

Multiframe 9.02**3rd February 2005
Release Note**

This release note describes the Windows version 9.02 of Multiframe, Steel Designer and Section Maker. This release contains many enhancements to this range of software.

Contents

Contents	1
Multiframe	2
Changes from v9.01 to v 9.02.....	2
Changes from v9.0 to v 9.01.....	2
Supported Operating Systems.....	2
Drawing and Dragging in 3D.....	2
Dynamic Line Constraints	3
Snapping and Drawing Options.....	3
Drawing Grid in 3D views.....	4
Drawing Depth Marker.....	4
Graphical Manipulation of Model Geometry	4
Semi-rigid Connections	5
Adding Sections.....	6
Advanced Member Orientation.....	6
Axial Member End Releases.....	7
Member Classification by Structural Component Type	7
Grouping of Design Members	7
Disconnect Members	8
Longer Loadcase Names.....	8
Global Distributed Loads Applied over Length of Member.....	8
Drawing of Loads	8
Labelling Enhancements.....	9
Customisable Label Fonts.....	9
Member End Properties Table	9
Panning with Mouse	10
Re-analyse Command	10
Modal Analysis - Mode Shape Scaling.....	10
Mode Shape Menu.....	10
Tables of Maximum Actions and Stresses.....	10
Rendering of Global Plot Diagrams.....	10
Dynamic reloading of Section's library.....	10
User numbering of Joint and Members.....	10
Unit and Number Formats	11
Resizable Dialogs	11
Tool tips for Joint Restraints and Springs.....	12
Generate Toolbar	12
New Buttons in Joint Toolbar	12
New Buttons in Symbols Toolbar.....	12
DXF Import/Export.....	12
Export Time History Results to Text File.....	12
Steel Designer	13
Column restraints in AS4100 and LRFD.....	13
Labelling of Design Data.....	14
ASD Deflection Limit.....	14

Section Maker	14
Groups Window	14
Group Properties	14
Dynamic Reloading of Section's Library	15
Resizable Dialogs	15
Problems Fixed	15
Problem Reports	16

Multiframe

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

Changes from v9.01 to v 9.02

A problem with display of the Shape toolbar in Section Maker has been fixed
Saved report files now have the correct file type.
A problem with display of text labels when exporting time history results has been fixed.
Renumbering when a frame contains grouped members no longer gets out of sync with the groups.

Changes from v9.0 to v 9.01

A problem which could cause a crash when placing or modifying grouped items in Section Maker has been fixed.

Supported Operating Systems

Microsoft no longer provides support for the following operating systems

- Windows 95 – Extended support ceased in December 2001
- Windows 98 – Extended support ceased in January 2004
- Windows NT 4.0 – Extended support ceased in June 2004

For some time Multiframe has not been supported on Windows 95 and version 8.6 will be the last to be supported on Window 98 and Windows NT 4.0. It is recommended that Windows XP, Windows 2000 or Windows 2003 be used for this and future versions of Multiframe.

This version will still operate under Windows 98 but problems may arise depending upon your computers configuration. Systems that have not been kept up to date with patches and updates from Microsoft will most likely experience difficulties. If this occurs we recommend that you upgrade your operating system to Windows XP.

Drawing and Dragging in 3D

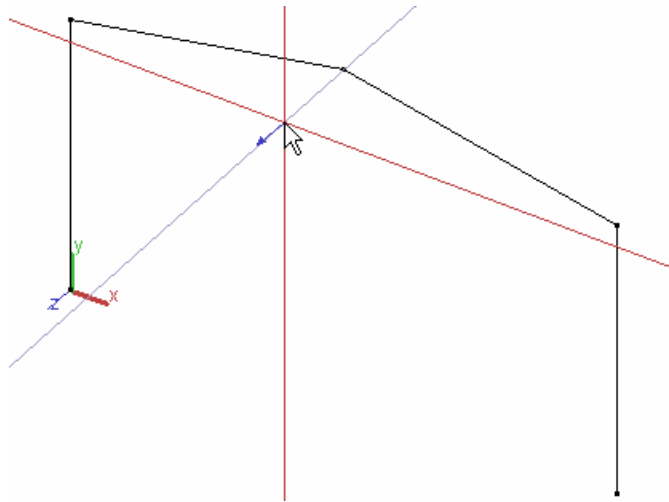
In previous versions of Multiframe drawing in 3D views has been limited to using snap points such as joints or grid points. This restriction has been removed so that arbitrary drawing can now be performed in 3D. In general drawing will occur on the current drawing plane which is the plane aligned with the global axes, that is most normal to the current viewing direction and passing through the current drawing depth. However, if the mouse is initially moved along the direction of the global axis that is perpendicular to the drawing plane (i.e. the global axis pointing out of the screen) then drawing will occur along this direction in 3D space. When drawing out of plane, the colour of the crosshairs will change and a small arrow will be drawn to indicate the direction to which drawing is constrained.

