

Multiframe

Windows and Macintosh Version 5.1

User Manual

License & Copyright

Multiframe Program

© 1985-99 Formation Design Systems

Multiframe is copyrighted and all rights are reserved. The license for use is granted to the purchaser by Formation Design Systems. As a single user license and does not permit the program to be used on more than one machine at one time. Copying of the program to other media is permitted for back-up purposes as long as all copies remain in the possession of the purchaser.

Multiframe User Manual

© 1985-99 Formation Design Systems

All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form or by any means, without the written permission of Formation Design Systems. Formation Design Systems, reserve the right to revise this publication from time to time and to make changes to the contents without obligation to notify any person or organization of such changes.

DISCLAIMER OF WARRANTY

Neither Formation Design Systems, nor the author of this program and documentation are liable or responsible to the purchaser or user for loss or damage caused, or alleged to be caused, directly or indirectly by the software and its attendant documentation, including (but not limited to) interruption on service, loss of business, or anticipatory profits. No Formation Design Systems distributor, or agent, or employee is authorized to make any modification, extension, or addition to this warranty.

Table of Contents

License & Copyright.....	iii
Table of Contents.....	v
About this Manual.....	1
Chapter 1 Learning Multiframe.....	3
2D Tutorial.....	6
3D Tutorial.....	29
Chapter 2 Using Multiframe	41
Creating a Structure	51
Applying Loads	87
Performing Analysis	106
Viewing Results.....	109
Calculations.....	121
Printing	124
Chapter 3 Multiframe Reference	129
Windows.....	130
Toolbars	131
Menus.....	133
Chapter 4 Multiframe Analysis	157
Appendix A Troubleshooting.....	165
Appendix B Macintosh Memory	171
Appendix C Analysing Trusses.....	173
Appendix D Importing And Exporting Data	175
Appendix E Text File Format.....	179
Appendix F Using Spreadsheets With Multiframe	187
Appendix G Quality Assurance	193
Index.....	197

About this Manual

This manual is about Multiframe, a structural analysis and design system.

Chapter 1 Learning Multiframe

Gets you started creating, editing, analysing and checking results for a structure and its loading. It is a series of exercises designed to help you learn the basics of Multiframe and become confident in using it to analyse structural behavior.

Chapter 2 Using Multiframe

Chapter 2 Using Multiframe includes step-by-step instructions in using the program. This will explain most of the tasks you will carry out in defining, analysing and designing a structure.

Chapter 3 Multiframe Reference

Gives an overview of the operations of Multiframe and a summary of the commands used.

Chapter 4 Multiframe Analysis

Discusses the numerical methods used by Multiframe. It is important for you to understand these methods and their limitations before using Multiframe for structural analysis and design. The chapter ends with a summary of the capabilities and limitations of Multiframe.

This manual describes the three versions of Multiframe, 2D, 3D and 4D. Where appropriate the manual will indicate features, which are only available in the 3D and 4D versions.

Chapter 1

Learning Multiframe

This chapter helps you get started installing and using Multiframe by working through a sample tutorial problem.

Getting Started

If you are not familiar with the concepts of structural analysis using the matrix stiffness method, it will be necessary for you to review the structural analysis concepts outlined in Chapter Four. If you are familiar with the analysis, you can proceed at your own pace.

Multiframe will run on any Macintosh computer with at least 8 Mb of memory, or any Windows 95/98/NT computer with at least 16 Mb of memory.

Installing Multiframe on Windows

If you are installing from floppy disk, insert Disk 1 into your floppy drive. Select Run from the Start menu (Windows NTv3.51 users choose Run from the File menu of the Program Manager). A dialog box will appear asking you to enter the name of the program to run. Type 'A:\SETUP' in the space provided, then click on OK, and then follow the instruction on screen.

If you are installing from CD, insert the CD into your CD drive and follow the instructions on screen.

After installation, Multiframe should be accessible through the Start Menu. Simply select 'Multiframe' from the Start menu. Multiframe is automatically installed in the Program Files directory unless you specify otherwise.

Installing Multiframe on Macintosh

If you are installing from floppy disk, insert the Multiframe disk and double click on the Installer program. This will prompt you to specify a location on your hard disk where Multiframe will be installed.

The Utilities disk which comes with Multiframe contains a number of folders with libraries containing the structural shapes used in different countries. You will need to copy the Section Library from the appropriate folder on the floppy disk into the same folder as the Multiframe program on your hard disk.

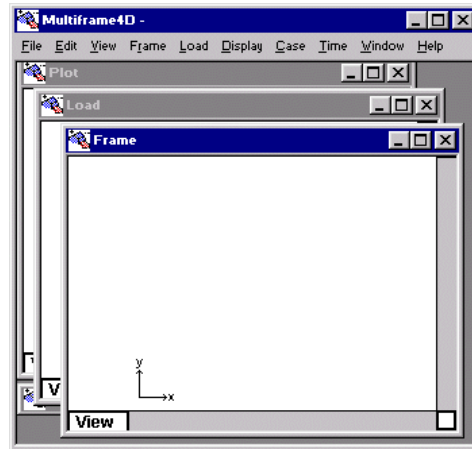
If you are installing from CD, insert the CD into your CD drive and follow the instructions on screen.

You can start Multiframe by double clicking on the Multiframe icon on your disk. If a dialog appears asking you to locate the Sections Library when you try to start Multiframe, this means that you have

not copied the Sections Library from the Utilities floppy disk into the same folder as the Multiframe program. On the CD you should run the Multiframe Utilities installation program.

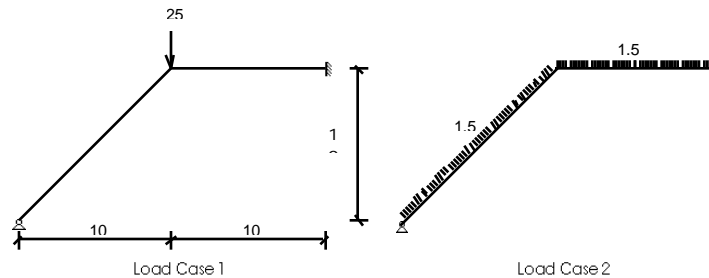
Starting Multiframe

After the copyright window has been displayed, a number of windows will appear on the screen including three titled Frame, Load and Plot.



Multiframe Tutorial

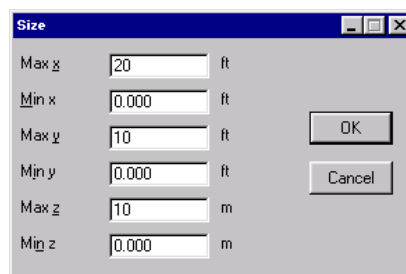
To introduce you to the concepts and techniques you will use in Multiframe, this chapter will describe a simple frame analysis step by step. The frame you will analyse is shown below. You will analyse it for the two different loading conditions shown and for a superposition of the loads.



First, you will draw the frame to scale, a member at a time, inside the Frame window. It is necessary to first set up the scale for drawing in the Frame window.

- **Choose Size... from the View menu**

A dialog box will appear with fields in it specifying the maximum dimensions of the frame.



- **Type 20 for the maximum x coordinate and then press the Tab key to move to the second field**
- **Type in 0 for the minimum x coordinate and then use the Tab key to move to the next two fields entering 10 and 0 for the maximum y and 20 for the minimum z coordinates**
- **Click the OK button when you have finished entering the values**

If you make a mistake entering the numbers, you can use the backspace or delete key to delete the character just to the left of the blinking cursor and you can use the Tab key to move from one field to the next.

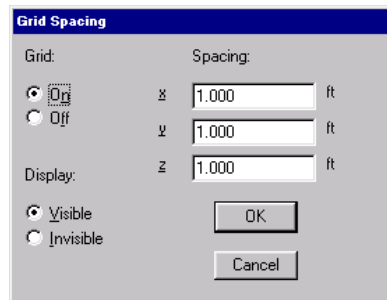
2D Tutorial

If you are using Multiframe 2D, you will notice a number of differences from the illustrations in this manual. In general, fields in dialogs associated with 3D e.g. Max z and Min z in the dialog will not be visible in the 2D version of the program. All of the commands in 2D work the same as the commands in 3D, the only difference is that items which relate only to dimensions, deflections or forces in the z direction will be omitted from the 2D version.

To make it easier for you to draw the frame, you can use Multiframe's Grid option to make drawing automatically align to a grid with regular spacing.

- **Choose Grid... from the View menu**

A dialog box will appear with values for the x (horizontal), y (vertical) and z spacing of the grid.

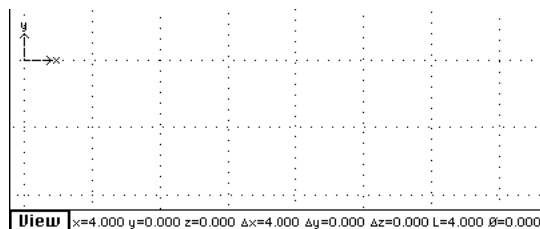


- **Type 1 for the x spacing, use the Tab key to move to the other fields and also enter 1 for the spacing there**
- **Click on the On radio button to make subsequent drawing align with the grid**

This will also turn on the Visible button to make the grid visible in the Frame window.

- **Click on the OK button to confirm your settings**

Now move the pointer inside the Frame window and notice that as you move the mouse, the coordinates of the pointer are shown at the bottom left hand corner of the window. The coordinates will automatically align to the nearest point on the grid.



You can now begin to draw the structure to scale in the Frame window.

Drawing the Frame

To draw the first member in the frame

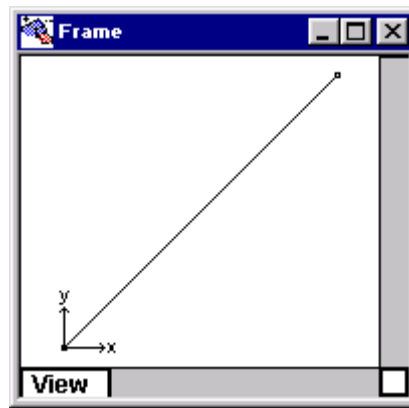
- **Select Add Member from the Frame menu**

A cross hair will appear in the Frame window.

- **Move the cross hair to the location of the first joint**

Move the cross hair towards the axes at the bottom left hand part of the window until the coordinates read $x=0.000$, $y=0.000$.

- Press the mouse button and hold it down



- Drag the crosshair to the position of the second joint

Note that Windows users may choose to click on the first joint and click on the second joint to draw a member instead of dragging the mouse.

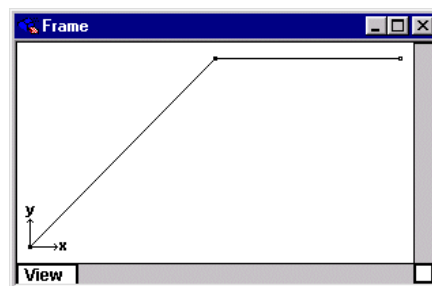
Move the crosshair up and to the right. Notice that as you move the crosshair, the member is continually re-drawn to show its current position. The coordinates, length and slope of the member are also shown. Move the crosshair until the coordinates read $x=10.000$, $y=10.000$.

- Release the mouse button (Windows users can click the mouse button)

When you click or release the button the member is drawn in its final position. Notice also that two small black squares are drawn showing the position of the joints marking the ends of the member.

You can now draw the second member, which runs horizontally across from the top right hand end of the first.

- Choose Add Member from the Frame menu
- Move the cross hair to the top right hand joint of the member you have drawn
- Click on the joint and drag to draw the member
- Release the mouse button when the coordinates read $x=20.000$, $y=10.000$



When you add a member by clicking on or very close to an existing joint, Multiframe will create a connection between the existing joint

and the new member. In this case, Multiframe has connected the two members together at the common joint.

If you make a mistake with the position of any joints, you can drag them to a new position using the mouse or double click on them to enter a new location.

This completes the setup of the geometry of the frame. You can now move on to applying the restraints and section types needed for the structure.

Selecting a Joint

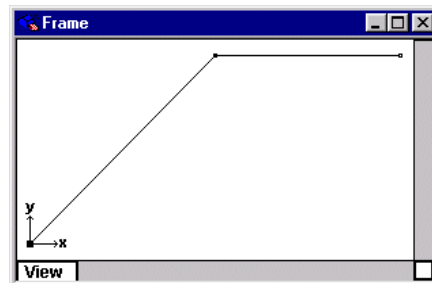
In order to set the restraints and section properties for the structure, you first need to select the members and/or joints that you want to work with.

You will find it helpful to turn off the grid now that you have finished drawing the structure.

- **Choose Grid... from the View menu**
- **Click on the Off radio button to turn off the grid**
- **Click OK**

The grid display and alignment features in the Frame window are now turned off. To select the first joint

- **Point to the bottom left hand joint of the frame and click**



Notice that a larger black square appears at the joint. This indicates that this joint is selected.

Restraining a Joint

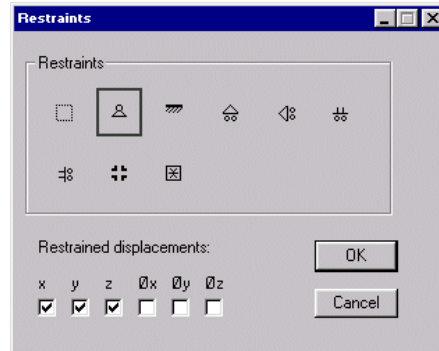
Joints, which cannot move freely in a structure, must be restrained in some way in the Multiframe model. This restraint may be horizontal, vertical, a restraint against rotation or a combination of these restraints.

To apply the pinned restraint to the joint you have selected

- **Choose Joint Restraint from the Frame menu**

Note that Windows users have the option of clicking the right mouse button to get a context sensitive menu.

A dialog box will appear with a number of icons in it.

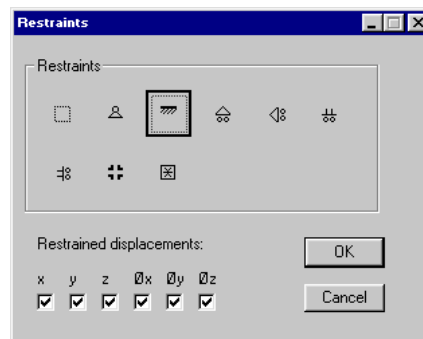


- Click on the pinned restraint icon
- Click OK

Notice that the pinned restraint icon is now displayed at the joint which was selected. A pinned restraint prevents the joint from moving horizontally or vertically but allows it to rotate freely.

Next select the top right hand joint of the frame by clicking on it.

- Click on the top right hand joint on the frame
 - Choose Joint Restraint from the Frame menu or right mouse click
- The joint restraint dialog will appear.



- Click on the fixed joint icon
- Click OK to apply this restraint to the selected joint

A fixed joint cannot rotate or move vertically or horizontally.

Section Properties

In order to compute deflections in the structure it is necessary to know the material properties and dimensions of the sections used in the structure. Multiframe has a built-in table of the most commonly used structural sections from which you can select the desired section type. Multiframe also allows you to add your own sections to the library.

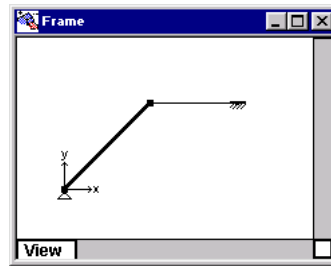
Selecting a Member

In order to specify the section properties for the members in the frame, it is necessary to select each of the members in turn.

To select the sloping member

- Point to the sloping member away from its ends and click

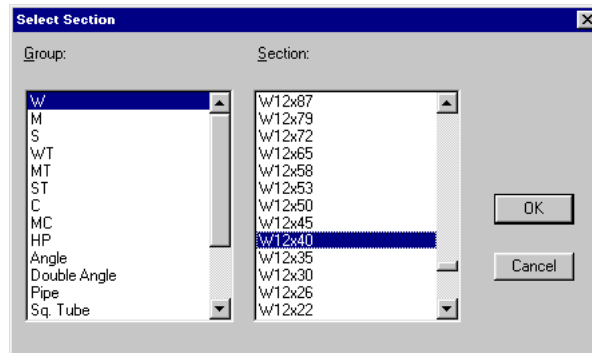
The member will be drawn with a bold line indicating that it is selected.



- **Choose Section Type from the Frame menu**

Note that Windows users have the option of clicking the right mouse button to view a short cut menu.

A dialog box will appear with a list of groups and sections.



- **Click on the name W in the list of section groups**

This will display, in the list on the right, the list of sections stored in this group.

- **Use the scroll bar to scroll down the list of sections until the W12x40 is visible**

- **Click on this name to select it**

- **Click on the OK button to confirm your choice**

Since the horizontal member uses a custom section that is not stored in the Sections Library, it is necessary to add the section to a custom group in the library.

To add the custom section to the library

- **Choose Add Section... from the Edit menu**

A dialog box will appear which contains a list of groups and a table of section properties.

New Section

Section Name:

Group:

- Custom1
- Custom2
- Custom3
- Frame**

Properties:

	Property	Value	Units
1	Mass	50.000	lb/ft
2	A	10.000	in ²
3	Ix	250.000	in ⁴
4	Iy	45.000	in ⁴
5	J	1.000	in ⁴
6	E	29000.000	ksi
7	G	11153.000	ksi
8	D	8.500	in

Shape:

OK Cancel

- Click on the group named **Frame** in the list of groups at the left

This indicates that this section will be stored in the same file as the frame rather than in the standard section library database.

At the top of the dialog, a name for the section can be entered.

- Type in **"Hbeam"** as the name of this section

You can now enter the properties for the section.

- Click on the first value in the table to select it

- Type in the value for the mass of the section

You can now enter each of the values in turn using the Down Arrow key to move from one value to the next (Do not use the enter key as this will close the dialog box).

For the section in this structure the values will be

Mass	50	lb/ft	Mass per unit length
A	10	sq in	Cross sectional area of the section
Ix	250	in ⁴	Moment of inertia about major axis of section
Iy	45	in ⁴	Moment of inertia about minor axis of section
J	1	in ⁴	Torsion constant
E	29000	ksi	Young's Modulus
G	11153	ksi	Shear Modulus
D	8.5	in	Depth
B	8	in	Width

tf	0.8	in	Flange thickness
tw	0.4	in	Web thickness

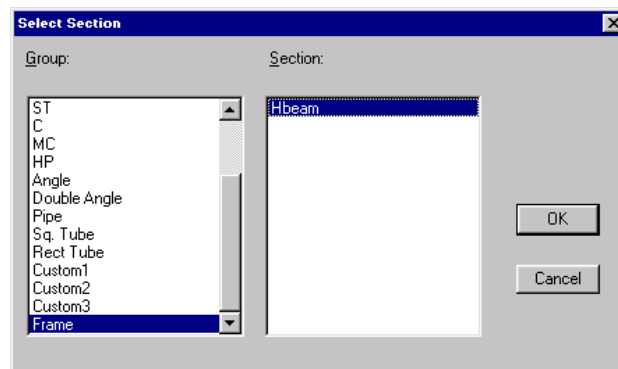
- Click "I" under Shape at the bottom of the dialog box

Leave all other fields set to the default values.

- Click the OK button when you have finished entering the section property values

Now select the horizontal member and set its section type.

- Click on the horizontal member to select it
- Choose Section Type... from the Frame menu or from the short cut menu



- Scroll down the list of member groups and click on the group named Frame

The section Hbeam that you stored in the library will be shown in the list of sections for this group.

- Click on the section named Hbeam
- Click the OK button

You have now completed the physical description of the structure including the geometry, restraints and section properties. You can now move to the Load window to apply the loads to the frame.

Loads

The loads acting on the structure can be set or changed in the Load window. Click anywhere in the visible part of the Load window to bring it to the front or choose Load from the Window menu if the Load window is not visible. Notice that the structure is shown in this window exactly as you drew it in the Frame window.

To change the loads on all or part of the structure you first select the members and/or joints that you want to work with. This is done using the same techniques you used in the Frame window by clicking on the joints or members to be selected.

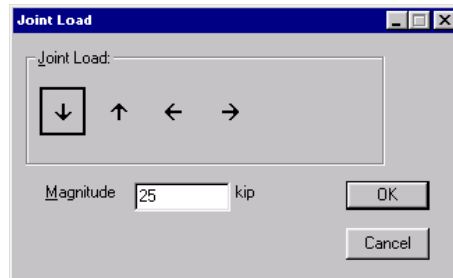
Loading a Joint

First apply the point load at the middle joint of the frame.

- Click on the middle joint to select it

- Choose Joint Load from the Load menu or from the short cut menu

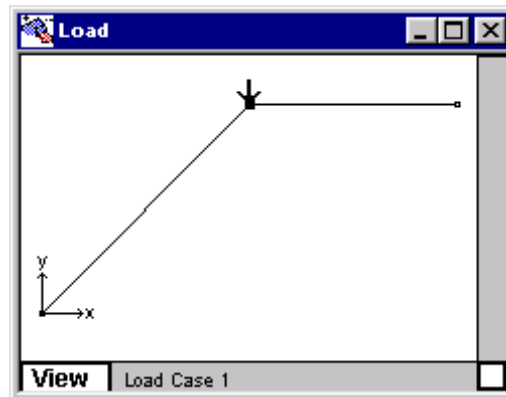
A dialog box will appear with a number of load icons in it.



- Click on the vertical load icon
- Type in a value of 25 for the load
- Click the OK button

Note that it is not necessary to enter a positive or negative sign to indicate the direction of loads in Multiframe. Your choice of icon, either upward or downward in this case, defines the direction of the applied loading.

The load you have applied at the middle joint of the frame will now be displayed in the Load window.



Loading a Member

The second loading condition for this frame includes two uniformly distributed loads acting on members. Since we wish to also check the combined effect of the two loading conditions, it will be useful to apply the distributed loads as a separate load case.

- Choose Static from the Add Case submenu under the Case menu

A dialog box will appear asking you to name the load case. Leave the name of the second load case set to Load Case 2.

- Click the OK button

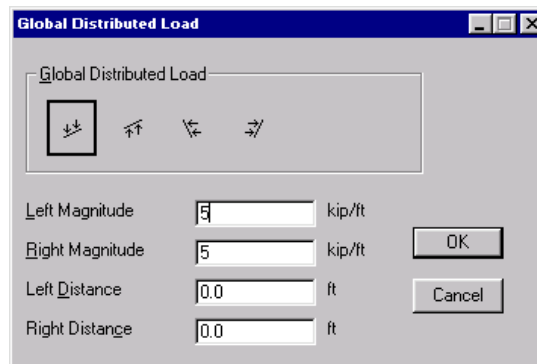
One load case at a time may be displayed and edited in the Load window. The name of the current load case is displayed in the lower left corner of the Load window. Load Case 2 is now the current load case.

Now apply the distributed load to the horizontal member.

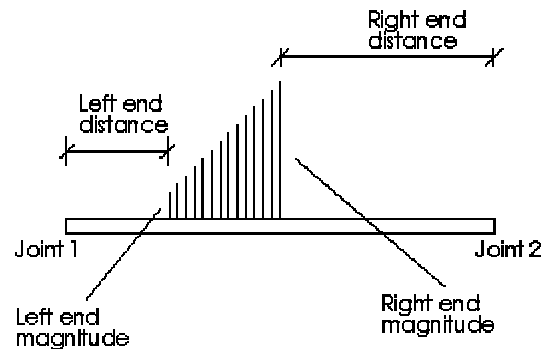
- Click on the horizontal member to select it

- Choose Global Dist'd Load from the Load menu or from the short cut menu

A dialog box will appear with a number of icons in it.



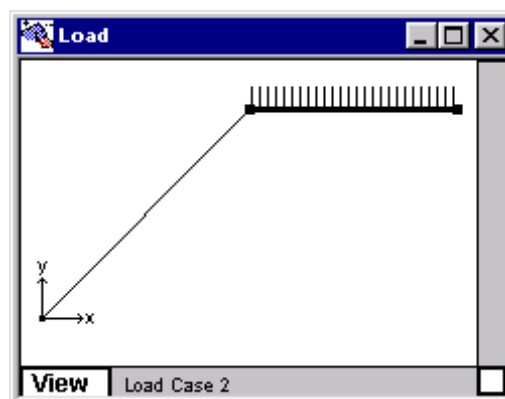
Since a distributed load can act over all or part of a member and may be uniform, triangular or trapezoidal in magnitude, it is defined by specifying a start and finish magnitude and a distance of the start and finish of the load from the ends of the member. So a uniformly distributed load of 5 kip/ft over the whole member will have a Left Magnitude of 5, a Right Magnitude of 5, a Left Distance of 0 and a Right Distance of 0.



- Click on the uniform vertical load icon
- Type a value of 5 for the left magnitude of the load
- Use the Tab key to move to the next number in the dialog

Notice that the right magnitude is automatically set to the same value as the left magnitude.

- Click OK to confirm the values you have entered

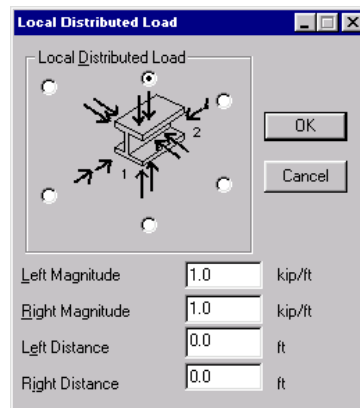


The load you have added will be drawn on the member in the Load window. Notice that the pattern indicates the direction of action of the load and the load is drawn on the side of the member on which it acts.

Now add the load acting on the sloping member.

- **Select the sloping member by clicking on it**
- **Choose Local Dist'd Load from the Load menu or from the short cut menu**

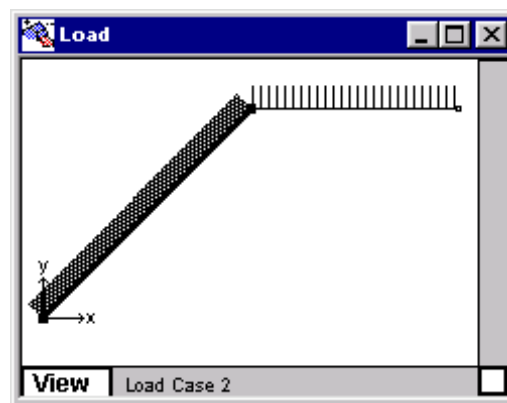
A dialog box will appear with a number of icons indicating different directions for the loading and numbers to specify the magnitude and position of the load on the member.



These loads act in a direction specified by the slope of the member. The loading can be either normal (shear) or tangential (axial) to the member.

- **Click on the normal load icon**
 - **Type in the magnitude of 1**
 - **Press the Tab key to move to the right hand magnitude**
- This will automatically be set to the same value of 1.
- **Click the OK button**

The normal load will be displayed on the sloping member.

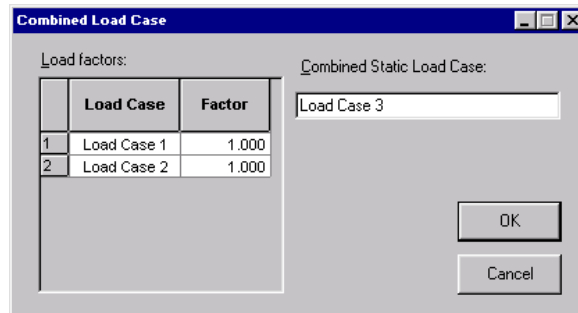


Combining Load Cases

Multiframe allows you to automatically combine load cases together. For our sample frame we wish to superimpose the two loading conditions.

- **Choose Static Combined... from the Add Case sub-menu under the Case menu**

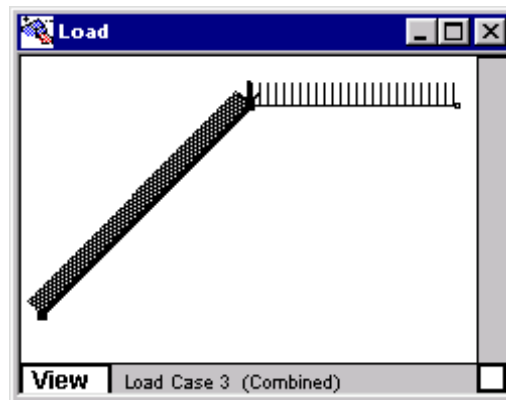
A dialog box will appear with a name for the new case and a table of load factors.



The factors in the table indicate the value each load case will be multiplied by before being added to the new case. In this case the factors for Load Case 1 and 2 will both be 1.0 since we are using a simple superposition of the loads without any other factoring.

- **Click on the first factor to select it**
- **Enter 1.0 then Down Arrow to the second factor**
- **Enter 1.0 for the second factor**
- **Click the OK button**

The third load case will now be drawn in the Load window with the three loads from the two load cases superimposed.

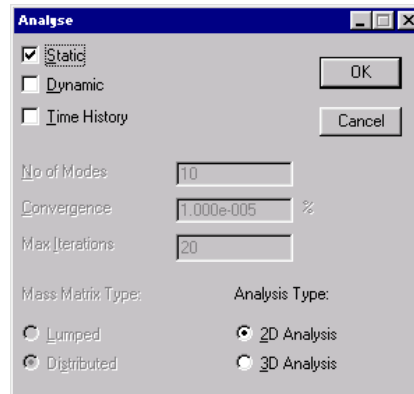


Analysis

Now that you have set up the structure and loading to your satisfaction you can analyse the structure.

- **Choose Analyse from the Case menu**

If you are using Multiframe4D, a dialog will be displayed to specify analysis options. Notice that 4D will automatically detect that the structure is 2D and set the analysis type accordingly. This makes analysis faster and minimizes memory requirements. Ensure that just the Static check box is selected and then click OK to start the analysis.

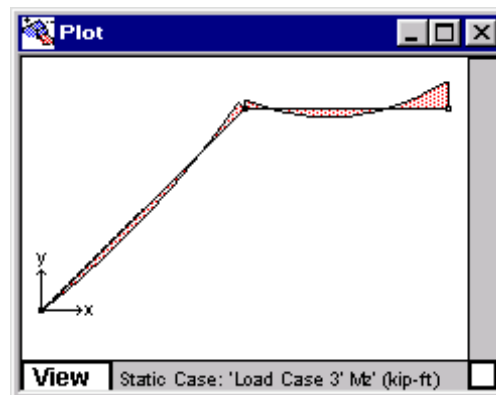


The analysis will commence. A progress bar will show the progress of the analysis. The progress bar will disappear and the cursor will turn back into an arrow once the analysis is complete.

Viewing Results

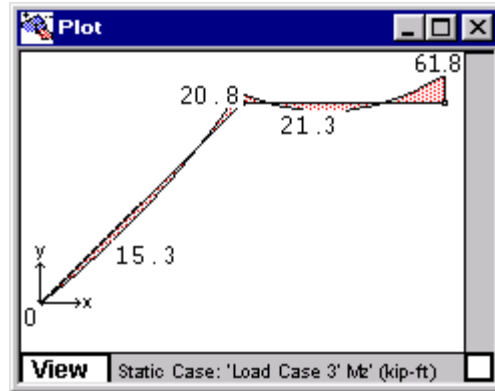
A graphical display of the forces and deflections within the structure is shown in the Plot window. Click anywhere in the visible part of the Plot window to bring it to the front. If the Plot window is not visible, choose Plot from the Window menu to bring it to the front.

The initial diagram displayed will be that of bending moments for the current load case. As in the Load window, the name of the current load case is displayed in the bottom left hand corner of the window. The bending moment diagram is drawn so that the moment diagram is drawn on the tension face of the member.



You can also display the magnitudes of the actions on the diagram.

