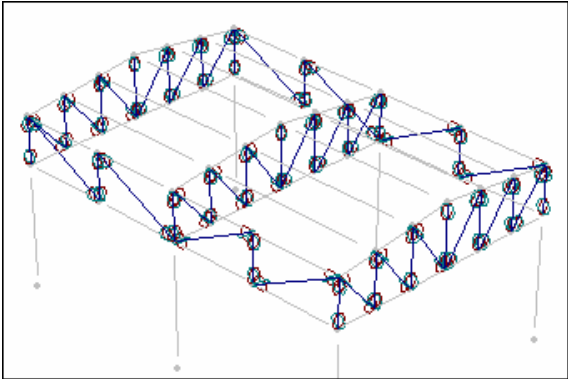
Hinges

Member	T. only	JM3	KM3	JM2	KM2	TOR	JV2	KV
2	0	1	1	1	1	0	0	0
5	0	1	1	1	1	0	0	0
17	0	1	1	1	1	0	0	0
18	0	1	1	1	1	0	0	0
19	0	1	1	1	1	0	0	0


Go to the Spreadsheet **Members/Hinges** (Releases) and press button 

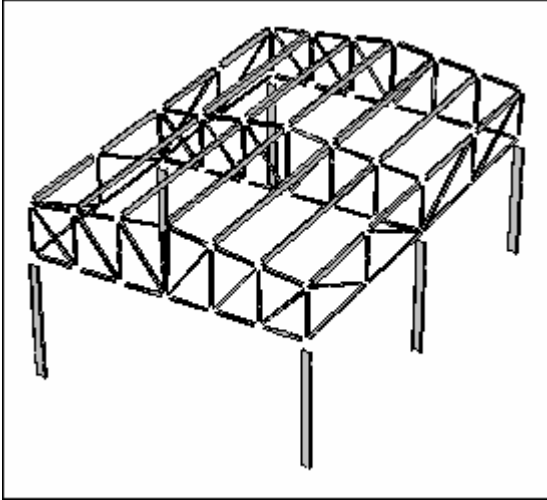
Note. - To rigidly connect pinned elements, press 




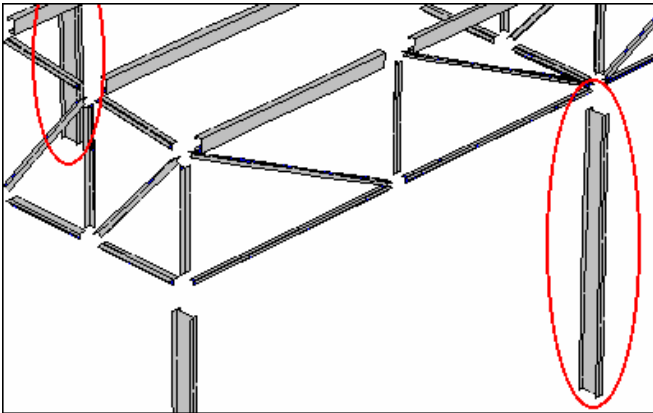
Elements have been pinned

19) Rotating columns

If you press  (in the Rendering toolbar) you'll see the elements with three-dimensional sections displayed. This allows you 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, we will rotate the middle columns by 90 degrees.

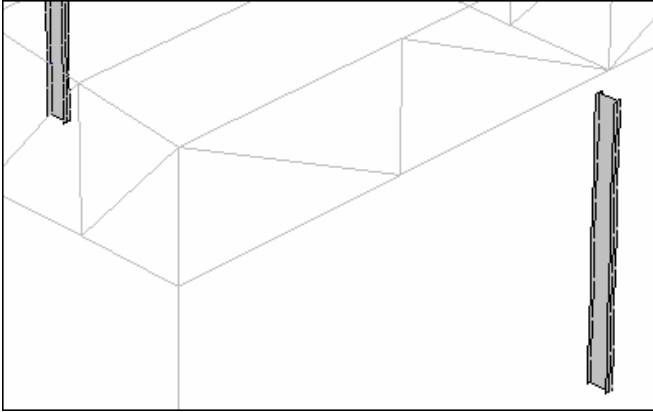


Press button  to see the element profiles in 3D.

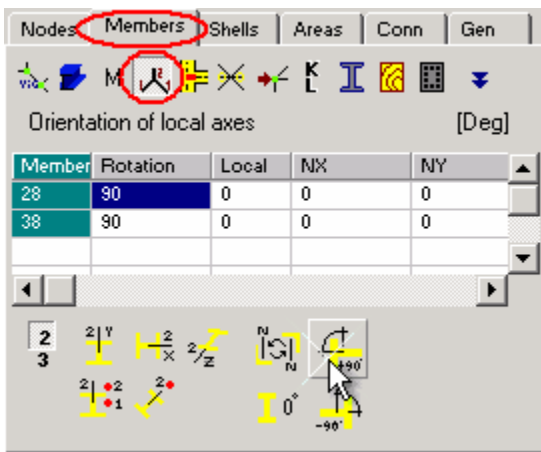


Columns in the middle will be rotated 90 degrees.