Dragging of joints and members has also been improved to allow similar manipulation in 3D. The only difference is that dragging will default to a plane aligned with the global axes, which is most normal to the current viewing direction and passing through the original position of the point being dragged.

Dynamic Line Constraints

The 3D drawing and dragging described above is in fact part of a more general concept for automatically constraining the direction the mouse moves in 3D space. When moving the mouse from a joint, if the mouse is initially moved in along the direction of one of the global axes then drawing will be constrained to that axis in 3D space. Furthermore, if the mouse is moved in the direction of a member attached to the joint then drawing will be constrained to the direction of that member in 3D space.

When movement is constrained the direction of a global axis or element, a small arrow will be drawn at the cursor pointing in the direction of the line of the constraint. The colour of the arrow indicates is the constraint is in the direction of a global axis or a member.



The drawing constraint will be deactivated if the cursor is moved to far away from the direction of the constraint. This can be avoided by holding down the shift key and this will force the current restraint to remain active.

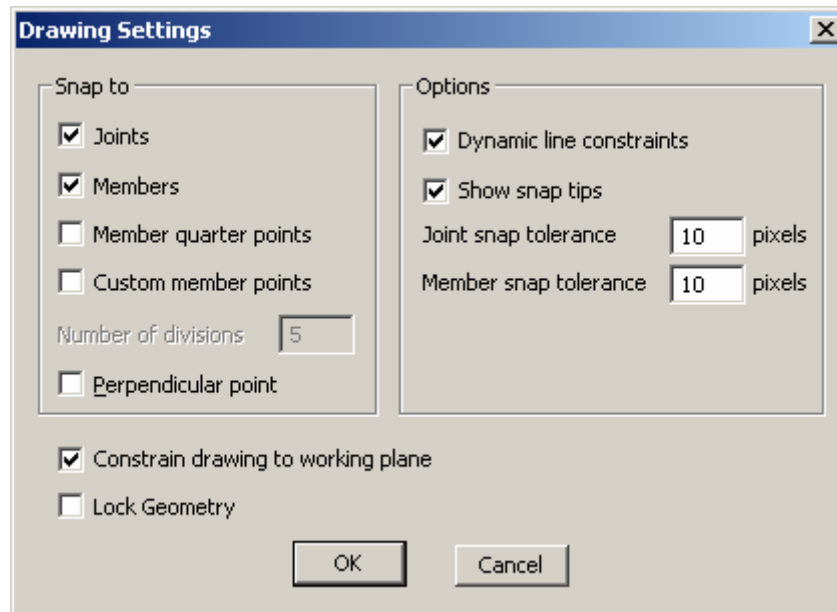
Dynamic line constrains can be disabled via the new Drawing Settings dialog described later in this release note.

Snapping and Drawing Options

A number of options have been introduced to give the user greater control over drawing and snapping. This includes options to

- Toggle snapping to joints
- Toggle snapping to members
- Toggle snapping to quarter points of members
- Toggle perpendicular snap to members
- Lock geometry of the model
- Constrain drawing to drawing depth
- Adjust snap tolerances

These options are control via the Drawing Settings dialog available via the Geometry menu.



The “Constrain drawing to working plane” option forces all drawing in 2D views to occur at the drawing depth. The cursor will still snap to joints and members out of the drawing plane and when this occurs the position of the cursor is simply projected back to the drawing depth. This is useful when working with 3D models as it allows new parts of the model to be drawn in a specific plane but ensures they are aligned to other parts of the structure.

A new drawing toolbar has also been added to Multiframe that provides quick access to some of the new drawing options. The Add member commands are now part of this toolbar.



Drawing Grid in 3D views

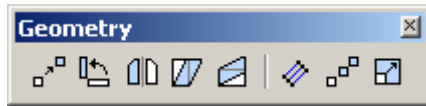
The drawing grid is now available in the 3D views. The grid is applied to the current drawing plane which is the plane aligned with the global axes, that is most normal to the current viewing direction and passing through the current drawing depth.

Drawing Depth Marker

The drawing depth markers, the little triangles displayed between the clipping bars, can now be dragged using the mouse to modify the drawing depth. Full support for snapping to the grid and to the model is provided to allow to depth to be set accurately.

Graphical Manipulation of Model Geometry

A new Geometry toolbar is available which allows you to modify the structure graphically.