- Choose Symbols from the Display menu
- Click the Plot Labels check box to turn it on
- Click the OK button

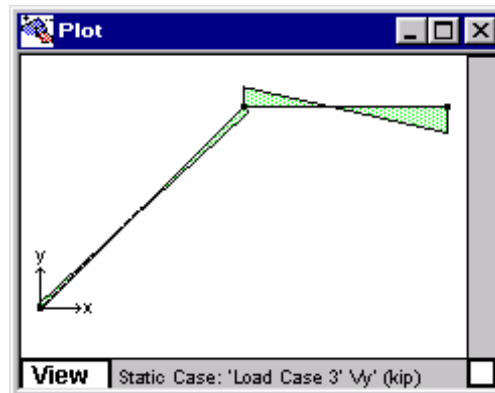


The values on the diagrams will now be displayed at the ends of the members and at any intermediate maximum or minimum points.

You can also view the shear forces in the frame.

- Choose Shear V_y from the Actions sub-menu under the Display menu
- Or Click on the Shear icon in the toolbar at the top of the Plot window

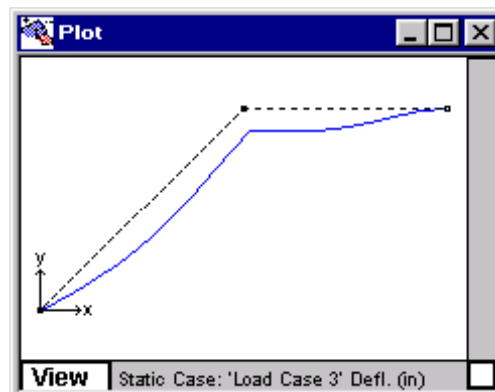
The shear force diagram for the whole structure will now be displayed in the Plot window.



The deflected shape of the frame may also be displayed.

- Choose Deflection from the Display menu or click the Deflection icon in the toolbar

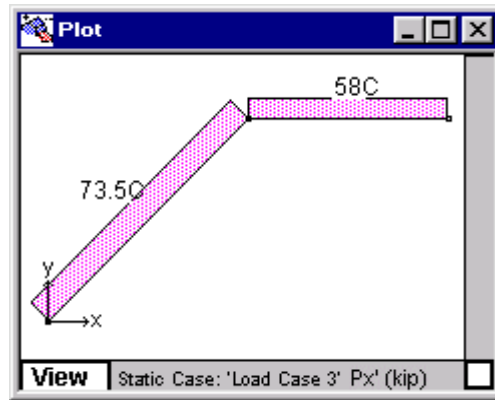
A diagram of the structure drawn in gray will be displayed with an exaggerated deflected shape drawn in blue over the top of it.



The axial forces resulting from the combined load case can be viewed also.

- **Choose Axial P_x from the Actions sub-menu under the Display menu or click the P_x icon in the toolbar**

This will display the axial forces in the frame with compressive forces being shown as positive.



To return to a display of the bending moment diagram for the structure

- **Choose Moment M_z from the Action sub menu under the Display menu**

Viewing Different Load Cases

To view the diagrams for a different load case, you can choose the name of the case you wish to view from the Case menu.

- **Choose Load Case 2 from the Case menu**

The bending moment diagram for this load case will be displayed in the Plot window. The bending moment diagrams for each of the load cases is drawn to the same scale. From this you can see that the moments induced by Load Case 2 are slightly less than those of Load Case 3 as would be expected.

- **Choose Load Case 1 from the Case menu**

You will see that the magnitude of the moments for this load case is smaller still.

- **Choose Load Case 3 from the Case menu**

This will return to the display of the results for the combined loading condition.

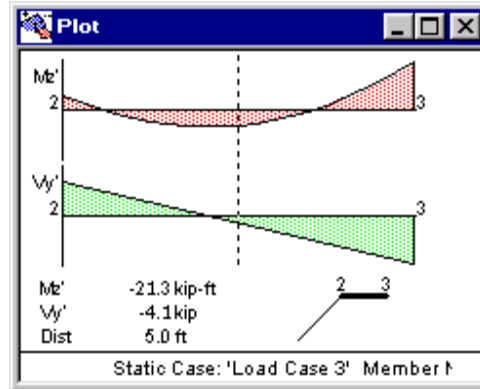
Viewing Member Diagrams

The diagrams of the whole structure you have viewed so far are useful for determining the overall behavior of the structure and as a quick check that you have created the structure correctly. To determine the exact magnitude of the forces and deflections however, it may be necessary to look at the diagrams for each member.

You can view the diagram for a member by clicking on it.

- **Point at the original position of the horizontal member and click**

The original position of the member is shown in gray. The local diagrams for this member will now be displayed in the window.



Notice that the values on the diagrams at the location of the crosshair are shown below the diagram. The number below this shows the current distance of the crosshair along the member (it will initially be halfway along the member). The diagram in the lower right hand corner of the window shows the member selected and its adjacent members and joints in the structure.

You can view the value on the diagram by dragging the crosshair with the mouse.

- **Point to the gray crosshair and press the mouse button and hold it down**
- **Drag the crosshair back and forth along the member**

As you move the crosshair you can view the values at any position.

- **Release the mouse button to fix the crosshair at a new position**

Try moving the crosshair back and forth a few times to get used to viewing member actions in this way.

To return to the diagram of the whole structure

- **Click on the drawing in the lower right hand corner of the window**

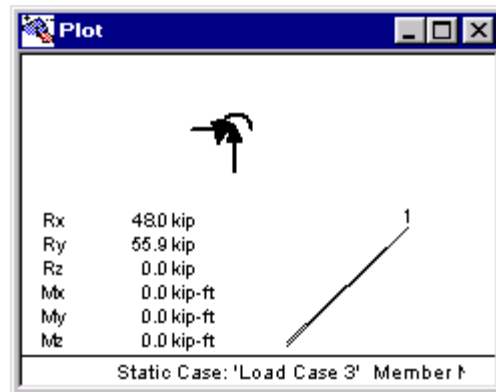
Viewing Joint Displacements and Reactions

You can use a similar technique to view the joint displacements and reactions in the structure. First, turn back on the display of the moment diagram.

- **Choose Moment Mz' from the Actions sub-menu under the Display menu**

Now you can check the reactions at a joint by clicking on it.

- **Point to the lower left hand joint of the frame and click**



The reactions for this joint will be shown graphically with the arrows pointing in the direction of the reaction. The magnitudes of the reactions will be shown in the bottom left corner of the window.

- **Click on the diagram at the lower right of the window**

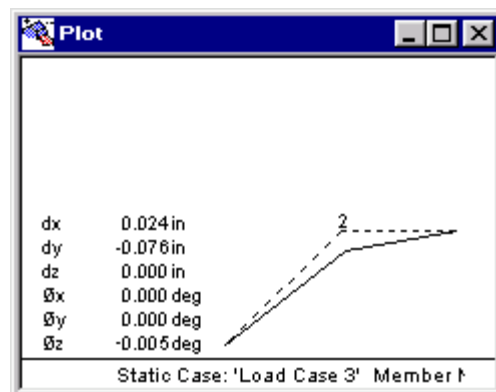
This will return to the diagram of the whole structure.

You can view deflections at a joint in a similar way.

- **Choose Deflection from the Display menu**

To view the magnitude of the deflection at the middle joint

- **Click on the middle joint of the frame**



The deflections of this joint will be displayed with the magnitudes of the deflection being shown on the left of the window.

- **Click on the drawing at the bottom right to return to a diagram of the whole structure.**

Tables of Results

If you wish to view the results of the analysis in numerical form, the Result window may be displayed which will show tables of results from the analysis.

To make the Result window visible

- **Choose Result from the Window menu**

When the Results window appears you will see a table showing the displacements of the joints in the structure and the joint reactions as computed in the analysis. If there are more joints than can be displayed in the window at one time, the scroll bars may be used to scroll through the table and view the results for any joint. The horizontal scroll bars can be used to view different columns in the

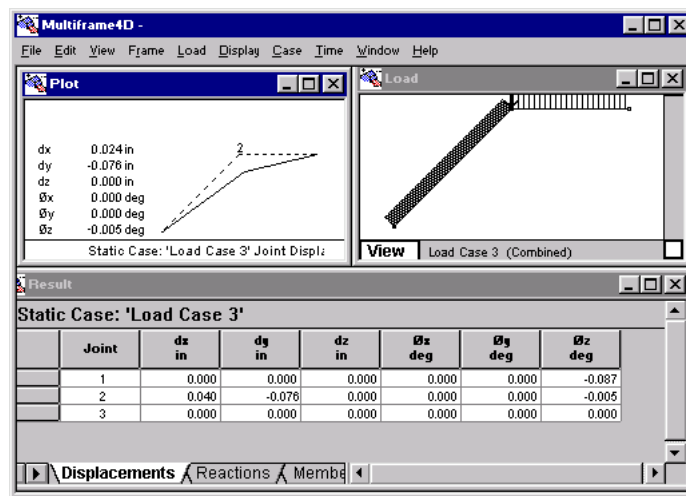
table. On Windows you can also Right Click in the column headings to sort and hide columns.

	Joint	dx in	dy in	dz in	θx deg	θy deg	θz deg
1	1	0.000	0.000	0.000	0.000	0.000	-0.087
2	2	0.024	-0.076	0.000	0.000	0.000	-0.005
3	3	0.000	0.000	0.000	0.000	0.000	0.000

You may find the window layout options useful for examining the results of the analysis.

- Choose Result Layout from the Window menu

The Plot, Load and Result windows will be laid out on the screen.



Notice that each joint in the results table is identified by a number. You can display these numbers on the graphics of the structure.

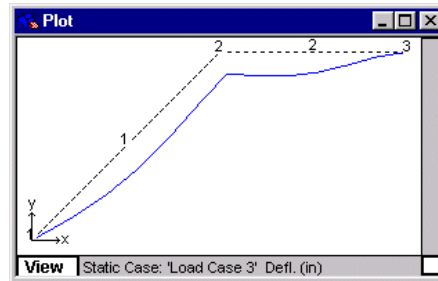
- Make sure the Plot window is in front

- Choose Symbols from the Display menu

A dialog box will appear with a number of check boxes in it.

- Check the Joint Numbers and Member Numbers boxes
- Click the OK button

Notice now that the numbers of the joints and members are displayed in the Plot and Load windows. You can refer to these diagrams while examining the results numerically.



The computed member actions may also be viewed in the Result window.

- Click in the Result window to bring it to the front

Remember that the front window at any time is the one that has its title bar highlighted.

- Choose Member Actions from the Results sub-menu under the Display menu (or on Windows, click the Member Actions tab at the bottom of the window)

A table of computed member end forces will be displayed.

	Memb	Joint	Px' kip	Py' kip	Vz' kip	Tx' kip-ft	My' kip-ft	Mz' kip-ft
1	1	1	73.470	5.603	0.000	0.000	0.000	0.000
2	1	2	-73.470	8.539	0.000	0.000	0.000	-20.758
3	2	2	57.989	20.913	0.000	0.000	0.000	20.758
4	2	3	-57.989	29.087	0.000	0.000	0.000	-61.627

Each row in the table displays the end forces for a member in the structure. You can refer to the member numbers displayed on the graphics in the Plot window to determine which member is which.

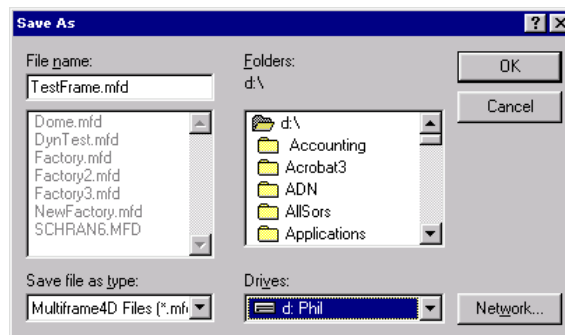
Saving Your Work

If you are using the Multiframe demo, you will not be able to save. Demo users may wish to skip to the Summary to continue reading.

Now that you have created and analysed a structure, it is probably a good idea to save the work you have done.

- Choose Save As from the File menu

A dialog box will appear which allows you to type in a name for your structure and to specify which disk or which folder it is to be stored in.



- Type in "TestFrame.mfd" for the name of your frame

Macintosh users: Including the .mfd suffix to your file names ensures that your files can be read by Windows users.

- **Click on the OK/Save button**

The file will be saved to disk. Later you can read this structure back in and look at the data and the results or perhaps make some changes to the structure and analyse it once more.

Printing

If you are using the Multiframe demo, you will not be able to print. Demo users may wish to skip to the Summary to continue reading.

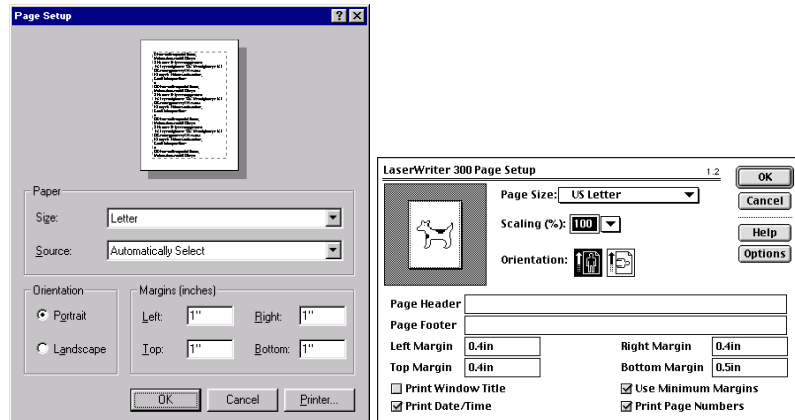
Now that you have some data describing the structure and its behavior, you may wish to print a hard copy of the work you have done. Multiframe allows you to print a report, the contents of any window or a list of member diagrams.

Setting up the Printer

First, ensure that your printer is attached to your computer with an appropriate cable, that it is switched on and has sufficient paper loaded. Refer to your printer manual if you have any problems.

- **Choose Page Setup from the File menu**

A dialog box will appear which allows you to choose the paper size and orientation, set the size of margins for printing and type in text to appear at the head and foot of each page of paper printed.



- **Select the paper size you have installed in your printer**

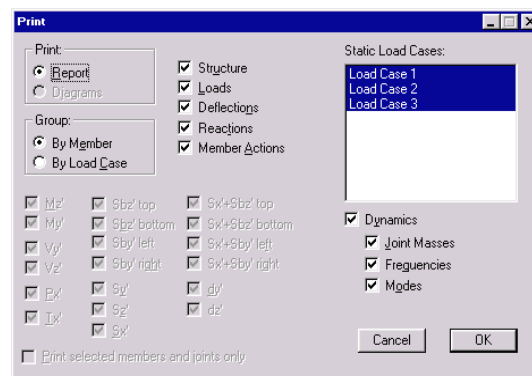
You can also choose to enlarge or reduce the printout and adjust the output with a number of other options. You will not usually require these more sophisticated options with Multiframe.

- **Click the OK button**

Multiframe allows you to enter values for header and footer titles to be printed at the top and bottom of each page printed. You can also set up the size of the margins you would like around the printed output. On Macintosh the two check boxes at the lower left of the dialog allow you to switch on or off the printing of page numbers and date and time information. On Windows these options appear under the Titles button in the Print Preview window.

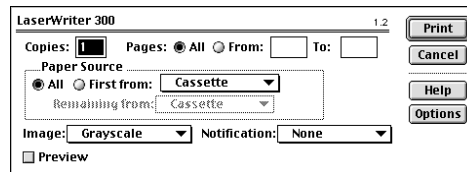
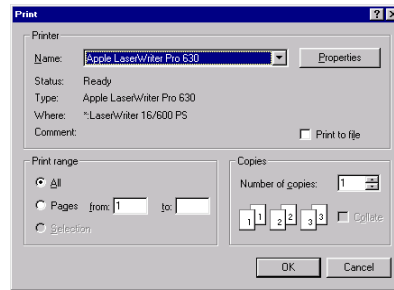
To print out the data describing the structure you have created and the results of the analysis

- A dialog box will appear with a number of printing options.



- **Click the OK button**

A second dialog box will appear which lets you set the printing details for the particular printer you are using.



- **Click OK to print**

A dialog box will be displayed while printing is in progress. Multiframe will print out the results on your printer.

Macintosh users: If you want to cancel the printing while it is in progress, hold down the **Command** key and type a period ".".

Windows Users: If you want to cancel the printing while it is in progress, press the "Esc" key.

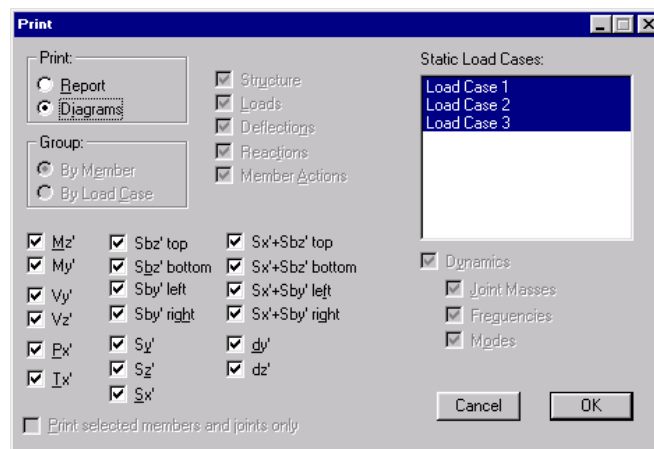
Printing Diagrams

(Macintosh Only)

Multiframe allows you to print the force or deflection diagrams for some or all of the members in the structure. First, make sure that the Plot window is frontmost. Multiframe lets you print the diagrams of the selected members in the Plot window. (Remember you can select all the members in the Plot window by choosing Select All from the Select sub-menu under the Edit menu.)

- **Choose Print Report... from the File menu**

The printing dialog box will appear.



- **Click on the Diagram button at the top left of the dialog**

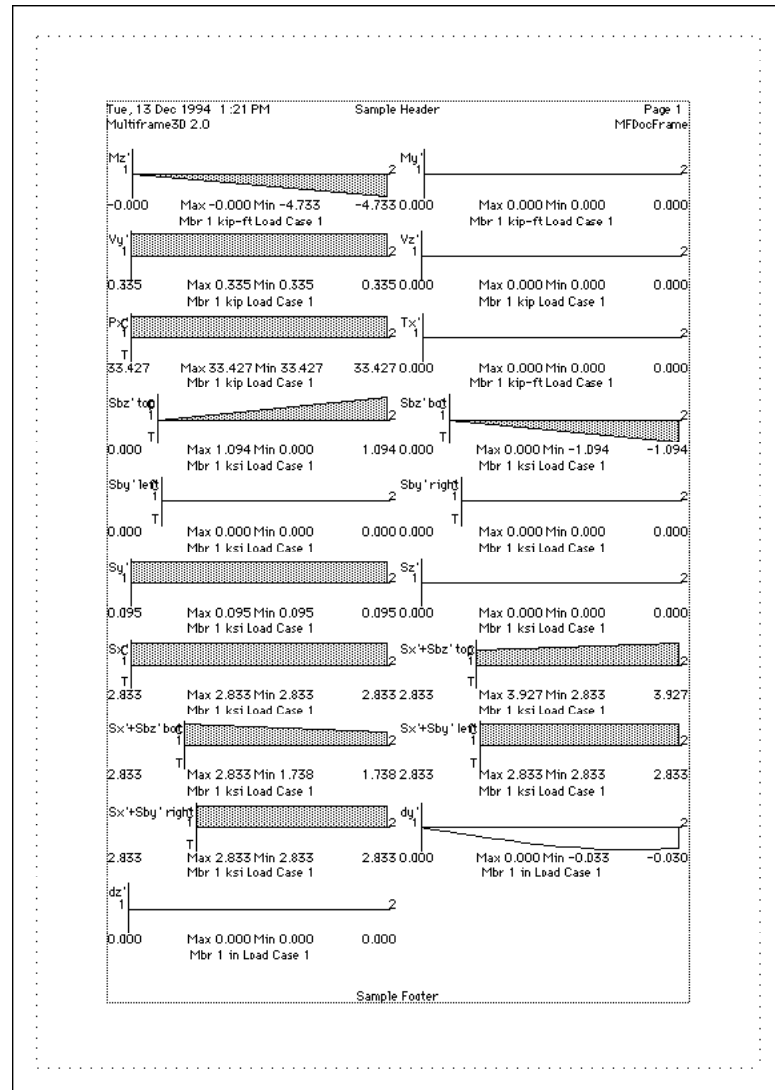
This tells Multiframe that you want to print the diagrams for the selected members rather than print all the data.

- **Click the OK button**

The usual print dialog will appear.

- Click OK in the print dialog to start the printing

Multiframe will print out the moment and shear diagrams for each member for the three load cases as shown below.



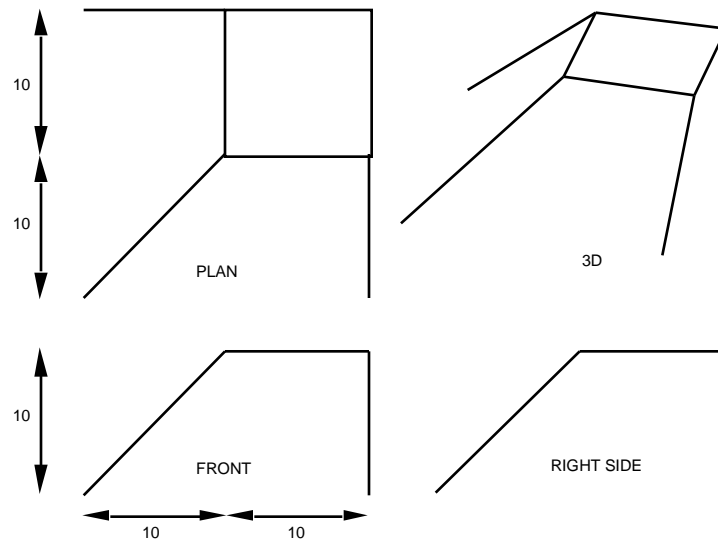
Summary

This completes our brief introduction to the capabilities of Multiframe. You should now be able to create and analyse 2D structures using Multiframe. Of course, you have only looked at the most basic methods available to the user of Multiframe. Multiframe has many powerful features to allow you to create and analyse structures quickly and effectively and with fewer errors. This means there is often more than one way to accomplish a given task. Take the time to investigate the techniques described in the next chapter and determine which methods are most suitable for your style of operation and your type of structures.

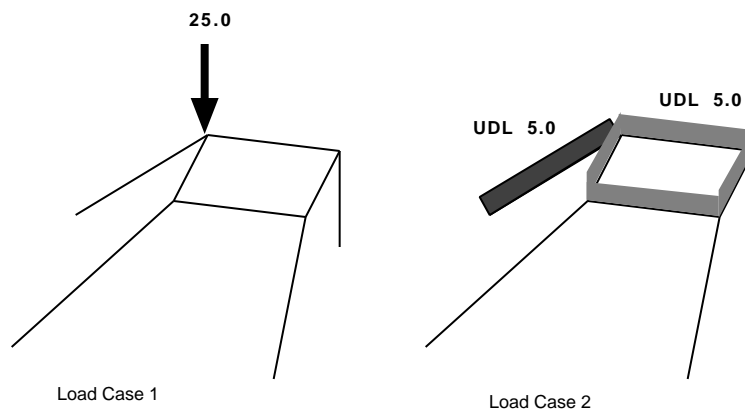
For 3D and 4D users, the following tutorial extends the above example to a 3D problem. 2D users may wish to skip to the next chapter to continue reading.

3D Tutorial

To extend the concepts and techniques you have learned, this section will describe how to extend your 2D structure into 3D. The frame you will analyse is shown below.



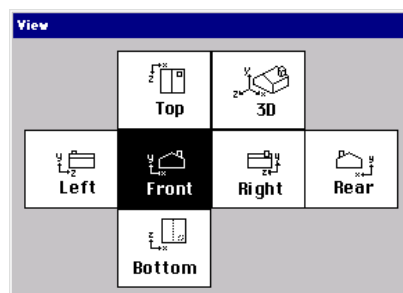
You will analyse it for the two different loading conditions shown and for a superposition of the two loads.



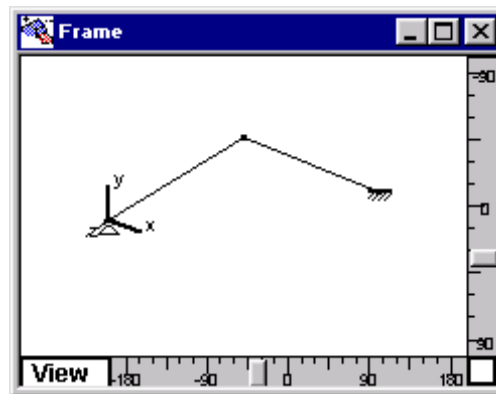
So far you have been drawing in a front view of the Frame. To change your view of the frame you use the View button in the bottom left hand corner of the window. First, make sure the Frame window is in front then

- Click on the View button

A dialog will appear with a number of icons showing the seven different views possible in a Multiframe window.



- Click on the 3D icon to change to a 3D view



Selecting Members

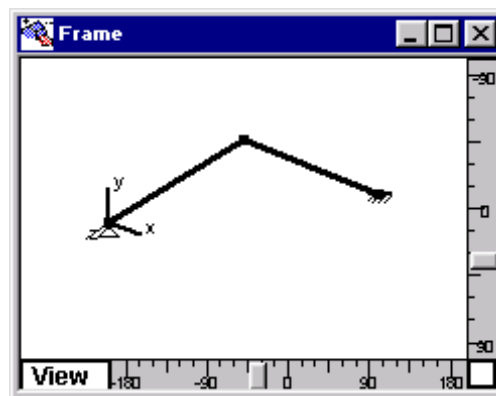
Rather than draw the additional members in the Frame, you can automatically create them by using the Duplicate and Extrude commands from the Frame menu. Before duplicating a part of a frame, it is necessary to select the members to be duplicated.

- Point to the sloping member, away from its ends, and click

Notice that the member is drawn with a bold line indicating that it is selected. Next, select the other member to be duplicated.

- Hold down the shift key and click on the second member

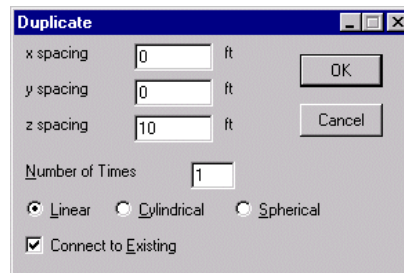
When you hold down the shift key while clicking on a member the member is added to the current selection if it not already selected or removed from the current selection if it is already selected. In this case, both members will be drawn with a bold line to indicate that they are selected.



Once you have selected the two members,

- Choose Duplicate from the Frame menu

A dialog box will appear with a number of fields, which allow you to specify the spacing of the duplicated members in each of the axis directions. In this case the spacing of the duplicated members is 0, 0, and 10.000 in the x, y and z directions respectively.

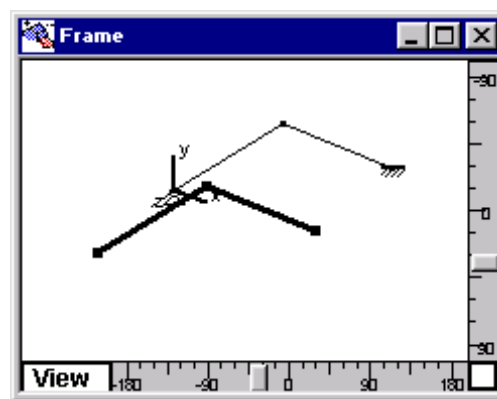


- **Type in 0, 0 and 10 for the spacing**

Since we will only be duplicating the selected members once, you can leave the Number of Times field set to 1.

- **Click the OK button**

The selected members will be duplicated to the new position.



Next, we will add the two right hand supports in the frame by using the Extrude command. Extrude allows you to construct a member by projecting it in space from an existing joint parallel to one of the coordinate axes. This provides an easy way to create beams and columns, which run horizontally or vertically.

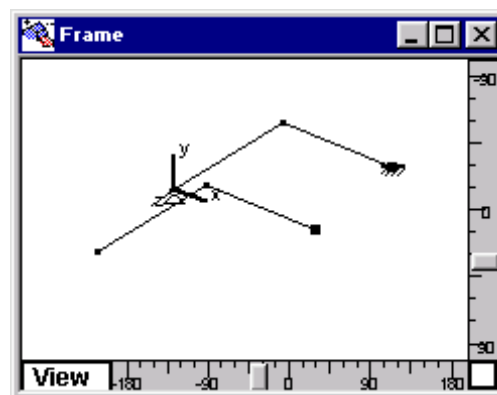
First, select the two joints at the right hand side of the frame

- **Click on the first joint**

This will select the first joint and indicate it is selected by drawing a bold box on top of it

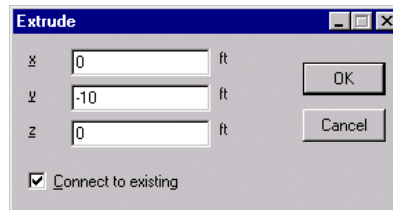
- **Hold down the shift key and click on the second joint**

This will select the second joint and also draw a bold box on top of it. In the same way that you selected more than one member by holding down the shift key when clicking, you can now select more than one joint.



- **Choose Extrude... from the Frame menu**

A dialog box will appear which allows you to specify how far and in which direction to extrude the members.

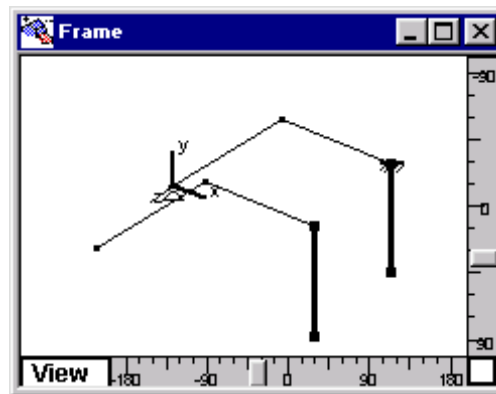


In this case we wish to project the members down by 10.

- **Enter 0, -10 and 0 for the extrusion dimensions**

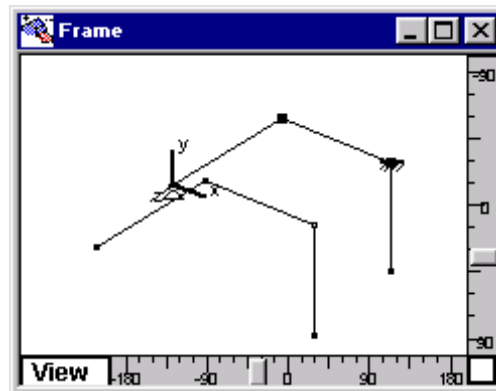
- **Click the OK button**

The two members will be added and connected to the two existing joints.



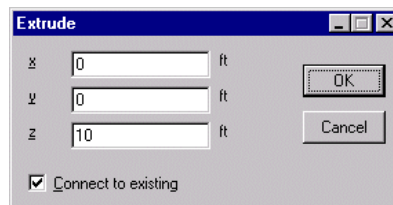
Next, we will add the two beams, which connect the two frames by using the Extrude command.

First, select the two joints at the top of the first frame.