To rotate 90 degrees, proceed as follows:



Select columns to be rotated

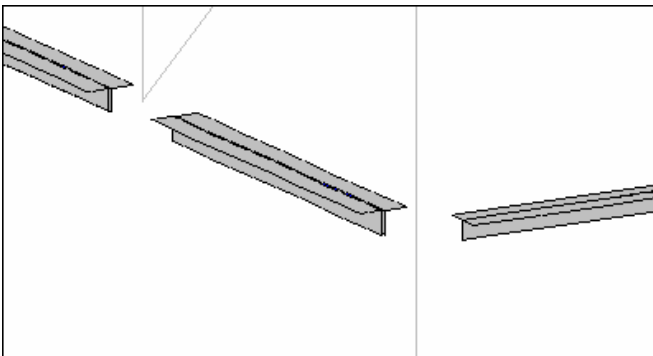


Go to the Spreadsheet Members/Local axes and press button



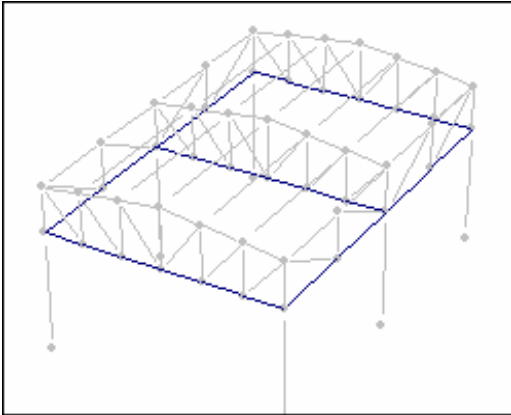
20) Rotating beams 180 degrees

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

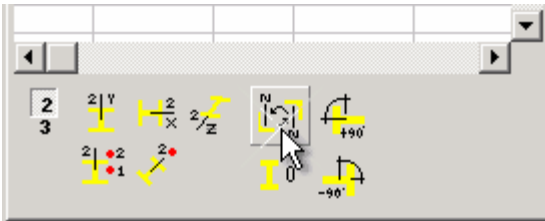



BEAM2, BEAM3 elements need to be rotated 180 degrees

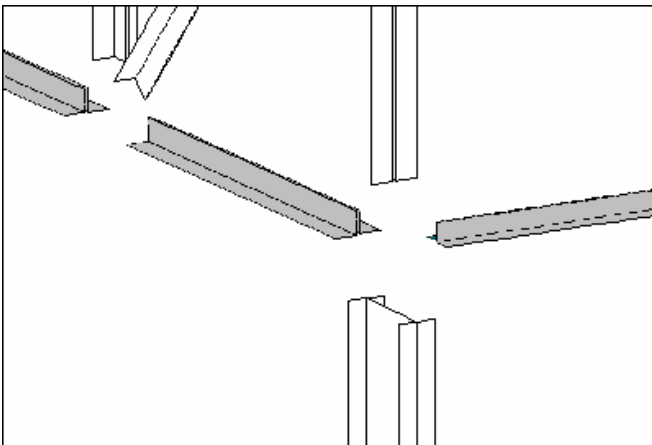
To do this, follow the next steps:




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



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



Elements have been rotated 180 degrees.

Note. - Notice that you can also rotate by entering the required angle in the spreadsheet and pressing  to fill the column with the entered value.

21) Entering loads

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

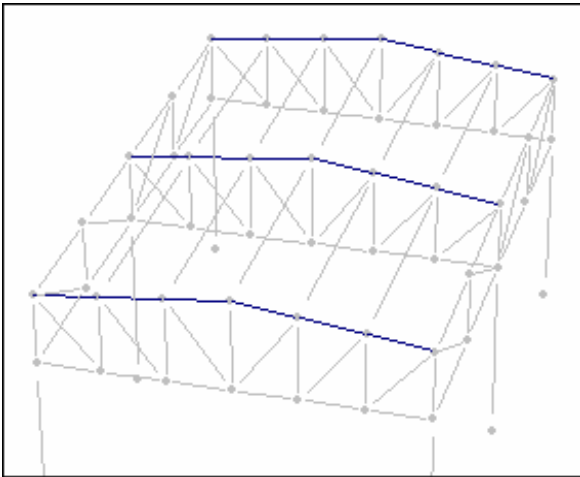
Notice that RAM Advanse automatically creates a load case named "Dead Load". Therefore, it isn't necessary to create it. Later we will see how to create a new load case and a load combination.

Before entering a load, you should determine if it is a:

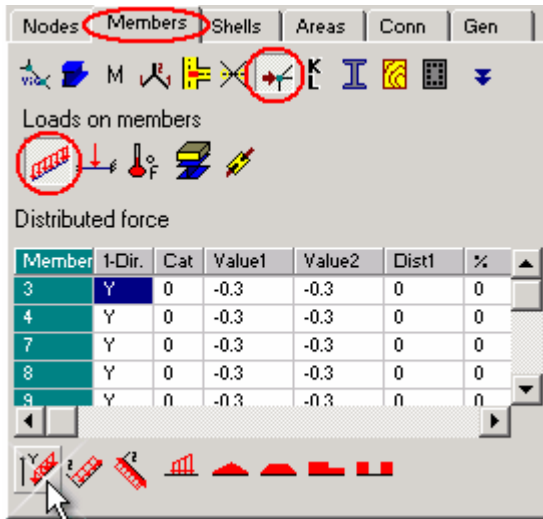
- 1) Load on node
- 2) Load on frame members, or
- 3) Load on shell elements.


Load on frame members

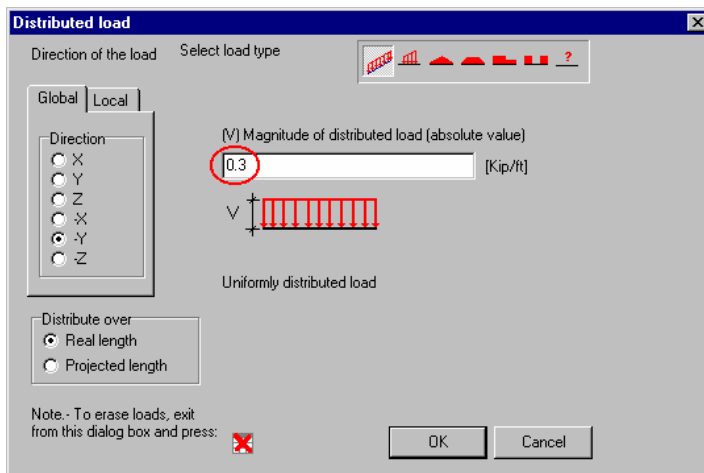
To enter loads on frame members, proceed as follows:



