

Multiframe 7.0 25th January 2000

Release Note

This release note describes the Windows 95/98/NT Version 7.0 of Multiframe, Steel Designer and Section Maker. This release contains numerous enhancements for each of these products.

Contents

Contents	1
Multiframe.....	2
Member Offsets.....	2
Joint Orientation	3
Joint Linking.....	4
Global Moment diagrams	5
Tension and Compression Diagrams.....	5
Overlay Plots.....	5
Plot Window.....	6
Graphical Reactions	6
Member Datasheets.....	6
Geometry Menu.....	6
Symbols Toolbar.....	6
Sums of Reactions	6
Section Colours	7
Colours	7
Selection Tools.....	7
Evaluating Expressions.....	7
Joint Variables	8
Member Variables.....	8
Example.....	9
Report View.....	10
Analysis	10
Steel Designer	10
Design Groups	10
Creating and Deleting Design Groups.....	10
Numbering of Design Groups	11
Editing Design Groups.....	11
Viewing Results Using Design Groups.....	13
Design Group Symbols.....	13
Rendering of Design Groups	13
Design Reporting.....	14
Lateral Restraints	14
Governing Load Cases.....	14
Constraints	15
Section Maker	15
Section Colours	15
Problems Fixed.....	15
Problem Reports.....	16

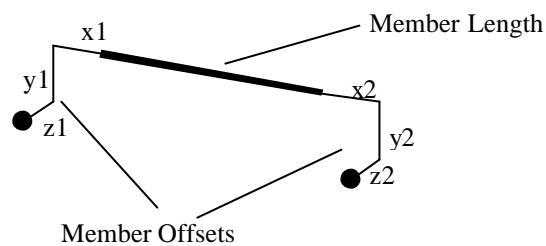
Multiframe

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

Member Offsets

When modeling a structure there are situations in which members do not extend directly between two joints but are instead offset from the joints. In other situations, the intersection between two members may be relatively large and can be considered as a rigid link. In all these instances, the geometry of the model can be specified to more accurately model the structure by using rigid connections between the joints and the members. In Multiframe, these rigid connections are applied using Member Offsets.

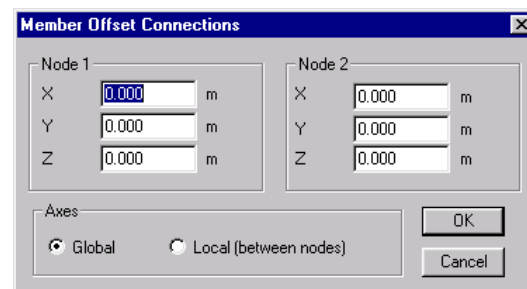
Member Offsets are rigid links between a joint and the end of a member. They are infinitely stiff and do not deform in bending, shear or axially. Loading on members with rigid offsets is applied to the flexible portion of the member only (see Member Length on the diagram below) and distances used to specify the position of loads are measured from the ends of the rigid offsets.



When Member Offsets are specified, Multiframe considers the length of the member to be the length of the flexible portion of the member. (Member Length in the diagram above)

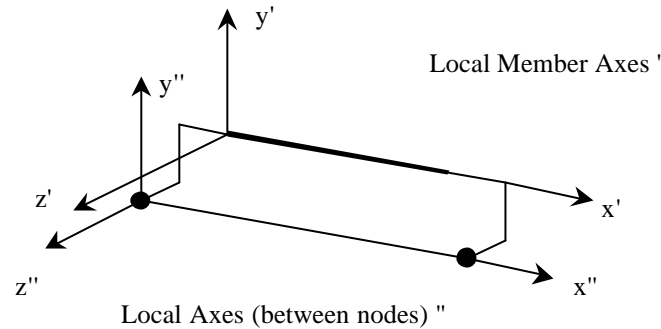
To set the Member Offsets on a member

- **Select the member or members to be modified**
- **Choose Member Offsets from the Frame menu**



- **Click on the radio button corresponding to the axes from which the offset connections will be measured.**
- **Type in the size of the offset connections at both ends of the members.**
- **Click on the OK button**

The size of the offset connections may be specified using global or local axes. If local axes are selected, the offset connections are measured relative to the local axes as defined by the nodes at the end of the members (see Local Axes between nodes below). Note that this is distinctly different from the true local member axes which are aligned with the actual position and orientation of a member after the Member Offset has been applied.