- **Choose Extrude... from the Frame menu**

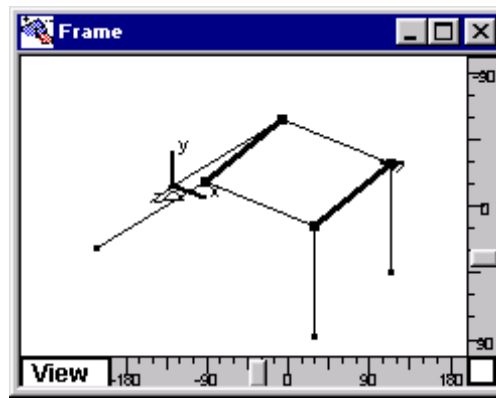
A dialog box will appear which allows you to specify how far and in which direction to extrude the members.



In this case we wish to project the members in the z direction by 10

- Enter 0, 0 and 10 for the extrusion dimensions
- Click the OK button

The two members will be added and connected to the two existing joints.

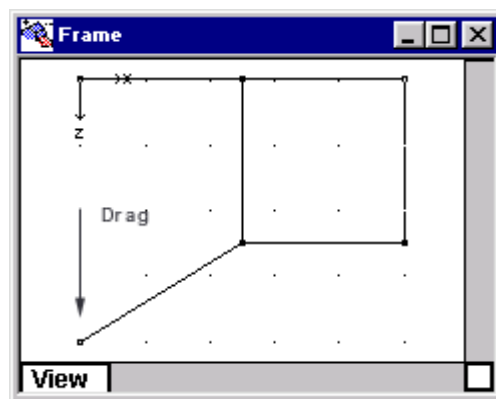


Next, we need to move the lower joint of the frontmost diagonal leg so that it is inclined to both the x and z axes.

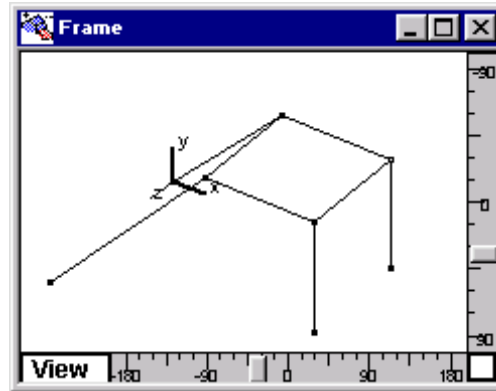
- Use the View button to change to a Top view

First, click away from any joints to turn off the currently selected joints. Next, turn the Grid on again to make it easier to drag the joint to the desired location.

Now, point to the bottom left joint with the mouse, click and hold down the mouse button, then drag the mouse to the new location of the joint where the coordinates will read $x=0.000$, $y=0.000$, $z=20.000$. Note that although the y coordinate shows the current depth (10.000 in this case) Multiframe does not change the depth of a joint when you drag it in a two dimensional view.



Change back to the 3D view to check that your frame now looks correct. If necessary, you can use the Size To Fit command from the View menu to make the display of the frame fit into the Frame window.



Clipping

One problem when working with framed structures in three dimensions is that it is difficult to tell which members are in front and which are behind when working in two dimensional views. Multiframe provides you with clipping controls which lets you control how much of the structure can be worked with at one time. Clipping involves defining a three dimensional box which encloses the part of the frame which is of interest to you.

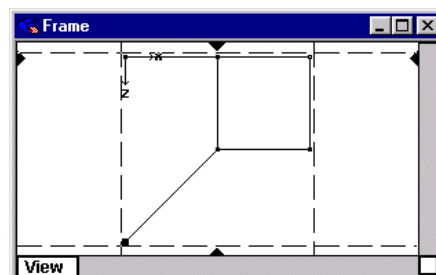
You will find it helpful to turn off the display of the grid while working with the clipping controls.

- **Choose Grid... from the View Menu**
- **Click on the Invisible radio button to turn off the display of the grid**
- **Click OK**

The grid is no longer displayed however the grid alignment features in the Frame window will still be active.

To start working with clipping, first make the Top view the current view by using the View button at the bottom left of the window. Now you will turn on clipping by initially positioning the clipping bars around the frame.

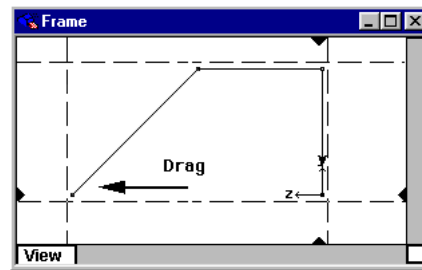
- **Choose Clip To Frame from the Clipping sub-menu under the View menu**



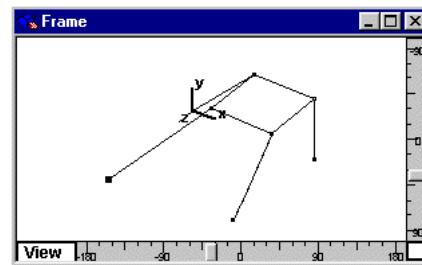
This positions the walls of the clipping box so that they just surround the frame in each direction. You can change the location of the walls of the clipping box by dragging its boundaries with the mouse.

- **Point to the left hand clipping bar and press the mouse button**
- **Drag this bar until it is just to the left of the center joint of the frame**
- **Release the mouse button**

Notice that members which are completely inside the clipping box are drawn in black or colored as usual while members which are not completely inside the box are drawn in gray. A member drawn in gray cannot be selected, moved or connected to. Now change to the Right Side view.



Click and drag the bottom left joint of the frame to a new location of $x=10.000$, $y=0.000$, $z=20.000$. With the clipping on there is no danger of accidentally connecting the joint to a joint behind it in space. Finally, use the View button to go back to a 3D isometric view of the frame to check that you have created it correctly. Use the Size To Fit command from the View menu to position the whole frame in the middle of the window.



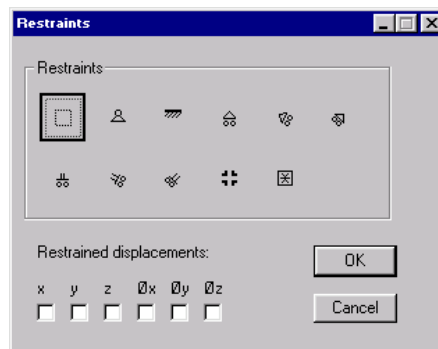
This completes the setup of the geometry of the frame. You can now turn off clipping by choosing No Clipping from the Clipping sub-menu under the View menu. You can now move on to applying the restraints and section types needed for the structure.

Restraining the Joints

You will now need to set the restraints for the 3D frame. First, remove the existing restraints from the structure.

To remove all the restraints from the frame

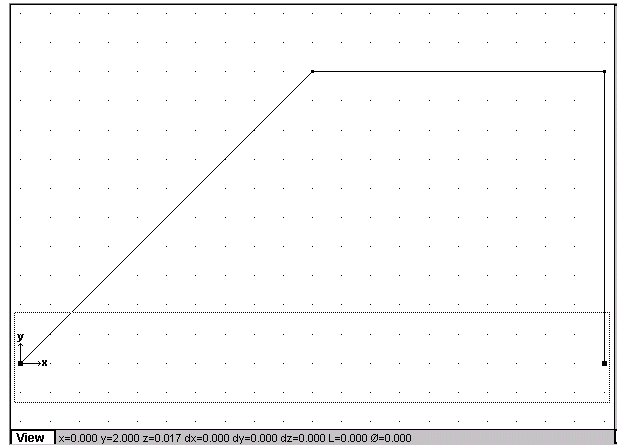
- Choose Select All from the Select sub menu under the Edit menu
- Choose Joint Restraint from the Frame menu or from the short cut menu



- Click on the No Restraint icon
- Click OK

You will find it helpful to turn off clipping and turn off the grid now that you have finished drawing the structure.

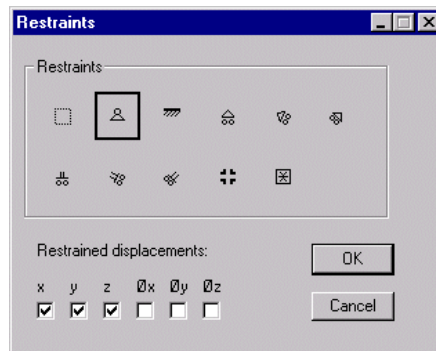
Next, change to the front view before setting the restraints. Now, drag a selection rectangle enclosing the joints at the bottom of the frame, using the mouse.



Now apply the pinned restraint to the joints you have selected.

- Choose Joint Restraint from the Frame menu or from the short cut menu

The joint restraints dialog will appear.

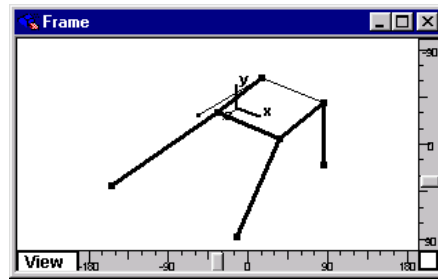


- Click on the pinned joint icon
- Click OK

This will apply a pinned restraint to the selected joints. A Pinned Joint cannot move, but is free to rotate. The check boxes at the bottom of the dialog indicate which degrees of freedom will be restrained by each icon.

Section Properties

It is now necessary to select the members that you have added to the structure and specify their section types.



- Select the 5 new members
- Choose Section Type from the Frame menu
- Click on the name W in the list of section groups
- Use the scroll bar to scroll down the list of sections until the W12X40 is visible
- Click on this name to select it
- Click on the OK button to confirm your choice

Loads

Next, you can apply additional loads in the Load window.

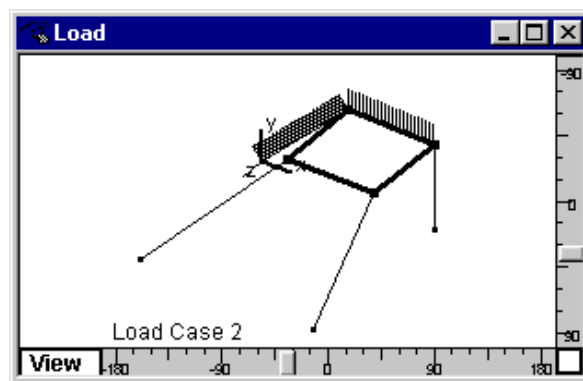
The point load we applied to the 2D frame will still be there so no further modification to Load Case 1 is required. Load Case 2 however has three additional distributed loads which you must now apply.

First, switch to Load Case 2.

- Choose Load Case 2 from the Case menu

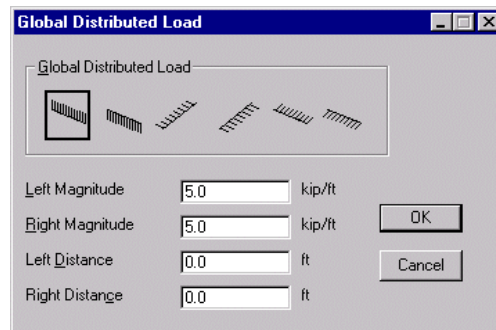
You can now add the distributed loads to the horizontal members in the frame.

- Select the horizontal members in the frame

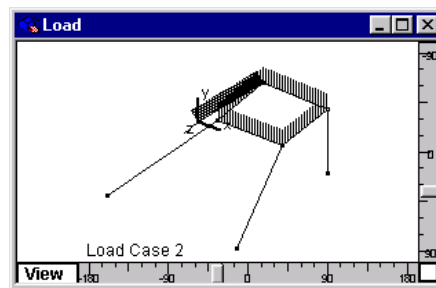


- Choose Global Dist'd Load from the Load menu or from the short cut menu

A dialog box will appear with a number of icons in it.



- Type a value of 5.0 for the left magnitude of the load
- Use the Tab key to move to the next number in the dialog
- Click OK to confirm the values you have entered



The loads you have added will be drawn on the members in the Load window.

Now, switch to Load Case 3.

- Choose Load Case 3 from the Case menu

Notice that the combined load case has been automatically updated with the loads you added to load case 2.

Analysis

Now that you have set up the structure and loading to your satisfaction, you can carry out the analysis of the frame.

- Choose Analyse from the Case menu

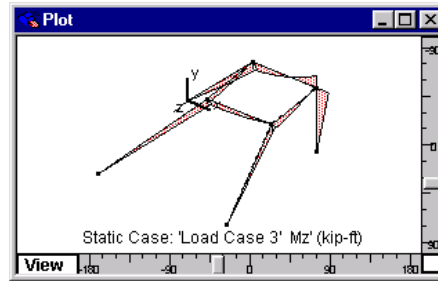
A progress bar will appear indicating how far along the analysis has progressed. The progress bar will disappear once the analysis is finished.

Viewing Results

You can now review the results of the 3D analysis in the Plot window. Click anywhere in the visible part of the Plot window to bring it to front or choose Plot from the Window menu if the Plot window is not visible.

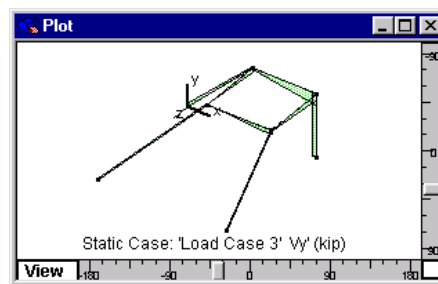
- Use the View button to change the 3D view
- Choose Moment Mz' from the Actions sub-menu under the Display menu

The bending moment diagram is drawn so that the moment diagram is drawn on the tension face of the member. The bending forces shown are those about an axis perpendicular to the web of the member.



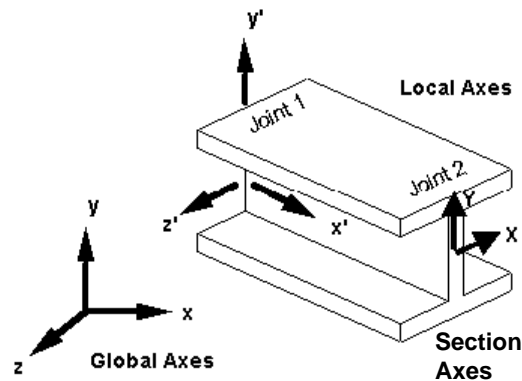
- Choose Shear $V_{y'}$ from the Diagram sub-menu under the Display menu

The shear force diagram for the whole structure will now be displayed in the Plot window. These shear forces are the shearing actions through the web or principal axis of each member.



Axes

The naming of member actions reflect the axis the actions apply to. There are three sets of axes in Multiframe

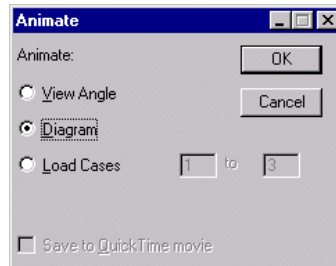


- Global Axes
are used to refer to joint coordinates and joint displacements.
- Member Axes
are used to specify the orientation of a member, the direction of perpendicular or tangential loads, the actions (moment shear and axial) acting on the member, stresses on the member and the member's deflection. Member axes always have a ' suffix. For example, $M_{z'}$ indicates that the moment is about the z' axis, $V_{y'}$ indicates shear through the y' axis and so on.
- Section Axes
are used to specify section properties such as moment of inertia, section modulus etc.

Animation

You can also animate your results while in the Plot window. Ensure that you are in the 3D view by clicking on "View" in the lower left-hand corner of the Plot window and ensure that Deflections are displayed by choosing Deflection from the Display menu.

- **Select Load Case 3 from the Case menu**
- **Choose Animate... from the Display menu**



Make sure that "Diagram" is selected and click OK. Multiframe will animate the deflected shape of your structure.

Summary

This completes our brief introduction to the capabilities of Multiframe. You should now be able to create and analyse structures using the range of commands and tools.

Of course, you have only looked at the most basic methods available to the user of the software. Multiframe has many powerful features to allow you to create and analyse structures quickly and effectively with fewer errors. This means there is often more than one way to accomplish a given task. Take the time to investigate the techniques described in the next chapter and determine which methods are most suitable for your style of operation and your type of structures.

Chapter 2

Using Multiframe

This chapter provides a more detailed description of the commands available in Multiframe.

Introduction

If you have read Chapter 1, you are now familiar with some of the features of structural analysis and design using Multiframe. This chapter presents a series of step-by-step instructions to the tasks covered in Chapter 1 as well as other procedures you will want to know about.

The chapter begins with a summary of basic computer skills, and this is followed by a description of the tasks involved in analysing and designing a structure using Multiframe. These tasks fall into three general categories; Creating the structure, specifying the loads and interpreting the results of analysis. The first three sections of this chapter reflect these categories. This is followed by detailed explanations of doing design calculations, printing and saving data and transferring data to other programs.

Summary of Mouse Techniques

You will use the following mouse techniques to do just about all of the tasks in this chapter.

- Click to select or activate something
- Press to cause a continuous action
- Drag to select, choose from a menu or move something
- Shift-Click to select or to extend or reduce a selection
- Ctrl-Click to select or to extend or reduce a selection (Windows only)
- Double-Click to get information about an object

To Click

Position the pointer on what you want to select or activate
Press and quickly release the mouse button

To Press

Position the pointer on something
Without moving the mouse, press and hold down the mouse button

The effects of pressing continue as long as the mouse button is held down. Pressing on a scroll arrow results in continuous scrolling. Pressing on a menu title pulls down the menu and keeps it down until you release the mouse button.

To Drag

Position the pointer on something
Press and hold down the mouse button and move the mouse
Release the mouse button

To Shift-Click

Summary of Keyboard Techniques

Shift-click is used to extend or reduce the selection of joints and members

Hold down the shift key and click on the joints or members you wish to add to the selection or which you wish to remove from the selection

To Ctrl-Click (Windows only)

Ctrl-Click is used to extend or reduce the selection of joints and members

Hold down the Ctrl key and click on the joints or members you wish to add to the selection or which you wish to remove from the selection

To Double-Click

Double click is used to get information about a joint or member in the Frame window

Point to the item you wish to double click and then click twice quickly in succession without moving the mouse.

Tab

You can use the Tab key to move horizontally within a table or to move from one field in a dialog to the next.

Return (Macintosh Only)

The Return key can be used to move to the next entry down in a table or the next line down in the CalcSheet and is the same as clicking on the OK button in a dialog.

Enter

The Enter key can be used to confirm the entry of numbers into a table and is the same as clicking OK in a dialog.

Arrow Keys

The keys may be used to move the selection in their respective directions in the Data or Result tables or in tables in dialog boxes.

Delete - Macintosh

Backspace - Windows and earlier Macintosh models

The Delete or Backspace key may be used to delete the current selection. If nothing is selected and you are typing text or numbers, it will delete the character to the left of the blinking cursor.

Command - Macintosh

Ctrl - Windows

The Command or Ctrl key may be held down while typing another key to choose a command from a menu without using the mouse. Menu items which have a key to the right of the name may be chosen in this way. For example, to choose Undo from the Edit Menu you could hold down the Command/Ctrl key and type Z.

Shift - Macintosh

Shift or Ctrl - Windows

You can hold down the Shift or Ctrl keys while clicking on something to add it to the current selection or remove it from the selection if it is already selected.

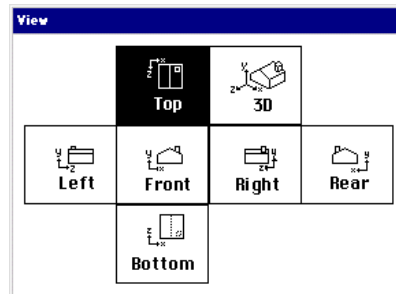
Holding down the shift or Ctrl while drawing a member, dragging a member or dragging a joint will constrain the movement to be horizontal, vertical or at a 45 degree angle.

Home

Takes you to the top of the table in the Data and Result windows.

Using Views

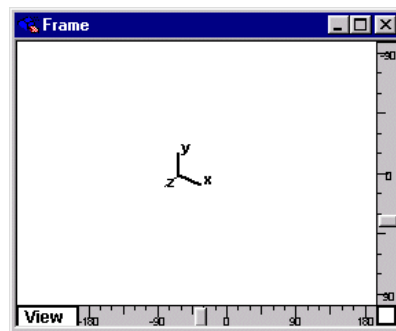
The View button in the bottom left hand corner of the Frame, Load and Plot windows can be used to display a range of two and three dimensional views of a structure. Each window has its own view and is controlled separately by its view button.



You can change the view by clicking on the View button and then clicking on the icon of the view you require.

Rotating a 3D View

When you have a 3D view displayed in the Frame, Load or Plot windows, you can control the angle of view by using the rotation controls at the bottom and right sides of the window. The axes indicator indicates the current angle of view.



The control at the bottom controls the rotation about the y axis while the control on the right hand side controls the rotation about the x axis. You can press in the arrows at the ends of the controls to gradually rotate the structure to a new position or you can click in the control to move the angle of rotation to the position of the mouse. If you hold down the mouse button after clicking in the control, you can rotate the structure back and forth until you have the desired angle of rotation. If your structure is too large to draw rapidly as you rotate it, Multiframe will draw a partial outline of the frame.

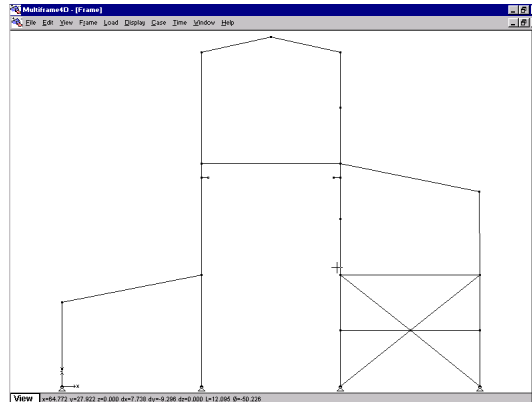
Zoom, Pan, Shrink and Size To Fit

The Zoom, Pan, Shrink and Size To Fit commands in the View Menu may be used to control the scale of the graphics displayed in the Frame, Load and Plot windows. Each view within a window has its own scale and center of interest. This means you can have a close-up plan view and a far away 3D view in the same window and simply switch from one to another using the View button.

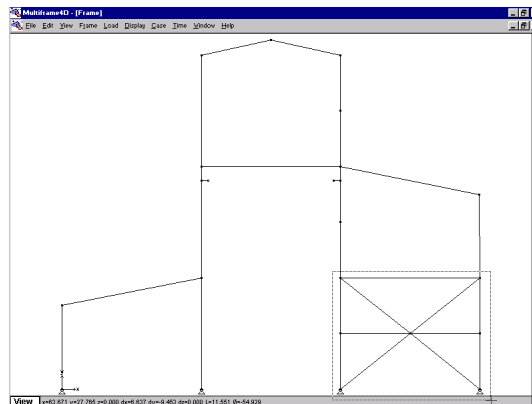
Zoom

Zoom allows you to increase the size of the drawing in the frontmost window.

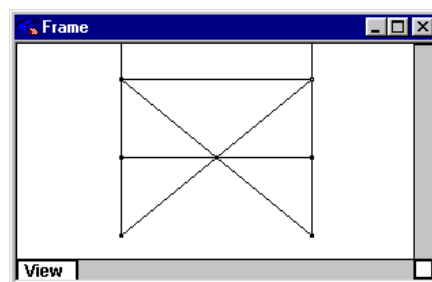
- Choose Zoom from the View Menu
- Move the pointer to the top left hand corner of the area you wish to view in close detail



- Drag a rectangle down and to the right which encloses the area of interest and release the mouse button



The window's contents will be re-drawn to display the part of the structure contained in the rectangle you have drawn.



You can use the Ctrl-E keyboard shortcut to use the Pan command.

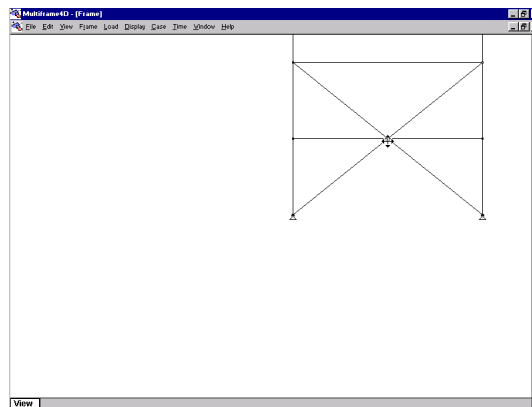
Pan

Pan allows you to shift the display of the structure within the window upwards, downwards, to the left or right.

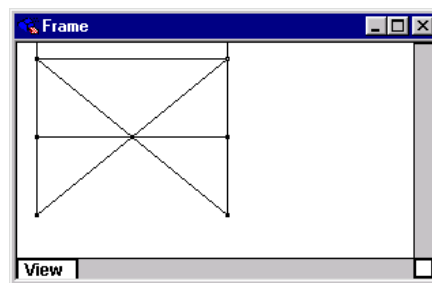
- Choose Pan from the View Menu

The cursor will change to a hand on the Macintosh or a box on Windows.

- Press inside the window and hold down the mouse button



- Drag the drawing to its new location
- Release the mouse button to re-draw the contents of the window.



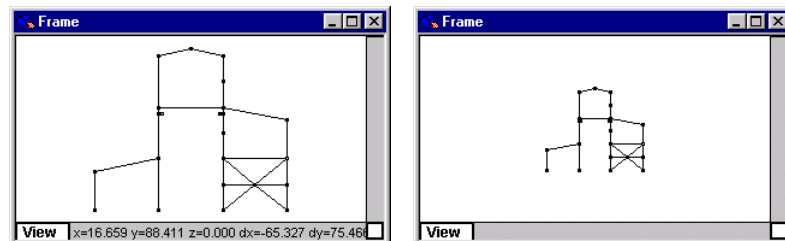
If not enough memory is available to move the image, a rectangle the size of the window will be moved around as you move the mouse.

Shrink

Shrink allows you to decrease the size of the drawing on the screen by 50%.

- Choose Shrink from the View Menu

The drawing will shrink down to 50% of its current size and be re-drawn.



Size To Fit

Size To Fit automatically resizes the drawing in the frontmost window so that the structure just fits inside the window in the current view. This is most useful after you have been zooming, panning or shrinking as it returns you to a viewing scale that just fits the frame inside the window.

Size To Fit Frame fits the whole frame inside the window.

Size To Fit Selection fits the selected members and joints inside the window.

Size To Fit Clipping fits the range of clipping inside the window.

Clipping and Masking

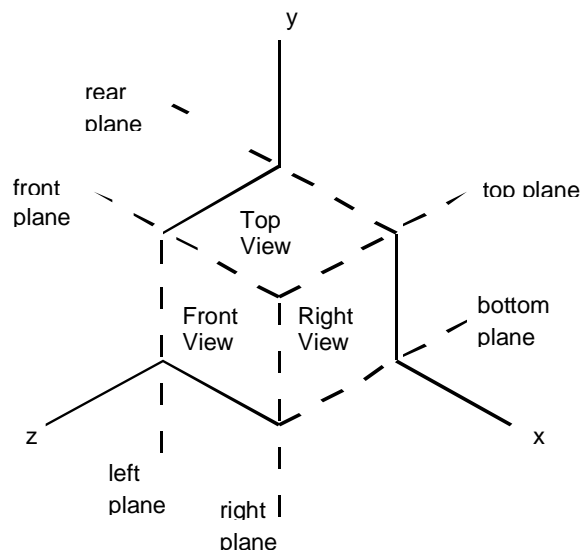
Multiframe allows you to control how much of the structure you wish to view at one time by use of two techniques named clipping and masking. Clipping allows you to define a three dimensional box which encloses the part of the structure you wish to work with while masking allows you to selectively show or hide any member or group of members in the structure.

For example, suppose you wish to view the bending moment diagram for a given column line in a frame. Without clipping the diagram is very difficult to decipher. However, if you select the column line and then choose Clip to Selection from the Clipping sub-menu under the Edit menu, the diagram is made much clearer. Masking may be used in a similar way. When a member is gray or hidden because of clipping or masking, it cannot be selected or moved with the mouse.

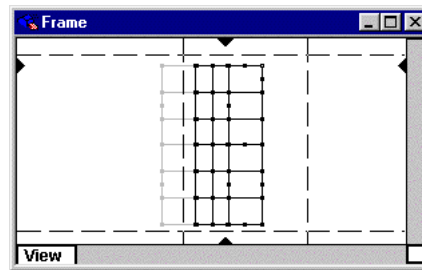
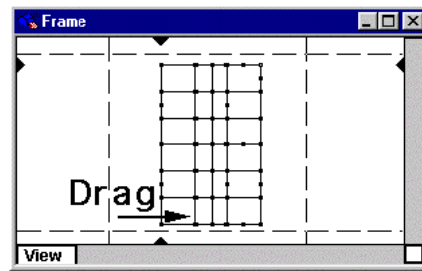
Clipping

You can control the clipping of the structure in any of the two dimensional views in a window. The clipping affects the display of graphics in the Frame, Load and Plot windows. This means that you can use the clipping controls in a view of one window to make it easier to see the graphics in a different view of another window. There are two types of clipping, clipping which draws the clipped out part of the frame in gray, and clipping which makes the clipped out members completely invisible.

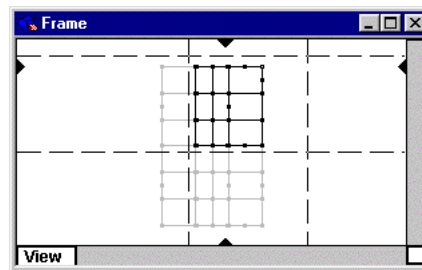
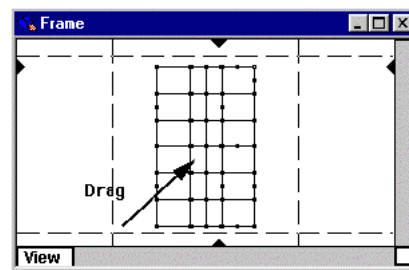
When clipping is turned on, the clipping bars are displayed as dotted lines in the two dimensional view. These dotted lines represent the boundaries of the clipping box.



To change the boundaries of the clipping box, and therefore change which members in the structure are visible, you can press and drag on the bars with the mouse.



If you want to move two bars simultaneously, you can press and drag on the intersection between the two bars and drag them at the same time.



Clip To Frame

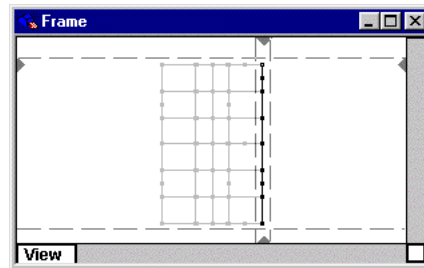
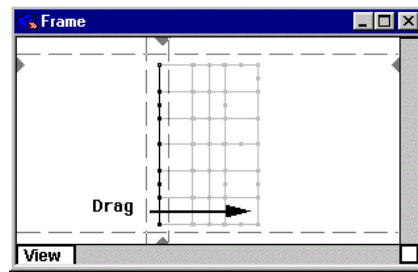
Usually you will find it convenient to start by choosing Clip To Frame from the Clipping menu. This positions the clipping bars so that they lie just outside the outer boundaries of the frame. You will then find it easy to move the appropriate bar to restrict your viewing to the part of the frame which is of interest to you.

Clip To Selection

You will also find it convenient to use the Clip To Selection command from the Clipping menu. This will position the clipping bars so that they lie just outside the farthest extents of the currently selected joints.

If you try to move a clipping bar past the position of its opposing bar (for example, move the bottom bar up past the top bar) the opposing bar will be moved to maintain a small distance between the two. This can be very useful when you want to move clipping from one floor to another or from one column line to another. You can clip on

the bar farthest from the direction you wish to move and drag the two bars together.