This uses the same Move, Rotate, Mirror, Shear, Extrude, Duplicate and Rescale commands available in the Geometry menu, however each of these toolbar commands acts on the currently selected nodes and elements in the Frame window. To use a command

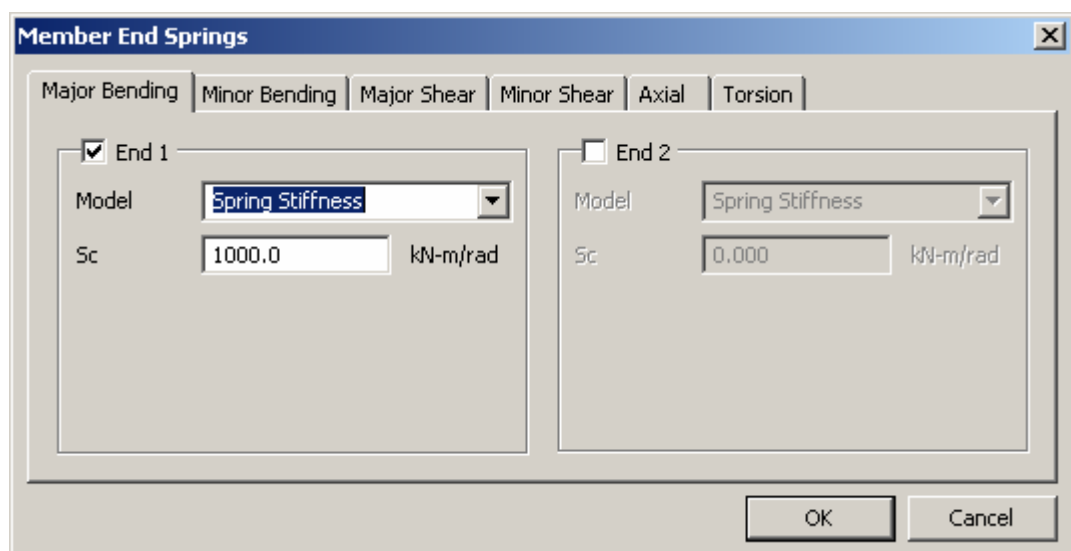
- Select the elements and nodes to be modified
- Click on the required tool in the Geometry toolbar
- Click on any node in the selected group
- For some commands you will then need to click to specify a reference point or line, a tool-tip will prompt you for this
- Drag the selection to the new location
- Click to specify the final location

While you are dragging you can right-click to use other modification options like adding multiple items.

When you are using these tools, Multiframe's new smart snapping options will help you position the crosshair on existing nodes or grid points.

Semi-rigid Connections

Semi-rigid connections can now be modelled in Multiframe using member end springs. The implementation of end springs in Multiframe utilises all six degrees of freedom at the end of an element. This allows the user to model the semi-rigid moment, shear and axial behaviour of a connection. Member end springs are added to a model using the Member End Spring command which prompt the user to specify which element degrees of freedom will include a spring connection and the associate stiffness of the spring.



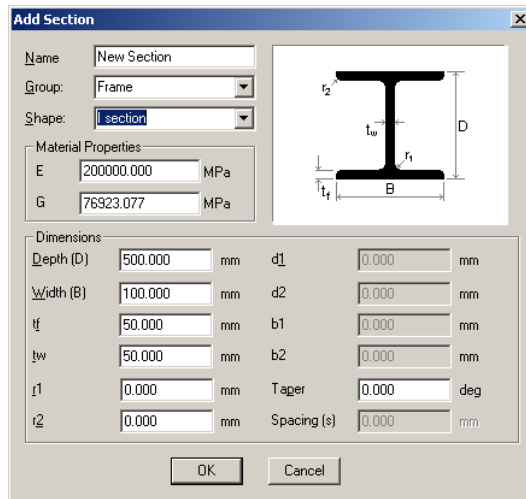
WARNING: This version of Multiframe does not support the analysis of members with both end offsets and end springs defined at the same end of a member.

Member end springs can also be used to model member end releases by activating a spring but specifying no stiffness in that degree of freedom. The use of member end releases should be used in preference to this technique as Multiframe can perform more checks on the structure and the analysis will be more efficient. However, this method provides a convenient way of modelling shear releases that are not supported in Multiframe as a member end release.

Each end spring at the end of a member is considered as a separate entity and as such each end spring can only be associated with a single end of a member. All end springs used in a model are displayed and can be edited via the Member End Spring table added to the Data window.

Adding Sections

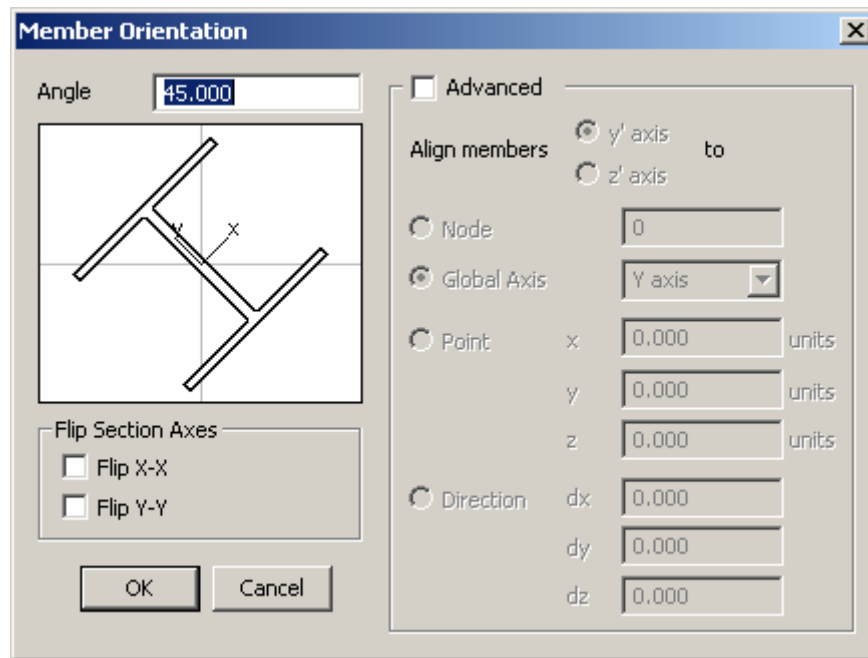
New sections can now be added to the sections library or to the Frame group by specifying the shape and dimensions of the section. The section properties associated with the shape will be computed by Multiframe and inserted into the library.