Member Offsets are not available in Multiframe2D.

Caution: Results of analyses including Member Offsets can be read into older versions of Multiframe. This could cause some confusion as the older version of Multiframe will not recognise the Member Offsets but will consider the results to be valid.

Joint Orientation

When modeling structures it is sometimes convenient to define restraints that are not aligned with or skewed from the global coordinate system. These types of restraints can now be specified using the new Joint Orientation feature that has been incorporated into this version of Multiframe. The Joint Orientation is used to specify an arbitrary local coordinate system at each joint. This local coordinate system is then used for applying boundary conditions and loading to the structure.

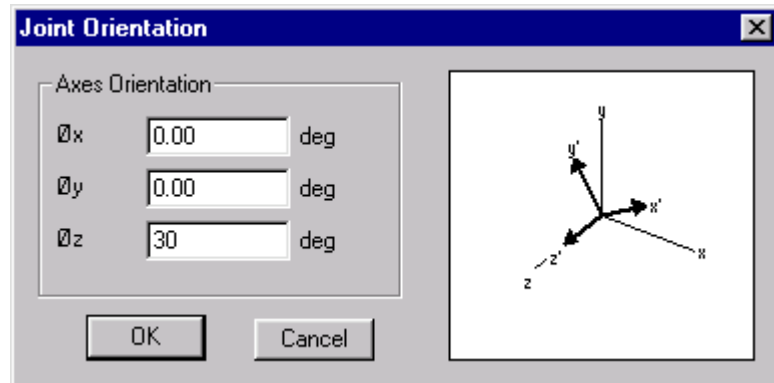
All restraints, springs and prescribed displacements that are applied to a joint are now specified in the local coordinate system. The local orientation of a joint can also be used for defining joint loads. As with member loads, joint loads can now be specified in either local or global coordinates. This is particularly useful when working with curve or cylindrical structures.

The orientation of a joint is specified by 3 angles that measure the rotation of the local coordinate system from the global coordinate system. The right hand rule is used to determine the direction of the rotations. By default, all joints are initially aligned with the global axes.

To set the orientation of a joint.

- **Select the joint**
- **Choose "Joint Orientations..." from Frame menu.**

- Enter the angles defining the orientation
- Press OK



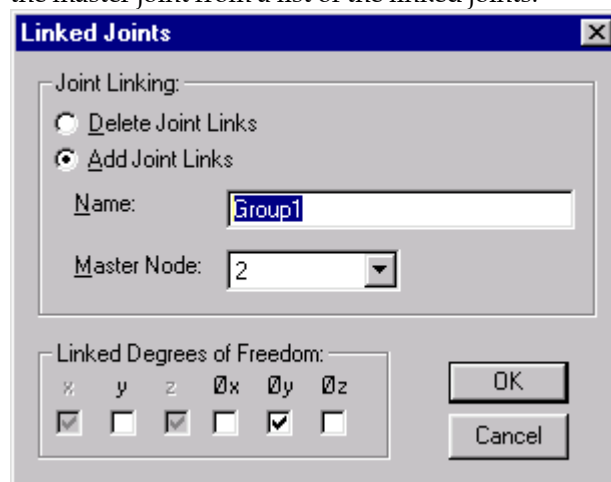
The orientation of a joint is displayed in the Frame and Load Windows. If the orientation of a joint is not aligned with the global coordinates then the axes of the local coordinates are displayed at the node. An option has been added to the Symbols Dialog to turn off the display of the local joint axes.