Masking

Masking allows you to control the visibility of the structure by selecting members and then choosing to show or hide them. If you use the Mask To Selection command in the Masking menu, this will hide all of the members in the structure except those, which are selected. If you choose Mask Out Selection, the selected members will be hidden and the remaining visible members will remain visible.

Like clipping, masking affects the display of graphics in the Frame, Load and Plot windows and can also be used in a view of one window to make it easier to see the graphics in a different view of another window. Masking can also be gray or invisible however Multiframe will always ensure that clipping and masking both use the same display method. i.e. either both will display in gray or both will make hidden members invisible.

Masking is useful for situations where the area you wish to view is not rectangular in shape and therefore is not suitable for clipping.

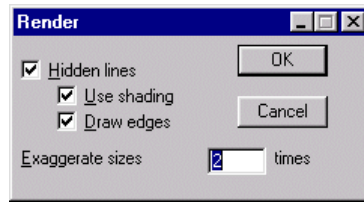
Rendering

Multiframe3D and Multiframe4D allow you to render the display of the frame in the Frame, Load and Plot windows as an aid to visualising the relative sizes and orientation of the sections in the structure. Rendering involves drawing a representation of the frame, complete with web and flange details, with hidden lines removed. Rendering can only be done in the 3D view in a window and only on the Deflection diagram in the Plot window. The number of segments drawn per member on the rendered, deflected shape is controlled by the precision set using the Plot... command from the Display menu.

To turn on rendering in the frontmost window

- **Choose Render from the Display menu**

A dialog will appear with the rendering options.

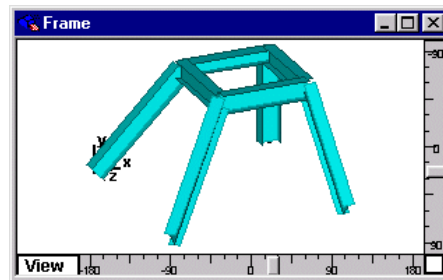


- **Check the Hidden Lines check box to turn on the rendering**

If you wish you can also choose whether to use shading on the sections and whether to draw separate lines for the edges of the sections. The Exaggerate sizes field can be used to exaggerate the sizes of the section shapes as they are drawn in the rendered view. Because the size of sections is usually very small compared to the size of the frame, this helps make the actual shapes more clearly visible.

- **Click the OK button**

The structure will be drawn using rendering until you turn rendering off. To turn off rendering, choose the Render command and turn off the Hidden Lines check box.



If you have clipping or masking turned on, rendering will only render the visible members. If you have drawn custom sections using Section Maker, rendering will display the actual shape of the custom section. Circular sections are displayed approximately as octagonal shapes.

Creating a Structure

Creating a structure with Multiframe involves defining the geometry, restraints, section types and the connections of the structure by drawing in the Frame window. You can also use the pre-defined generation aids to automatically construct portal frames or geometrically regular structures.

This section begins with a description of the drawing techniques you can use to create a structure. This is followed by a summary of how to select joints and members in the structure and how to use Multiframe capabilities to assist in generating commonly used structures. A description of the commands, which can be used to specify data defining restraints, section properties and members with pins or moment releases, completes the section.

Drawing

Members of the structure may be drawn directly by using the mouse to define the beginning and end points of a member. The scale at which this drawing is carried out may be specified by choosing the Size... command from the View Menu. The maximum and minimum coordinates to be used in the x, y and z directions may be entered and Multiframe will scale these coordinates to the current size of the Frame window. All movements in the window are accompanied by a display of the current pointer coordinates in the lower left hand corner of the window. All coordinates are shown in the current length units as set using the Units command from the View menu.

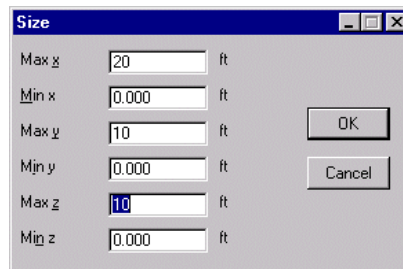
Setting the Size

Before starting drawing, you will need to set up the drawing area for the size of frame you intend working with. To do this

- **Choose Size... from the View Menu**

A dialog box will appear with the dimensions of the structure.

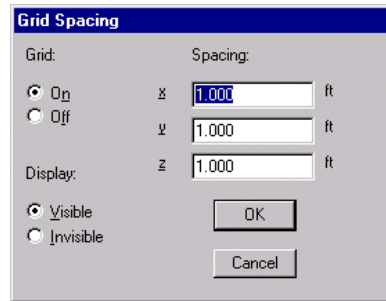
- **Enter the maximum and minimum coordinates you wish to use in each direction**
- **Click the OK button**



Using the Grid

Multiframe has a built-in facility to allow you to have your drawing automatically align with an evenly spaced grid. You can control the spacing, display and use of this grid with the Grid... command from the View Menu.

- **Choose Grid... from the View Menu**



- Enter values for the x, y and z spacing of the grid
- Click on the On button if you want drawing to align to the grid
This will switch on the Visible button to make the grid visible.
- Click the OK button

If you want to have the joints automatically align with the grid, but do not want to see the grid, simply click on the Invisible button before clicking OK. Similarly you can click on the Visible button to have the grid displayed as a visual guide, but click on the Off button to disable the automatic alignment with the grid.

All drawing and dragging in the Frame window will align to the grid while the grid is switched on.

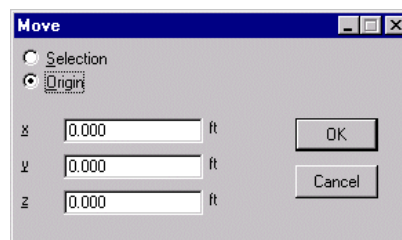
Setting the Origin

Multiframe has an origin for its coordinate system indicated by a symbol of x, y and z axes drawn in the Frame window. You can control the display of these axes by using the Axes command from the View Menu.

You can also change the location of the origin, relative to a structure.

- Choose Move ...from the Frame menu

A dialog box will appear allowing you to enter x, y and z coordinates for the new position of the origin.



This command is useful for creating structures, which are made up of larger sub-structures. You can shift the origin to a convenient location prior to doing the drawing for each sub-structure.

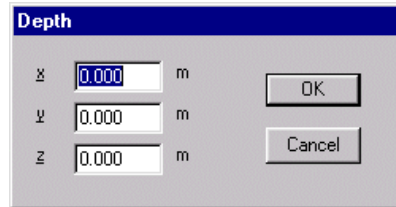
Drawing Depth

When you draw in a two dimensional view it is necessary to specify the depth at which you are drawing. For example, when drawing in the Top view, the y coordinate must be specified for any members drawn.

To change the current drawing depth

- Choose Depth... from the View menu

A dialog box will appear with the current depth selected.

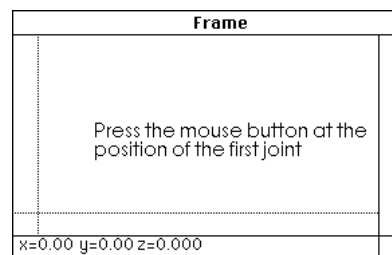


Type in a new value for the depth and click OK to set the new depth. If you have clipping turned on, you can set the depth by dragging the depth indicators which are the small black triangles drawn on the sides of the window between the clipping bars. The depth must always lie in between the clipping planes. If you try to drag the depth outside of this range, the clipping bars will be moved to allow the indicators to be moved to the new position. You can double click on the depth indicators to bring up the Depth dialog if you wish.

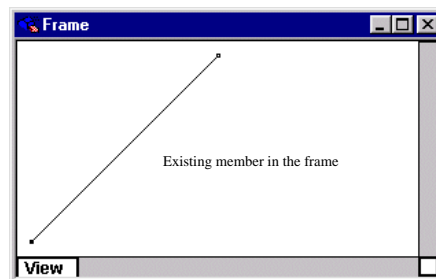
Adding a Member

To add a member to the structure

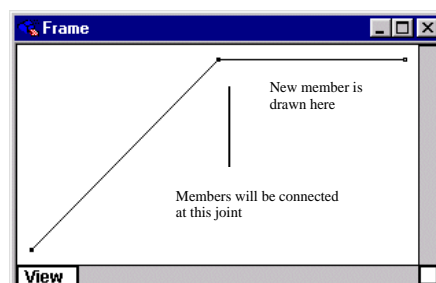
- Choose Add Member from the Frame menu
- Move the pointer to the position of the first joint



- Press the mouse button and drag to the position of the second joint (Windows users have the option of just clicking on the first joint instead of dragging)



- Release the mouse button (Windows users have the option of just clicking on the second joint to complete the new member)



Holding down the shift key while drawing a member constrains the new member to be either vertical or horizontal or at 45 degrees.

When you draw the new member, the coordinates of the mouse will be displayed in the bottom left corner of the window. Also, the distance and slope of the current position of the mouse from the last point will be displayed.

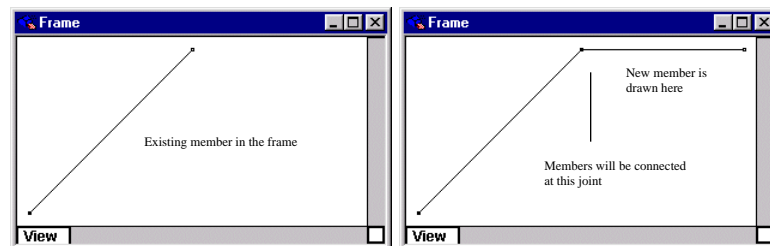
View | x=2.705 y=-0.058 z=0.000 dx=-10.361 dy=-13.123 dz=0.000 L=16.720 θ =-51.710

When drawing in the 3D view, you can only draw a member between two existing joints.

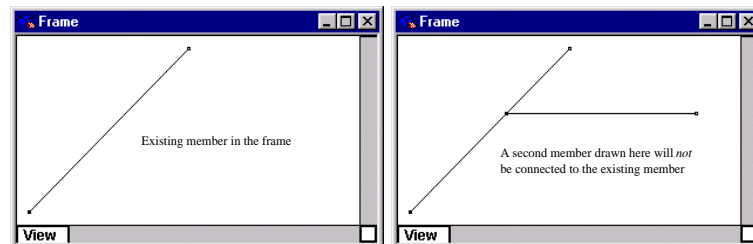
If the grid is turned on while you draw the member, the position of the joints will snap to the grid. The depth of the member will depend on the setting of the depth for the current view. See Depth above for an explanation of how to set this depth.

Connections Between Members

If the position of the first or second joint of the member you have drawn coincides with the position of an existing joint, the new member will be connected to the existing member at the existing joint. Note that this will happen even if the existing joint is at a different depth from the current drawing depth. For this reason, you will probably find it necessary to use the clipping and masking controls when drawing members which lie in front of or behind other members in the two dimensional views. When drawing in the 3D view, you can only draw a member between two existing joints.



The new member will not connect with an existing member unless you start or finish drawing on an existing joint.



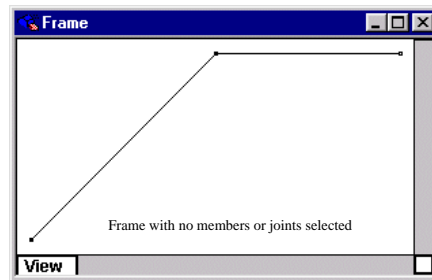
The connections between the members are set to be rigid. You can change this using the Member Type command if you require a connection between members, which cannot support moments.

Selections

In order to specify the properties, restraints and loads associated with joints and members, it is necessary to be able to identify which parts of the structure the various properties will be associated with.

You do this by graphically selecting joints and members prior to choosing menu commands. When you choose a command from a menu, to apply restraints at joints for example, the command will act

on the joints that are currently selected. Note that you cannot select a joint or member, which has been hidden using the clipping or masking commands.

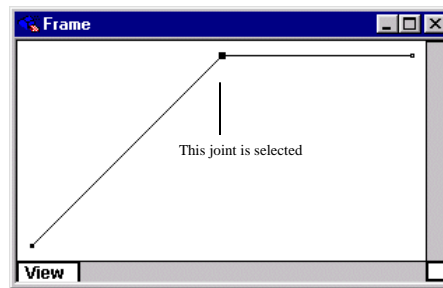


Selecting Joints

To select a joint

- **Click on the joint**

A selected joint is indicated by a solid black box around the joint.



To select a group of joints

- **Drag a rectangle which encloses the joints to be selected**

To select all the joints

- **Choose Select All from the Select sub-menu under the Edit menu**

- **Click the OK button**

To extend or reduce the selection

- **Shift-click to add a joint to or remove a joint from the current selection**
- **Shift-drag to add a group of joints to or remove a group of joints from the current selection**

If you know the number of the joint you wish to select, you can select it using the Select Joint command from the Select sub-menu under the Edit menu.

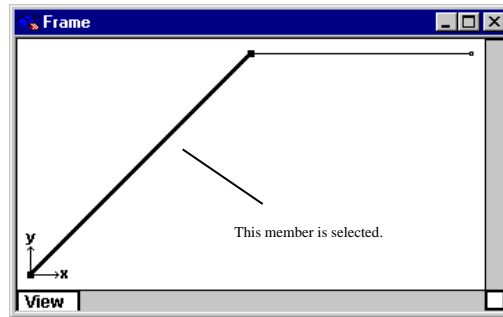
If more than one joint lies under the point where you click, Multiframe will select the joint closest to you.

Selecting Members

To select a member

- **Click on the member away from its ends**

A selected member is drawn with a heavy black line. A selected member always has the two joints at its ends selected.



To select a group of members

- **Drag a rectangle which encloses the members to be selected**

To select all the members

- **Choose Select All from Select sub-menu under the Edit menu**

To extend or reduce the selection

- **Shift-click to add a member to or remove a member from the current selection.**
- **Shift-drag to add a group of members to or remove a group of members from the current selection**

If you know the number of the member you wish to select, you can select it using the Select Member command from the Select sub-menu under the Edit menu.

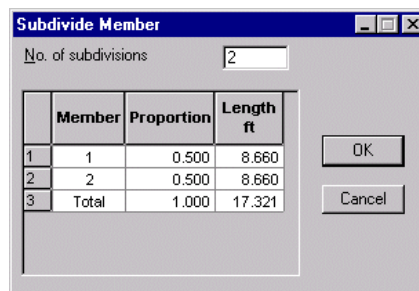
If more than one member lies under the point where you click, Multiframe will select the member closest to you.

Subdividing a Member

To subdivide a member in the structure

- **Select the member or members to be subdivided**
- **Choose Subdivide Member from the Frame menu**

A dialog box will appear with a field for the number of members you wish to subdivide the member into and a table listing the lengths of the new members.



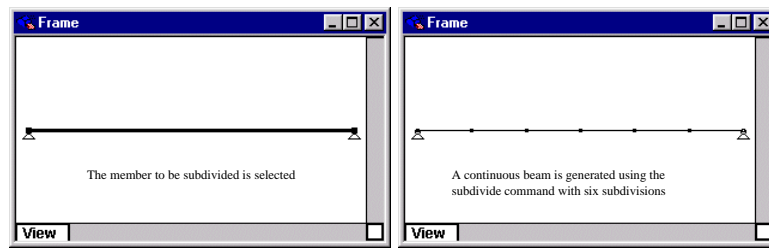
- **Enter the number of subdivisions to be created**

If you want the member to be subdivided evenly, leave the table of lengths unchanged otherwise, use the Tab key to move to the table and change the length or proportion of the new members.

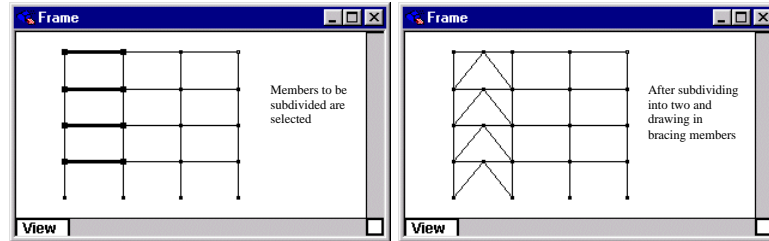
- **Click the OK button**

A number of new members connected end to end will be generated to replace each of the selected members. This command is particularly useful for generating continuous beams or for subdividing beams for the insertion of angle bracing.

Generation of a continuous beam using the subdivide command.

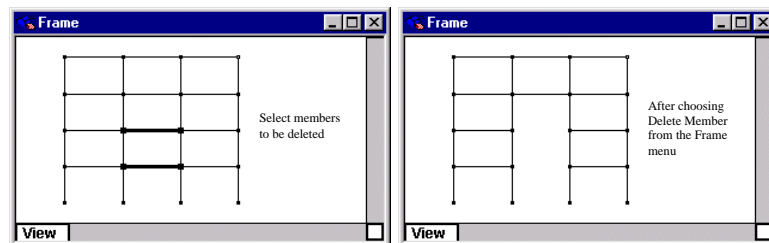


Insertion of bracing in a frame by subdividing floor beams.



Deleting a Member

- In the Frame window, select the member or members to be deleted
- Choose Delete Member from the Frame menu or..
- Type Delete (or Backspace on Windows machines and earlier Macintosh models)



Automatic Generation

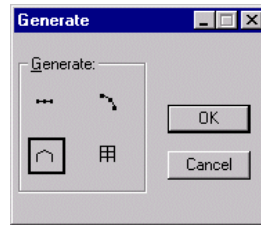
Multiframe includes capabilities for automatically generating frames, which occur frequently. The four facilities provided are for standard bay and storied frames such as those found in multi-story buildings, portal frames as used in many buildings with sloping roofs, continuous beams and curved structures.

Generating a Portal Frame

To generate a portal frame

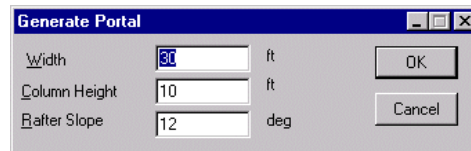
- Choose Generate... from the Frame menu

A dialog box will appear with the various generation icons in it.



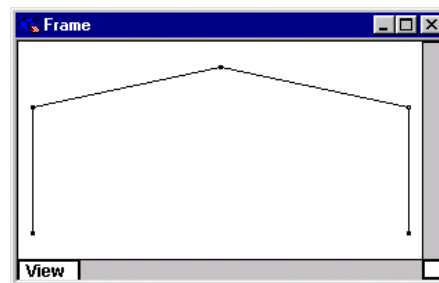
- Click on the portal frame icon and click OK

A dialog box will appear allowing you to enter the dimensions of the frame.



- Enter the values for the width, column height and the slope of the rafters
- Click the OK button

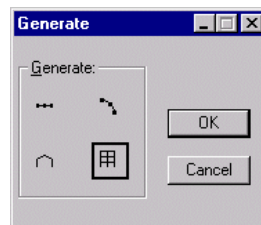
The pitched portal frame you generate will be shown in the Frame window.



Generating a Multi-story Frame

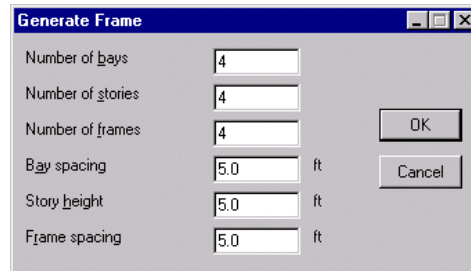
- Choose Generate... from the Frame menu

A dialog box will appear with the various generation icons in it.



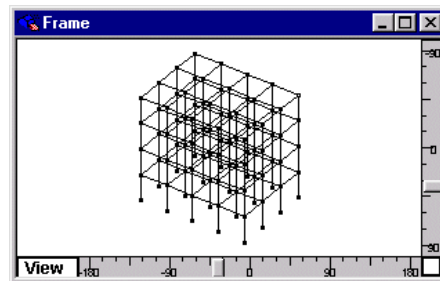
- Click on the multi-bay frame icon and click OK

A dialog box will appear with fields for the spacing in each direction. Bays run in the x direction, stories in the y direction and frames in the z direction.

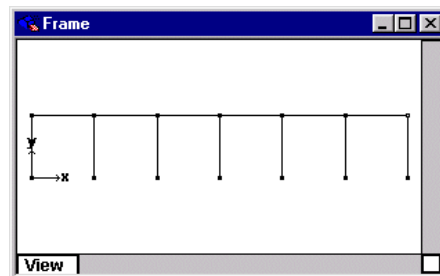


- Enter the values for the number of structural elements and the dimensions in each direction.
- Click the OK button

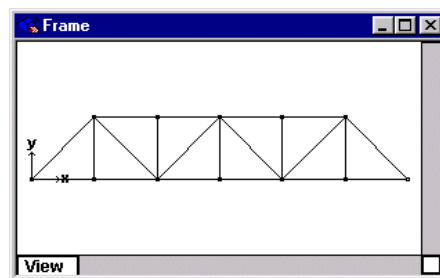
The frame you generate will be shown in the Frame window.



You may also find this command useful for generating the initial geometry for other structures such as trusses. You can start by generating the appropriate number of bays, 1 frame and 1 story.



You can then use the other drawing and duplication tools to create the structure you require e.g.



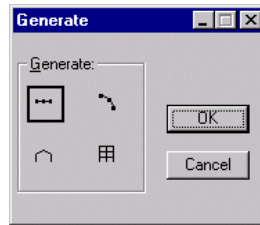
Generating a Continuous Beam

Multiframe allows you to quickly generate a continuous beam, which is made up of a number of even or varying length spans.

To generate a continuous beam

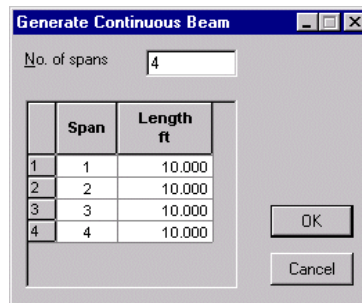
- Choose Generate... from the Frame menu

A dialog box will appear with the various generation icons in it.

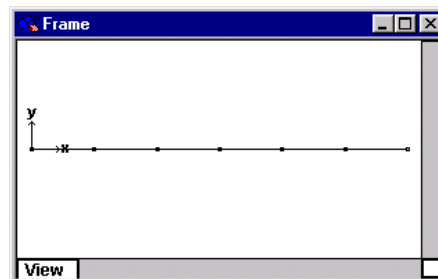


- Click on the continuous beam icon and click OK

A dialog box will appear with a table in it.



- Type the number of spans in the beam
- Click on the first length value in the table to select it
- Enter the lengths for each of the spans using the Down Arrow key to move down the table
- Click the OK button



Multiframe will automatically generate the beam with the dimensions you have specified.

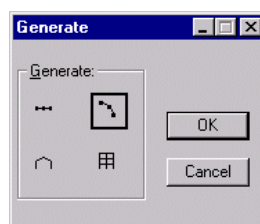
Generating a Curved Member

Multiframe allows you to quickly generate an approximation to a curved member by generating a number of short straight members.

To generate a curved member

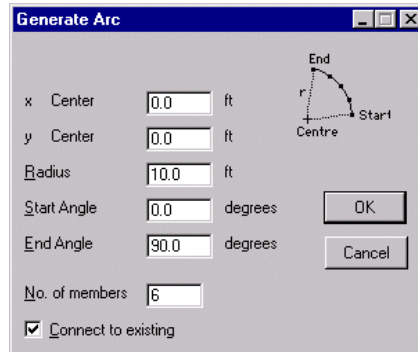
- Choose Generate... from the Frame menu

A dialog box will appear with the various generation icons in it.

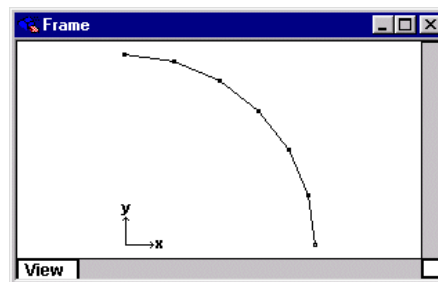


- Click on the curved beam icon and click OK

A dialog box will appear allowing you to enter the angles and radius of the beam.



- Type the coordinates of the center of the arc, radius and angle of sweep using the Tab key to move from number to number
- Type in the number of members you would like to use to approximate the curve
- Click the OK button



Multiframe will automatically generate the assembly of members with the dimensions you have specified. The direction of the arc will depend on the current view in the Frame window. The arc will always be generated about the axis, which is perpendicular to the screen or in the case of the 3D view, about the axis, which is most perpendicular to the direction of view. This axis is not drawn in bold on the axis indicator.

Normally, you will want the new members to connect with any existing members in the structure. If you do not want this to happen, uncheck the check box at the bottom of the dialog.

It is probably a good idea for you to test the accuracy of this approximation to a curve, by generating arcs with different numbers of members and examining the effect this has on the results of analysis.

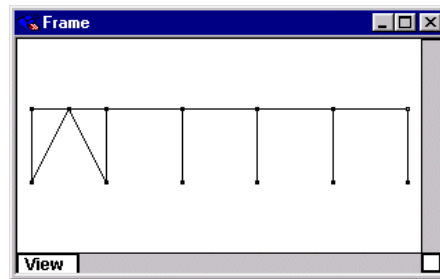
Generating a Regular Frame

Multiframe allows you to quickly generate a regular frame, which is made up of a number of evenly spaced, similarly shaped sub-structures. Typical examples of this would be high-rise buildings, trusses or multiple bay portal frames. Multiframe also allows you to duplicate shapes in cylindrical and spherical coordinates.

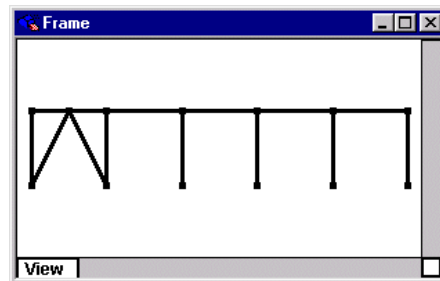
Duplicate

To duplicate a structure

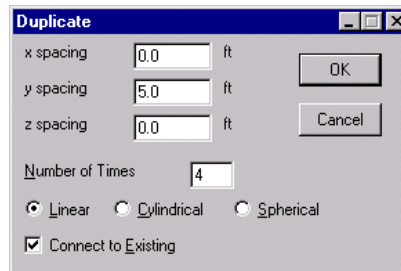
- Draw or generate the sub-structure you wish to duplicate



- Select the sub-structure to be duplicated

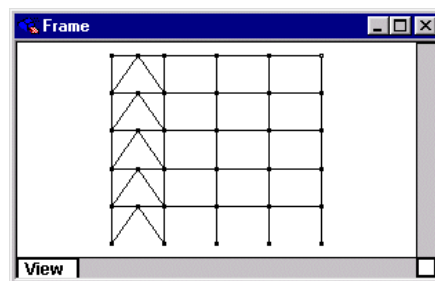


- Choose Duplicate... from the Frame menu



A dialog box will appear allowing you to specify the type of duplication, the spacing in each direction, the number of times the selection is to be duplicated and whether the duplicated members should be connected with the existing structure.

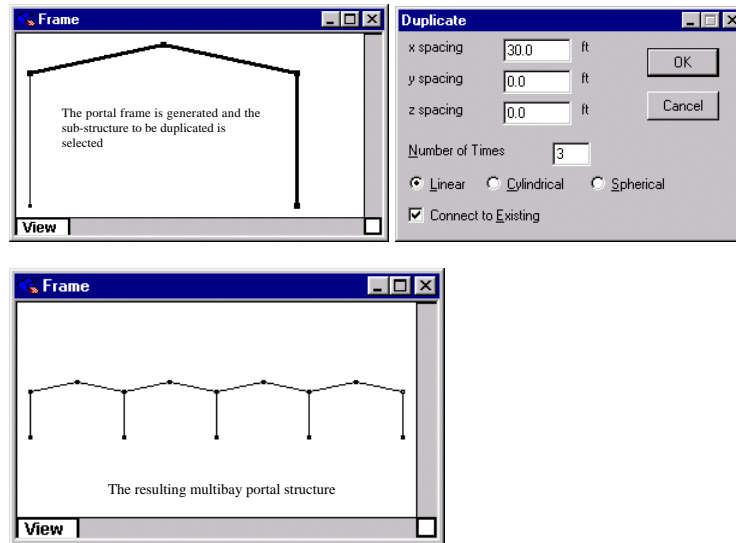
- Choose which type of duplication you require
- Enter the spacing for your sub-structure
- Enter the number of times you wish to repeat the sub-structure
- Click the OK Button



If you do not want connections to be generated between adjacent members, then switch off the check box 'connect to existing'. Normally you will want this option left on. A connection will be

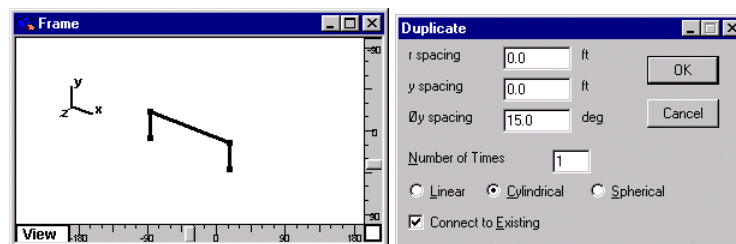
made between generated members and existing members if the end joints are within 0.2 inches (5 mm) of each other.

An example of the use of the Duplicate command to generate a multiple bay portal frame is shown below.

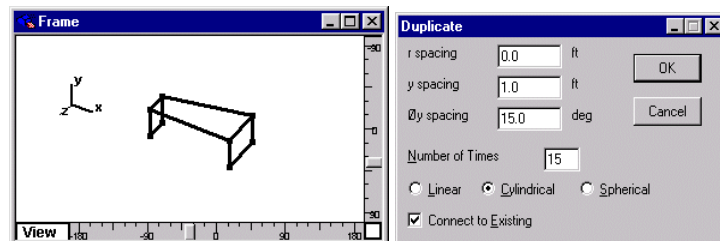


Cylindrical Coordinates

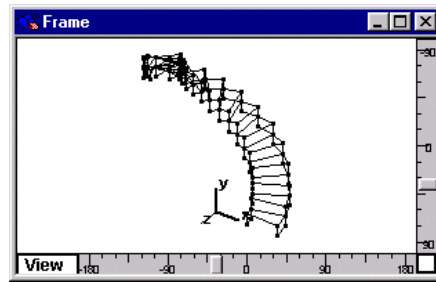
If you wish to generate a structure in cylindrical coordinates, you can select the cylindrical option. This then allows you to space the duplication radially from the y axis, angularly about the y axis, or linearly along the y axis. As an example, to generate a circular staircase you would first draw half of one step and then duplicate it to form the other half.



This would form one step of the staircase.



Then you would select the whole step and duplicate it to form the entire staircase.



Spherical Coordinates

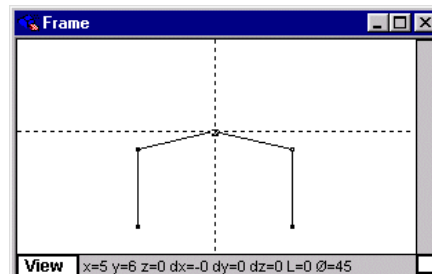
If you wish to generate a structure in spherical coordinates, you can select the spherical option. This then allows you to space the duplication radially from the origin, angularly about the z axis, and angularly about the y axis. As an example, to generate a cylindrical vault you could first draw a continuous beam parallel to the z axis and then duplicate it angularly about z to form the vault.