The new functionality is accessed via the Add Standard Section... command in the Section submenu. Sections with shapes not supported within the program can still be added to the library using the existing Add Section dialog.

Advanced Member Orientation

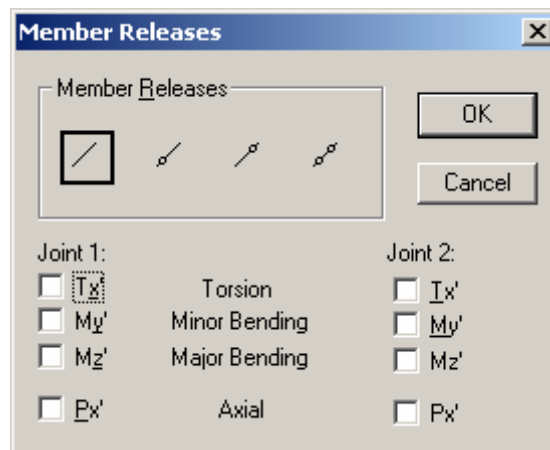
The orientation of members can now be defined by specifying the plane containing the local y-axis or local z-axis of the member. The orientation plane is defined by the local x-axis of a member and either a point in space or a direction. The point in space can be defined by its coordinates or by specifying a joint in which case the orientation is parametric and will update if the position of the joint changes. Similarly, the direction used to define the plane can be specified by a vector or by specifying a global axis direction.



A warning message will be displayed prior to analysis if the orientation of a member is not well defined due to the point or vector used to define the orientation plane being coincident with the local x-axis of a member.

Axial Member End Releases

Axial end releases can now be defined at the ends of members. These are set using the existing Member Releases dialog.



Member Classification by Structural Component Type

Members can now be identified as representing various structural components in a frame. A member can be identified as a column, primary beam, secondary beam, tertiary beam, rafter, tie, brace, truss chord, truss post, or a truss brace.

Grouping of Design Members

The Create Design Member command has been enhanced to automatically group multiple design members. The command will search the current selection for all colinear members that have matching sections, which are rigidly connected and are of the same

structural component type and will group them into a design member. Furthermore, if member component types have been assigned, the grouping shall take into account the dominant member(s) at a joint and use this to determine when to terminate design members.

Disconnect Members

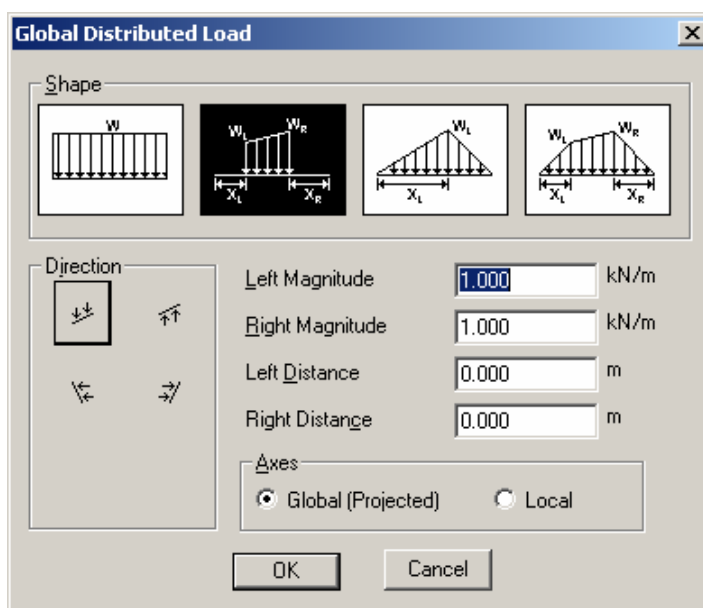
A command to disconnect a selection of members from the rest of a model has been added to the Advanced submenu in the Geometry menu. The command creates new joints at all the joints that connect the selection to the rest of the model. These new joints are then used to define the topology of the disconnected part of the frame.

