

# Multiframe 11.03 Windows

**26 Mar 2009  
Release Note**

---

This release note describes the Windows version 11.03 of Multiframe, Steel Designer and Section Maker. This release will run on Windows XP, Windows 2003/Vista/Windows Server 2008/Windows 7.

## Contents

---

Changes between 11.02 and 11.03.....	2
Changes between 11.01 and 11.02.....	3
Changes between 11 and 11.01.....	4
Multiframe .....	4
Load Panels.....	4
Adding Load Panels.....	5
Selecting Load Panels.....	7
Editing Load Panels.....	7
Load Panel Display.....	9
Load Panel Loads .....	9
Bug fixes and minor changes .....	12
Section Maker .....	12
Steel Designer .....	12
Problem Reports.....	13

## **Changes between 11.02 and 11.03**

---

- Support for SDF version 3 has been added
- Distributed member loads now display correctly when the loads are relatively small
- A problem which could cause bending moment diagrams to display incorrectly when loading a frame with load panels from a file has been fixed
- DXF files containing LINES with OCSs are now read correctly
- Local loads on load panels are now always displayed correctly
- Mbly is now calculated correctly in cold formed design for back-to-back sections when no bending moments are applied
- A problem with incorrect mass of sections generated from tapered sections has been fixed
- Section Maker shortcut in the Start menu now points to the correct path
- Some of the symbol font characters were corrupted when using the inbuilt report window
- Fixed a crash after reading SDF and choosing analyse
- Corrected errors in LRFD code for Slenderness Reduction Factors
- Added MF project file backup before being overwritten
- Fixed a problem with AddLoad in the Automation interface

## **Changes between 11.01 and 11.02**

---

- Node or member renumbering doesn't work properly for load panels.
- In member property dialog window, section groups combo contains wrong section types;
- Steel design menu/command doesn't work properly due to the wrong deflection checks;
- Add some tension  $f_y$  checks into AS4600 and AISI when calculating  $M_{sy+}$  or  $M_{sy-}$ ;
- Correct an error of calculating  $\beta_y$  in AS4600 and AISI of Steel Designer;
- Fix the duplications of thermal loads in print summary;
- Fix a bug of auto-detecting 3D loads for 2D structures during analysis;

## **Changes between 11 and 11.01**

---

- Load panel types were not copied correctly during load panel mirroring.
- Load panel corners can only be allowed to set on the nodes of the structures.
- Minor updates to mfd file I/O reading and writing.
- Placing a double angle section caused crash while taper is not zero.
- Fix to print bug while doing a “Print Summary” printing.
- Removed warning messages about invalid numbers that sometime arise when entering numbers in dialog fields.
- Generation of trusses occasionally failed to add vertical members of truss.
- Incorrect behaviour of pre-solve check for zero stiffness spring restraints.
- Error in adding local distributed loads to design members when using projected load lengths.
- Section combo in toolbar was not populated with sections.
- Some design options were missing from the Symbols Dialog.
- Reordering of static load cases corrupts load cases associated with envelope load cases.
- For envelope load cases the crosshair prints in an incorrect position when printing the Plot window with a member plot displayed.
- Layout of member plots can become corrupted when printing the Plot Window for envelope load cases.

## **Multiframe**

---

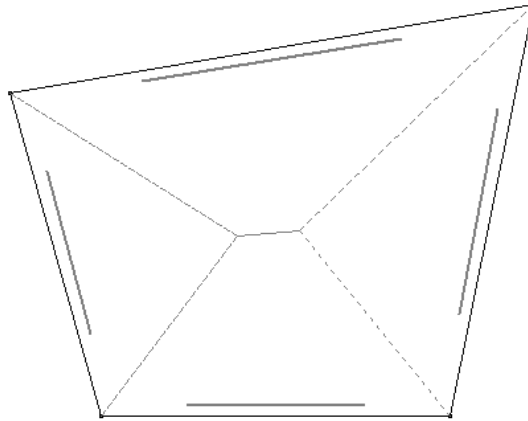
The following features have been modified or added to Multiframe in this release.