The joint displacements and joint reactions displayed in the Results Window are now displayed in local joint coordinates. Global results for each of the displacement of reaction components are displayed in the tool tips in these tables.

Caution: Multiframe files for structures using local joint loads are not compatible with older versions of Multiframe. These files may be read into an older version of Multiframe but analysis of these structures will cause the program to crash unexpectedly.

Joint Linking

The implementation of Joint Linking has been revised so that linked rotational degrees of freedom now impose rigid body rotation of the linked joints. As part of this implementation, the user can now specify the master joint from a list of the linked joints.



The properties of a linked group of joints can be edited by double clicking on the group in the Frame window.

The following restrictions apply to linked joints and the restraints, or supports, applied to the linked joints.

- Joints may be common to several groups of linked joints provided that the master joint in all of the intersecting groups is the same joint.
- Prescribed displacements can only be applied to linked joints that are master joints.
- Restraints to linked degrees of freedom must be applied to the master node
- The reactions in the direction of the linked degrees of freedom for a group of linked joint linked are computed at the master joint.

The compliance of the frame to these rules is tested when the user analyses the frame.

Global Moment diagrams

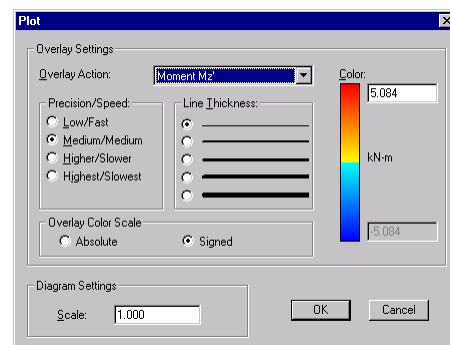
New plots have been added to Multiframe which display moments about each of the three global axes. These diagrams are useful when viewing the moments in rectangular frames in which the members are aligned with the global axes but the orientation of some members is non-zero. The resulting diagrams show the moments in the members without accounting for the orientation of the member.

Tension and Compression Diagrams

Further plots have been added to Multiframe to display axial tension and axial compression. Complementary plots showing axial tensile and compressive stresses have also been added. These plots can be accessed via the Actions and Stresses sub-menus in the Display menu.

Overlay Plots

Overlay plots can now be displayed in two colour scales. The original scale displayed the overlay plot using shades of colour from blue to red to represent the magnitude of the overlay action. Red represents large magnitudes while blue is used to signify small magnitudes. A new colour scale has been added which displays positive values in shades of red and negative values in shades of blue.



The user can select the type of colour scale used to display the overlay plot via the Plot dialog.

Plot Window

The speed of drawing in the plot window has been improved. This improvement is most noticeable in the drawing of the displacement diagrams for which the drawing speed has been improved by a factor of 4 to 5.

Graphical Reactions

The joint reactions from an analysis can now be displayed graphically within the Plot Window. The reactions are drawn as arrows with the tail of the arrow scaled according to the size of the reaction. A minimum tail length is enforced so that small reactions are always visible.

A number of options are available via the Symbols Dialog that can be used to customise the display of reactions. The user can choose to display either or both the force and moment reactions. A convenient shortcut button is provided in the Plot Toolbar to turn on or off the display of these reactions. The Symbols dialog also provides options for displaying the label associated with each reaction and for shading the reactions so that smaller reactions are shown in lighter shades of colour.

Reactions at joints restrained by a Joint Restraints, Spring or Prescribed Displacement are displayed in the local nodal coordinate system.

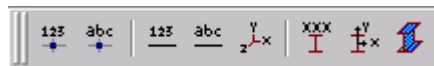
Member Datasheets

The Members datasheet in the Data Window has been split into the Member Properties and Member Geometry pages. The Member Geometry page contains the data describing the size and location of the members while the Member Properties page contains data describing properties of the members such as the section types, orientation and member releases.