Moving a Joint

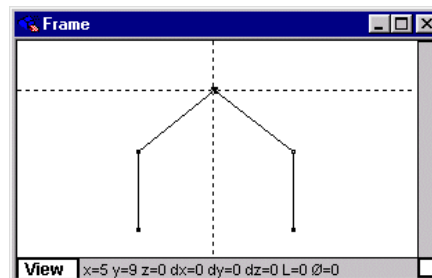
You can only move joints in the Frame window. Before moving a joint, be sure that there are no joints selected in the frame.

To move a joint

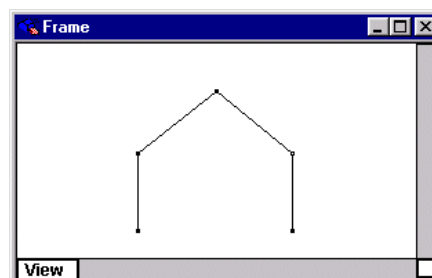
- Point to the joint, press the mouse button and hold it down



- Drag to the new position of the joint



- Release the mouse button to fix the new location



Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the joint will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the joint will align with the grid as you move it.

If you drag a joint on top of an existing joint, Multiframe will create a connection between the members, which meet at the common point.

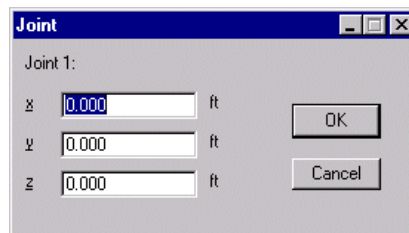
You can only drag joints in two dimensional views. When you drag a joint, Multiframe will not change the depth of the joint.

Typing Joint Coordinates

You can also move a joint by typing in new coordinates for its position.

- **Double click on the joint**

A dialog box will appear with the joint's coordinates.



- **Type in the new values for the coordinates**
- **Click the OK button**

You can also change a joint's location by typing or pasting new values in the X, Y and Z columns of the Joint table in the Data window.

Moving a Group of Joints

Multiframe also allows you to move a group of joints together either by dragging with the mouse or by typing in a distance to move.

To move a group of joints with the mouse

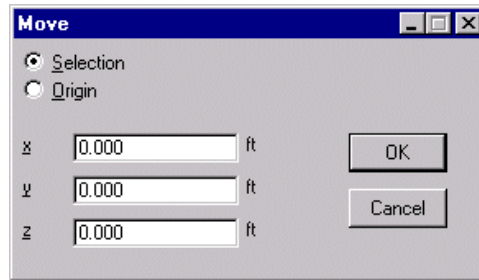
- **Select the joints to be moved**
- **Point to any of the selected joints and press the mouse button**
- **Drag the joints to the new location**
- **Release the mouse button**

Multiframe will connect together any members that share a common end point after you have dragged the joints. Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the pointer will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the joints will move in increments of the grid spacing as you move.

To move a group of joints numerically

- **Select the joints to be moved**
- **Choose Move... from the Frame menu**

A dialog box will appear with fields for the distance you wish to move the joints in each direction. The Selection radio button will be selected indicating that you wish to move the selected joints rather than the origin.



- Enter the distance the joints are to be moved in each direction
- Click on the OK button

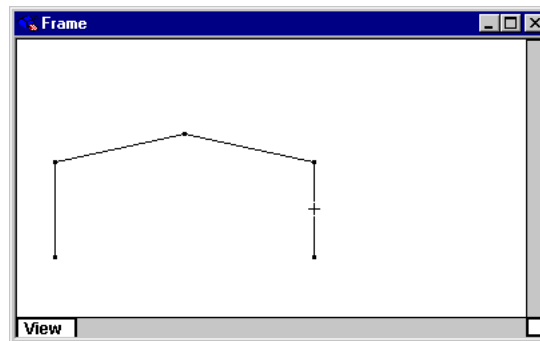
Multiframe will create a connection between joints that you superimpose on top of each other using this command. You can also move a single joint using this command. Simply select one joint before using the command.

Moving a Member

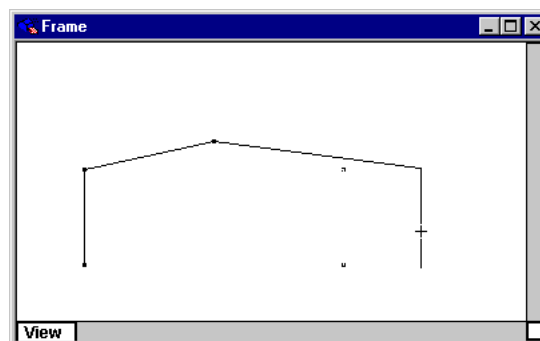
You can only move members in the Frame window. Before moving a member, make sure that there are no members selected in the frame.

To move a member

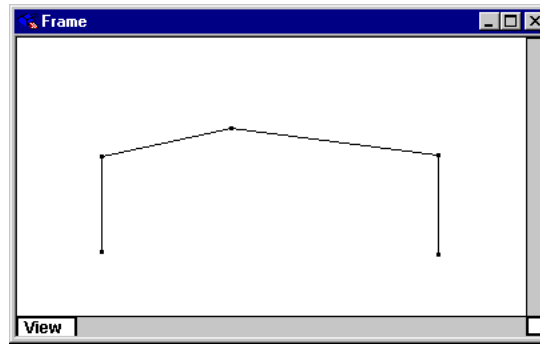
- Point to the member away from its ends and press the mouse button



- Drag the member to its new position



- Release the mouse button to fix the new position of the member



The slope and length of the member will remain constant as you move the member. You can only move members in the two dimensional views.

Holding down the shift key while dragging will constrain movement vertically, horizontally or to 45 degrees. The coordinates of the pointer will be displayed in the bottom left hand corner of the window as you drag. If you have turned on the Grid option, the member will move in increments of the grid spacing as you move.

If you drag a joint on top of an existing joint, Multiframe will create a connection between the members, which meet at the common point.

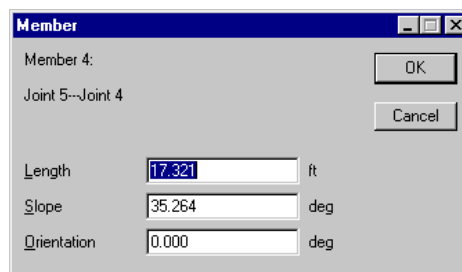
Resizing a Member

You can change the length or slope of a member in the Frame window by typing in new values.

To change the length or slope of a member

- **Double click on the member**

A dialog box will appear with the members name, end joint numbers, section type, length, slope and orientation.



- **Type in new values for the properties you wish to change**
- **Click the OK button**

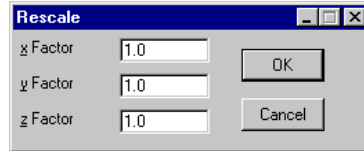
Joint 1 will stay at its original location, joint 2 will be moved to match the entered length and slope.

Rescaling the Structure

You can use the Rescale... command from the Frame menu to rescale the selected joints in the Frame window. This command is useful for investigating structural alternatives where you may wish to change the overall aspect ratio of a structure or part of a structure.

- **Select the part of the structure to be rescaled**
- **Choose Rescale... from the Frame menu**

A dialog box will appear with the scaling factors in each of the axis directions.



- Enter the scaling factors for the x, y and z directions
- Click the OK button

Rescaling is done relative to the origin of the axes, so you may wish to use the Move... command to move the origin prior to rescaling, to ensure that rescaling occurs from the correct point. Coordinates are rescaled by multiplying their value by the scaling factor. For example, a joint with coordinates $x=2$, $y=2$, $z=2$ and rescaled using scale factors of 1.5, 2.0 and 3.0 for the x, y and z directions respectively, would move to the new location of $x=3$, $y=4$, $z=6$.

Using a scaling factor of -1 will reflect the selected joints about the appropriate axis.

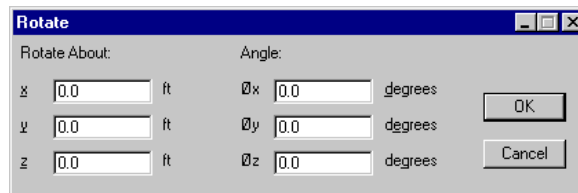
Multiframe will not create a connection between joints that you superimpose on top of each other using this command.

Rotating Members

You can use the Rotate... command from the Frame menu to rotate the selected joints in the Frame window.

- Select the part of the structure to be rotated
- Choose Rotate... from the Frame menu

A dialog box will appear with the rotation centers and angles.



- Enter the coordinates of the axis you wish to rotate the selection about
- Enter the magnitude of the rotation for the appropriate axis
- Click the OK button

Rotating is done relative to the axis you specify with the coordinates. For example, suppose you wanted to rotate the whole frame by 30° about a line parallel to the z axis passing through the point $x=10$, $y=20$. You would first select the whole frame, and then you would enter $x=10$, $y=20$, $z=0$, $\text{Ø}x=0$, $\text{Ø}y=0$, $\text{Ø}z=30$ in the Rotate dialog.

Multiframe will not create a connection between joints that you superimpose on top of each other using this command.

When rotating the selected joints you should enter only one non-zero angle. That is, Multiframe can only do one rotation at a time.

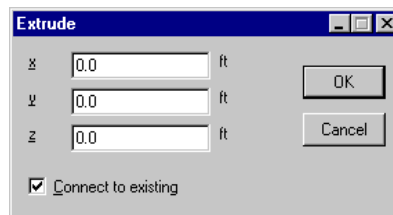
Extruding Beams or Columns

When you wish to add beams or columns to an existing frame, you may find it easier to use the Extrude command rather than drawing the new members with the mouse. The Extrude command allows you to project a member or group of members in any of the global axis directions from existing joints. For example, you could draw a floor plan and then extrude the columns up from it or you could extrude the beams out from a wall frame you have drawn.

To extrude members from the frame

- **Select the joints in the frame to be extruded**
- **Choose Extrude... from the Frame menu**

A dialog box will appear with fields for the direction and dimension of the extruded members.



- **Enter the length of the extruded members in the appropriate direction**
- **Click the OK button**

If you leave the Connect to existing check box checked, Multiframe will automatically create a connection between the selected joints and the extruded members. It will also connect the extruded members to any existing members in the frame if they lie within a 0.2 in (5mm) tolerance of the ends of the existing members.

Editing Coordinates Numerically

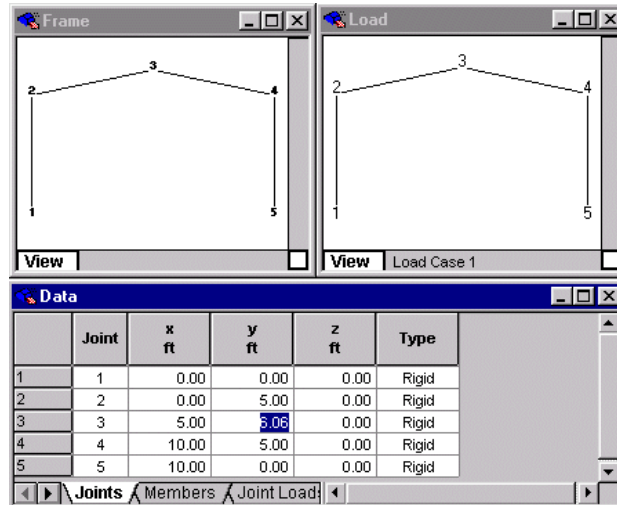
Multiframe allows you to change the positions of joints in the structure numerically rather than graphically. You will find this option useful for "fine tuning" the coordinates of your structure after drawing it.

Each joint in the structure is identified by a unique number. You can display these numbers in the Frame, Load and Plot windows by choosing Symbols... from the Display menu, checking the Joint Numbers check box and clicking the OK button.

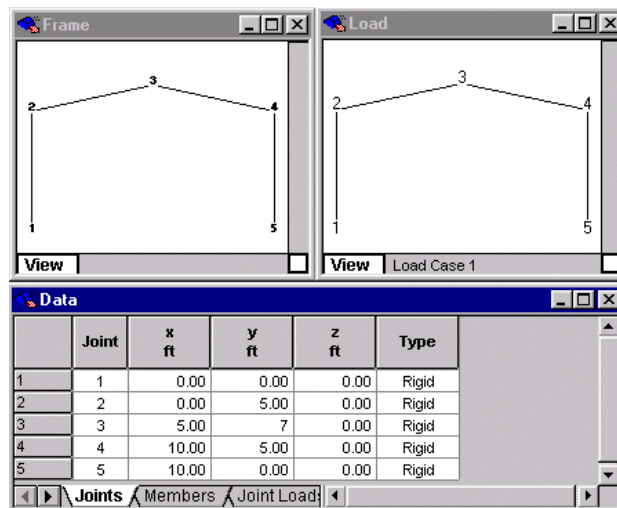
Coordinates are changed by typing values in the Data window or by double clicking on the joint in the Frame window. You will probably find it helpful to use the Editing Layout from the Window menu while you are typing in coordinates, as this will allow you to see the changes to the structure as you make them. The coordinates are displayed in a table in the Data window. You can resize the columns in the table by dragging the lines, which separate the columns. You can change the text font and size used in the table by using the Font... command from the View Menu while the Data window is frontmost on the screen. You can control the format of the numbers displayed in the table by using the Numbers... command from the Format window while the Data window is frontmost.

To change a coordinate of a joint

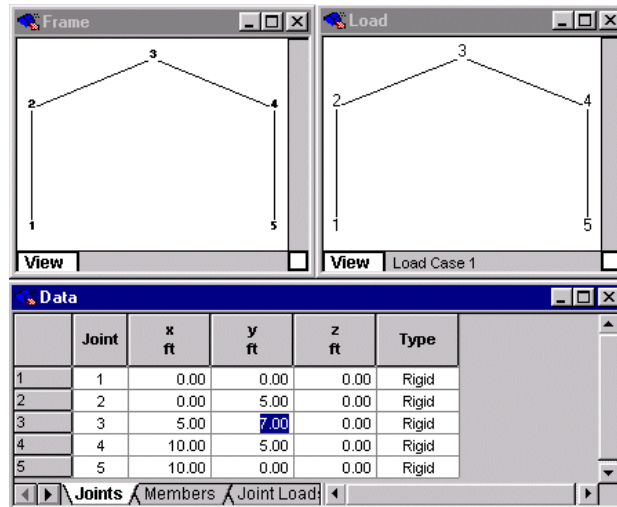
- Make sure the Data window is frontmost
- Click on the coordinate to be changed



- Type in the new value for the coordinate



- Press the Enter key on a Macintosh, or click inside the Frame or Load window on Windows



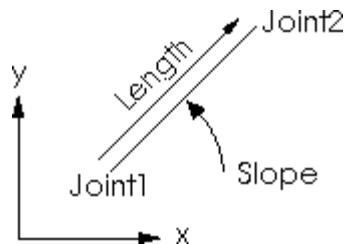
The structure will be re-drawn in the Frame and Load windows to reflect the changes you have made. You can repeat this process for any of the other joints in the structure. You can use the Tab Return keys or the arrow keys to move from number to number in the table or you may simply click on the number you wish to change.

Note that changing the coordinates of a joint so that it is the same as another joint in the structure will not create a connection between the members at that joint. A connection can only be made by dragging one joint onto the other in the Frame window.

If you double click on a joint in the Frame window, Multiframe will display a dialog, which allows you to change the coordinates of the joint.

Changing the position of a joint will change the length and/or slope of any members, which are connected, to it. The length and slope of the members in the structure may also be changed directly. The Member lengths and slopes may be displayed by choosing the Members command in the Data sub-menu under the Display menu.

Simply click on the numbers to be changed and type in the new values. The drawing in the Frame and Load windows will be updated to reflect your changes. Remember that the slope of a member is measured in degrees with positive angles being measured from a zero angle on the horizontal plane passing through the lower joint. Joint 1 will be the leftmost joint in the case of a horizontal or sloping member as viewed in the front or right side views or the bottom-most joint in the case of a vertical member.



You can also change the length and slope of a member by double clicking on it in the Frame window and typing in the new values in the dialog which appears.

If you change the length of a member, Joint 1 will be held fixed and Joint 2 will be moved to give the member the required length. The

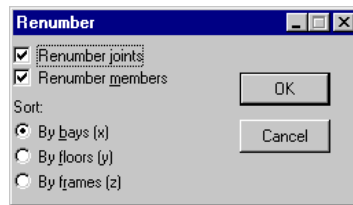
slope will not be changed. Similarly, changing the slope of a member will move Joint 2 and will leave Joint 1 in the same position while leaving the length of the member unchanged.

Joint and Member Numbers

As you add members and joints to a structure, Multiframe will automatically assign numbers to them. If you wish to renumber the joints and or members, you can do so using the Renumber item from the Frame menu.

To renumber the structure

- **Choose Renumber from the Frame menu**



You have three options for the directions, in which the renumbering will be done,

by bays	left to right,
by stories	bottom to top,
or by frames	front to back.

Note that you can renumber after analysis and your results will be preserved but displayed using the new joint and member numbers.

You can display the joint and member numbers on the members in the drawing windows by using the Symbol command from the Display menu and turning on Joint Numbers and/or Member Numbers.

Restraints

A structure must be sufficiently restrained for analysis to be carried out. A minimum requirement is that the structure be restrained against movement in each of the x, y and z directions and that it should not have any mechanisms caused by collapsible pinned structures.

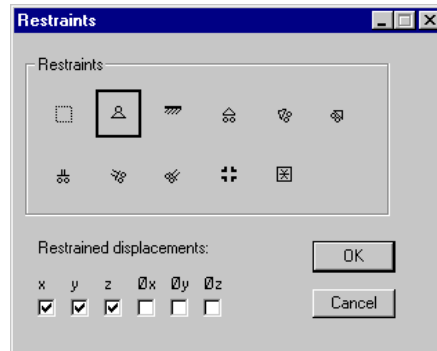
The restraints on the structure are specified by first selecting the joint or joints to be restrained and then selecting the appropriate type of restraint. The restraint may take the form of a zero displacement restraint, a spring or a prescribed displacement. A joint may have either a restraint, or a prescribed displacement or a spring for each degree of freedom but not combinations of these restraints for any degree of freedom. Thus a joint could have a vertical prescribed displacement and a horizontal spring but not a horizontal prescribed displacement and a horizontal spring or a vertical prescribed displacement and a vertical restraint.

Zero Displacement Restraint

To prescribe a restraint at a joint

- **Select the joint or joints to be restrained**
- **Choose Joint Restraint from the Frame menu**

A dialog box will appear with a range of icons for the various types of restraint



- Click on the appropriate restraint icon
- Click on the OK button

Selecting the "no restraint" icon (the first icon in the list) will remove all joint restraints from the selected joints. Selecting any other icon will remove any prescribed displacements or springs from the selected joints and replace them with the selected restraint.

Each of the possible restraints is indicated by an icon. As you click on the icons, the check boxes at the bottom of the dialog will display the degrees of freedom, which will be restrained if you select that icon.

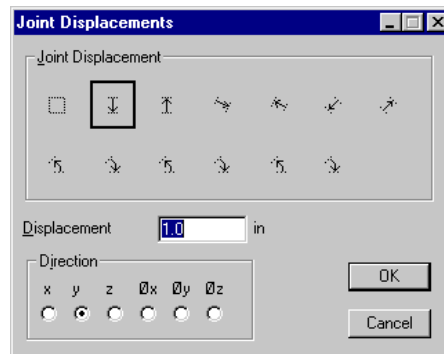
You can also click on the check boxes directly to choose which degrees of freedom you wish to restrain. If the combination of restraints you select is not one of the standard icons, Multiframe will display the special restraint icon (the last icon in the list).

Prescribed Displacement

To prescribe a displacement at a joint

- Select the joint or joints to undergo the prescribed displacement
- Choose Joint Displacement from the Frame 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. Since a prescribed displacement is equivalent to a restraint with a non-zero value, selecting any other icon will remove any restraints or springs which share the same degree of freedom from the selected joints and replace them with the prescribed displacement.

Prescribed displacements act in the global coordinate system of the structure. 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.

Note that a prescribed joint displacement affects the geometry of the structure and therefore applies to all load cases.

Springs

To attach a spring support to a joint

- **Select the joint or joints supported by springs**
- **Choose Joint Spring from the Frame menu**

A dialog box will appear with icons for the spring directions.



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

Selecting the "No spring" icon (the first icon in the list) will remove all springs from the selected joints. Selecting any other icon will remove any restraints or prescribed displacements with the same degree of freedom (if any) from the selected joints and replace them with springs. The rotational spring icons each have an axis drawn through them in the 3D view, which indicates the axis about which they provide rotational stiffness.

Editing Restraints Numerically

You can display and edit tables of springs and restraints in the Data window. You can use the commands from the Data sub-menu under the Display menu to control which table is on display at any time.

Restraints such as pinned or fixed joints will have restraint values of zero while prescribed displacements such as a settlement condition will have non-zero displacement values.

To change the value of a prescribed displacement

- Click on the number you wish to change
- Type in the new value
- Press the enter key

You can edit spring supports in exactly the same way as prescribed displacements. You can edit the stiffness coefficients of the springs by clicking and typing in new values.

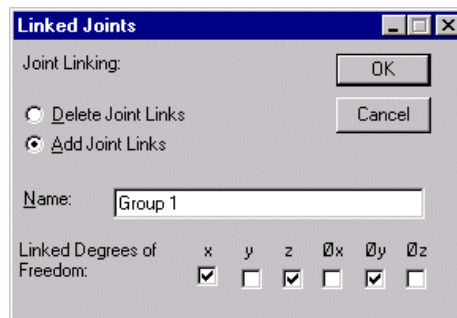
While editing numbers in the Data window you can use the scroll bars to scroll through the list of data, horizontally or vertically, to find the number you want to alter. If you wish, you can use the Return or Enter, Tab or arrow keys to move to another number rather than pressing enter. You can resize the columns in the table by dragging the lines, which separate the column titles. You can change the text font and size used in the table by using the Font... command from the View Menu while the Data window is in front on the screen. The format of the numbers displayed in the table can be controlled by using the Numbers... command from the Format window while the Data window is in front.

Linking Joints Or Master-Slave

Multiframe allows you to link groups of joints together so that they move together in response to either static or dynamic loads. This has a twofold benefit. Firstly, it allows you to simulate rigid structural elements such as a floor slab, and secondly it significantly reduces the size of the stiffness matrix resulting in lower memory requirements and faster analysis times.

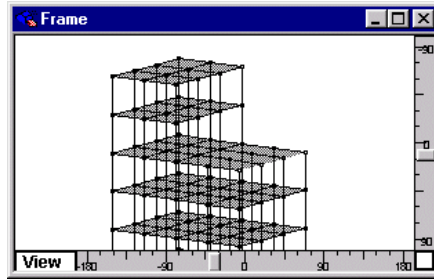
Joints can be linked together on a degree-of-freedom basis. This means some degrees-of-freedom can be linked together and others left independent. In the case of a rigid floor slab for example, all the joints on that level could have their x, z and $\emptyset y$ deflections linked together to simulate the rigid translation and rotation of the slab while still allowing bending of the floor.

- Select the joints to be linked in the Frame window
- Choose Joints Links from the frame menu



- Type in a name for the group and select the degrees of freedom to be linked

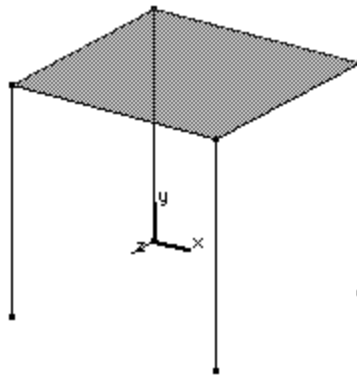
The linked group of joints is displayed as a shaded area in the Frame window.



The linked joints are also displayed in tabular form in the Data window. Choose Linked Joints from the Data sub-menu to display this table.

	Group	Joint	x	y	z	ϕ_x	ϕ_y	ϕ_z
1	Group 1	29	Linked	-	Linked	-	Linked	-
2		30	Linked	-	Linked	-	Linked	-
3		31	Linked	-	Linked	-	Linked	-
4		60	Linked	-	Linked	-	Linked	-
5		61	Linked	-	Linked	-	Linked	-
6		62	Linked	-	Linked	-	Linked	-
7		91	Linked	-	Linked	-	Linked	-
8		92	Linked	-	Linked	-	Linked	-
9		93	Linked	-	Linked	-	Linked	-
10		122	Linked	-	Linked	-	Linked	-
11		123	Linked	-	Linked	-	Linked	-

The linking is defined for the global XYZ directions. If you link two or more nodes together in the X direction then they will be locked together for any movement in the global X direction. Referring to the diagram below the two loads in the X direction will cancel each other out without inducing a moment about the Y axis. This is because the nodes at which the loads are applied are linked in the X direction prohibiting any rotation about the Y axis which would result in differential X axis movement.

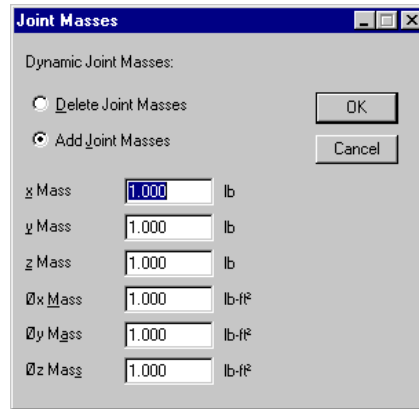


Joint Mass

If you are performing a dynamic analysis using Multiframe4D, you may wish to add additional masses to the structure to simulate the effects of equipment or construction loading which will affect the inertia of the frame. The Joint Mass command from the Frame menu allows you to add these additional masses at joints in the structure.

To add a mass at a joint or joints

- Select the joint or joints
- Choose Joint Mass from the Frame menu



- Enter the mass values to be associated with each global direction of movement

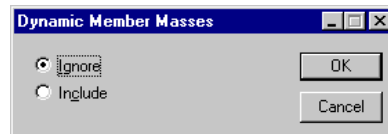
If you have already added masses that you wish to remove, you can click the Delete Joint Masses radio button to remove Joint Masses from the selected joints.

Member Masses

If you are performing a dynamic analysis using Multiframe4D, you may wish to include or ignore the masses of the members in the frame and the effect they will have on the inertia of the frame. The Member Mass command from the Frame menu allows you to ignore or include the effect of member mass in dynamic analysis.

To specify whether member mass is to be included

- Select the member or members
- Choose Member Masses from the Frame menu



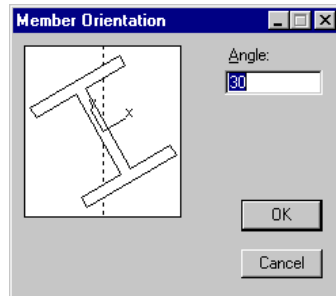
- Click Ignore or Include as appropriate

Section Orientation

When analysing a three dimensional frame it is necessary to know the orientation of the structural section used for each member relative to the global coordinate system. Initially, Multiframe assumes that the web or direction of principal strength of the section is aligned so that the principal direction lies in a vertical plane passing through the member. In the case of a vertical column, it is assumed that the principal direction passes through the global x axis. If you wish to have the section oriented at another angle (referred to by some engineers as the beta angle) you can use the Orientation... command from the Frame menu.

To change the orientation of a member or members

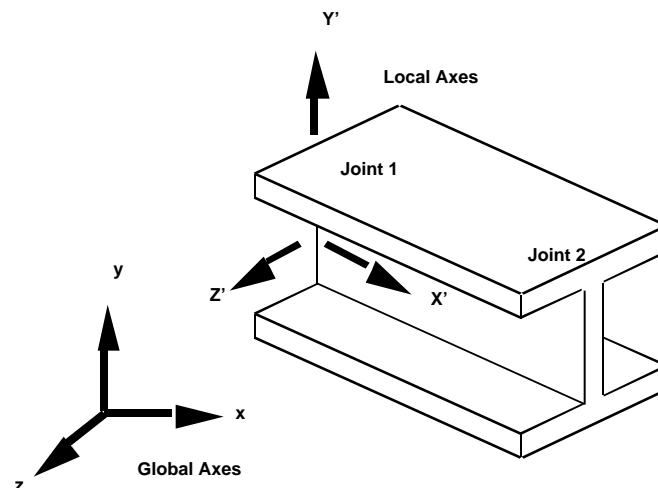
- Select the member or members to be specified
- Choose Orientation from the Frame menu



A dialog box will appear which displays the section's shape. The view of the section shape in the dialog is the view you would see if your eye was at joint 2 and you were looking down the member towards joint 1.

- Click and drag on the shape to rotate it to a new orientation or type in a new orientation angle
- Click OK to set the new orientation

When you set the orientation you are effectively defining the direction of the local y' axis of the member. The angle you type in is the angle between the y' axis and a vertical plane passing through both ends of the member. As the angle increases the y' axis will rotate towards the z' axis.



You can display the section axes on the members in the Frame window by using the Symbol command from the Display menu. You can also show the member axes in this way.

Section Properties

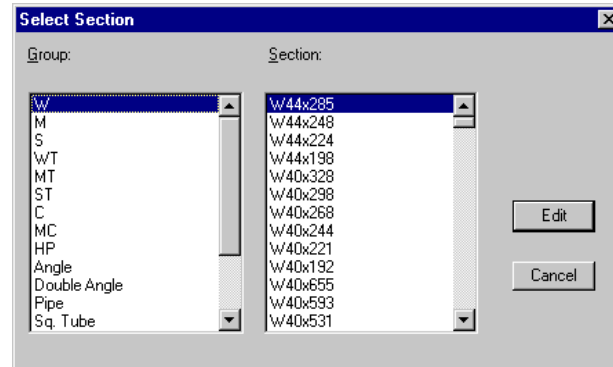
To compute the deflections in the structure it is necessary to know the geometric and material properties of the members, which make up the structure. Multiframe has a library of pre-defined section properties for the most commonly used steel sections. If you wish to use structural sections other than those pre-defined in the Sections Library, see "Adding a custom section" below.

Note that Multiframe uses the word "Member" to describe a geometric component of a structure such as a beam, column or strut and the word "Section" to describe a particular structural shape or section with its own material and geometric properties. Each member of the structure is constructed using a particular section.

To specify the section for a member

- **Select the member or members to be specified**
- **Choose Section Type from the Frame menu**

A dialog box will appear with a list of the groups and sections available in the Sections Library.



- **Click on the name of the group you wish to choose from**
- **Click on the name of the section you wish to choose**
- **Click the OK button**

You must specify the section type for every member in the structure before carrying out the analysis.

You can display the sectional shapes on the members in the Frame window by using the Symbol command from the Display menu and turning on Section Shapes.

Adding a Custom Section

If the structural section you require is not contained in the Sections Library, you can define your own section and store it in the library or store it with the structure. You may find it easier to use the Section Maker application to create and install your section however this section describes how you can add a section by simply typing in the key sectional properties.

The sections are arranged in the library in groups. Each of the groups usually consists of a range of sections of a similar type. This corresponds to the various tables of sections found in engineering handbooks. At the end of the list of groups are a number of groups labeled Custom1, Custom2, Custom3 and Frame. These groups are designed to be used to store special sections in.

To add a custom section

- **Choose Add Section from the Edit menu**

A dialog box will appear with a list of group names, a table of section properties, a field for the section's name and a list of icons to indicate the sections shape.