### **Load Panels**

---

Multiframe now includes functions to simplify the application of pressure loads to an area which is supported by members of the structure. This is most useful for wind, snow and distributed dead loads. Multiframe provides this functionality by way of load panels. A load panel is a triangular or quadrilateral planar area supported along 1 or more edges by members of the structure.

When the pressure load is applied to a load panel, Multiframe automatically calculates and applies the appropriate distributed loads to the supporting members. This is done using the usual tributary area method where the angles at each corner of the load panel are bisected to create load regions alongside each supporting member. Each member receives the load corresponding to the panel area which is adjacent to it.



Load panel tributary areas

Load panels can also be used for purely visualization purposes, ie. a user could add a panel but never apply any loads to it. This would allow visualization of cladding, floor slabs and shear walls.

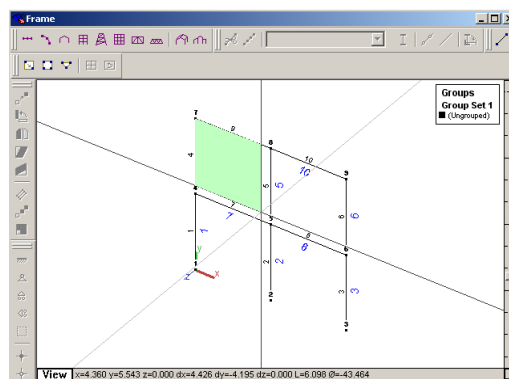
Load Panels are created in the Frame window, and then have loads applied to them in the Load window. The following functions are available for adding, editing and deleting load panels.

### Adding Load Panels

There are several ways for users to add a load panel to the structure. These include rectangular panels, triangular panels, quadrilateral panels, and automatically created a group of panels in a floor or frame. These commands are available from the Geometry menu or from the Load Panel toolbar.

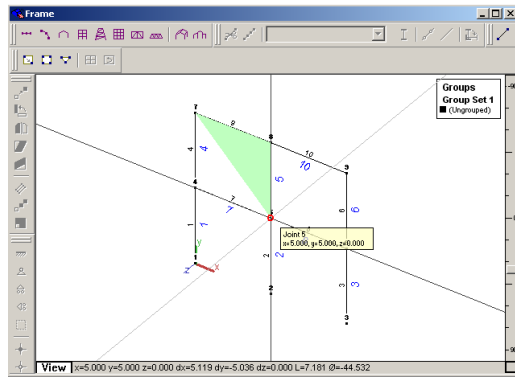
To Add a rectangular load panel

- **Choose Add rectangle Load Panel from the Geometry menu**
- **Click on two nodes at diagonally opposite corners of the panel.**



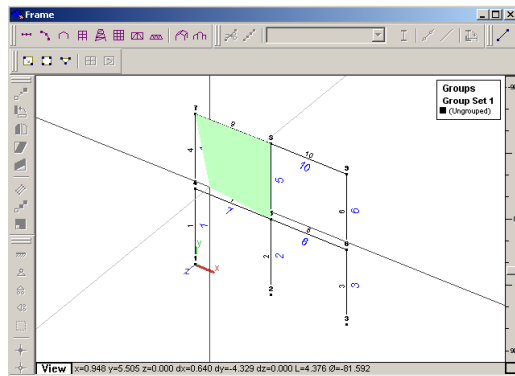
To add a triangular load panel

- **Choose Add 3-node Load Panel from the Geometry**
- **Click in anti-clockwise order the three nodes that define the corners of the load panel**