Geometry Menu

A new Geometry menu has been added to the main menu bar. This menu contains the functions for manipulating the geometry of the frame including adding, deleting or subdividing members as well as the functions for duplicating, rescaling and rotating parts of the frame. These items used to be contained in the Frame menu.

Symbols Toolbar



A Symbols toolbar has been added to Multiframe that provides a quick and simple way to turn on or off symbols such as joint numbers, joint labels, member number, member labels, section names. A button has also been included to toggle rendering of the section shapes.

Sums of Reactions

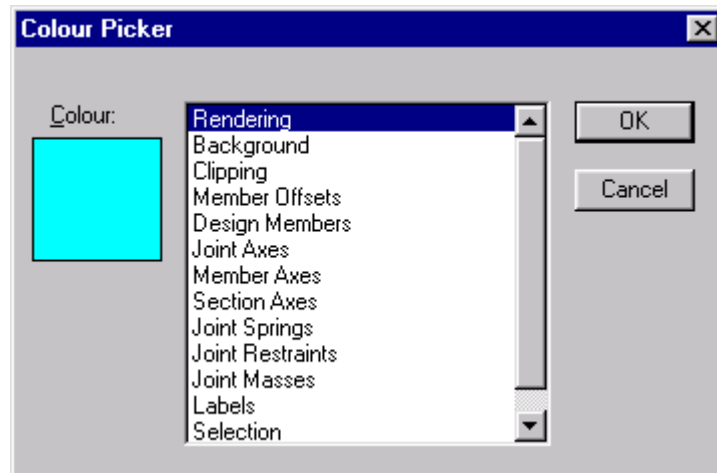
A summation of the reaction components is now displayed at the bottom of the Reactions datasheet in the Results Window. Only the three force reactions are summed, moment reactions are ignored.

Section Colours

When the Section Colours are displayed in the Frame Window, the names of the sections in the Sections datasheet are drawn in the colour associated with that section.

Colours

Multiframe now allows the user to customise the colours used to draw many parts of the frame. This includes the colours of the restraint icons, member axes, clipped members, rendering and even the colour used to highlight selected members. The colours of these items can be customised via the "Color..." menu item in the View menu.



The colour settings for the action diagrams in the Plot Window have been removed from this dialog. A separate dialog has been provided for these colour settings which is accessed via the "Plot Colors..." item in the Display menu.

Selection Tools

All Multiframe users are familiar with the rectangular selection tool used to select multiple members within a region. A new line selection tool has been added which allows the user to drag a line across the screen. All members intersecting this line will be selected. The user can swap between the rectangular and line selection tools using the buttons added to the View toolbar.

Importing Structures

A new option has been added to the Import menu to insert an existing Multiframe structure into the current frame.

Evaluating Expressions

Multiframe can evaluate mathematical expressions in some dialog fields and in the CalcSheet Window. Some new functions have been added to this facility for evaluating arc tangents, arc sines and arc cosines. The ATAN function evaluates the arc tangent of a number and returns the angle in degrees. This compliments the existing ARCTAN function that evaluates the arc tangent in radians. The ASIN and ACOS functions compute the arc sine and arc cosine of a value and

return the angle in degrees. ARCSIN and ARCCOS functions have also been added which return the resulting angle in radians.

These new functions are useful for setting the Joint Orientation or Member Orientation based on the position of the joint or member. See the example later in this release note.

Joint Variables

In certain dialogs that act upon joint loads, restraints or the properties of a joint, Multiframe can evaluate expressions that include variables representing the coordinates of a joint. These variables are as follows:

Variable	Description	Units
X	x coordinate of joint	Length
Y	y coordinate of joint	Length
Z	z coordinate of joint	Length

When the dialog is accepted, each joint in the current selection is considered in turn, the above variables substituted by the location of the joint, and the appropriate values set for the load, restraint or joint property. The following dialogs currently support parsing these variables:

- Joint Spring dialog
- Joint Displacement dialog
- Joint Orientations dialog
- Joint Load dialogs
- Joint Moment dialogs
- Dynamic Load dialog