Longer Loadcase Names

The names of load cases can now be up to 46 characters. The width of the load case combo in the Load Case Toolbar has been widened to support this.

Global Distributed Loads Applied over Length of Member

The application of global distributed loads has previously been based upon the projected length of the member normal to the direction of the load. A new option has been added so that global distributed loads can now have either a global or local distribution along the member. Loads with a global distribution will be applied to the projected length of the member while those applied with a local distribution will apply the load with a variation based on the local geometry of the member.



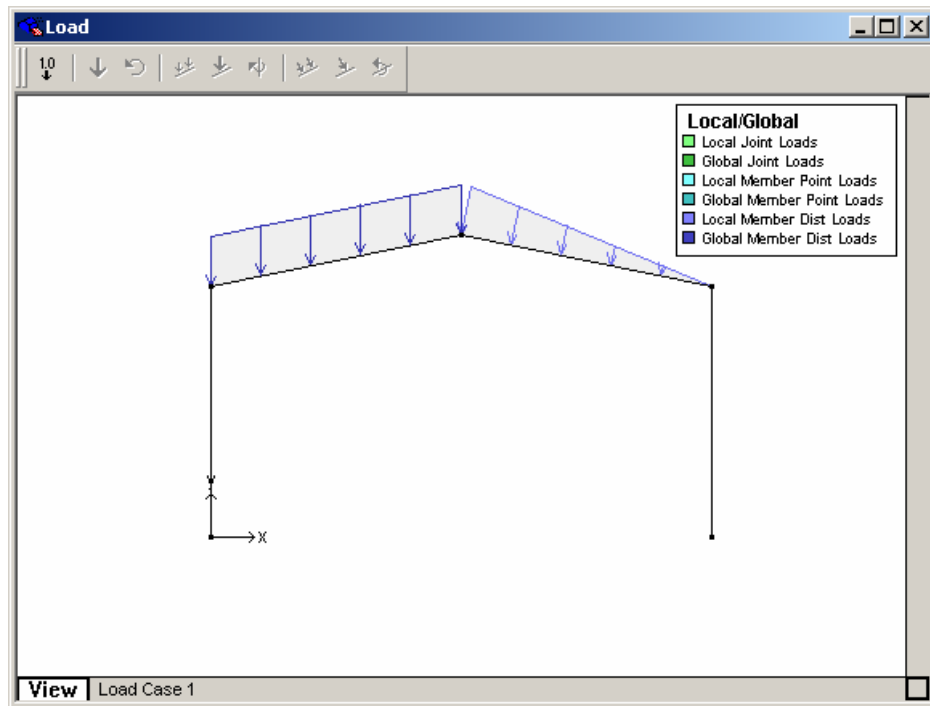
Loads with a local distribution are identified as a separate type of load and are denoted with a subscript 'l' (e.g. W_{xl} , W_{yl} or W_{zl}).

Drawing of Loads

The drawing of distributed loads has been improved. They are now represented as a shaded polygon containing arrows showing the direction of the load (see image below).

The drawing of loads has also been enhanced by the better use of colour. A number of colour schemes can now be used to represent loads. This colour scheme, which is selected via the Symbols Dialog, allows loads to be displayed in colours based upon their

direction, type or their source load case. A legend showing the meaning of each colour can also be displayed via the Symbols dialog.



Labelling Enhancements

A new field has been added to the Symbols dialog to allow the user to display properties of a joint that are not available as separate symbols. When selected, the user can choose from a list of properties associated with a joint including a couple of compound labels for displaying the coordinates of a joint.

Two new fields for labelling the properties of members have also been added to the Symbols dialog. In this case the properties of the member are split between two fields in the dialog with those properties associated with an end of a member (e.g. releases, offsets) listed separately. The labels associated with ends will not be displayed at the centre of the member but will instead be drawn justified to the end of the member to which they apply.

Similarly a field for labelling various properties of design members has also been included.

Customisable Label Fonts

Different fonts can now be used to display labels associated with joints, members and design members. The user can customise the font used to draw these labels via the Customise Font dialog that has replaced the old Font dialog.

Member End Properties Table

A new table displaying the properties of members that are related to the end of a member (e.g. member releases) has been added to the Data window. This table differs from the Member Properties table in that the data for each end of a member is represented on separate lines in the table. Each row displays the joint, releases, offsets and end springs associated with the end of a member.

Panning with Mouse

In the graphical windows the model can be panned by depressing the middle mouse button and dragging the model to a new position within the window. This works the same way as in Autocad.

Re-analyse Command

A command to reanalyse the model using the previous analysis settings has been added to the program. This command is only available via the Load Case Toolbar

Modal Analysis - Mode Shape Scaling

Options for scaling mode shapes such that the maximum displacement is unity or the modal mass is unity were made available via the automation interface in version 8.6. This option has now been exposed in the user interface and is now available via the Analysis dialog.

Mode Shape Menu

The mode shape names in the Mode Shape submenu now include the frequency of the mode shape.