New Section

Section Name:

Group:

- Custom1
- Custom2
- Custom3
- Frame

Properties:

	Property	Value	Units
1	Weight	0.000	lb/ft
2	A	0.000	in^2
3	Ix	0.000	in^4
4	Iy	0.000	in^4
5	J	0.000	in^4
6	E	0.000	ksi
7	G	0.000	ksi
8	D	0.000	in

Shape:

☐ I ☐ L ☐ T ☐ □ ☐ ○ ☒ ?
☐ C ☐ L ☐ □ ☐ ○ ☐ □

OK Cancel

- Click on the name of the group you wish to store the member in
 - Type in a name for the section and press the Tab key
 - Fill in the fields in the table with appropriate values using the Tab key to move from field to field
 - Click on the icon which indicates the shape of the custom section
- If your section does not have one of the shapes shown in the list, leave the question mark icon selected.
- Click on the OK button

Note that it is not necessary to fill in all the fields if you do not wish to use the variables later in the CalcSheet. However, the following fields must be filled in:

A	Cross sectional area
Ix	moment of inertia about the major(x-x) axis
Iy	moment of inertia about the minor (y-y) axis
J	Torsion constant
E	Young's Modulus
G	Shear Modulus

Use the units indicated for each field.

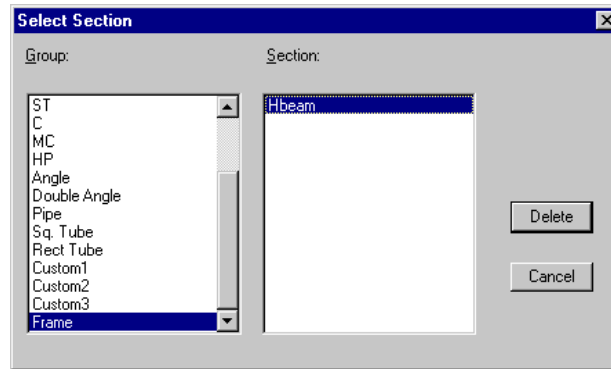
Only the groups in the library, which are not locked, will be shown in the list of groups. If you wish to store the section in the library and have it available for use in other structures, store the section in one of the groups other than the group named Frame. Sections stored in the Frame group will be stored with the structure and will not appear in the list unless you are using this structure. You will probably find it convenient to store most of your sections in the Custom group to avoid cluttering up your library with sections, which are only used in one or two structures.

Removing a Custom Section

To remove a custom section from the library

- Choose Delete Section from the Edit menu

A dialog box will appear with a list of groups and sections.



- Click on the group and section to be deleted
- Click on the Delete button

Editing a Custom Section

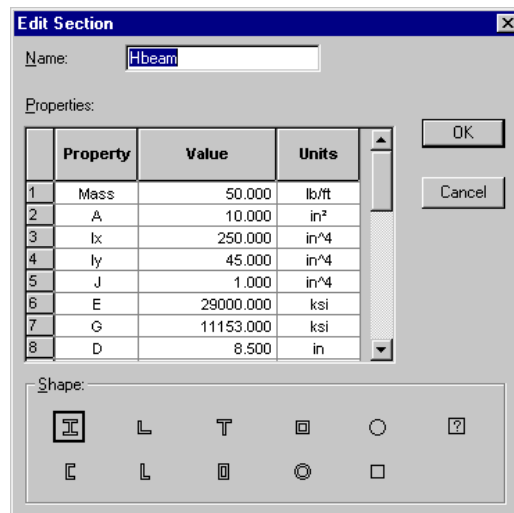
To change the section properties of a custom section

- Choose Edit Section from the Edit menu

A dialog box will appear with a list of groups and sections.

- Click on the group and section to be edited
- Click on the Edit button

A dialog box will appear with the data for the section shown in a table.



- Enter the information you wish to change using the Tab key to move from field to field
- Click on the OK button

You can also use this procedure to look at the actual values stored in the library for any section without changing them. Simply click the Cancel button after viewing the properties of interest to you.

If the OK button is drawn in gray this indicates that the group this section is stored in is locked and the section properties cannot be changed.

Section Types

You can review the sections used in a structure by displaying the Sections Table in the Data window. Choose Sections from the Data sub-menu under the Display menu to display this table. The table of sections displays details all of the sections used in the frame. It contains information about the sectional properties as well as the dimensional information about the sections. This is particularly useful when copying information to a spreadsheet for further processing (see Appendix F for more information).

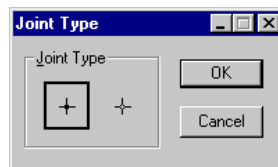
Joint Types

Multiframe allows you to use two basic types of joints. Either joints, which are completely "rigid", i.e. they can transmit moments, or joints, which are "pinned" i.e. they cannot transmit any moment.

To set the joint type

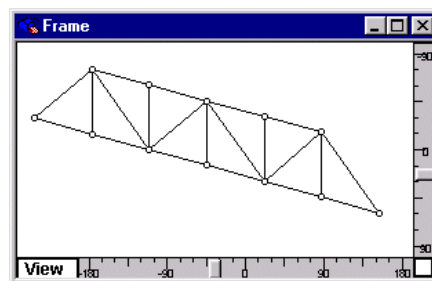
- **Select the joint or joints to be pinned**
- **Choose Joint Type from the Frame menu**

A dialog box will appear with icons indicating the two types of joints.



- **Click on the icon which represents the type of joint you require**
- **Click on the OK button**

The joints in the Frame window which are now pinned will be drawn with a circle on top of the joint to indicate that it is pinned.



Pinned Joints are most useful in situations where all or most of the joints in a structure cannot support moment transfer. If you wish to model a situation where some of the members connected at a joint can transmit moment and some cannot, then you should use the Member Type command explained below.

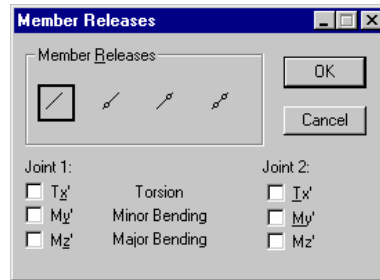
Member Types

Multiframe allows you to use four basic types of members. These are members, which are pinned or rigid at one or both ends. If "rigid", they can transmit moments at the end, or if "pinned" they cannot transmit moments (sometimes referred to as member releases).

To set the member type

- **Select the member or members to be pinned**
- **Choose Member Type from the Frame menu**

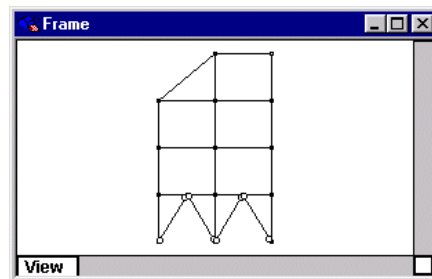
A dialog box will appear with icons indicating the four types of members.



- Click on the icon, which represents the type of member, you require
- Click on the OK button

When you specify the type of a member you have control over which degrees of freedom are released at the pinned ends of a member. You can release any of the three rotations at each end of a member. This helps prevent the problem of torsional mechanisms being created when a group of connected members all have the $\theta x'$ rotation released. To help prevent this, the default settings for a pinned end only release the major and minor rotations ($\theta y'$ and $\theta z'$)

The members in the Frame window with pinned ends will be drawn with a circle near the ends, which are pinned. Although the pins are shown to be a small distance from the end of the member in the Frame window, for the purposes of analysis they are in fact infinitely close to the ends of the member. A pinned end releases all the bending and torsional moments about each of the local member axes.

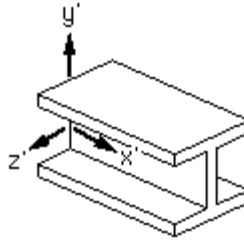


Note that the assumed member type is initially a rigid/rigid member and it is not necessary to explicitly define any member to be of this type, unless you have previously set the member type to rigid/pinned, pinned/rigid or pinned/pinned.

Shear Area

Multiframe allows shear area to be input and used in analysis. Using shear area can be switched on or off on a member by member basis.

To use shear area in a Multiframe analysis, a member must have a section type that has the shear area fields defined. A_{sx} and A_{sy} are the two fields in the Sections Library group that define the shear area for the sections local x and y axes, where x is the major axis. For all other sectional values, y is equivalent to local member axis y' and x is equivalent to local member axis z' in the negative direction.



The shear area values must be greater than zero, and the member must have the appropriate shear area flag set.

- **Choose Member Shear Area...** from the **Frame** menu



- **Select the local member axes for which shear area is to be used in analysis**

Remember A_{sx} corresponds to the z' axis and A_{sy} to the y' axis.

The members that use shear area in analysis are indicated in the last column in the Member Data sheet.

Applying Loads

Changes to the loads, which are applied to the structure, may be made in the Load window. Loads may be added to the structure or removed from it and can be applied as a number of different load cases. These load cases may be factored together to produce combined load cases if desired.

Loads are entered in the current load units as controlled by the Units... command from the View Menu.

All of the commands under the Load menu operate on the current load case. The name of the current load case is displayed in the bottom left hand corner of the Load window. Also, the name of the current load case has a check mark beside it in the Case menu. You can set the current load case by choosing the appropriate name from the list of load case names under the Case menu.

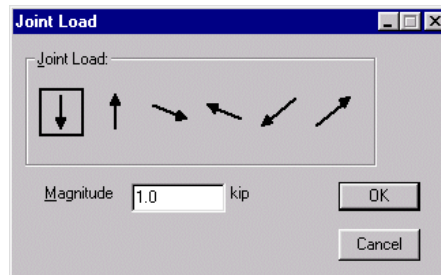
Joint Load

A joint load is a load, which acts at a joint in the structure and acts in a direction parallel to one of the reference x, y or z axes.

To apply a load to a joint

- **Select the joint or joints to be loaded**
- **Choose Joint Load from the Load menu or the short cut menu**

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



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 global 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 magnitude of the load**
- **Click on the OK button**

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

If you wish to remove the joint loads from a joint, select the joint and choose Unload Joint from the Load menu.

You can superimpose loads in the same direction on a single joint. Each time you use the Joint Load command, the new loads will be superimposed on any other joint loads you may have previously applied.

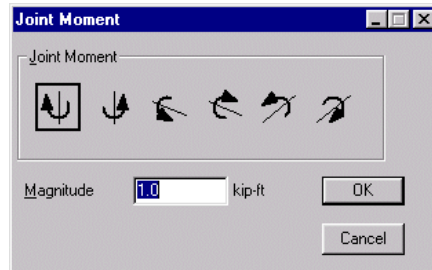
Joint Moment

A joint moment is a bending moment, which acts at a joint in the structure, and acts about one of the reference x, y or z axes.

To apply a moment to a joint

- **Select the joint or joints to be loaded**
- **Choose Joint Moment from the Load menu or the short cut menu**

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



In a two dimensional view, there will be four icons indicating the four possible moment directions. In the 3D view, all six possible icons will be displayed with the icons showing the direction of the action of the moments about the global axes in the current view. The line through the center of each icon indicates the direction of the axis the moment is acting about.

- **Click on the icon which shows the direction in which the moment is to act**
- **Type in a value for the magnitude of the moment**
- **Click on the OK button**

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

If you wish to remove the moments from a joint, select the joint and choose Unload Joint from the Load menu.

Global Distributed Load

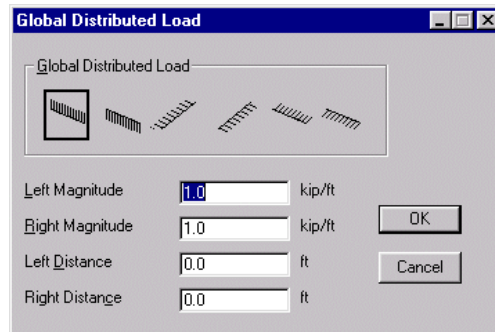
Multiframe allows loading on members to be applied relative to the direction of the global coordinate system or relative to the direction of the local member coordinate system. Loads, which are applied at an angle to either of these systems, can be modeled by using vector components of the loads.

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

To apply a global distributed load to a member

- **Select the member or members to be loaded**
- **Choose Global Dist'd Load from the Load menu or short cut menu**

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



In a two dimensional view, there will be four icons indicating the four possible loading directions. In the 3D view, all six possible icons will be displayed with the patterns in the icons showing the direction of the action of the loads. Each icon has a pattern with lines running parallel to the direction of action.

- Click on the icon which shows the direction in which the load is to act
- Type in values for the magnitude of the load at each end

The magnitude of a global distributed load refers to its load per length where the length is measured perpendicular to the direction of action of the load. This means a distributed load applied to an inclined member will apply a total load equivalent to the magnitude of the load times the horizontal projected length of the member.

- Click on the OK button

Note that when working in dialog boxes, you can use the Tab key to move from one editing box to the next. If you do this in this case, the right hand end magnitude will automatically be set to the same value as the left hand end magnitude. You may type over this second value if you wish to have a non-uniform load.

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

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third span you can enter $L/3$ for its start position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

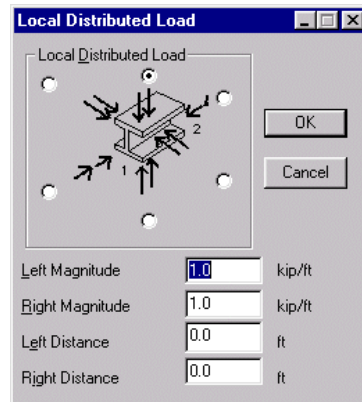
Local Distributed Load

A local distributed load is a load which is distributed along all or part of the member and acts in a direction either normal (shear) or tangential (axial) to the member.

To apply a local distributed load to a member

- Select the member or members to be loaded
- Choose Local Dist'd Load from the Load menu or short cut menu

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



- Click on the icon which shows the direction in which the load is to act
- Type in values for the magnitude of the load at each end
- Click on the OK button

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

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to start at one third span you can enter $L/3$ for its start position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

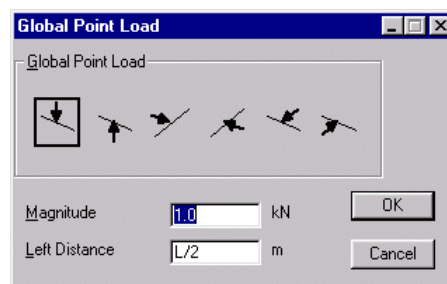
Global Point Load

A global point load is a concentrated load, which acts at a position part way along a member and acts in a direction parallel to one of the reference x, y or z axes.

To apply a global point load to a member

- Select the member or members to be loaded
- Choose Global Point Load from the Load menu or the short cut menu

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



In a two dimensional view, there will be four icons indicating the four possible loading directions. In the 3D view, all six possible icons will be displayed with the direction of the arrows in the icons showing the direction of the action of the loads as they will be displayed in the current view.

- Click on the icon which shows the direction in which the load is to act
- Type in the value for the magnitude of the load
- Press Tab and type in the position of the load measured from joint 1
- Click on the OK button

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

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter $L/2$ for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

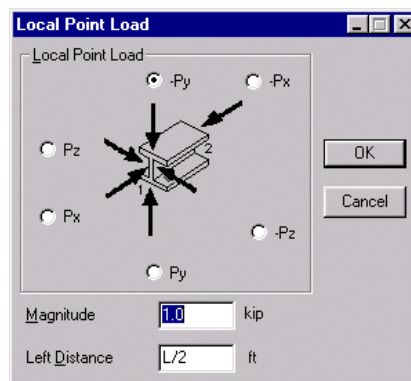
Local Point Load

A local point load is a concentrated load, which acts part way along a member and acts in a direction normal (shear) or parallel (axial) to the member.

To apply a local point load to a member

- Select the member or members to be loaded
- Choose Local Point Load from the Load menu or the short cut menu

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



- Click on the icon which shows the direction in which the load is to act
- Type in the value for the magnitude of the load

- Press Tab and type in the position of the load measured from joint 1
- Click on the OK button

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter $L/2$ for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

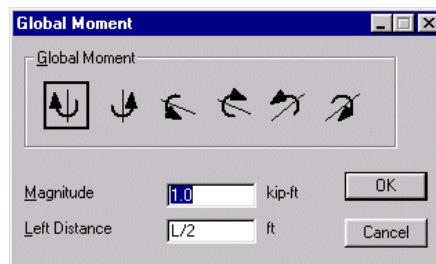
Global Moment

A global moment is a bending moment, which acts part way along a member and acts about one of the reference x, y or z axes.

To apply a global moment to a member

- Select the member or members to be loaded
- Choose Global Moment from the Load menu or the short cut menu

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



In a two dimensional view, there will be two icons indicating the two possible moments which can be applied. In the 3D view, all six possible icons will be displayed with the direction of the arrows in the icons showing the direction of the action of the loads, as they will be displayed in the current view.

- Click on the icon which shows the direction in which the moment is to act
- Type in the value for the magnitude of the moment
- Press Tab and type in the position of the load measured from joint 1
- Click on the OK button

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

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter $L/2$ for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different

lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

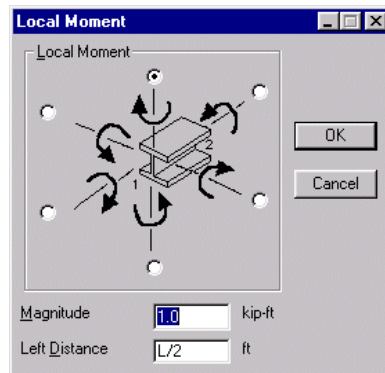
Local Moment

A local moment is a bending moment, which acts part way along a member and acts about one of the local axes.

To apply a local moment to a member

- **Select the member or members to be loaded**
- **Choose Local Moment from the Load menu or the short cut menu**

A dialog box will appear with icons to indicate the direction of the moments relative to the member.



- **Click on the icon which shows the direction in which the moment is to act**
- **Type in the value for the magnitude of the moment**
- **Press Tab and type in the position of the moment measured from joint 1**
- **Click on the OK button**

When you enter positions of loads in the member loading dialogs or the Data window, you can enter calculation expressions for the position. For example, if you want a load to be at mid span you can enter $L/2$ for its position. If you enter this in a load dialog, this expression will be calculated for all the selected members. This means you can apply this load to a number of members of different lengths simultaneously. You can also enter more complicated expressions such as $2*L/3$ or $1.35+(L-4)/2$. The variable L is always available and contains the length of the member, the syntax of the expressions is the same as that used in the CalcSheet.

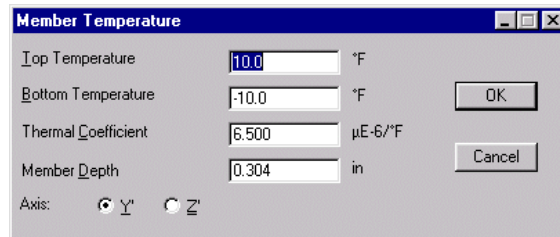
Thermal Load

A thermal load is a load resulting from a temperature differential in the structure between a member and the ambient temperature. A thermal load may also result from a temperature gradient through the depth of a member.

To apply a thermal load to a member

- **Select the member or members to be loaded**
- **Choose Thermal Load from the Load menu or the short cut menu**

A dialog box will appear with fields for the magnitude and depth of the thermal load.



The dialog box titled "Member Temperature" contains the following fields and controls:

- Top Temperature:** A text box containing "10.0" followed by a unit label "°F".
- Bottom Temperature:** A text box containing "-10.0" followed by a unit label "°F".
- Thermal Coefficient:** A text box containing "6.500" followed by a unit label "μE-6/°F".
- Member Depth:** A text box containing "0.304" followed by a unit label "in".
- Axis:** Two radio buttons labeled "Y'" and "Z'". The "Y'" button is selected.
- Buttons:** "OK" and "Cancel" buttons are located on the right side.

- Choose whether the temperature varies through the depth (web or y' axis direction) or the breadth (flange or z' axis direction) of the member

- Type in the temperatures at the top and bottom of the member

These two temperatures will be the same if the member is at a constant temperature. If the load is applied in the z' direction top refers to the positive z' side while bottom refers to the negative z' side.

- Type in a value for the thermal coefficient for the member material

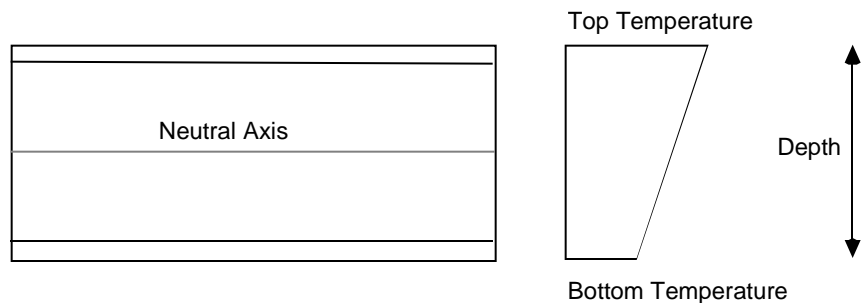
The thermal coefficient is entered in units of microstrain per degree.

- Type in a value for the depth of the thermal load

- Click on the OK button

All the temperature values are variations in degrees from the ambient temperature. For example, a bridge deck with the top of the beams at 45° and the bottom of the beams at an ambient of 20° would have top and bottom temperatures of 25° and 0° respectively.

The thermal gradient is assumed to be symmetric about the neutral axis of the member. That is, the top temperature occurs at half the depth above the neutral axis and varies linearly down to a point half the depth below the neutral axis.



Editing Loads Numerically

You can edit the values and positions of loads numerically in tables of data, which may be displayed in the Data window. These tables display Joint Loads, Member Loads and Thermal Loads. You can choose which table to display by choosing the appropriate item from the Data sub-menu under the Display menu.

To change the value or position of a load

- Click on the number to be changed

- **Type in the new value for the load**
- **Press the enter key**

If you change the position or direction of a load, the drawing in the Load window will be updated to reflect the change.

Note that the loads for the current load case only will be displayed in the Data window. You cannot change loads for a combined load case, a dynamic load case or a self weight load case.

Self Weight

Multiframe allows you to automatically include the self weight of the structure as a separate load case.

- **Choose Self Weight... from the Load menu or the short cut menu**

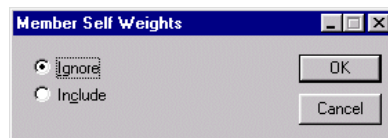
A new load case named Self Weight will be created. Only one self weight load case is allowed. You should make sure that you have set all of the section types for the structure before you use the self weight command. If any of the sections used in the structure have zero mass, Multiframe will prompt you with a dialog to go ahead with or cancel the use of self weight. If you wish to use self weight with custom sections, make sure you enter an appropriate value for the mass when adding the section to the library.

Member Self Weight

If you add a self weight case, you can specify whether or not a member's weight is to be included in the Self Weight by using the Member Self Weight command from the Load menu.

To specify whether a member's weight should be included in self weight

- **Select the member(s) to be changed in the Load window**
- **Choose Member Self Weight from the Load menu**



- **Click the Ignore or Include radio button as appropriate**
- **Click OK**

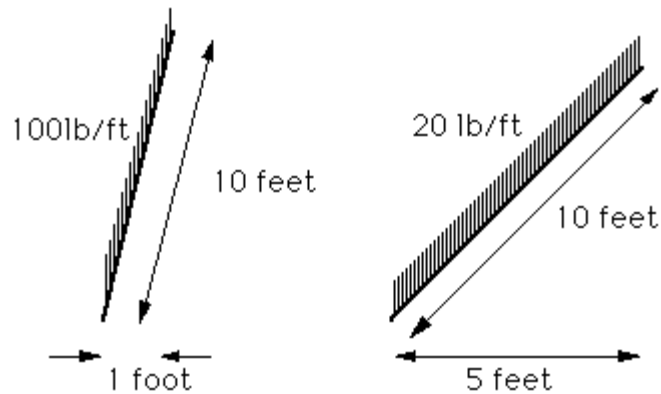
The Member Self Weight command only affects the self weight for static analysis. If you are doing a dynamic analysis in 4D, the member's weight will always be included in the mass matrix of the dynamic analysis.

A self weight load case can be combined with other load cases in the usual way. The numerical values of the self weight loads are not displayed in the Data window. These values will be updated just prior to analysis to ensure that the weight for any members you have changed are included correctly.

Self Weight Loads

The loads for self weight are included in two ways. Members which are vertical are loaded with an axially distributed load, Wz' , acting in the vertical direction, from joint 2 to joint 1. You will see values for this load in printouts or in factored load cases, which are made up from a self weight load case. Members, which are not vertical, are

loaded with a vertical distributed load scaled to take into account the horizontal projected length of the member (see Global Distributed Load above). Because global distributed loads are applied in units of force/horizontal length, the magnitude of the distributed load must be scaled to take into account.



For example, a 10ft long member weighing 100 pounds with a horizontal projected length of 1 foot would require a distributed load of 100lb/ft to apply the correct self weight. The same 10 foot long member with projected length of 5 feet would only require a distributed load of 20lb/ft to correctly apply the self weight. For this reason, if you have members, which are very nearly vertical and therefore have a very small projected length, you will see very high values for the distributed load applied due to self weight.

Multiple Load Cases

You can apply up to fifty load cases to a structure at a time. You may define some of these load cases to be factored combinations of existing load cases. If you make a change to a load case, which is used by other load cases, the factored load cases will automatically be updated to reflect the change. You can automatically construct a load case, which includes the self weight of the structure as outlined in the previous section.

Initially, one load case is defined titled 'Load Case 1'. You can edit one load case at one time. The current load case is indicated by a check mark on the appropriate item in the lower part of the Case menu and is shown in the lower left hand corner of the Load window and the Plot windows when two dimensional views are displayed. If the load case is a factored combination of other cases, it will have (Combined) displayed after the name of the case in the Load window. You cannot edit or remove loads from a factored load case. If you change loads in a load case, which is used by a combined load case, the combined case will be automatically updated.

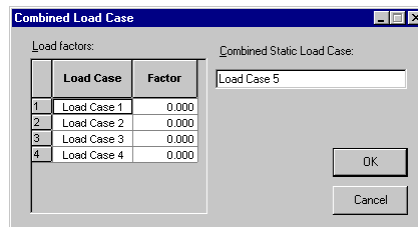
Factored load cases are identified in the Case menu by being underlined.

Note that a factored load case can be a factored combination of a factored load case, but cannot include a factored combination of a load case higher in the load case list. For example, load case 3 can include combinations of load cases 1 and 2 but could not include load case 4.

Adding a Load Case

To add a new load case

- Choose Add Case from the Case menu
- Select what type of load case you wish to add from the Add Case sub-menu
- Type in a name for the load case (or leave the default name if you wish)
- If you wish to create a factored load case (Combined Load Case) select Static Combined from the Add Surface sub-menu



- Enter the load factors for the existing load cases
- Click on the OK button

The name you give to the load case must not contain any of the following characters; ^ ! < / (

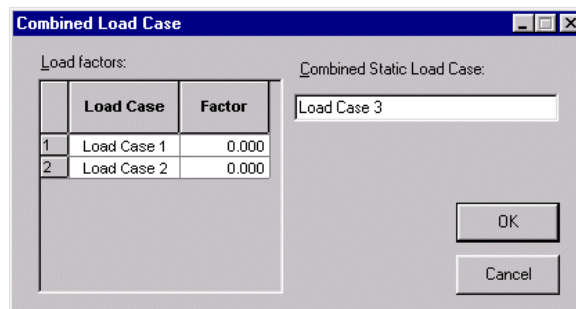
If you turn on the "Duplicate current load case" check box, the new load case will initially contain the loads in the current load case. This is useful if you have two load cases, which are very similar, and one can be made from a small variation on the other.

Editing a Load Case

To change the name or load factors of a load case

- Choose Edit Case from the Case menu

If the current load case is a factored load case, the following dialog box will appear.



If the current load case is not a factored load case, the following dialog box will appear.



- Type in a new name for the load case if required

- If the current load case is a factored case then type in the new load factors if desired
- Click on the OK button

The name you give to the load case must not contain any of the following characters ; ^ ! < / (.

If you use the Edit Case command to change the name of a load case after analysis, your results will not be lost if you only change the name and not any load factors.

Editing Load Cases Numerically

You can edit the names and factors of load cases numerically in the Load Case table, which may be displayed in the Data window. This table displays load case names and load case factors.

To change a load case name or factor

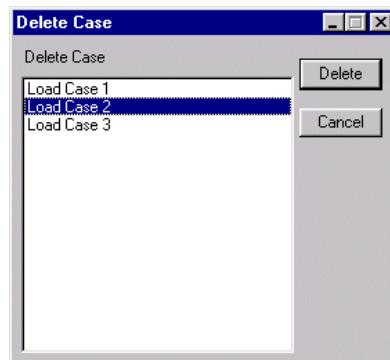
- Click on the value to be changed
- Type in the new value
- Press the enter key

Deleting a Load Case

You can remove a load case from the structure. This will remove the case and all its loads. To delete a load case

- Choose Delete Case from the Case menu

A dialog box will appear with the current load cases in a list



- Select the load case you wish to remove
- Click the Delete button

You select a load case in the list by clicking on its name. If you wish to remove more than one load case, hold down the shift key while clicking on the names. Holding down the shift key while clicking in the list adds the name to the selection or removes it if it is already selected.

Load Library

Multiframe4D includes a Load Library, which contains commonly used dynamic loads. These loads may be spectra from earthquake measurements (several common earthquakes are provided) and may also be dynamically varying forces or accelerations which can be applied at joints in the structure.

When Multiframe4D starts up it will now look for a file called "Load Library" or "Load Library.llb" in the same way that Multiframe looks for "Sections Library", first looking in the same directory as Multiframe4D, then looking on the current volume. Then if the Load Library is not found, Multiframe4D will prompt you for the current load library.

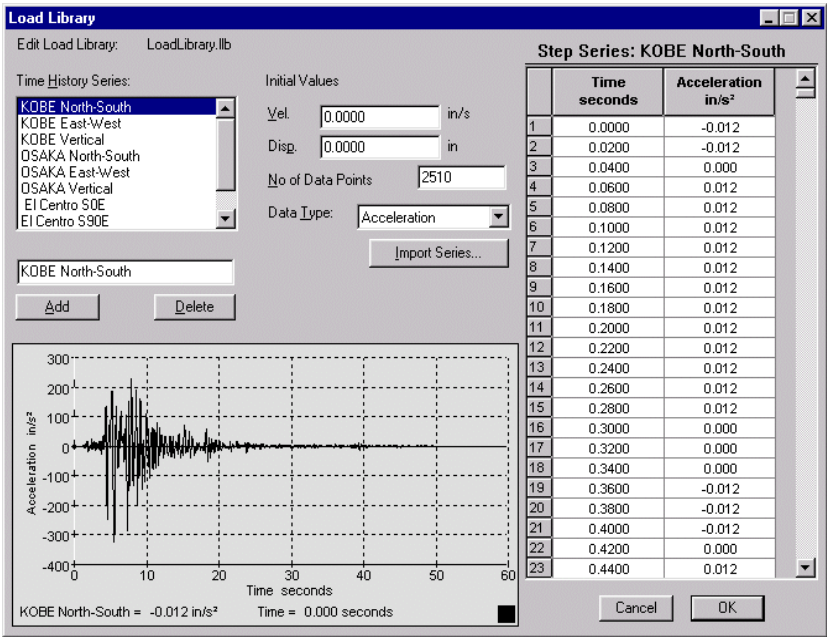
The Load Library shipping with Multiframe4D contains 9 earthquake spectra taken from 3 well known earthquakes. This includes accelerations in 3 orthogonal directions for each earthquake. The earthquakes are El Centro, Kobe and Osaka.

Editing the Load Library

All dynamic loads are stored in the Load Library as either a force series or acceleration series. You can add, edit or delete loads in the Load Library.

- **Choose Edit Load Library... from the Edit menu**

The Load Library dialog will appear which displays the load and acceleration series currently stored in the library.



The list at the top left of the dialog lists all of the series stored in the Load Library. The name of the currently selected series is displayed below the list and may be edited there.