Member Variables

Most users would be familiar with the ability of many dialogs within Multiframe to interpret the letter "L" as the length of a member. Multiframe can now evaluate expressions that contain variables that represent the geometric properties of a member or it's section properties. These variables include:

Variable	Description	Units
L	Length of member	Length
X1	x coord of joint at start of member	Length
Y1	y coord of joint at start of member	Length
Z1	z coord of joint at start of member	Length
X2	x coord of joint at end of member	Length
Y2	y coord of joint at end of member	Length
Z2	z coord of joint at end of member	Length
B	Width of section	Displacement
D	Depth of section	Displacement
Tf	Flange thickness	Displacement
Tw	Web thickness	Displacement
A	Area of section	Area
theta	Member Orientation	radians

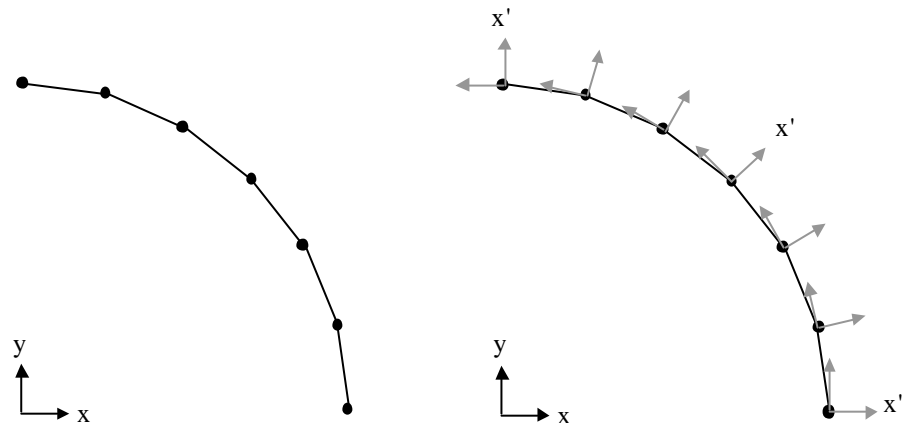
In addition, the section properties as described in the CalcSheet section of the manual can also be evaluated. When a dialog is accepted, each member in the current selection is considered in turn, the above variables are substituted by the appropriate value, and the expressions

evaluated before setting the value of field. The following dialogs currently support parsing member variables:

- Member Offsets dialog
- Member Orientation dialog
- All Member load dialogs
- Bending dialog (SteelDesigner)
- Tension dialog (SteelDesigner)
- Compression dialog (SteelDesigner)
- Constraints dialog (SteelDesigner)

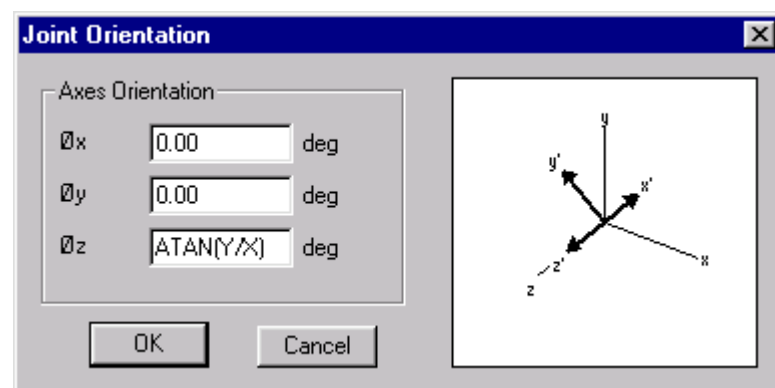
Example

Consider a circular beam shown below that lies in the x-y plane with its centre at the origin. A simple technique for applying Joint Loads which are perpendicular to the beam is to set the orientation of the joints along the beam so that the local x' axes are aligned away from the origin. Loads on the joints can then be applied as Local Joint Loads which will be aligned with the local joint axes.



