

Multiframe 7.5 8 December 2000

Release Note

This release note describes the Windows 95/98/NT/2000 version 7.5 final release of Multiframe, Steel Designer and Section Maker. This release contains numerous enhancements for each of these products.

Contents

Contents	1
Access Code.....	1
Multiframe.....	1
New File Format	2
Frame Window	2
Load Window.....	2
Plot Window	2
Project Details.....	2
Self Weight Load Cases.....	3
Distributed Loads.....	3
Load dialogs	4
Prescribed Displacement Loads.....	4
Space Gass Import and Export.....	5
Microstran Import and Export	6
DXF Import/Export.....	6
VRML Export	6
Non-linear Analysis	6
Dynamic Analysis Nomenclature.....	7
Reporting.....	7
Tensile/ Compressive Plots	7
Steel Designer.....	7
NZS3404	7
Section Maker.....	7
Problems Fixed.....	8
Problem Reports.....	8

Access Code

Please note that this release requires you to enter a new access code. The first time you run the new version, you will be prompted to enter a new code. Please enter the code from the Access Code sheet provided.

Multiframe

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

New File Format

The format of Multiframe files (*.mfd) has been updated for this release of Multiframe. While we try and avoid such upgrades, the current file format has been around for over 3 years and needs to be revised in order to support new features both now and in the future.

To provide compatibility with older versions of Multiframe an option has been provided in the Export menu to save a frame in the old file format. This will also allow Macintosh users to read files from the new version.

The format of the sections libraries (*.slb) has also been updated. When a sections library is saved by Multiframe it is saved in its original format. However, in Section Maker, the library will automatically be saved in the latest file format. An option has been provided in Section Maker to export a sections library in the old file format.

Frame Window

The properties of items displayed as icons in the Frame window can now be edited by simply double clicking the mouse over the icon. Items such as joint springs, restraints and joint masses as well as member releases and member masses can all be edited in this manner.

Load Window

The loads on nodes or elements can now be deleted using the delete or backspace keys. The loads on all joints or elements currently selected in the Load Window will be removed from the structure when either of these keys is pressed.

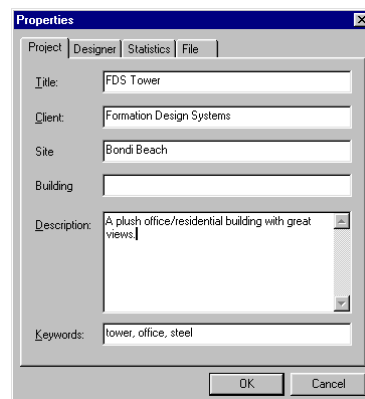
Plot Window

The behaviour of the Plot window has been modified so that a single mouse click is used to select items in the window. A double click of the mouse is now used to view the diagrams for a particular joint or member.

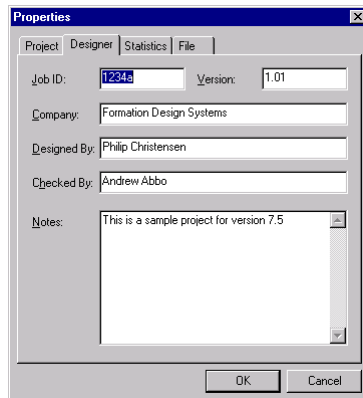
Project Details

Details of the project and designer can now be recorded and stored with the frame. To set the properties of a project

- **Choose Properties from the File menu**



- Type in the properties of the project.
- Select the Designer tab.



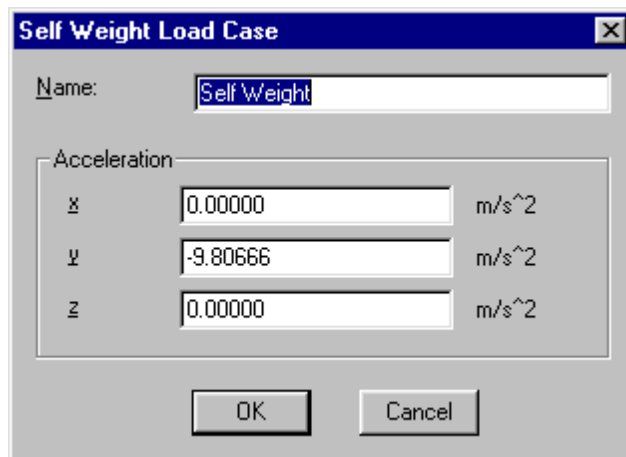
- Type in the details of the designer.
- Click on the OK button

This dialog also contains two additional pages that display properties of the file and statistics about the frame.