Each series has a number of attributes including start velocity and displacement, number of points and type of series (either force or acceleration).

To add a new series to the library

- **Click the Add button**
- **Type in a new name for the series**

Enter the attributes of the series and type the values for the series data in the table at the right hand side of the dialog. You can also copy and paste data into the table by selecting the cells to be pasted and using Paste in the Edit menu.

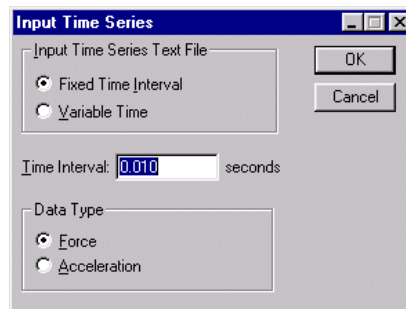
Remember that the final time value must be greater than any analysis time. Otherwise, the loading would be undefined. For a set load, the time could be any large number, as the load value does not vary.

Importing Load Data

You can import load data from a text file to the load library in addition to entering it by hand. To import load data

- **Choose Import... from the File menu**

A dialog will appear allowing you to choose the format of the data in the text file.



If you select Variable Time, the data should be in tab delimited format with two values per data point, the first value being the time in seconds and the second value being the load value (force or acceleration as appropriate).

If you select Fixed Time Interval, the data should be in tab delimited format with one force or acceleration value per data point.

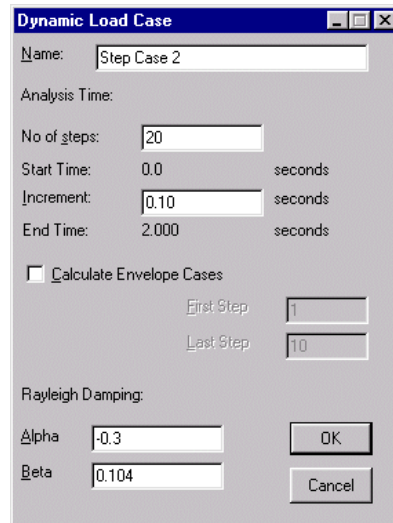
Adding Dynamic Load Cases

There are a number of different load cases, which can be added in Multiframe. These include a self weight static load case, a normal static load case, a factored combination of static load cases, a dynamic load case which can contain dynamically varying forces at one or more joints, and a seismic case which applies up to three orthogonal ground accelerations to any restrained joints.

To create a Dynamic Load Case

- **Select Dynamic... from the Add Load Case sub-menu**

A dialog will appear allowing you to enter the name, number and duration of the time steps for this load case along with other relevant information.

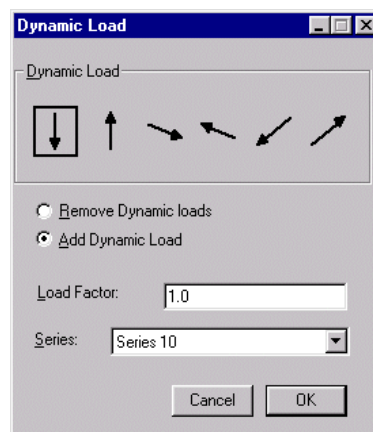


Applying Dynamic Loads

To add a dynamic load to the current dynamic load case

- Go to the Load window
- Select the joints to which you want to add the load
- Choose Dynamic Load from the Load menu

A dialog will appear allowing you to specify the direction of the load, a factor which will be multiplied by the values in the load series, and a pop-up to select the load series from the library.



Note that only force series may be applied to joints in a dynamic load case.

- Enter a load factor, if any
- Select the series you want to add and the global direction in which it is to act

Viewing Applied Time History Loads

To view the loads applied above while still in the current Time History load case

- Choose Step Loads from the Data sub-menu under Display

Data

	Node	Direction	Factor	Time History Series	Series Type
1	96	x	-1.000	Step Load	Force
2	25	z	-1.000	Step Load	Force
3	21	y	1.000	Step Load	Force

Adding Seismic Load Cases

In addition to creating dynamic load cases, which allow you to apply dynamically-varying forces to the structure, Multiframe4D allows you to create load cases, which apply dynamically changing ground accelerations. The most common use for this is applying an earthquake spectrum to a frame, but it may also be used for any other ground-based acceleration. Analysis is carried out by applying the specified acceleration to all of the restrained joints in the structure. The assumption is that all restrained joints move with the ground acceleration. The results of the analysis are displayed relative to the ground position at each point in time. This means that the restrained joints all show zero deflection and the deflections of the frame are relative to the global axes as usual.

To create a Seismic Load Case

- **Select Seismic from the Add Load Case sub-menu under the Case menu**

A dialog will appear allowing you to enter the name and number and duration of time steps, acceleration series to be applied, and the direction of the acceleration.

- **Enter the name of the load case**
- **Enter the number of time steps to perform**
- **Enter the duration of each time step**

Multiframe will display the total duration of the analysis below these values.

- **Select the orientation of the axes along which the ground acceleration will be applied**

The normal convention for these axes is to label them **North**, **East** and **Up** for the three directions. You can specify the orientation of the North axis relative to the global x axis by entering an angle (counter-clockwise about the y axis) in degrees from the x axis. You can also reverse the direction of any of the load axes by choosing the appropriate radio button beside the axis name. The drawing in the bottom right hand corner of the dialog will display your selection.

Finally, you can choose which acceleration series from the Load Library will be applied to which axes by choosing the appropriate names from the pop-up menus.

Choosing the Time Step

The time step you choose for a time history analysis (Δt) has the effect of including or excluding the effects of natural frequencies modes in response of the structure. As Δt increases more modes are effectively excluded from the response behavior as the time step. As a general rule if T is the period of the highest numbered mode that is to be included in the response calculation you should choose Δt such that

$$\frac{\Delta t}{T} < 0.1$$

Rayleigh Damping Factors

When adding either a dynamic or seismic case the user can specify if proportional damping is to be used in the analysis. To do so you need to specify non-zero values for the Rayleigh damping coefficients¹ Alpha and Beta.

Proportional damping assumes that the damping matrix is proportional to the mass and stiffness matrices M and K respectively.

$$C = \alpha M + \beta K$$

To calculate these values you need to look at the first two modes and the corresponding critical damping values required at those modes. Then using the equations

$$\begin{aligned} + \frac{2}{\omega_1} &= 2 \zeta_1 \omega_1 \\ + \frac{2}{\omega_2} &= 2 \zeta_2 \omega_2 \end{aligned}$$

where ω_1 and ω_2 are the approximate first and second natural frequencies of the frame and ζ_1 and ζ_2 are the critical damping values corresponding to these frequencies.

Combining the equations gives

¹ If the user is uncertain as to how to find the appropriate value for Alpha and Beta please refer to texts on the subject.

Finite Element Procedures in Engineering Analysis Bathe, Klaus-Jürgen © 1982 by Prentice Hall Inc. (pages 528-531) is a good reference.

$$= \frac{2 \left(\begin{array}{cccc} 2 & & & 2 \\ 2 & 1 & 1 & - \\ & 2 & & 2 \\ & 2 & - & 1 \end{array} \right)}{2 \left(\begin{array}{cccc} 2 & & & 2 \\ 2 & 2 & - & 1 \\ & 2 & & 2 \\ & 2 & - & 1 \end{array} \right)}$$

$$= \frac{2 \left(\begin{array}{cccc} 2 & & & 2 \\ 2 & 2 & - & 1 \\ & 2 & & 2 \\ & 2 & - & 1 \end{array} \right)}{2 \left(\begin{array}{cccc} 2 & & & 2 \\ 2 & 2 & - & 1 \\ & 2 & & 2 \\ & 2 & - & 1 \end{array} \right)}$$

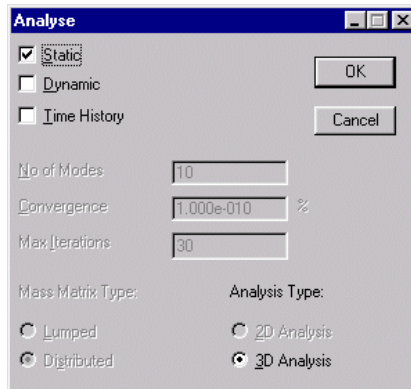
Performing Analysis

When you have finished setting up your structure and its restraints and loading, you use the Analyse command from the Case menu to carry out the analysis of the structure. While analysis is in progress, a progress bar will also be displayed.

Multiframe will check the structure prior to analysis and alert you if there are any problems. You must have specified the section type for all of the members. If you have missed any members, Multiframe will alert you with a dialog and will select all of the undefined members in the Frame window. You will also be alerted if there are any mechanisms or there are unrestrained degrees of freedom. In each case the offending part of the structure will be selected in the Frame window.

In Multiframe4D you can carry out a modal analysis of a structure and calculate up to the first 20 natural frequencies and mode shapes. The method used is 'sub-space iteration' and mode shapes are generated from the lowest natural frequency upwards.

- Choose Analyse from the Case menu



You are given the choice of performing a static analysis, a dynamic analysis, a time history analysis, or a combination of these three. If dynamic or time history analysis is chosen, then the user is able to modify three values required by the analysis. The three user input values are

1. No of Modes which is the number of natural frequencies and mode shapes required. This number must be between 1 and 20.

2. Convergence, which is the minimum value, required for the 'Convergence Error'. When the convergence error falls below the 'Convergence' the solution has completed successfully. The convergence error is calculated at the end of each iteration. The convergence error is defined as

$$\text{Convergence Error} = \frac{\sum_{i=1}^{\text{Max Modes}} \frac{\omega_{i,k+1}^2 - \omega_{i,k}^2}{2}}{\omega_{i,k}^2}$$

where $\omega_{i,k}$ = Natural Frequency i after iteration k .

Typically values for the convergence error from $1.0\text{e-}3$ to $1.0\text{e-}6$ will give adequate results.

3. Max Iterations is the number of iterations carried out before the analysis fails. The number must be greater than 0.

Note that if you are unsure of what values to choose, leave the current default values, which are suitable for most analyses.

Mass Matrix Type

The buttons Lumped and Distributed allow the user to decide on whether to use a distributed or lumped mass matrix when doing the modal analysis. Both matrices are valid so the choice is left up to the user. If you are short of memory the lumped mass matrix uses far less memory for storage. This is because the matrix is de-coupled and thus only the diagonal needs to be stored. In general, a lumped mass matrix will give slightly lower natural frequency values than an analysis using the distributed mass matrix.

2D/3D Analysis

Two more buttons appear at the bottom right of the dialog. These are 2D Analysis and 3D Analysis.

For most structures, the 2D Analysis button will be inactive and thus grayed out. The user will only be able to choose a 2D Analysis if either the structure and all loads including joint displacements are entirely in the x-y plane, or the structure is entirely in the x-y plane and only Dynamic or Seismic analysis is chosen.

The advantages of analysing in 2D include decreased analysis time combined with decreased memory requirements, and the ability to study only the in plane natural modes when using dynamic analysis.

Dynamic Analysis

Once you have defined your dynamic loads or accelerations, you can perform a Time History analysis to compute the response of the structure at each of the time steps you have specified.

To perform a Time History Analysis

- **Choose Analyse from the Case menu**

The Analyse dialog contains three options for the type of analysis you wish to perform. You can choose any of static, modal or time history analysis. If performing a modal analysis, you should enter the number of modes. The choice of mass matrix applies to both modal and time history analysis.

Precision and Memory Requirements

The solver will automatically switch from double precision to single precision arithmetic if there is not enough memory. Single precision arithmetic is less precise than double (it has about 7 digits of precision as opposed to 15 digits for double), but has the advantage of requiring less storage space for the stiffness matrix. This allows you to solve larger problems or have more load cases for the same amount of memory. The only problem, which can arise from the use of single precision arithmetic, is that if the stiffness matrix is badly conditioned, round-off errors may accumulate during analysis and may make results less accurate. Multiframe will warn you if it switches to single precision and give you the option of analysing using single precision or canceling the analysis.

Macintosh users: If you are using Multiframe under System 7.0 or later and you have virtual memory turned on, Multiframe will use the virtual memory if necessary to store the stiffness matrix used for analysis.

The solver also includes some additional error checking routines to detect problems with the structure. If you have drawn any members, which are not connected, to the rest of the structure, Multiframe will alert you to this fact. The section properties of all the sections will also be checked to ensure that they are non-zero where appropriate.

Canceling

A progress indicator is displayed while analysis is in progress. To cancel analysis

- **Hit the escape key on Windows**
- **Type command-period on Macintosh**

The cursor will change to a watch and analysis will be terminated.

You can switch from Multiframe to another program while analysis is in progress.

Viewing Results

Multiframe carries out a stiffness matrix analysis to determine the forces and displacements in the structure. The forces computed are Bending Moment, Shear Force, Torque and Axial Force. The displacements computed are the displacements and rotations of the joints and the displacement along the members as they deflect. These results can be viewed in numerical form in the Result window or in graphical form in the Plot window.

Result Window

Tables of numerical results may be viewed in the Result window. Click in the Result window to bring it to the front or choose Result from the Window menu if the window is not visible. If the structure has not been analysed since you made changes to it, no numbers will be displayed in the window.

There are five tables of results, which can be displayed in the Result window. These are a table of joint displacements, a table of joint reactions, a table of member actions or end forces, a table of member details and a table of dynamic results. Only one of these tables can be displayed at a time. The results can be viewed from the Results sub-menu under the Display menu to control which table is on display at any time. The current table is indicated with a check mark beside the appropriate menu item. On Windows you can also use the tabs at the bottom of the window.

The results for one load case at time can be viewed in the Result window. You can control which load case is currently on display by using the load case items at the bottom of the Case menu. The current load case is indicated with a check mark to the left of its name in the menu.

You may find the Result Layout command from the Window menu useful when viewing your results. This command arranges the Plot, Load and Result windows on the screen so that you can easily view graphical and numerical information at the same time. Remember that the Symbol... command under the View Menu will allow you to display the joint and/or member numbers on the graphics in the Plot and Load windows. This will allow you to refer to the text and graphics more conveniently.

In all of the tables drawn in the Result window, you can resize the columns by dragging the lines which separate the column titles. You can also change the text font and size used in the table by using the Font... command from the View Menu. Using the Numbers... command from the Format window can control the format of the numbers displayed in the table.

On Windows you can Right-Click on any column heading to sort or hide that column. Select and range of headings and use Right-Click to show hidden columns.

You can copy data from the Result window to the clipboard for use with other applications. To copy a single number from the table,

- **Click on the number to be copied**
- **Choose Copy from the Edit menu**

To copy a column from the table

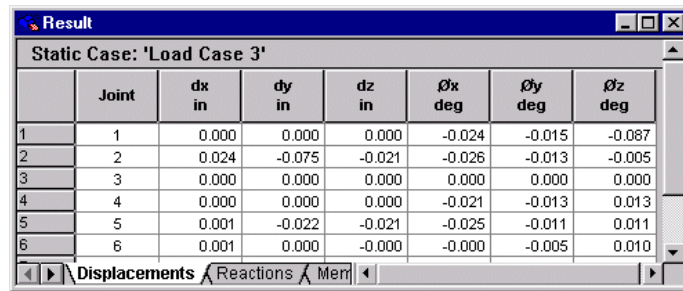
- Click on the title of the column to select it
- Choose Copy from the Edit menu

In a similar way you can select a row by clicking on the row number at the left hand end of the row or you can select the whole table by clicking in the box at the top left corner of the table.

Joint Displacements

To display the joint displacements

- Choose Joint Displacements from the Results sub-menu under the Display menu



The screenshot shows a software window titled 'Result' with a sub-header 'Static Case: 'Load Case 3''. It contains a table of joint displacements. The table has 8 columns: 'Joint', 'dx in', 'dy in', 'dz in', 'θx deg', 'θy deg', and 'θz deg'. There are 6 rows of data, numbered 1 through 6 in the first column.

	Joint	dx in	dy in	dz in	θx deg	θy deg	θz deg
1	1	0.000	0.000	0.000	-0.024	-0.015	-0.087
2	2	0.024	-0.075	-0.021	-0.026	-0.013	-0.005
3	3	0.000	0.000	0.000	0.000	0.000	0.000
4	4	0.000	0.000	0.000	-0.021	-0.013	0.013
5	5	0.001	-0.022	-0.021	-0.025	-0.011	0.011
6	6	0.001	0.000	-0.000	-0.000	-0.005	0.010

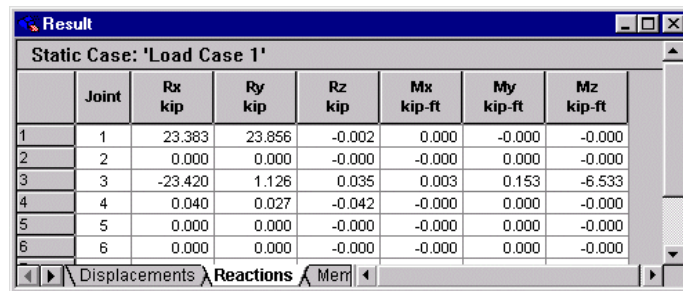
At the bottom of the window, there is a navigation bar with buttons for 'Displacements', 'Reactions', and 'Member Actions'. 'Displacements' is currently selected.

The table of joint displacements displays the number of each joint at the left of each row and the deflections and rotations in the direction of each global axis appear in the six columns. The units for each variable are shown underneath the title of the column in the table.

Joint Reactions

To display the joint reactions

- Choose Joint Reactions from the Results sub-menu under the Display menu



The screenshot shows a software window titled 'Result' with a sub-header 'Static Case: 'Load Case 1''. It contains a table of joint reactions. The table has 8 columns: 'Joint', 'Rx kip', 'Ry kip', 'Rz kip', 'Mx kip-ft', 'My kip-ft', and 'Mz kip-ft'. There are 6 rows of data, numbered 1 through 6 in the first column.

	Joint	Rx kip	Ry kip	Rz kip	Mx kip-ft	My kip-ft	Mz kip-ft
1	1	23.383	23.856	-0.002	0.000	-0.000	-0.000
2	2	0.000	0.000	-0.000	-0.000	0.000	-0.000
3	3	-23.420	1.126	0.035	0.003	0.153	-6.533
4	4	0.040	0.027	-0.042	-0.000	0.000	-0.000
5	5	0.000	0.000	-0.000	-0.000	0.000	-0.000
6	6	0.000	0.000	-0.000	-0.000	0.000	-0.000

At the bottom of the window, there is a navigation bar with buttons for 'Displacements', 'Reactions', and 'Member Actions'. 'Reactions' is currently selected.

The table of joint reactions displays the number of each joint at the left of each row and the reactions in the direction of each global axis appear in the six columns. The units for each variable are shown underneath the title of the column in the table.

Member Actions

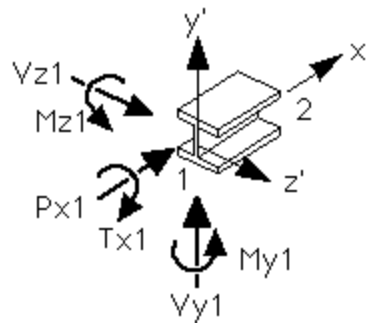
To display the member actions

- Choose Member Actions from the Results sub-menu under the Display menu

Result									
Static Case: 'Load Case 1'									
	Membr	Joint	Px' kip	Vy' kip	Vz' kip	Tx' kip-ft	My' kip-ft	Mz' kip-ft	
1	1	1	33.403	0.334	-0.002	0.000	-0.000	-0.000	
2	1	2	-33.403	-0.334	0.002	-0.000	0.035	4.730	
3	2	2	23.397	-1.126	-0.029	-0.002	0.055	-4.731	
4	2	3	-23.397	1.126	0.029	0.002	0.233	-6.533	
5	3	4	0.063	-0.011	-0.001	0.000	-0.000	-0.000	
6	3	5	-0.063	0.011	0.001	-0.000	0.022	-0.195	

The table of member actions displays the number of each member and the joints in the first two columns and the six end forces in each of the six remaining columns.

The member actions displayed in the Result window are shown in the diagram below. The actions at joint 2 follow the same sign convention as the actions at joint 1. Moments follow the right hand rule i.e. if the thumb on your right hand points in the direction of the positive axis, the direction of curl of your fingers will indicate the direction of positive moment.



Member Details

To display a detailed table of member actions for the member currently selected in the Plot Window

- Click on a member in the Plot window to display its local diagrams
- Choose Member Details from the Result sub-menu under the Display menu

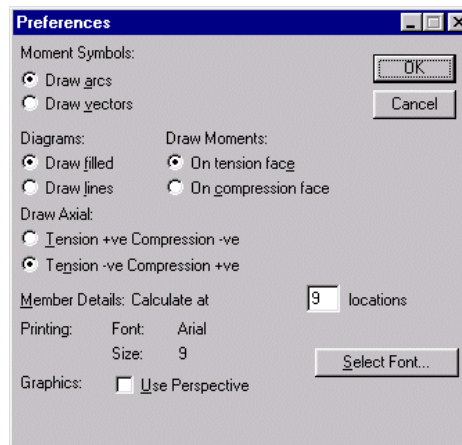
Result									
Static Case: 'Load Case 1' Member No 1 W12x40									
	x/L	x ft	Mz' kip-ft	My' kip-ft	Vy' kip	Vz' kip	Px' kip	Tx' kip-ft	
1	0.000	0.000	0.000	0.000	0.334	0.002	33.403	0.000	
2	0.125	1.768	0.591	0.004	0.334	0.002	33.403	0.000	
3	0.250	3.536	1.183	0.009	0.334	0.002	33.403	0.000	
4	0.375	5.303	1.774	0.013	0.334	0.002	33.403	0.000	
5	0.500	7.071	2.365	0.017	0.334	0.002	33.403	0.000	
6	0.625	8.839	2.956	0.022	0.334	0.002	33.403	0.000	

This table gives member actions at a number of points along a member. The table is displayed for the current member you have clicked on in the Plot window. The signs of the values displayed correspond with the diagram of the member and is controlled using the Preferences item (see Plot Window, Sign Convention below).

The member actions and deflections are initially displayed at a number of evenly spaced locations. You can vary both the number of points and the position of each point.

To specify how many points the details will be displayed at

- **Choose Preferences from the Edit menu**



- **Type in the number of points you want to display detailed information**
- **Click the OK button**

The number of locations must be between 2 and 64.

You can also change the location of any intermediate point by clicking in the first or second column and typing in a new distance or a proportional distance respectively. The first column in the table displays the distance of the point from joint 1 of the member as a proportion of the length of the member. For example, a value of 0.333 indicates the point is approximately one third of the distance along the member. The second column in the table shows the actual distance of the point from joint 1.

To change the position of a point as a proportion of the member's length

- **Click on the number in the first column of the point's row**
- **Type in a new proportion of the distance**
- **Press the Enter key**

The row's values will be re-calculated and re-displayed to show the values at the new location.

To change the distance of a point from the left hand end of the member

- **Click on the number in the second column of the point's row**
- **Type in a new distance of the point from the left hand joint**
- **Press the Enter key**

The row's values will be re-calculated and re-displayed to show the values at the new location.

Dynamic Results

In Multiframe4D, you can display a table showing the results of a dynamic analysis.

- **Choose Dynamic Results from the Result sub-menu under the Display menu**

Each row shows the frequency and period for that mode shape. The units for each variable are shown underneath the title of the column in the table.

Time History Results

The time history results are stored on disk in a file called “<FileName>. Dynamics” on Macintosh or “<Filename>.mth” on Windows.

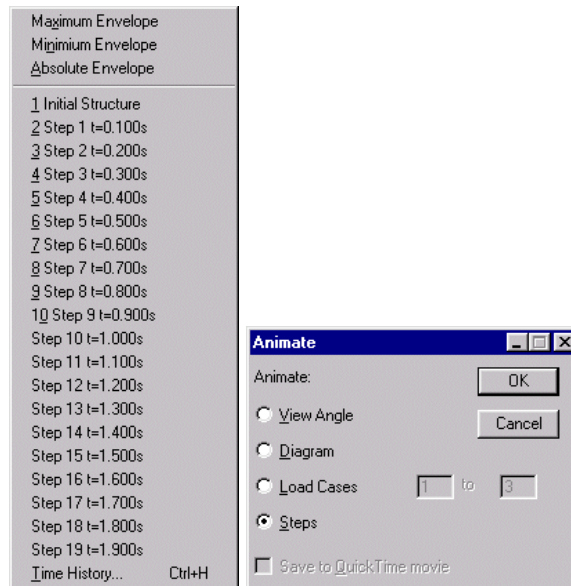
Do not delete this file while Multiframe4D is running and the associated frame is loaded.

To view the results for each time step in a dynamic or seismic load case

- **Choose the step from the Time... menu**

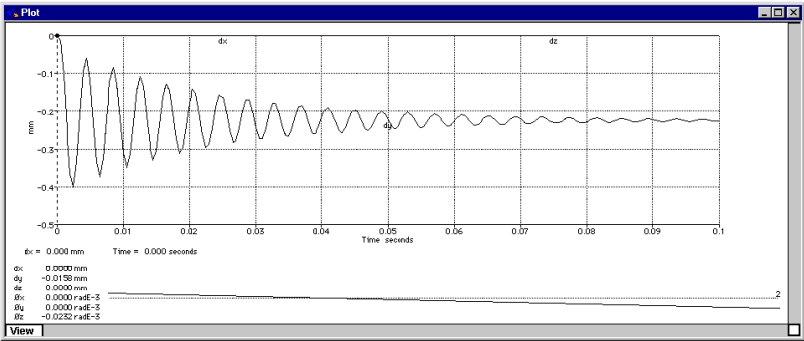
To best see the results

- **Choose Animate... from the Display menu**
- **Select Steps**



This will show you an animation including all the time steps.

When an increment in a Dynamic or Seismic load case is active you are able to view all results exactly as for a Static load case. In addition, if you select a node in the Plot window you get a graph of the behavior of the x y z degrees of freedom over the entire analysis.



Viewing Envelope Cases

Envelope cases are created for the Time History analysis if Calculate Envelope Cases check box is selected in the Dynamic or Seismic case edit dialogs. The range of cases to envelope can also be input.

Dynamic Load Case

Name: Step Case 2

Analysis Time:

No of steps: 20

Start Time: 0.0 seconds

Increment: 0.100 seconds

End Time: 2.000 seconds

☒ Calculate Envelope Cases

First Step: 1

Last Step: 10

Rayleigh Damping:

Alpha: 0.000

Beta: 0.000

OK

Cancel

The envelope cases store the absolute maximum displacement values in all cases along with the maximum, minimum and absolute values for the end forces. You can use the envelope cases to view the range of values encompassed by a dynamic or seismic analysis.

When viewing the results in the Results data window, the title bar will identify the current joint degree of freedom and the step increment that the value is associated with.

Result

Time History Case: 'Step Case 1' - Maximum Envelope

	Joint	dx in	dy in	dz in	Øx deg	Øy deg	Øz deg
1	1	0.00	0.00	0.00	-0.02	0.01	0.00
2	2	-0.00	0.00	-0.03	0.01	-0.01	-0.00
3	3	0.00	0.00	0.00	0.00	0.00	0.00
4	4	0.00	0.00	0.00	-0.01	0.01	0.01
5	5	-0.02	-0.01	-0.03	0.00	-0.01	0.00
6	6	-0.02	0.00	0.00	-0.00	-0.01	-0.00
7	7	0.00	0.00	0.00	0.00	-0.01	0.02
8	8	0.00	0.00	0.00	0.00	0.00	0.00

Displacements Reactions

Plot Window

Four different types of diagrams of forces and deflections can be viewed in the Plot window. These are diagrams of the whole

structure, diagrams for an individual member, reactions at a joint and deflections at a joint.

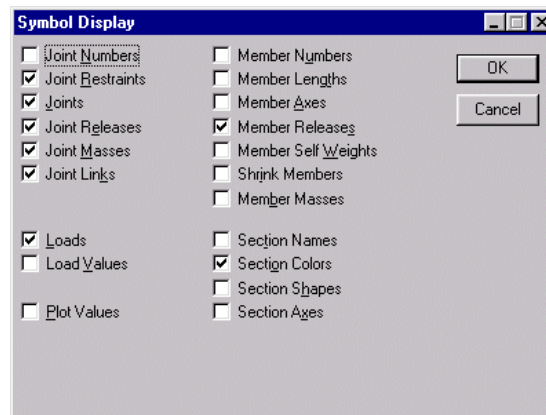
The results for one load case at time can be viewed in the Plot window. You can control which load case is currently on display by using the load case items at the bottom of the Case menu. The current load case is indicated with a check mark to the left of its name in the menu. The name of the current load case is also displayed at the bottom left hand corner of the Plot window except when viewing the 3D view of the structure.

When displaying diagrams of the whole structure or an individual member, you can use the items from the Diagram sub-menu under the Display menu to control which variable is to be displayed at any time. The variable currently being displayed will be indicated with a check mark to the left of the item in the menu and will also be shown at the bottom of the Plot window.

You can also use the Symbols command from the Display menu to control which symbols are displayed on the diagrams.

To turn on and off the display of symbols

- **Choose Symbols from the Display menu**



If you turn on the display of Plot values, the labels on the Plot diagrams for axial force and stress diagrams will have a T & C marked on the axes to indicate which is Tension and which is Compression.

Structure Diagrams

The diagrams of the whole structure can display bending moments, shear forces, axial forces, torque or deflection. When you have a diagram of the whole structure drawn in the Plot window and forces on display, the force diagram for each member will be superimposed on the member. With deflections on display, a diagram of the exaggerated deflection of the whole structure will be drawn. If you have performed a dynamic analysis and you are currently viewing a dynamic case, the deflected shape will represent the mode shape for that case. For all diagrams, Multiframe will choose a scale which best suits the size of your structure and the magnitude of the forces and displacements. All load cases are drawn to the same scale.

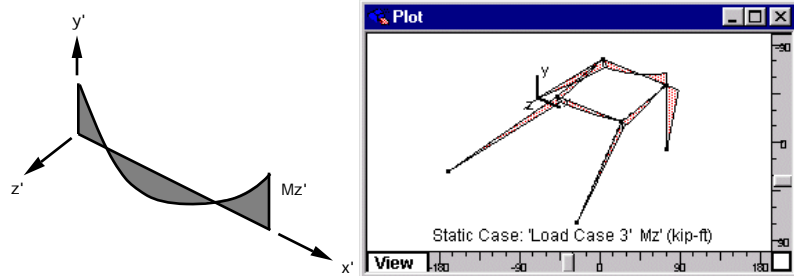
You can exaggerate the scale of a diagram

- **Choose Plot... from the Display menu**

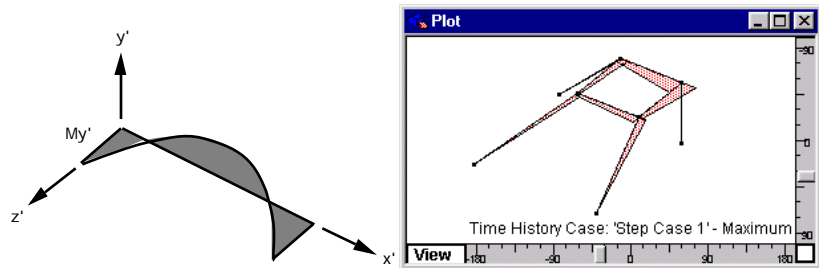
The Zoom, Shrink, Pan and Size To Fit commands can be used in the Plot window to change your view of the structure. The rotation bars can also be used to change the view point in the 3D view.