The orientation of all the joints can be set together using Multiframe's capacity for evaluating expressions containing nodal variables. To set the orientation of all the joints so that the local x' axes point directly away from the origin:

- **Select all joints along the curved beam.**
- **Choose "Joint Orientation..." from the Frame menu.**
- **Enter $ATAN(Y/X)$ in the fz field**
- **Press OK.**



Multiframe computes the angle for each joint in the selection by substituting the corresponding joint coordinates for X and Y. Similar

expressions can also be used to orient a joint to any point. This is particular useful when working with cylindrical or curved structures.

Report View

The Report Window is now activated within Multiframe as all times. It was previously only displayed when Steel Designer was running.

Analysis

The matrix solver at the heart of Multiframe analysis engine has been further optimised to reduce analysis times. A reduction in analysis time of up to 40% has been experienced for linear analyses. For nonlinear analyses, reductions of up to 60% have been recorded.

Convergence and iteration data from a nonlinear analysis is now output to the Report Window. This data includes the displacement and force norms at the end of each iteration as well as warnings and error messages in relation to the analysis. The amount of information output during a nonlinear analysis can be specified via the Analysis dialog which now includes options to specify the reporting level for an analysis as either None, Brief or Detailed.

Steel Designer

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

Design Groups

To improve the design capabilities of Multiframe we have added Design Groups. A design group is a series of connected members that can be considered as a single member for design purposes.

Before a group of members can be included in a design group and then used for design the following conditions must be met

- All members must be connected with the local x' axis facing the same direction
- All members must have the same section type
- All members must have the same orientation
- All members must be rigidly connected internally (end may be released)
- All members must be approximately co-linear
- Members may have rigid offsets at internal joints but the flexible portions of the members must be continuous within the design group.
- There must not be any restraints on the internal connecting nodes

Creating and Deleting Design Groups



Members can be combined into design groups in the Frame Window. To create a design group,

- **Select the members to be grouped**
 - **Choose "Group Members" from Design menu.**
- or**
- **Press Ctrl+G**

Note that the members to be grouped must satisfy the rules described above. The new design group is given a default label of "D" + the number of the group.

To break up design groups, select members that are part of the group(s) and choose "Ungroup Members" from the Design menu.

Numbering of Design Groups

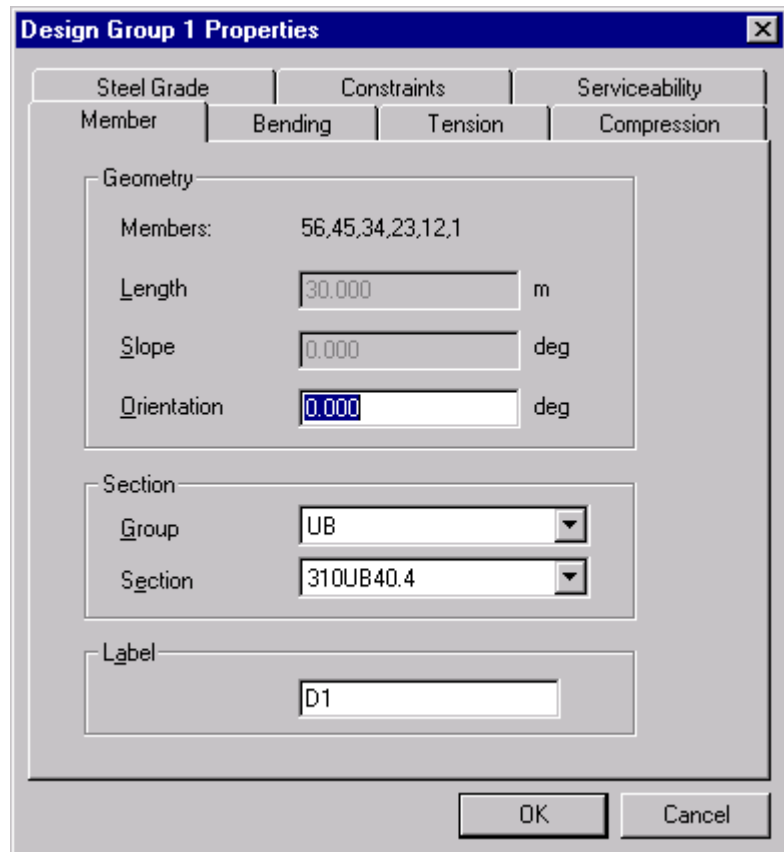
To distinguish design groups from the individual analysis members, a separate numbering system is adopted where every member has its own design group number at creation. Conceptually each member is a design group of one. When members are grouped into a multi-member design group the design group numbering is modified to reflect the reduced overall number of design groups.