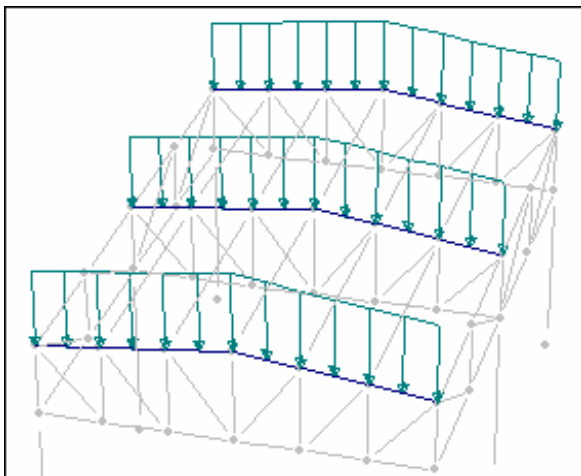
Select frame members where the load is acting. In this case, select beams on top of the truss.



Go to the Spreadsheet Members/Loads on members and press the  button



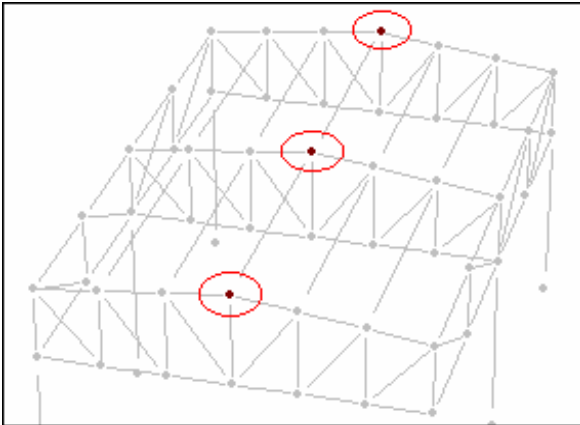
Enter the value of the distributed load (do not enter the minus sign). Then press OK.



The load has been entered.


Load on nodes

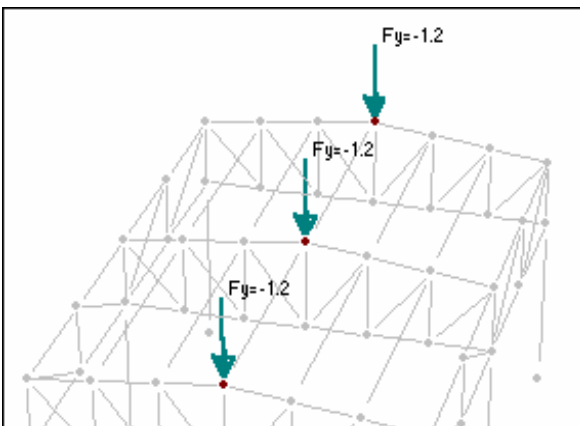
To enter load forces on nodes, follow the next steps:



Select the nodes on which the force is acting.

Node	FX	FY	FZ	MX
7	0	-1.2	0	0
25	0	-1.2	0	0
27	0	-1.2	0	0

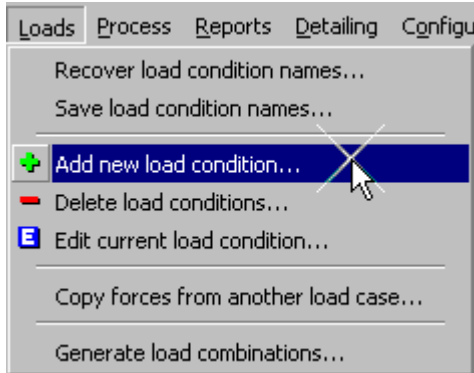
Go to the Spreadsheet Nodes/Forces and moments on nodes, enter a force value (enter the -1.2 value) and press  to fill-in the column



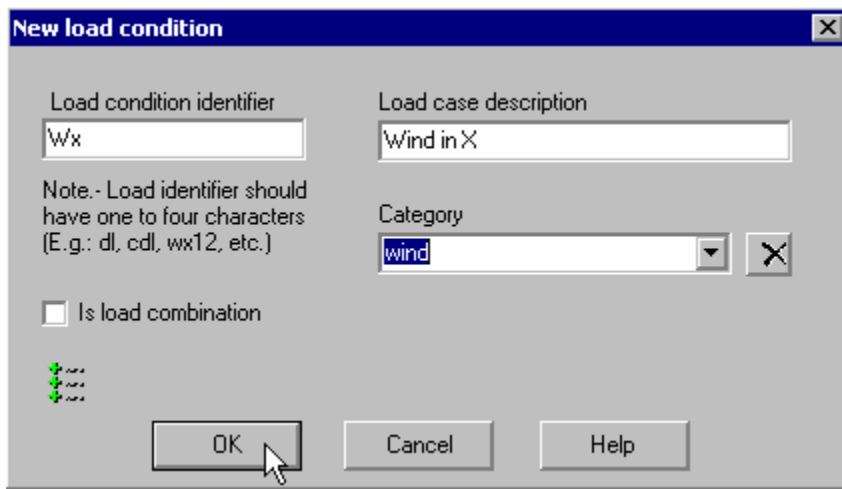
Notice that the force should include its sign. Forces on nodes have been entered.

22) Creating Wind in X load case

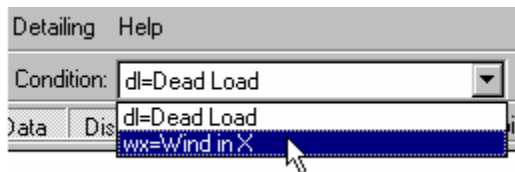
The Second load case acting on the structure is due to the wind force in the X direction. These steps show how to create a new load case:



Select the option to enter a new load case from the menu.