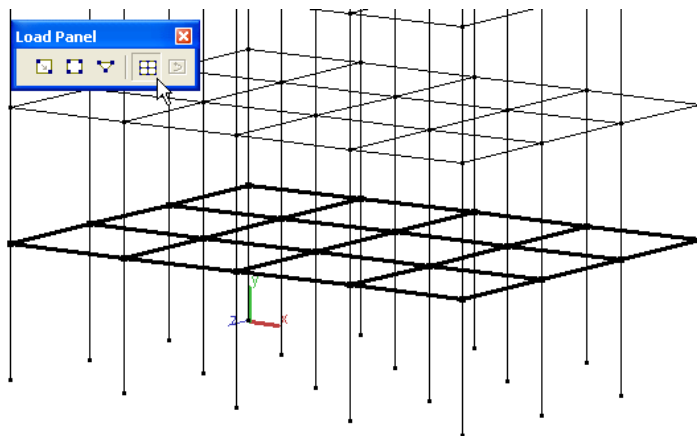
To add a quadrilateral load panel

- **Choose Add 4-node Load Panel from the Geometry menu**
- **Click in anti-clockwise order the four nodes that define the corners of the panel.**

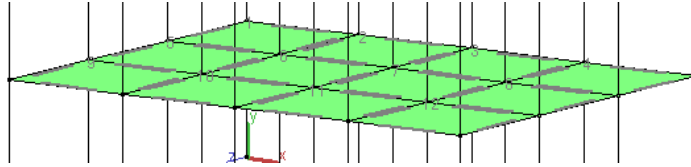


To automatically add load panels between all of the members in a planar region (eg a floor or frame)

- **Select the members which define the planar region**



- **Choose Auto-generate Load Panels from the Geometry menu**



### Selecting Load Panels

---

There are many techniques and commands that are used to selecting load panels within a graphical window. The most common way of selecting items is by using the mouse. To select a single item

- **Click on the item**

The selected panel is drawn with the current selection colour to make it appear highlighted. A different colour for drawing the selection can be set via the Colour command in the View menu.

To extend or reduce the selection

- **Shift-click on an unselected panel to add it to the current selection**
- **Shift-click on a selected panel to remove it from the current selection**
- **Shift-drag to invert the selection in the selection rectangle**

To select a group of items

- **Drag from left to right a rectangle which encloses the panels to be selected**
- **Drag from right to left to select all load panels which intersect or are contained inside the selection rectangle**

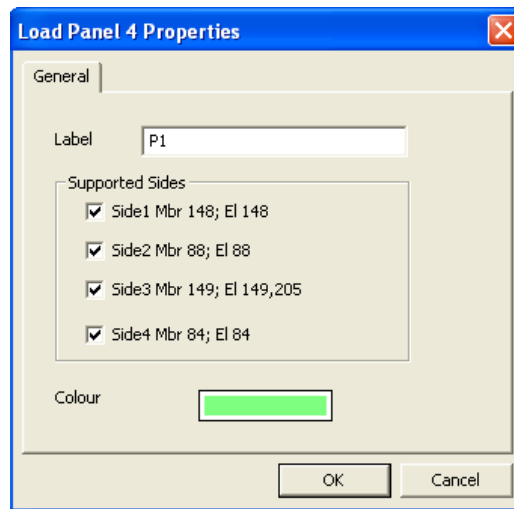
When performing a drag selection there are two techniques for items. The first is by using a rectangular box as described above. An alternative to this is to use a line selection in which the user drags the end of a straight line across the screen. In this case all members that intersect this line are selected and well as all joints connected to these members.

### Editing Load Panels

---

To change the properties of a load panel

- **Double click on the panel in the Frame window, or**
- **Select the item and choose properties from the Edit menu**



You can edit which sides the panels is supported on, enter a label for the panel, and also edit the colour of the panel.