Tables of Maximum Actions and Stresses

Two new tables have been added to the Results window that display the maximum action and stresses within members for the current load case. The values for each member are represented on three rows of the table that display the maximum positive, maximum negative and maximum absolute value of the actions or stresses within the member.

Rendering of Global Plot Diagrams

Rendering of the model in the Plot window is no longer restricted to deflection diagrams. Rendering can now be used with any global plot in which case the plot is automatically displayed as an overlay plot on the un-deflected structure.

The overlay action specified via the Plot dialog now specifies the action that will be used to display an overlay plot on the deflection plot.

Dynamic reloading of Section's library

Multiframe now checks for changes to current sections library and prompts the user to reload the sections library if a modification is detected. This check is performed each time the Multiframe application gains focus on the windows desktop.

User numbering of Joint and Members

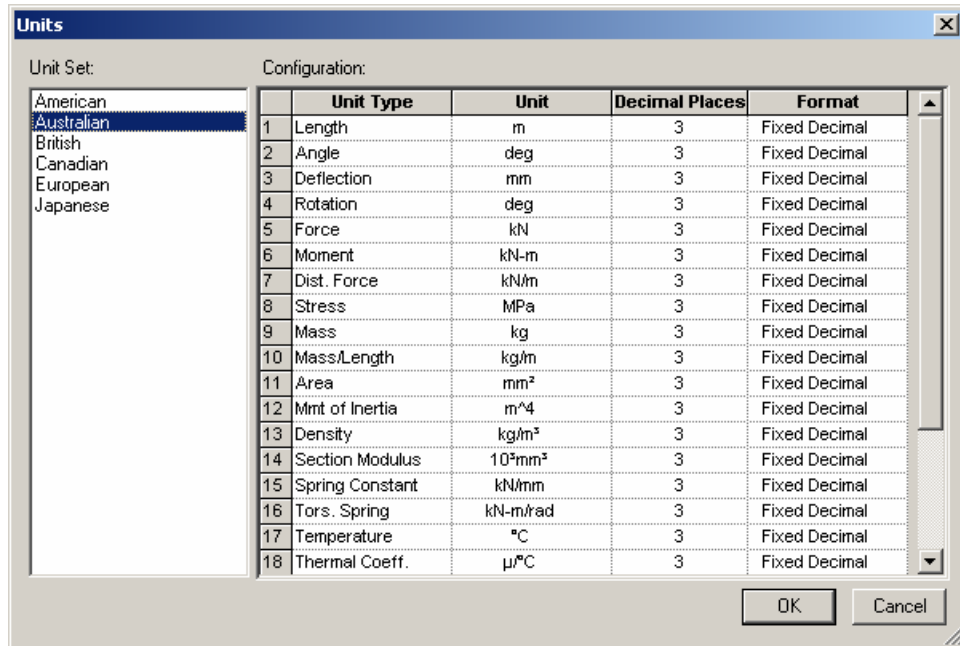
The automation interface has been modified to allow the user to specify the numbering of joints and elements via the Number method. This user numbering can be arbitrary so that the numbering need not be a continuous range. Default numbering of joints and member currently starts at one and is incremented each time an item is added to the model.

To ensure compatibility of all existing scripts it is recommended that the Number method be replaced by the Index method.

Please note that all interaction with the model via the user interface or via the automation interface is still specified using the index of the item and not the numbering.

Unit and Number Formats

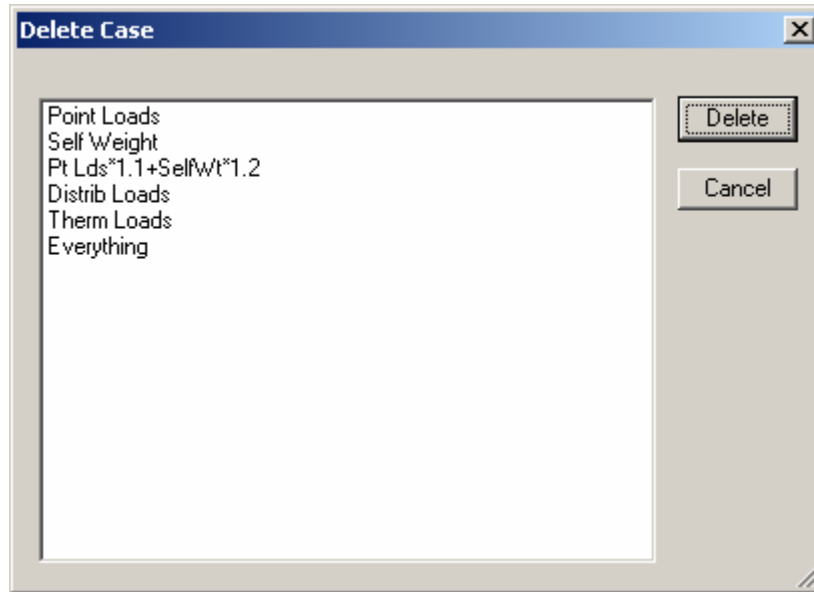
The format of real numbers is now associated with the units of the number. To facilitate this the Units Dialog has been redesigned as a table containing options to set the real number format for each type of unit.