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.



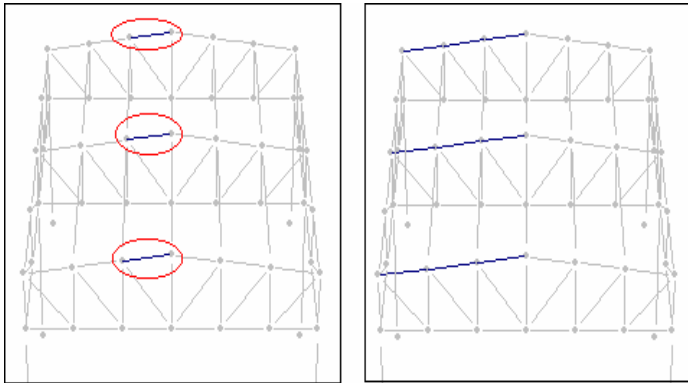
Press OK and you'll see the new load case in the drop-down list.


Note that you have to select a category. This feature is very useful to generate load combinations based on their categories. The engineer 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 has example files (ACIloadfactors.txt, ASDloadfactors.txt and LRFDloadfactors.txt located at main RAM Advanse directory/combo)

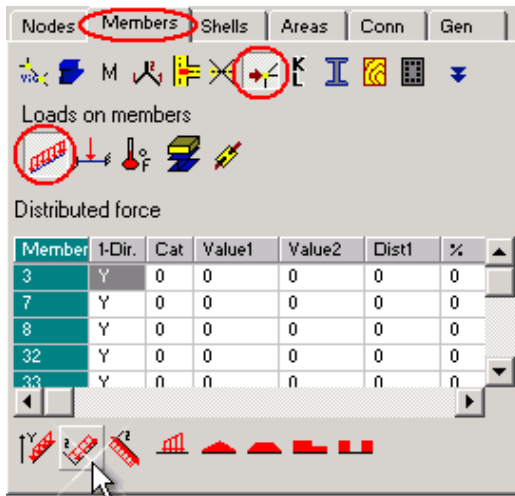
which have the basic load combinations to consider for the different codes. For more details see the chapter of Other Advanced Subjects in the manual.

23) Entering wind loads

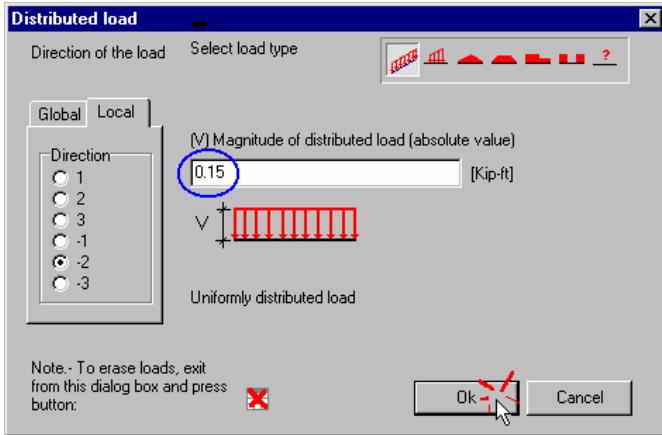
In this case, wind loads are to be applied perpendicular to the roof. There is a pressure of 150 Lb/ft on the left side of the roof, and a suction of 200 Lb/ft on the right side of the roof. Wind load entry is similar to the entry made before in the dead load condition. Notice however that the distributed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



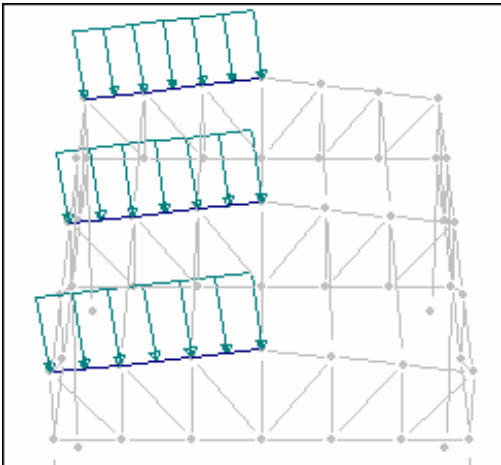
Select the elements on which the load acts. In this case, select one member of each portal and press  to select the aligned elements



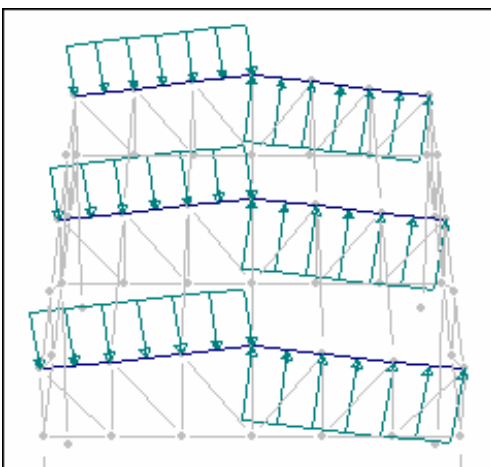
Go to the Spreadsheet *Members/Loads on members* and press button 



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.

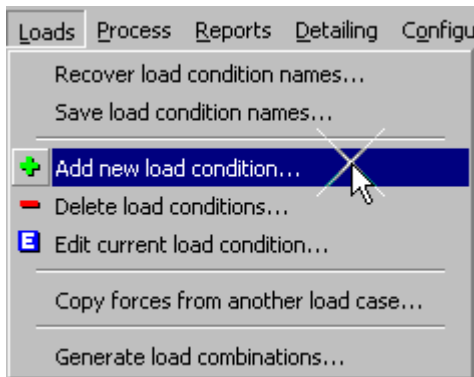
Notice that you should press button  instead of button  to enter suction.