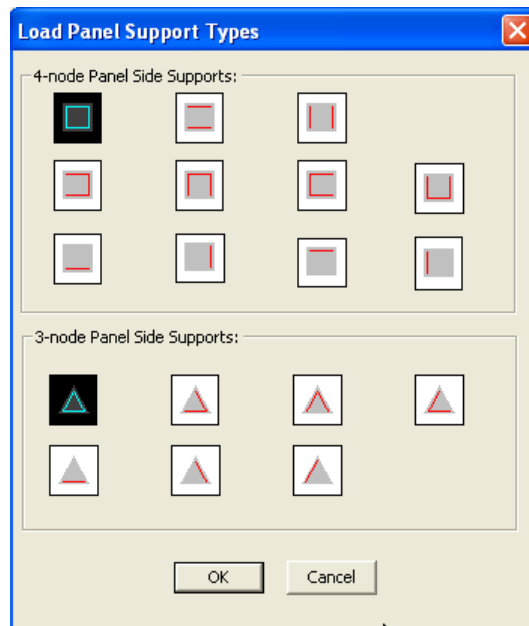
You can also edit which members support the edges of the load panel by using the Rotate Panel Supports icon in the Load Panel toolbar.



This will change the supports o the panel so that they all move one place anti-clockwise. This is most useful when used twice to change the supporting direction of a 2-way load panel.

You can also change the supports of the a group of load panels

- **Select a group of panels to be changed in the Frame window**
- **Right click and choose Load Panel Supports from the pop up menu**
- **Click on an icon to change which edges of the panels are supported**

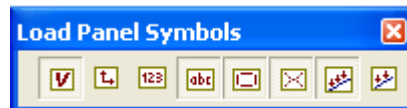


You can also use the same technique to change the labels and colours of a group of load panels by choosing the corresponding items from the pop up menu.

### Load Panel Display

---

Multiframe allows you to control the display of a range of symbols associated with load panels. These options are available via the Symbols command in the Display menu or more commonly via the icons in the Load Panel Symbols toolbar.



These icons control

- **Visibility of load panels**
- **Visibility of panel local axes**
- **Visibility of panel numbers**
- **Visibility of panel labels**
- **Visibility of panel edge supports**
- **Visibility of panel tributary area boundaries**
- **Visibility of panel pressure loads**
- **Visibility of the distributed member loads generated as a result of the panel loads**

### Load Panel Loads

---

Multiframe allows loading on panels to be applied relative to the direction of the global coordinate system or relative to the direction of the local panel coordinate system.

#### Global Panel Load

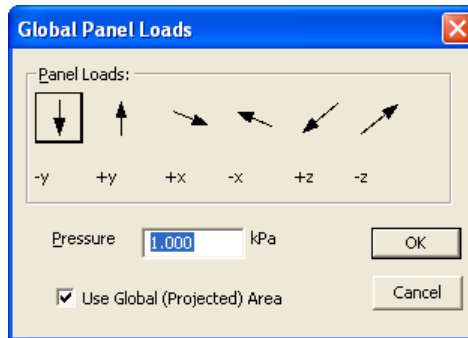
---

A global panel load is a load which is uniformly distributed along all areas of a load panel, and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global distributed load to a panel

- **Select the load panel or panels to be loaded**
- **Choose Global Panel Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



- **Click on the icon which shows the load direction**

In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate axes in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in a value for the pressure**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load values. The directions are determined from the icon that you select.

If you wish to remove the panel loads from a panel, select the panel and choose Unload panel from the Load menu. You can also double click on a panel to view a table of all the loads on the panel.

The axis of the load determines how the load is projected in the panel areas. With Global axes, the magnitude of a global panel load refers to its load per square meters where the area is measured perpendicular to the direction of the load. This means a vertical distributed load applied to an inclined panel will apply a total load equivalent to the magnitude of the load times the horizontal projected area of the panel. For loads applied in local axes, the magnitude of the distributed load refers to its load per square meters where the area is the actual area of the load measured along the panel plane.