The default number format, which is used for displaying dimensionless values, is now set via the Preferences Dialog.

Resizable Dialogs

Some of the dialogs in Multiframe can now be resized by simply dragging the border of the dialog. These dialogs are identified by a small icon in the bottom right hand corner of the dialog (see picture below). In Multiframe, dialogs that can be resized are generally those containing a table or a list.



Tool tips for Joint Restraints and Springs

Tool tips are now displayed in the Frame Window when the cursor is over a joint restraint icon or a spring icon. .

Generate Toolbar

The Generate toolbar has been expanded to give direct access to many of the new structural types added in version 8.6.

New Buttons in Joint Toolbar

New buttons have been added to the Joint Toolbar which specify vertical and horizontal roller restraints.

New Buttons in Symbols Toolbar

Two new buttons have been added to Symbols Toolbar to toggle the display of joint restraints and member releases.

DXF Import/Export

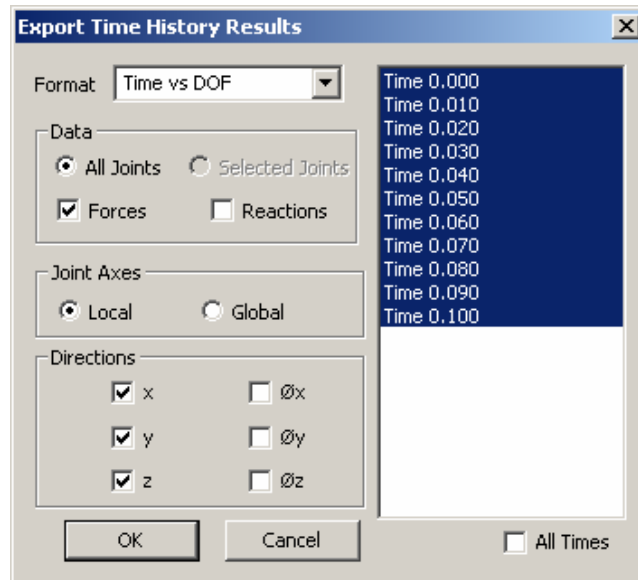
When importing DXF files, Multiframe now labels the new members using the drawing layer from which the member originated.

DXF files exported from Multiframe now include the colours of the members using the current colour settings of the Frame window. Furthermore, if the Frame Window has focus and the Member Legend is visible, the members will be exported using drawing layers, each layer representing an item in the legend.

Export Time History Results to Text File

The results of a time history analysis can now be exported to a text file. The new export command stores the results of the current time history load case. The data to be exported can be controlled by the user who can select which joints, degrees of freedoms and time steps to be saved in the text file. The displacements and/or reactions to be exported can be written to the text file in one of the following 4 formats:

- Time vs DOF – data is exported as a single table with each row representing a single time step and each column representing a single degree of freedom
- Node vs DOF – data is exported with a separate table for each time step, each row in the table contains the data for a single joint.
- Nodal Database – data is exported as a single table with each row representing the data for a particular time step for a single joint
- DOF data base – data is exported as a single table with each row representing the data for a particular time step for a single DOF