This dialog will automatically be displayed when saving a frame for the first time. It will also be displayed when an older file is saved to the new file format.

Self Weight Load Cases

The implementation of Self Weight load cases has been enhanced. Multiple self weight load cases can now be added to the frame and the gravitational accelerations can now be specified for each direction.

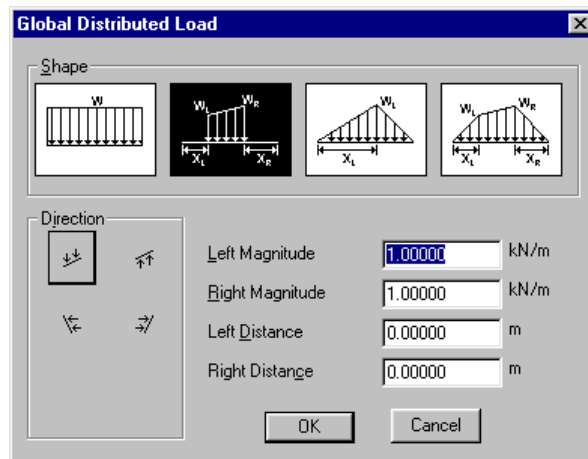


This may be useful when defining load cases based on inertia or seismic effects.

Distributed Loads

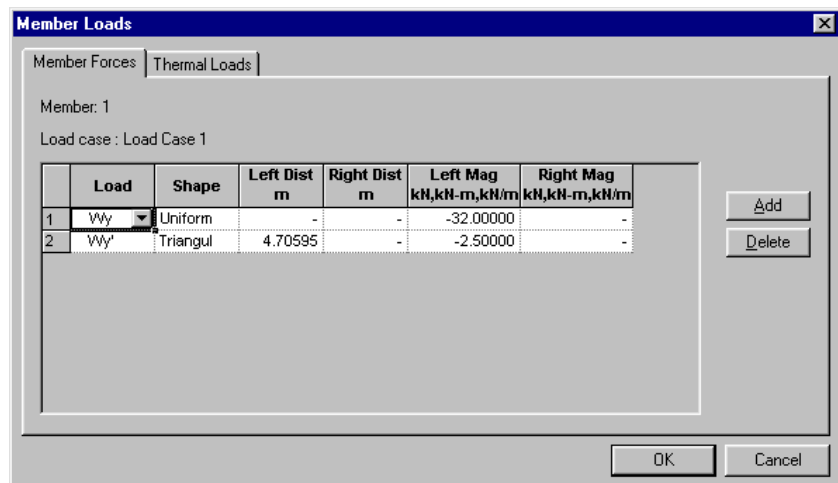
When adding a distributed load to a member, the user can now specify the distribution or shape of the loading. The shape of the

load may be specified as either uniform, linear, triangular or trilinear.



Load dialogs

The loads applied to a member can be edited by double clicking on the member in the Load Window. The dialog for editing the member loads has been enhanced to include pages for both force and thermal loads. In addition, loads can now be added via the dialog by using the button on the right hand side of the dialog.



The equivalent dialog for editing joint loads has also been enhanced in the same manner.

You can also now change the shape and type of loads in the tables in the Data window by selecting from the pop-up menu in the appropriate columns in the table.

Prescribed Displacement Loads

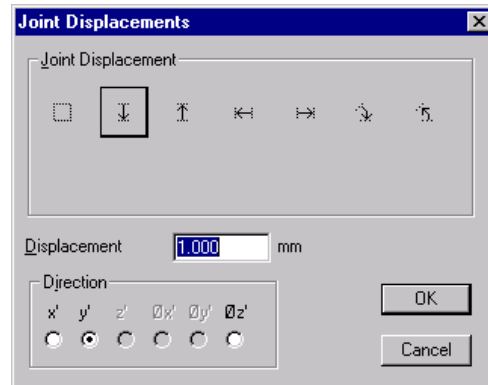
The implementation of prescribed displacements has been changed so that they are now applied in the current load case rather than to all load cases. For this reason, application of and display of

prescribed displacement now occurs in the Load window rather than the Frame window.

To prescribe a displacement at a joint

- **Select the joint or joints in the Load window**
- **Choose Prescribed Displacement from the Load menu**

A dialog box will appear with icons for the various prescribed displacements.



- **Click on the icon which shows the direction the displacement is to be applied**
- **Type in a value for the displacement**
- **Click on the OK button**

Selecting the "No prescribed displacement" icon (the first icon in the list) will remove all prescribed displacements from the selected joints. Prescribed displacements act in the local coordinate system of the node to which they are applied. Each of the icons represents a direction for the displacement relative to one of the reference axes. As you click on the icons, which indicate the various displacements, the directions of each displacement will be indicated by the radio buttons at the bottom of the dialog. You can also click on the radio buttons to choose which direction you wish the displacement to act in.