- **Click on the OK button**

#### **Local Panel Load**

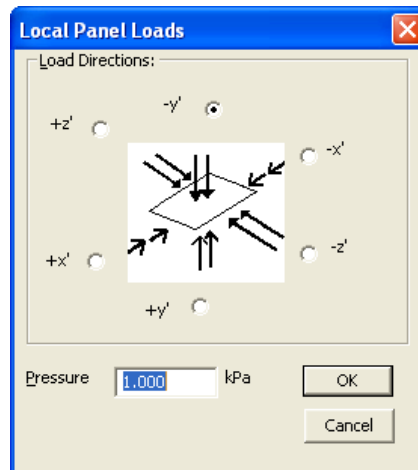
---

A local panel load is a load which is distributed along full area of the panel and acts in a direction either normal (shear) or tangential (friction) to the panel.

To apply a local distributed load to a panel

- **Select the load panel or panels to be loaded**
- **Choose Local Panel Load from the Load menu or short cut menu**

A dialog box will appear with icons to indicate the direction of loading.



- **Click on the icon which shows the load direction**

In a two dimensional view, there will be four icons indicating the four possible load directions. In the 3D view, all six possible icons will be displayed with the icons pointing in the direction of the appropriate local axes of the panel in the current view.

- **Click on the icon which shows the direction in which the load is to act**
- **Type in a value for the pressure of panel loading**
- **Click on the OK button**

There is no need to enter '+' or '-' signs for your load magnitudes. The directions are determined from the icon that you select.

If you wish to remove the panel loads from a panel, select the panel and choose Unload panel from the Load menu. You can also double click on a panel to view a table of all the loads on the panel.

## Bug fixes and minor changes

---

- Fix to improve the performance of NL analysis.
- Fix to analysis of end springs.
- Fix to apply user sign convention to max actions.
- Minor improved to SDF file format for orientation of channel and angle sections.
- Fix to plot window to draw the plot labels correctly.
- The sign of enveloped actions and stress were not consistent with the sign convention preferences.
- Joint linking is now available when the geometry is locked.
- A crash that occurred when a linear analysis was performed with the results of a buckling analysis displayed has been fixed.
- Member end actions for a design member are now available via the Calc window.
- A crash when zooming using a laptop trackpad shortcut has been fixed
- A problem using Convert Member to Arc which resulted in intermediate nodes connecting to existing nodes has been fixed

## Section Maker

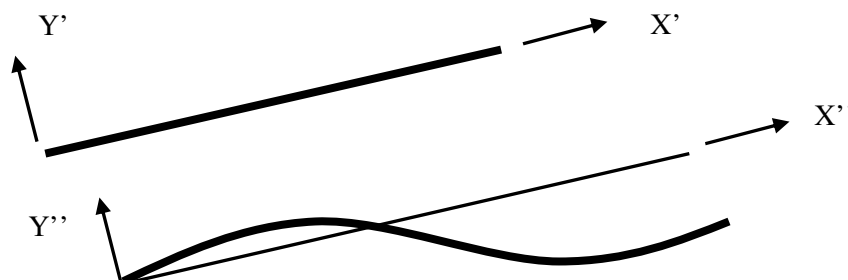
---

- A problem with the display of BetaX labels has been fixed
- An incorrect calculation of  $I_y$  for back to back channels has been corrected
- A dialog asking for a thickness when adding a tube no longer contains a spurious Angle field
- An overly tight restriction on non-zero properties has been relaxed to allow small round bars to be added to the library
- An error in section properties when copying and pasting sections with 5 shapes has been fixed

## Steel Designer

---

The deflection checking for the AII code has been changed to always use the deflection adjusted for the minimum deflection of the end nodes. This means the deflection used will be the deflection relative to the original member axis minus the smaller of the deflections at the ends of the member. Ie the deflection is  $Y''$  as shown below.  $X''$  is parallel to  $X'$  and adjusted for the smaller of the two  $Y'$  nodal deflections.



- **A problem with incorrect reporting of width/thickness ratio checks has been fixed.**

## **Problem Reports**

---

We greatly appreciate any bug reports or suggestions you may have. Please report any bugs or anomalies you find to:

Fax: +61 8 9335 1526

Email: [support@formsys.com](mailto:support@formsys.com)