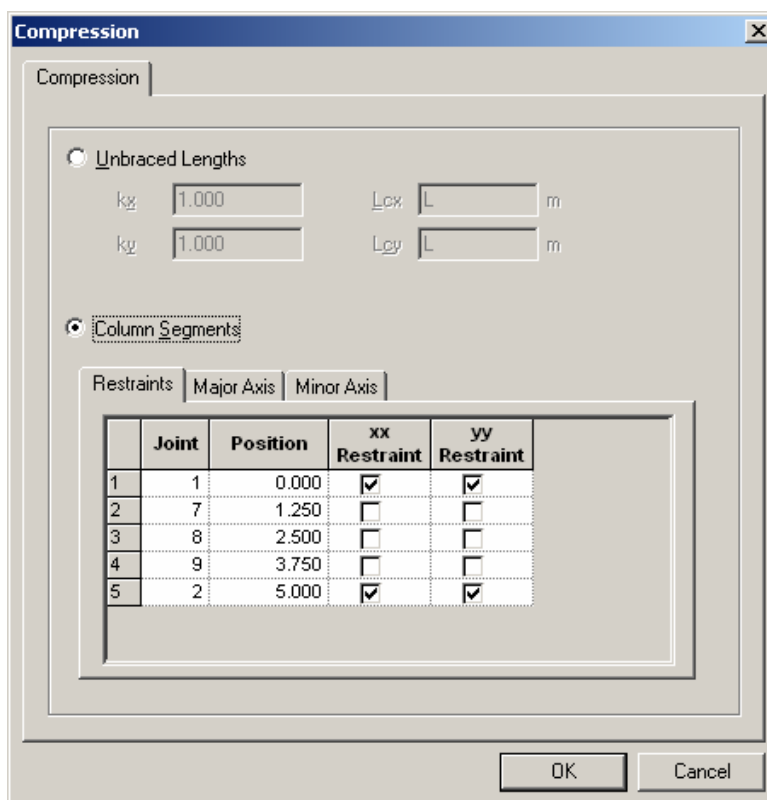


Steel Designer

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

Column restraints in AS4100 and LRFD

The definition of effect lengths for column buckling has been improved to allow the user to specify a number of column buckling segments along the member. The segments are specified by identifying the location of restraints against column buckling. The restraints must be located at joints along the member. While this may be onerous it ensures the model is suitably discretised for a 2nd order analysis, particularly those considering p-delta effects. Each of the column buckling segments will be designed individually in order to determine the capacity of the member for column buckling.



From the restrained nodes the program will determine the unbraced segments for which the user can specify an appropriate effective length factor.

Labelling of Design Data

Further labelling enhancement to those listed above have been added to the Symbols dialog for labelling members with design properties. A new field has been added to the dialog which, when selected, allows the user to choose a design parameter to be displayed with each member.

ASD Deflection Limit

The deflection limit can now be specified via the bending dialog.

Section Maker

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

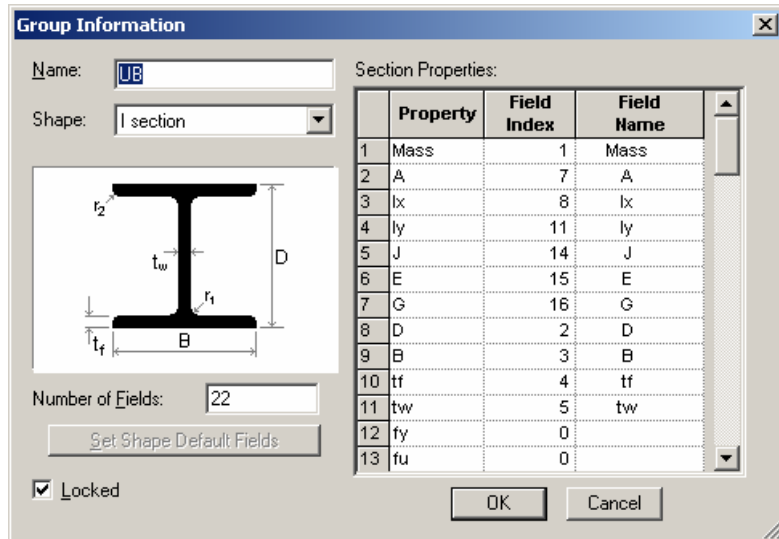
Groups Window

The Group Window has been replaced by a new Groups Window that lists all of the groups, and some of the properties of the groups, in the current sections library.

The functionality that was provided by the old window is now available via a new Edit Fields dialog that is available from the Groups submenu.

Group Properties

The table in the Group Information dialog has been modified so that the field associated with a section property can be selected by name.



Dynamic Reloading of Section's Library

As with Multiframe, Section Maker checks for changes to current sections library and prompts the user to reload the sections library if a modification is detected. This check is performed each time the Section Maker application gains focus on the windows desktop.

Resizable Dialogs

As in Multiframe some dialogs can now be resized by simply dragging the border of the dialog. The dialogs that can be resized are identified by a small icon in the bottom right hand corner of the dialog.

Problems Fixed

This version fixes the following problems experienced with previous versions of Multiframe, Steel Designer or Section Maker.

- Orientation of some section shapes icons was incorrect.
- Indentation problems with design report when checking multiple members.
- Incorrect DOF associated with buttons in Restraint dialog (Multiframe2D only)
- SetSection method of element object in automation interface would not accept single section by name.
- Units used to display torsion in Calc Window were incorrect.
- A crash generating a frame with more than 27 floors has been fixed
- A situation where an newly drawn element could snap on to a node that was not directly under the crosshair has been fixed
- Incomplete import of some DXF files.
- When printing windows the output sometimes prints to a small region at top of page. Typically only when printing rendered views.
- Grid label bubbled where not offset from model when printing.

- **Incorrect calculation of I_{xy} and plastic moduli for open polygons.**
- **Incorrect behaviour of some Boolean properties of objects in the Multiframe automation interface.**
- **LRFD – Moment capacity for section shapes not subject to web buckling now computed correctly.**
- **LRFD –Slenderness checks now based upon column buckling lengths instead of upon full member length.**

Problem Reports

We greatly appreciate any bug reports or suggestions we receive, please keep them coming. Please continue to report any bugs or anomalies you find:

Fax: +61 8 9335 1526 Email: support@formsys.com

You can also lodge your problem reports via our web site at the following location:
<http://www.formsys.com/Multiframe/MFSupport/MFProblemReport.html>

When emailing reports, please attach the frame and sections library with your message.