The design group numbering is used in both the Design Details and Design Efficiency data tables.

	Design Group	Members	Label	Section
1	1	56,45,34,23,12,1	D1	310UB40.4
2	2	2		310UB40.4
3	3	58,47,36,25,14,3	D2	310UB40.4
4	4	4		310UB40.4
5	5	60,49,38,27,16,5	D3	310UB40.4
6	6	6		310UB40.4
7	7	7		310UB40.4
8	8	8		310UB40.4
9	9	9		310UB40.4

Editing Design Groups

When a member is double clicked in the Frame window and that member is part of a design group, the details for the entire design group are available to be edited, rather than for the individual member clicked on.



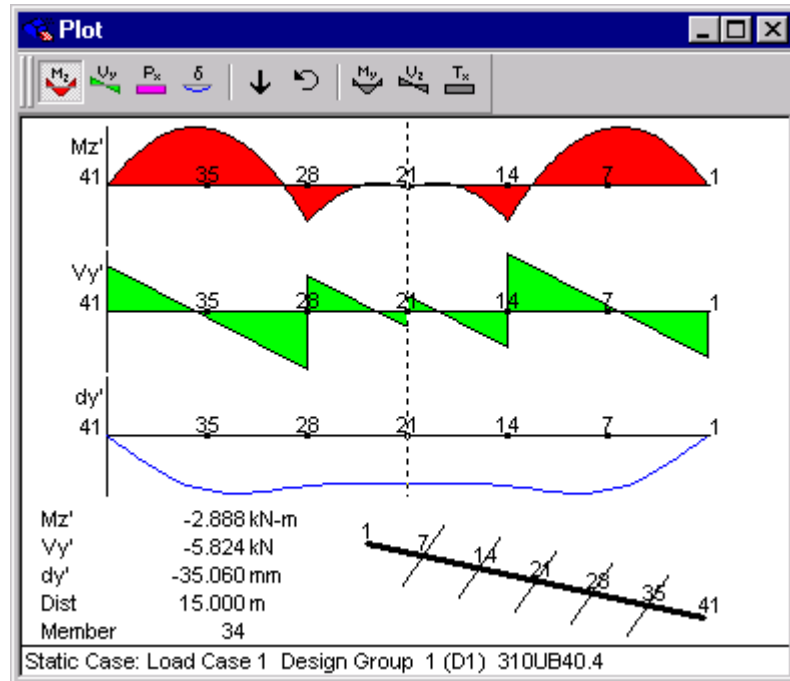
Note that the Length and Angle fields are disabled as you are not able to edit these directly for the entire design group.

To edit the details for an individual member, select the member and then select the "Member Properties..." item from the right click popup menu.



Viewing Results Using Design Groups

In the plot window if you click on a member that is part of a multi-member design group, you will get a local member view that includes all the members in the design group. The details of a single member within this group can be examined by simply clicking on a member within the diagram.



Design Group Symbols

In the Symbols dialog there are three check boxes dedicated to viewing design groups.



If **Design Groups** is checked then multi-member design groups are displayed in the Frame window by a patterned blue overlay.

If **Labels** is checked the labels of the multi-member design groups are displayed in all the drawing windows.