24) Creating load combinations

In this example, we will create the following load combination:

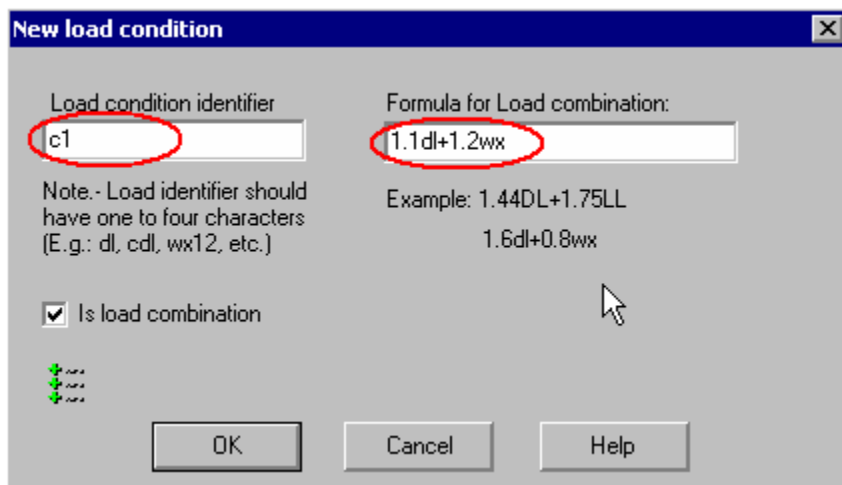
1.1dl + 1.2wx (1.1 times dead load plus 1.2 times wind in X)

To create it proceed as follows:



Select the option to enter a new load case from the menu.

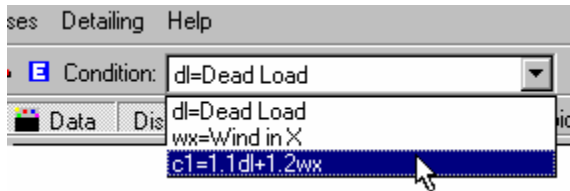
Click on the “Is load combination” check box, a new dialog window will be open.



In the dialog window, enter the following information (in this case enter the values 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 that describes the load combination (1.1dl+1.2wx).



Then press OK and you'll see the new load combination in the drop-down list.

Notice that the formula can contain the minus sign. E.g. "1.1dl -1.2wx" is a valid combination formula.

Note- When working with an allowable stress design code, the engineer must multiply all load cases factors by 0.75 (1/1.33) when creating load combinations that include lateral load cases. This is to account for the 1/3 allowable stress increase, which is not currently accounted for in the member capacity calculation.

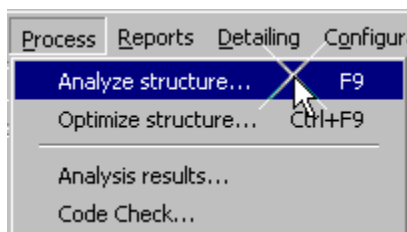
Note. - It is not possible to enter or edit loads data while "a load combination" is selected as the current load condition. Notice that the introduction of loads is disabled.

25) Analyzing the structure

After you entered the structure, you are ready to analyze, design, optimize and view the results.

It is suggested that after you analyze the structure that you entered, you compare the results with the Example1.AVW file to be sure that the data was entered without subtle errors.

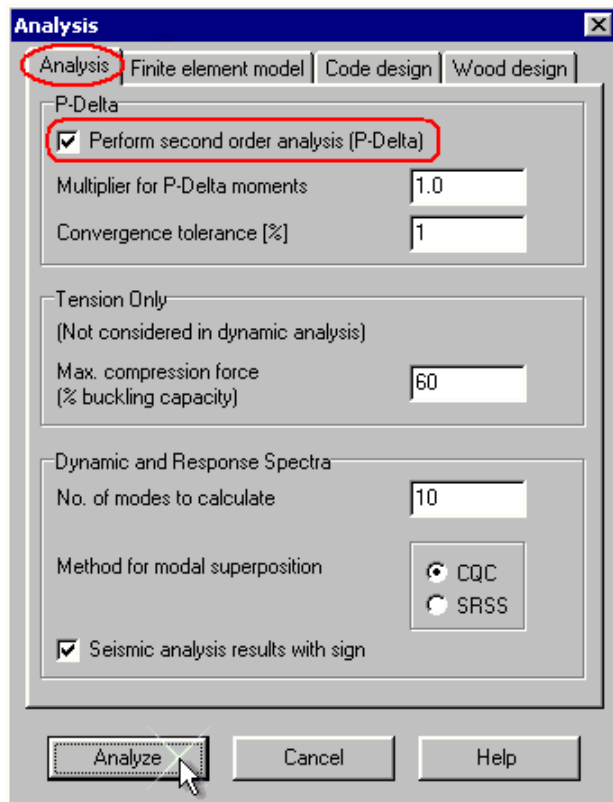
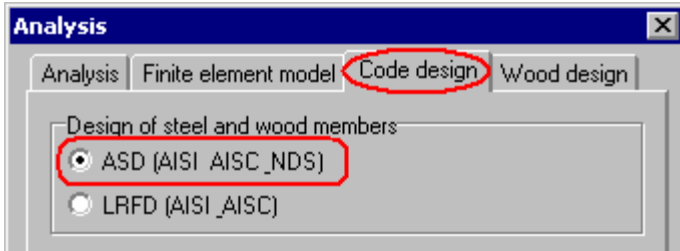
To analyze your structure proceed as follows:



Execute Analyze Structure

In this example we will perform a Second Order Analysis (P-Delta). This analysis takes longer to analyze a structure as it involves iteration but 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.

Note that you can specify the code to be used in the design of steel members. You can choose between ASD (Allowable Stress Design) and LRFD (Load and Resistance Factor Design). For this example select ASD (AISI_AISC_NDS).