The properties of a prescribed displacement may be edited by double clicking on the icon representing the prescribed displacement.

Space Gass Import and Export

Multiframe now supports importing and exporting the Space Gass text file format. While not all the features in Multiframe and Space Gass are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

When importing a Space Gass text file, Multiframe attempts to match the names of the sections in the text file to those in the current sections library. If a matching section cannot be found then the user is prompted to manually find an appropriate section. If the user does not choose a section then the section is added to the Frame group using the properties stored in the Space Gass file.

Space Gass supports discontinuous numbering of nodes and elements. When importing a text file, Multiframe adds nodes and elements in the order in which they appear in the text file. The original node or member number is used to label the corresponding node or member in Multiframe.

Microstran Import and Export

Multiframe now supports importing and exporting the Microstran archive file format (version 4). While not all the features in Multiframe and Microstran are compatible, the majority of the geometry, topology and loading can be exchanged via this file format.

As with the Space Gass file format, Multiframe attempts to match section names and prompts the user to manually locate a section if a matching section cannot be found. Similarly nodes and members are labelled with the original Microstran numbering.

DXF Import/Export

A DXF file may now be imported into Multiframe and appended to the frame currently open in the program. The imported structure will automatically connect to the existing structure at common joints while line segments corresponding to existing members will be ignored.

The exporting of DXF files has been modified such that only active members (i.e. not clipped or masked) are included in the exported data. In addition, nodes and members can now be labelled with their number and/or label. These will be written to the DXF file if they are visible within Multiframe.

VRML Export

An option has been added to the Export menu to save a frame as a VRML file. This option is only available when the frame is rendered in the active window. VRML is a format used to view 3D models in an Internet web browser. You can also embed these models in web pages to provide online display of your projects.

Non-linear Analysis

The non-linear analysis engine has been improved and now models the P-DELTA effect using a finite displacement formulation. This better represents the change in geometry of a member by accounting for the rotation of the member from its original position.

Support has also been added to model the axial shortening of a member due to flexure of the member. This non-linear effect may be included in an analysis by selecting the appropriate option in the Non-linear Analysis dialog. The implementation in Multiframe only considers shortening of the member due to the moments acting at the ends of the member. No account is made for the effect of loads applied to the member.

Dynamic Analysis Nomenclature

The nomenclature used in Multiframe4D to describe the dynamic analyses and the associated load cases has been changed. A dynamic analysis to determine the mode shapes of a structure is now referred to as a Modal Analysis while Dynamic Load Cases have been renamed to Time History Load Cases.

Reporting

The reporting capacity of Multiframe has been enhanced by allowing the user to redirect output usually displayed in the Report Window to a Microsoft Word document. This option can be selected via the preferences dialog and requires that either Microsoft Word 97 or Microsoft Word 2000 be installed on the computer. Multiframe will automatically run Microsoft Word when it is required.

Another option has also been added to the preferences dialog that buffers output to the report window. Using the option will improve the speed at which the design calculations are output to the Report window.

Tensile/ Compressive Plots

The plots of tensile forces have been refined so that it is scaled according to the maximum tension action in the frame. Previously this plot was scaled using the maximum axial force which considered both tensile and compressive forces. Plots of compressive force, tensile stress and compressive stress have also been modified in the same way.

Steel Designer

- Steel Designer now prints a summary of lateral restraint information in the design report.

NZS3404

Design of steel structures to the New Zealand design code NZS3404 has been incorporated into this version of Steel Designer. Details of the implementation and scope of design to NZS3404 are contained in the manual and online help.

The implementation of this design code includes the changes described in the draft of Amendment No. 1 to the standard.

Section Maker

- A problem redrawing the Shape view has been fixed
- A problem importing shapes on European systems has been fixed.

Problems Fixed

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

- Zero length line segments in DXF files are now ignored.
- Sections can now be changed in the Design Details table in the Data Window.
- 2D Analysis of a 2D frame with 3D loading has been disabled.
- An incorrect unit scaling factor for J values in the CHS350 group in the Australian sections library has been corrected.
- Efficiency plots scales now drawn correctly.
- Rendering of efficiency plots is improved
- Rendering Design Members now works correctly in all cases
- Font in graphical windows now saved as preference.
- Non-linear analysis was failing to converge for some 3D analyses, this is now fixed.

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

Also visit our Web site to get all the latest information at:

<http://www.formsys.com/BetaTest>

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.