If **Numbers** is checked the numbers of all the design groups used in design are displayed in all the drawing windows.

Rendering of Design Groups

Design groups with more than one member are rendered in the Frame and Load windows as a single member.

Design Reporting

When Checking or Designing members you are now actually working with Design groups. So if you select a member that is part of a multi-member design group then the Check/Design will be carried out on the design group.

This is reflected in the reporting which will display the design group number being check and below it the member(s) that make up that group.

Checking Design Group 1 (D1)

Members: 34,23,12,1

Group: UB
Section: 310UB40.4

Compute section properties

Lateral Restraints

The location and type of lateral restraints can now be displayed graphically in the Frame and Plot windows. The display of lateral restraints can be turned on or off via the Symbols Dialog which now contains options for displaying lateral restraints and labeling these restraints.

The restraints are draw as a short line in the plane of the major axis of the member. These lines extend each side of the member for a distance that is roughly the scale of a purlin or girt. Lateral restraints are also displayed in the rendered view of the frame in which they are draw to extend from each flange by approximately the size of a purlin. The restraints may be labeled using a one or two letters to indicate the type of restraint (e.g. F - fixed, P - partial).

Note that lateral restraints at the end of a member are draw slightly offset from the node so that restraints at the ends of connected members may be more readily distinguished.

Governing Load Cases

The governing load case associated with the design of a member is now recorded when designing or checking a member. The governing load case associated with each member is displayed in the Efficiency Datasheet.

The governing load cases associated with each of the individual design checks are also stored as part of a design. The governing load case for a particular design check is displayed as a cell tooltip in the Efficiency Datasheet or as a member tooltip in the Plot Window when a plot of the efficiency is displayed in the PLOT window.

Constraints

The implementation of constraints has been modified so that any design requirement that is not dependent upon the design actions can be testing independently of the load cases. The concept of Design Constraints is now used to encompass the existing constraints that can be imposed upon the dimensions of a member as well as any constraints that may be imposed by various design checks. (I.e. a slenderness check that may be required as part of a bending design). For most of the design codes no additional constraints have been added and the only Design Constraints will be the constraints on the size of the member. For the AIJ design code, a check on the Width/Thickness ratio of the section is performed with several of the design checks. Design Constraints are applied when Designing and Checking a member. Previously, the constraints on the size of a member were only applied when Designing the member.

The calculations associated with Design Constraints are output to the design report. These calculations are performed at the start of the design before considering the design checks for each load case. When using Brief Reporting, the calculations for failed design constraints are output to the report. With detailed or full reporting, the calculations for all Design Constraints are shown in the report.

The status of Design Constraints which were tested when Designing or Checking a member are displayed in a new "Constraints" column in the Design Efficiency datasheet. If no constraints were checked for a particular member, a dash is shown in this column. Otherwise, this column displays the number of Design Constraints that were not satisfied as part of the design.

Section Maker

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

Section Colours

A column has been added to the Sections Window that displays the colour associated with section. Double clicking over a cell in this column allows the user to select a new colour using the Colour Picker dialog. The colours in this column may be cut and pasted or duplicated to other sections using the fill down option.

Problems Fixed

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

- Drawing of Tension/Compression only members in Displacement diagram.
- Analysis of Tension/Compression only members sometimes failed to converge when analysing multiple load cases.

- Deflection checks in Steel Designer are now performed using local deflections measured relative to displaced position of the members end.
- In Steel Designer, design actions about the major and minor axes of the member were switched when using Multiframe2D. This only occurred for members with non-zero member orientations.
- Incorrect behaviour of Size to Fit in 3D views.
- Spurious messages when duplicating joint loads and restraints
- Section colours set in Multiframe are now saved to section library
- Incorrect load cases names used for nonlinear results when designing to ASD or AIJ design codes
- Action diagrams for Tension/Compression only elements with applied loads are now computed correctly.

Problem Reports

We greatly appreciate the bug reports and suggestions we have had so far, 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>