Select the same options of the graphic above

26) View results graphically

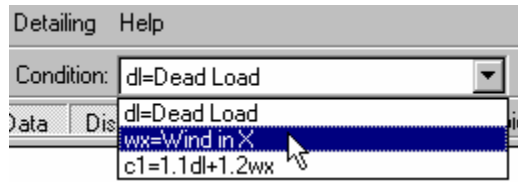
As you can see, several buttons (in the Analysis toolbar) are enabled once the structure has been analyzed. These newly enabled buttons allow you to select what results to display.



Result buttons from the Analysis toolbar are enabled when the structure has been analyzed.

In order to see results graphically, simply click on the button corresponding to those items you wish to see, and then select the elements for which you want to see the results.

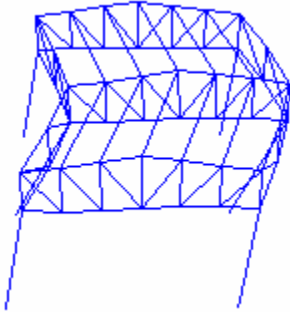
Notice that the selected display options will only be seen on the selected elements.





Select the load condition for which you want results displayed.

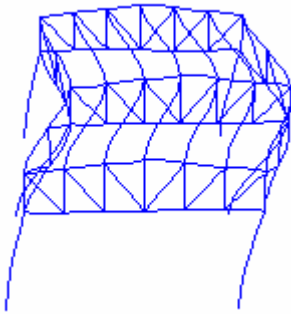
27) Deformed shape



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





To see the deformed shape, press . The graphic shown corresponds to the Wind in X load case

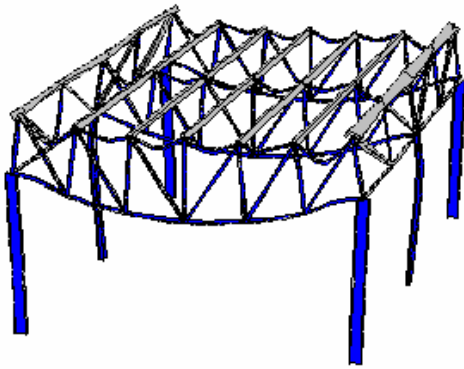
In this view the elements are drawn as lines. To see a more realistic member curvature also depress .





To see the deformed plot with curvilinear lines, depress both  and . The graphic shown corresponds to the Wind in X load case

28) 3D Sections Deformed shape

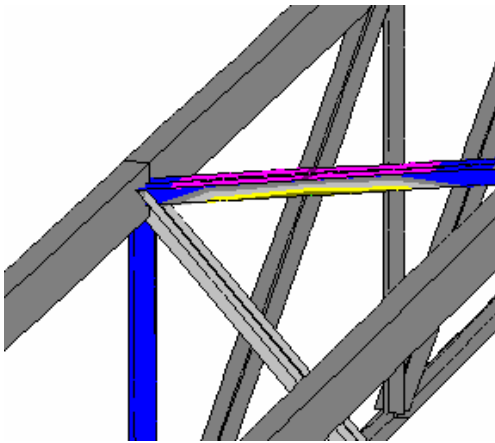
You can also see the deformed shape with the extruded sections. To do this activate (push down) buttons  and  of the Rendering toolbar. Notice that this view may take a longer time to draw.






Activate button  and  to see the deformed shape with three-dimensional section profiles. The graphic shown corresponds to the Dead load case

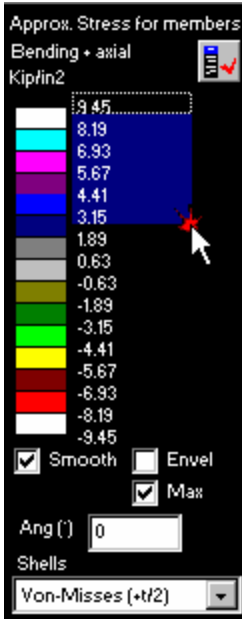
29) Stress


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.

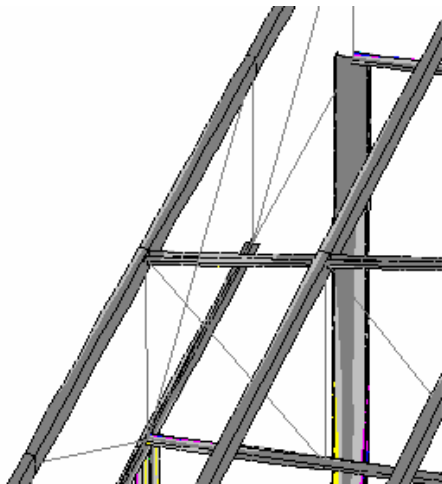


Press buttons  and  to see frame member stresses.


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



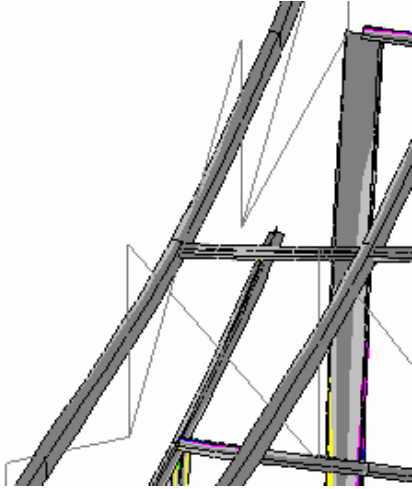
To see element stresses within a certain range, mark the range and press 




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

Note. – If you wish to see only the axial stress (without bending moments, press ).

30) Stress and deformation



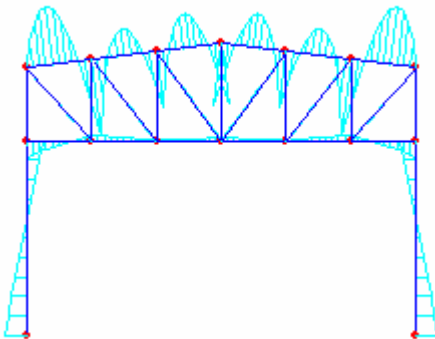
To view stress and deformation of the elements, activate these three buttons 

31) Forces diagrams

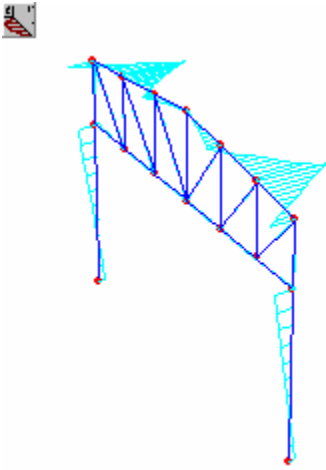


The following buttons allow you to see the forces diagrams of the frame members:

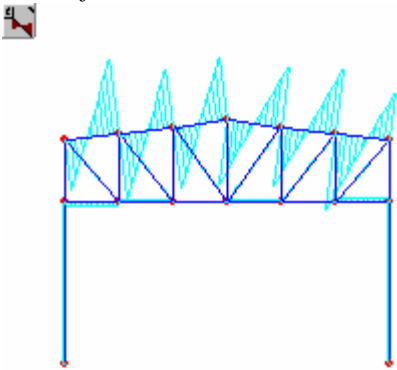
Bending moment around element axis 3 (Typically strong axis bending)



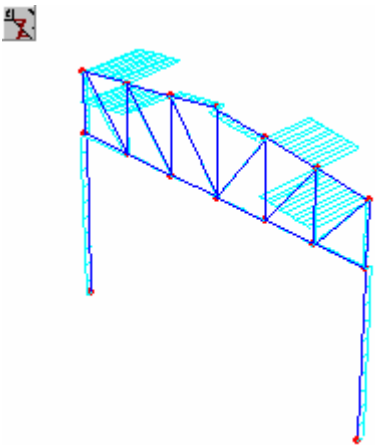
Bending moment around element axis 2 (Typically weak axis bending)



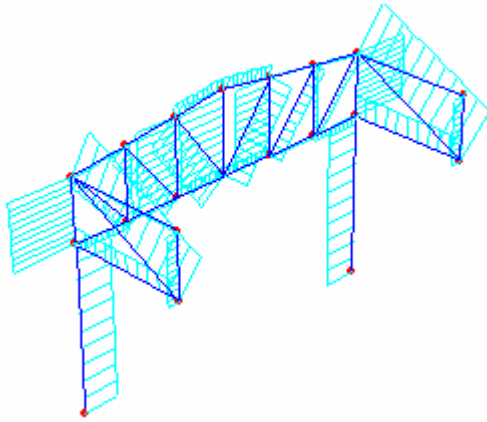
Shear forces in element axis 2 (typically weak axis) (Dead load case)



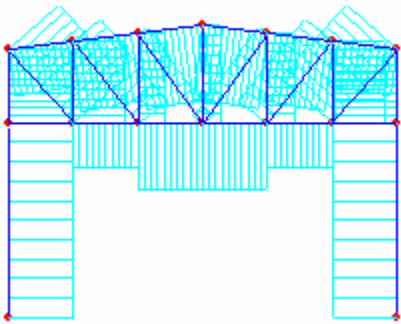
Shear forces in element axis 3 (typically strong axis) (Dead load case)





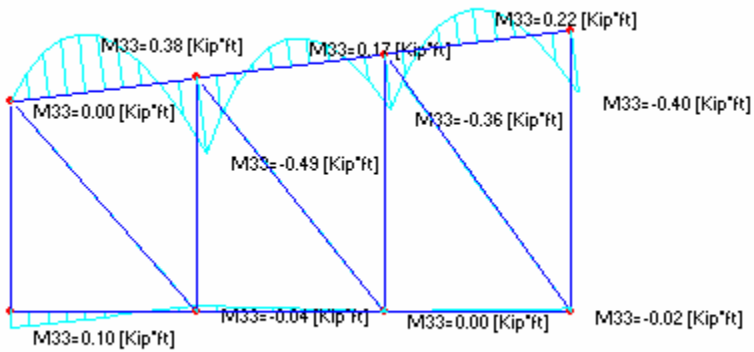
Torsion (Wind in X load case)



Axial forces (Dead load case)




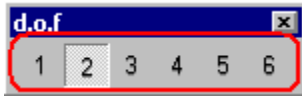
Select  to simultaneously display the magnitude of the forces. Select  to display the units.



Press  to display the force magnitudes. Press  to view units

32) Displacements of nodes

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




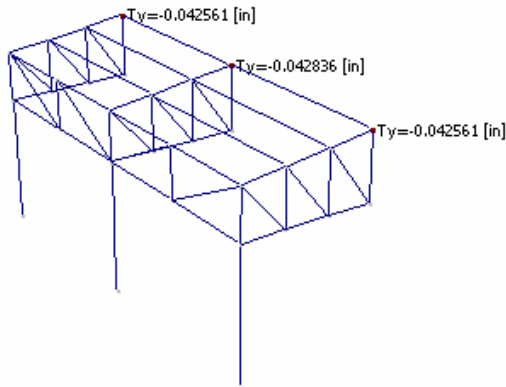
The relation between a degree of freedom and its respective displacement in the global coordinate system is as follows:

- 1: X translation
- 2: Y translation
- 3: Z translation
- 4: Rotation about X
- 5: Rotation about Y
- 6: 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 



Press  and the degree of freedom corresponding to the displacement that you wish to view.

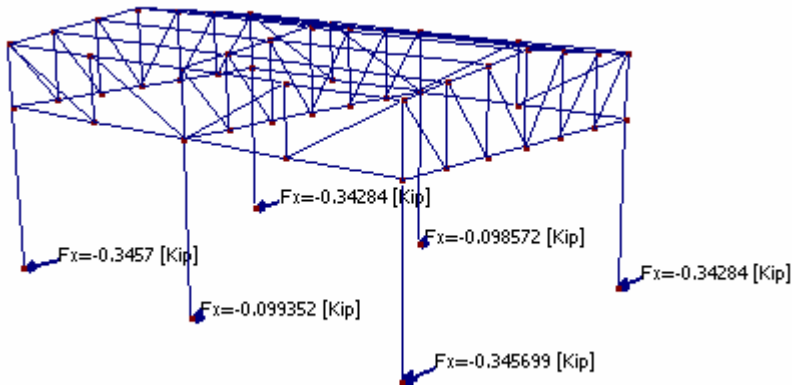
Notice that you can see displacement units by pressing . **Hint:** You can select the specific units of several groups of data and results with the option *Configuration/Units* of the main menu.


33) Reactions

To see reactions, press  and the degree of freedom corresponding to the action that you wish to see.


This is the relation between degree of freedom and force:

- 1: X force
- 2: Y force
- 3: Z force
- 4: Moment about X
- 5: Moment about Y
- 6: Moment about Z



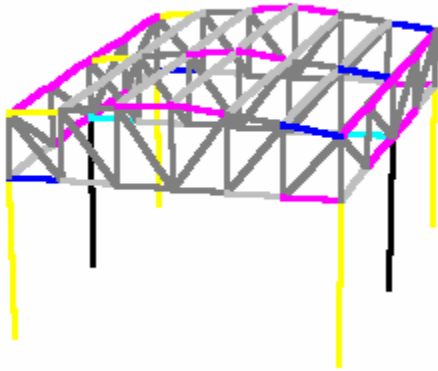
Press  and the degree of freedom that corresponds to the reaction you want to view. (Case: Wind in X)

34) 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 .

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

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





Press  to see the element colored Defl/L ratios.

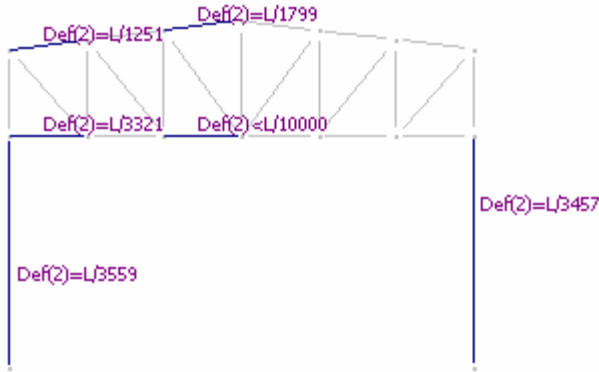


In this panel mark a range of Defl/L ratios and press  to select the elements that have slopes within the marked range.

35) Deflection values

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


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

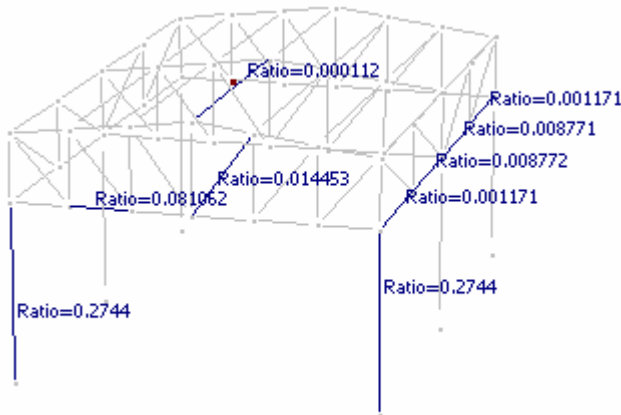


Deflection in function of L for the Load combination C1.

Press  or  to see the Deflection values in local axis 2 and 3, respectively.


36) Design: Interaction Values

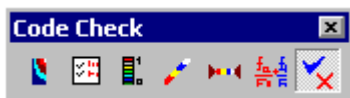
To see interaction values for the currently selected load condition, press .



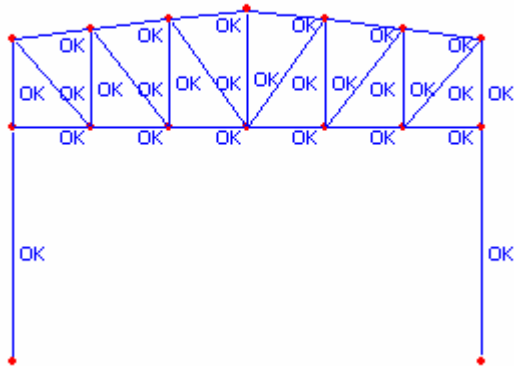
Press  to view interaction values for the current load condition

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

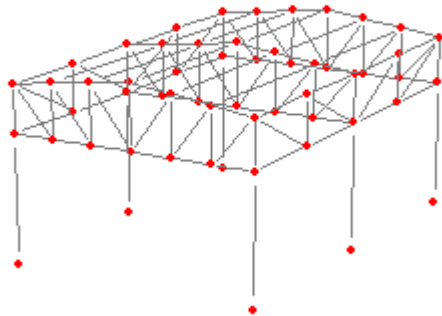
To view the elements that failed code check (for the current load condition), press  in the Code Check toolbar:



Press to view elements that failed code check



Press button to see elements that pass code check



Press button to quickly select all elements that failed code check. A legend with the cause of failure is also shown

The user can print the results of the steel design in a report. To print them, go to the Menu *Reports/Steel Design...* There are two types of reports, Summary Output and Detail Output. 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 engineer 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 and that is located above the original section in the list of sections specified for the optimization. To use the

optimization feature goes to the Menu *Calculation/Optimize structure...* For more details see the chapter of Steel Structure Optimization and Code Check.