The six force diagrams, which can be displayed in the Plot window, are

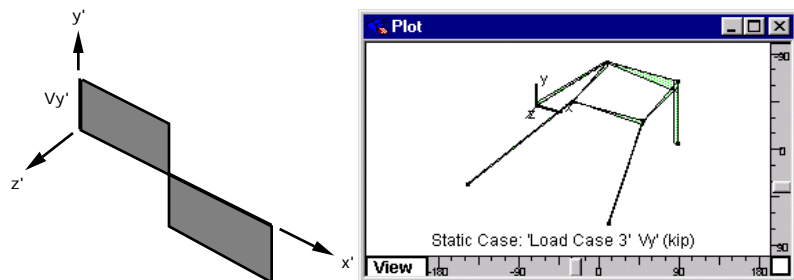
$M_{z'}$ bending moments about the local z' axis of the members (in plane bending)



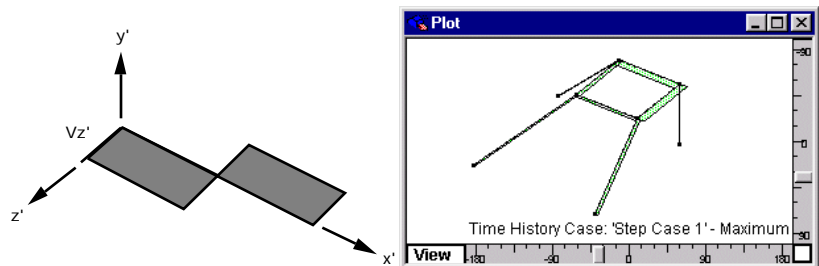
$M_{y'}$ bending moments about the local y' axis of the members (out of plane bending)



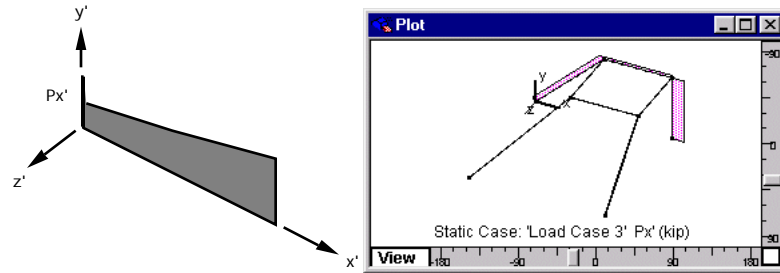
$V_{y'}$ shear force through the local y' axis of the members (in plane shear)



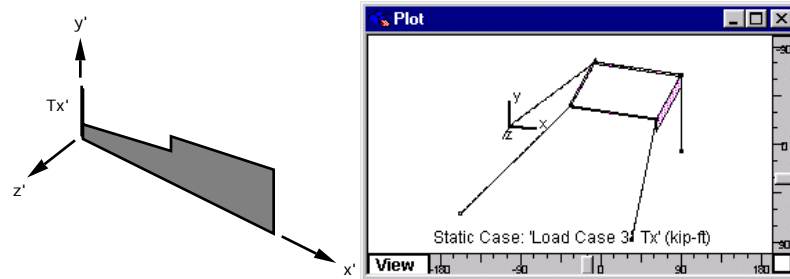
$V_{z'}$ shear force through the local z' axis of the members (out of plane shear)



$P_{x'}$ axial force along the local x' axis of the members (tension or compression)



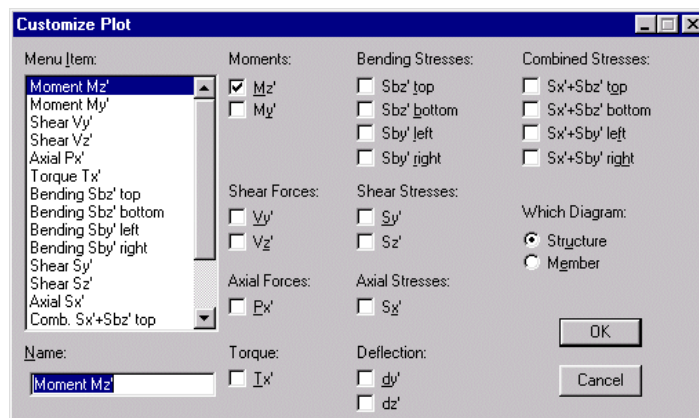
Tx' twist or torque about the local x' axis of the members



To choose which diagrams are displayed in the Plot window

- **Choose Customise Plot... from the Display menu**

A dialog will appear listing the menu items controlling the plot display and the actions, stresses and deflections which can be displayed.

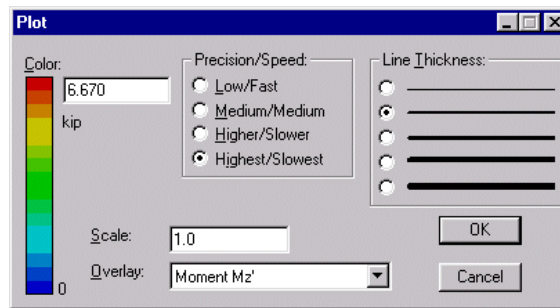


- **Click on the name of the menu item you wish to change**
- **Click on the Member button**
- **Set the check boxes of the actions you wish to display in the local member diagrams**
- **Click on the Structure button**
- **Set the check boxes of the actions you wish to display on the diagram of the whole structure**
- **Click the OK button**

You can control the values associated with the colors used when overlaying an action onto the structure deflection diagram. These colors and values are controlled using the Plot command under the Display menu.

To change the value associated with the overlaid colors

- Choose Plot... from the Display menu



- Type Tab to move to the field containing the maximum threshold value
- Type in a new maximum threshold value to be used with red to indicate the highest actions or stresses
- Click the OK button

Every part of the structure with an action value greater than or equal to the value you specified will be colored red. Other parts of the structure will be colored in an even gradient down to blue for those parts of the frame with an action of zero.

Stresses

You can display stresses as well as forces and deflections in the Plot window. Which data to display is controlled by two hierarchical menus and the deflection item under the Display menu.

To display a force diagram

- Choose the desired item from Actions sub-menu under the Display menu

To display a stress diagram

- Choose the desired item from Stress sub-menu under the Display menu

To display deflections

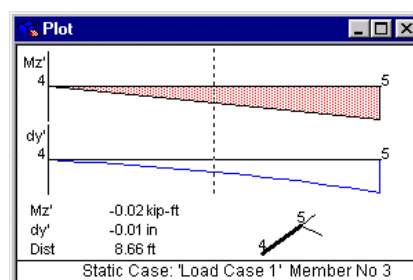
- Choose the Deflection item from the Display menu

You can choose to display a number of diagrams simultaneously in the Plot window. This applies to both structure and member diagrams.

Member Diagrams

The diagrams of individual members can also display bending moments, shear forces, axial forces, torque or deflections. To obtain a diagram for an individual member

- Click on the member you wish to view in detail



The local diagram for that member will then be drawn in the Plot window.

The maximum and minimum values on the diagram are displayed below the graph, and below these two numbers indicate the value of the diagram at the position of the gray crosshair and the distance of that crosshair from the left hand end of the member.

You can drag the crosshair up and down the length of the member to determine the value of the diagram at any position. The crosshair will remain in the position where you release the mouse button after dragging.

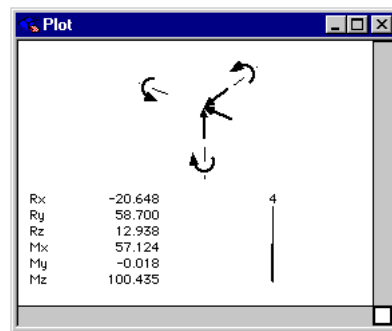
Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Joint Reactions

The diagram of a joint's reactions shows the direction and magnitudes of the reactions at the joint. Joint reactions are displayed relative to the global coordinate system.

To display the reactions for a joint

- **Ensure the moments, shear, torque or axial forces are on display**
- **Click on the original position of the joint in the structure diagram**



The original position is marked by the point where the gray lines of the members connecting at the joint meet. The diagram of the structure will be replaced by a diagram of the joint reactions.

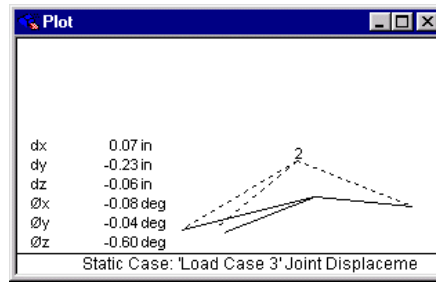
Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Joint Displacements

The diagram of a joint's displacements indicates the vertical, horizontal and rotational displacements of the joint.

To display the displacements for a joint

- **Choose Deflection from the Display menu**
- **Click on the original position of the joint in the structure diagram**



Click on the drawing at the bottom right of the window to return to a diagram of the whole structure.

Dynamic Results

After a dynamic analysis has completed successfully each mode is stored as a separate load case. The mode shapes are normalised and stored in the same way as displacements in a static analysis.

To view the natural frequencies and mode shapes

- **Choose the mode shape from the Case menu**

The frequency and period are printed in the bottom left corner of the Plot window.

To view the mode shape

- **Select the Plot window**
- **Choose Deflection from the Display menu**

There are no moments, shears or other forces associated with the dynamic results.

Calculations

As well as carrying out an analysis of your structure, Multiframe allows you to prepare your own design calculations by making use of the CalcSheet facility. Design calculations can be prepared and evaluated in the CalcSheet window. The calculations may be entered line by line, terminating each line by pressing return or enter. The calculations should follow the same general syntax as the Basic programming language, however, only simple calculations are supported. There is no support for looping or IF statements or comparison (< , >) operators.

If the member diagram for a member is currently displayed in the Plot window, the section properties and results of analysis for the member will be automatically included as variables in the CalcSheet. These variables are listed below.

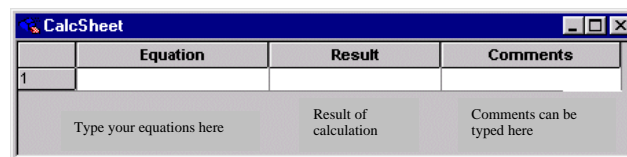
The calculations can be saved to and read from disk or printed by choosing the appropriate items from the File menu while the CalcSheet window is frontmost.

Calculation Sheet

The CalcSheet facility in Multiframe allows you to prepare your own design calculations and evaluate them without leaving Multiframe. The calculations can use any number of variables defined by you and can also access a number of variables from within Multiframe, which contain the results of analysis and section properties for a member.

Calculations are prepared and evaluated in the CalcSheet window. To make the CalcSheet window visible

- Choose CalcSheet from the Window menu



The window is divided into three columns titled Equation, Value and Comment. Calculations may be entered into the equation column and when the expressions are evaluated the results will be displayed in the Value column. The Comment column may be used by you for entering any information, which is relevant to the line on which it is entered. The Comment column is especially useful for entering the units for the results of calculations you carry out.

Calculations are evaluated by choosing the Calculate command from the Case menu. The results of the calculations are displayed in the Value column. The value shown is the value of the expression evaluated at each line.

You can change the equations in the CalcSheet at any time and then re-evaluate the expressions using the Calculate command.

Pre-defined CalcSheet Variables

Multiframe provides you with a number of pre-defined variables for use within the CalcSheet. These variables relate to a single member

in the structure at a time. You specify which member in the structure to use by clicking on that member in the Plot window and displaying its member diagram. Multiframe will automatically extract the appropriate data from that member and store it in the variables in the CalcSheet.

The pre-defined variables include the length, slope, moments, forces, displacements and section properties. Remember that these variables are only available if there is a member diagram on display in the Plot window.

A complete list of the variables is

Name	Description
dx1	x displacement of joint 1
dy1	y displacement of joint 1
dz1	z displacement of joint 1
Øx1	rotation of joint 1 about the x axis
Øy1	rotation of joint 1 about the y axis
Øz1	rotation of joint 1 about the z axis
dx2	x displacement of joint 2
dy2	y displacement of joint 2
dz2	z displacement of joint 2
dx	x' deflection at the crosshair
dy	y' deflection at the crosshair
dz	z' deflection at the crosshair
Øx2	rotation of joint 2 about the x axis
Øy2	rotation of joint 2 about the y axis
Øz2	rotation of joint 2 about the z axis
Mz	bending moment about z' at crosshair
My	bending moment about y' at crosshair
Vy	shear force through y' at crosshair
Vz	shear force through z' at crosshair
Px	axial force at crosshair
Tx	torque at crosshair
Length	Length of member
Slope	Slope of member
Dist	distance to the crosshair
MaxMz	Absolute value of the max Mz'
MaxMy	Absolute value of the max My'
MaxVy	Absolute value of the max Vy'
MaxVz	Absolute value of the max Vz'
MaxAT	Absolute value of the max axial tension
MaxAC	Absolute value of the max axial compression
MaxTx	Absolute value of the max torque
Maxdx	Absolute value of the max x' deflection
Maxdy	Absolute value of the max y' deflection
Maxdz	Absolute value of the max z' deflection
Pi	3.14159

The pre-defined variables in the CalcSheet will be in the units currently specified using the Units item from the View menu (See Units below).

Section Properties Variables

Multiframe will automatically extract all the section properties for the section used for the member from the Sections Library. These properties will be stored in variables with the same names as the properties in the sections library.

To view the names of these properties

- **Choose Edit Section... from the Edit menu**

Edit Section

Name: W44x285

Properties:

	Property	Value	Units
1	Weight	285.000	lb/ft
2	A	83.800	sq in
3	D	44.020	in
4	tw	1.025	in
5	B	11.810	in
6	tf	1.770	in
7	Ix	24600.000	in^4
8	Sx	1120.000	in^3

Shape:

☒ I ☐ L ☐ T ☐ □ ☐ ○ ☐ ?
☐ C ☐ L ☐ □ ☐ ○ ☐ □

OK Cancel

When you view the table of properties for a section, the name of each property will be shown in the left hand column of the table.

For example, if you wished to compute the shear stress in the web of a member you could enter a calculation such as

$$\text{ShearStress} = V_y / (D * tw)$$

Where V_y is the shear force variable and D and tw are the section depth and web thickness taken from the properties for the section.

The constant π is also provided as a variable in the CalcSheet.

Saving Calculations

To save the calculations you have created

- **Ensure the CalcSheet window is frontmost**
- **Choose Save As from the File menu**

You can then save the calculations file on disk. The calculations in Multiframe are stored independently from the structure you are working on, so you can use the same calculations on a number of different structures.

Printing

Multiframe offers a number of options for controlling the output of data or results to a printer. Multiframe can print text, numbers and graphics.

Page Setup

When printing data or diagrams from Multiframe, it is first necessary to set up the page size and the print quality. The Page Setup command from the File menu can be used to set up the page size you will be using in the printer. This set up need only be done once and all subsequent printing will use this format.

Setting up the Printer

First, ensure that your printer is attached to your computer with an appropriate cable and that it is switched on and has sufficient paper loaded. Refer to your computer's owner guide or your printer manual if you have any problems.

- **Choose Page Setup... from the File menu**

A dialog box will appear which allows you to choose the paper size and orientation, set the size of margins for printing and type in text to appear at the head and foot of each page of paper printed.

Select the paper size you have installed in your printer. You can also choose to enlarge or reduce the printout and adjust the output with a number of other options.

No matter what type of printer you have, Multiframe allows you to enter values for header and footer titles to be printed at the top and bottom of each page printed. You can also set up the size of the margins you would like around the printed output. The two check boxes at the lower left of the dialog allow you to switch on or off the printing of page numbers and date and time information. The Autoset margins check box can be used if you want to use the maximum amount of printable area (i.e. the minimum margin size) that your printer will allow.

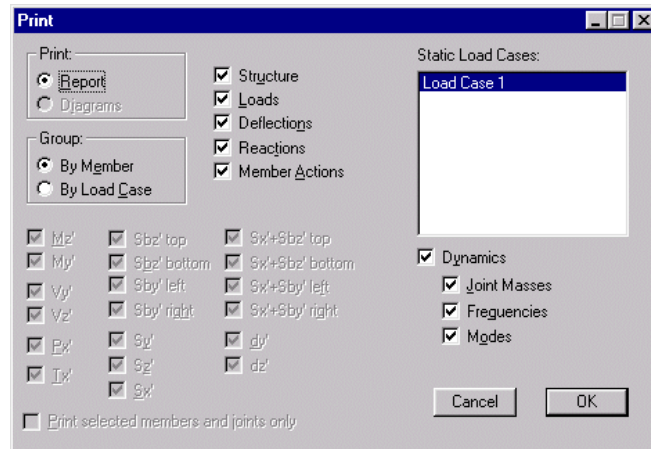
Color printing is supported on Postscript printers, which provide color output.

Printing Results

When printing a report summary of Multiframe data, you can restrict the output of results to just the members selected in the frontmost window.

To print results for a range of members

- **Select the members in the frontmost window**
- **Choose Print from the File menu**



- Turn on the "Print selected members only" check box
- Click the OK button

Only the results for the selected members will be printed.

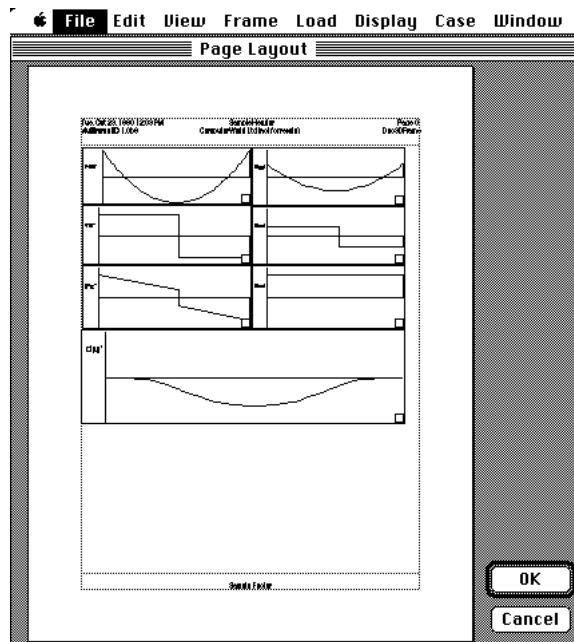
Page Layout

(Macintosh Only)

The Page Layout command under the File menu allows you to control the layout of member diagrams when printing.

To adjust the layout of diagrams on the page

- Choose Page Layout... from the File menu



A window will appear titled Page Layout

- Drag and resize the diagrams to the desired positions on the page
- Click the OK button

Drag the position of a diagram by pressing inside it and dragging to a new location. Resize a diagram by pressing in the bottom right hand corner of the diagram and dragging it to its new shape. You should keep the diagrams inside the dotted printable area of the page if you want them to be printed. The printable area will vary

depending upon the mechanical limitations of the printer you are using. You can drag a diagram off the page to prevent it being printed.

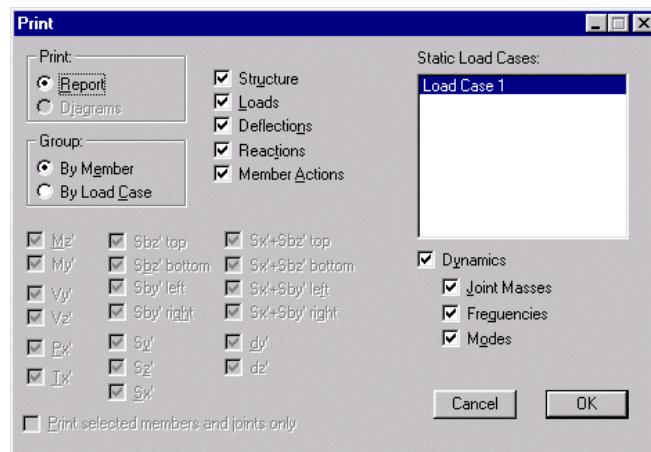
The layout will be stored and used by Multiframe for laying out your force and deflection diagrams on the page when using the print selection option. Multiframe will repeat the layout as many times as will fit down and across each page.

Print Command

To print a report on the structure

- **Choose Print from the File menu**

A printing options dialog will appear.



- **Choose the data you want to print and which load cases to print**
- **Click the OK button**

The standard print dialog will appear

- **Select the print quality, number of copies and any enlargement or reduction of the printing**
- **Click OK to start the printing**

The printing options dialog allows you to control what data will be printed. If you wish to turn off the printing of some data, click on the appropriate check boxes. If you only wish to print some load cases, hold down the shift key and click on the cases you don't wish to print.

The standard printing dialog allows you to set the print quality, page range to print, number of copies to print and the type of paper feed. The page range, number of copies and paper feed may be set to any appropriate setting. To cancel printing while it is in progress, hit the Escape key on Windows or hold down the command key and type period (.) on Macintosh, or click on Cancel.

Printing Diagrams

You can print the force and deflection diagrams for any or all of the members in a structure by using the Print Selection option available by using the Print... command from the File menu. First, ensure that the Plot window is frontmost on the screen. Then select the members to be printed, by clicking or dragging a rectangle, in the usual way.

To print the diagrams for the selected members

- **Choose Print... from the File menu**

The printing options dialog will appear.

- **Click on the Selection radio button**
- **Select which diagrams and load cases you wish to print**
- **Click the OK button**

The standard print dialog will appear.

- **Select the print quality, number of copies and any enlargement or reduction of the printing**
- **Click OK to start the printing**

The printing options dialog allows you to control what data will be printed. If you wish to turn off the printing of some data, click on the appropriate check boxes. If you only wish to print some load cases, hold down the shift key and click on the cases you do not wish to print. If you want to turn the printing of a load case back on, shift-click on the load case once more.

The standard printing dialog allows you to set the print quality, page range to print, number of copies to print and the type of paper feed. The page range, number of copies and paper feed may be set to any appropriate setting. To cancel printing while it is in progress, hit the Escape key on Windows or hold down the command key and type period (.) on Macintosh, or click on Cancel.

Chapter 3

Multiframe Reference

This chapter summarises the overall structure, windows, toolbars and menu commands of Multiframe. (The Toolbars are only available in the Windows version of Multiframe.)

Windows

Multiframe uses a range of graphical, tabular, graph and report windows.

Frame Window

This window is used for preparing a physical description of the structure. This includes geometry, connections, section types, section orientation and restraints. You can also modify joint and member masses for dynamic analysis.

Data Window

This window is used for viewing and editing the data describing the structure and its loading. It allows you to edit data numerically rather than graphically. A number of different tables of data can be displayed describing joints, members, joint loads, member loads, restraints and prescribed displacements, springs, sections and load cases.

Load Window

This window is used for setting up the loading conditions you wish to apply to the structure. You can also add a self weight load case and control whether member's self weight is included in static or dynamic analysis.

Result Window

This window is used for viewing the results of the analysis in numerical form. It can display tables of joint displacements and reactions and member actions or end forces. Results of any dynamic analysis are also displayed here in the form of a table of frequencies and periods of vibration.

Plot Window

This window is used for viewing diagrams of the forces and deflections in the structure. It allows you to view diagrams of the whole structure, individual members, joint reactions and joint displacements. You also view the mode shapes for dynamic analysis in this window.

CalcSheet Window

This window is used for preparing and evaluating design calculations, which use the results of the analysis as a basis for design checks.

Toolbars

Users of the Windows version of Multiframe can use the icons on the toolbars to speed up access to some commonly used functions. You can hold your mouse over an icon to reveal a pop-up tip of what the icon does.

Generate Toolbar



The Generate Toolbar is found in the Frame Window and performs the same functions as the Generate command in the Frame menu. From left to right, the buttons generate the following geometry:

Continuous Beam
Curved Member
Portal Frame
Multi-story Frame

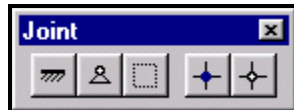
Member Toolbar



The Member Toolbar is found in the Frame Window and performs some of the functions found in the Frame menu. From left to right, the buttons perform the following functions:

Add Member
Delete Member
Sub-divide Member
Section Type
Pinned/Pinned Member Release
Rigid/Rigid Member Release
Member Orientation

Joint Toolbar



The Joint Toolbar is found in the Frame Window and performs some of the functions found in the Frame menu. From left to right, the buttons perform the following functions:

Fixed Joint Restraint
Pinned Joint Restraint
Free Joint Restraint
Rigid Joint Type
Pinned Joint Type

View Toolbar



The View Toolbar is found in the Frame, Load, and Plot Windows and performs some of the functions found in the View menu. From left to right, the buttons perform the following functions:

Zoom
Shrink
Pan
Size To Fit
Toggle Clipping
Toggle Masking
Toggle Grid
Toggle Axes

Actions Toolbar



The Actions Toolbar is found in the Plot Window and performs some of the functions found in the Display menu. From left to right, the buttons perform the following functions:

Bending Moment about the local z' axis
Shear forces in the local y' axis
Axial forces for the current load case
Display Deflection
Bending Moment about the local y' axis
Shear forces in the local z' axis
Torque about the local x' axis

Load Case Toolbar



The Load Case toolbar can be used as a shortcut for changing the current load case. You can select a load case from the drop down list or click the right and left arrows to go to the next or previous load case.

Menus

Multiframe uses the standard set of Macintosh or Windows menu commands for File, Edit and Windows operations. It also has a range of menus for:

File Menu

The file menu contains commands for opening and saving files, importing and exporting data, and printing.

New

(Windows: Ctrl N / Macintosh: command N)
Erases the structure and all loads. Use this to start work on a new structure. If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before starting the new work.

Open

(Windows: Ctrl O / Macintosh: command O)
Open a file, which has previously been saved on disk. If the CalcSheet window is frontmost, it will read in a new calculations file. Otherwise, it will read in a frame you have previously saved to disk.

If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before opening the new file. Multiframe can read Multiframe, Multiframe text and DXF files.

Close

Select Close when you wish to finish with the current design. Before closing, a dialog box will appear asking whether you wish to save the current design. If you select Yes the current design will be saved to the disk.

Save

(Windows: Ctrl S / Macintosh: command S)
Save work so far with the same name you saved with last time.

Save As

Save work so far in a file with a new name. If the CalcSheet window is frontmost, this will save the current calculations in a file. Otherwise, it will save the structure. You can also choose to save the geometry of the frame in a number of different file formats.

Import

See “Import Submenu”

Export

See “Export Submenu”

Open Library

The Open Library command opens a Sections Library or Load Library.

Page Setup

Set up the printer for printing. Allows you to set margins, header and footer titles and options on printing the date, time and page numbers.

Print Report

(Windows: Ctrl P / Macintosh: command P)

Windows users: Print out the data and results of the analysis or print out member diagrams for the selected members in the Plot window.

Print Window

Print the contents of the frontmost window on the screen

Page Layout

(Macintosh Only)

Define your preferred layout for printing of member diagrams. You can drag the diagrams and resize them to get the layout you desire.

Print

(Macintosh Only)

Print out the data and results of the analysis or print out member diagrams for the selected members in the Plot window.

Quit / Exit

(Macintosh: command Q)

(Macintosh Only: Quit)

Leave Multiframe and go back to the Finder. If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before quitting.

(Windows Only: Exit)

Leave Multiframe and go back to the Desktop. If you have any work unsaved, Multiframe will prompt you to save any changes to the structure or calculations before quitting.

Import Submenu

(Windows Only)

Import data from another program into Multiframe.

DXF

Files can be saved in either 2D or 3D format. This is to allow compatibility with older CAD programs, which only accept 2D DXF. The members in the frame are saved as LINE entities.

Multiframe will allow DXF file input from any CAD program. We have tested with AutoCAD, MiniCAD, Microstation and Claris CAD, and we expect input from all other systems to work also.

When reading in DXF files, Multiframe will interpret each line or segment of a polyline as a member. Any members, which have ends within 0.2 in (5mm) of each other, will be connected together. Multiframe can read 2D or 3D DXF files compatible with AutoCAD release 10 and higher.

Fortran Text

See "Appendix E"

Export Submenu

(Windows Only)

Data can be exported from Multiframe in file formats compatible with other programs.

2D DXF

This is to allow compatibility with older CAD programs, which only accept 2D DXF. The members in the frame are saved as LINE entities.

3D DXF

You can export 3DFACE entities into a 3D DXF file. This is useful for input to rendering or 3D presentation or modeling programs.

To export 3DFACE entities in the DXF file:

- **Ensure rendering is turned on in the frontmost window on the screen**
- **Choose Save As from the File menu**
- **Choose 3D DXF from the File Format pop-up menu**
- **Type in a name for your file**
- **Click the Save button**

If you do not have rendering turned on in the frontmost window when you save in 3D DXF format, the lines representing the frame will be saved as 3DLINE entities.

Spreadsheet Text

There are now two types of text output. FORTRAN style text is the same text output as in version 1.0 and is best for input to a post-processing program. The new Spreadsheet text output is a summary of maximum actions for each member in the frame and is in a format suitable for input to a spreadsheet such as Excel or Lotus.

Day Star Text

Multiframe outputs a tabular summary of member actions in a format suitable for input to the Day Star Text AISC steel code checking program available from DayStar Software Inc., <http://www.daystarsoftware.com>

The American steel library included with Multiframe uses a section naming convention consistent with that used by DS Steel.

Edit Menu

The Edit menu contains commands for copying and pasting data and working in tables.

Undo

(Windows: Ctrl Z / Macintosh: command Z)
Undo the last action you carried out. The name of this item will change to reflect the command that can be undone.
Windows users: If your frame has less than 500 members you can undo up to 10 steps.

Redo

(Windows: Ctrl Y)
(Windows Only)
Return to the last action you carried out before selecting Undo from the Edit menu.

Cut

(Windows: Ctrl X / Macintosh: command X)
Remove the current selection and place it on the clipboard.

Copy

(Windows: Ctrl C / Macintosh: command C)

In the Data, Result or Graphics window, this command copies the current selection to the Clipboard

If you hold down the Shift key on Windows, or the Option Key on Macintosh in the Data or Result window while choosing the Copy command from the Edit menu, the column titles will be included in the text placed on the clipboard.

If you hold down the Shift key on Windows, or the Option Key on Macintosh in a graphics window while choosing the Copy command from the Edit menu, you will be presented with a dialog offering options to copy to a Pict, DXF, Renderman, or Postscript File.

Paste

(Windows: Ctrl V / Macintosh: command V)

Paste the contents of the clipboard into the current selection.

Clear

Remove the current selection without placing it on the clipboard.

Show/Hide Clipboard

(Macintosh Only)

Make the clipboard visible/invisible.

Select

See "Select Submenu"

Find

(Windows Only)

Allows you to search through the structure to find members with actions, deflections or stresses in excess of limits you may enter. For example, you could find and select all of the members with a tensile stress greater than 21ksi.

Add Section

Store the data for a custom section type in the member properties file.

Edit Section

Edit the data for a custom section type in the member properties file.

Delete Section

Delete the data for a custom section type from the member properties file.

Section Colors

Allows you to set the colors to be used for display of section types in the Frame window.

Change Section Library

(Macintosh Only)

Open a new Section Library.

Edit Load Library

Allows you to view, edit and add data in the load library.

Preferences

Allows you to set how moment diagrams are drawn and what type of moment symbols to use. There is also an option to use perspective when displaying graphics in the 3D views. You may find this useful for presentation purposes. However, you will usually find it more convenient to work with the standard orthographic projection. If you do turn on the perspective option, it uses a perspective angle of 30° corresponding to the natural perspective of the human eye.

You can control the font and size of text used when printing by using the pop-up menu options in the Preferences dialog.

The sign convention for axial forces and stresses can be controlled using the Preferences item. You can choose to either display tension or compression as positive when drawing member diagrams.

Stresses are also displayed according to the sign convention you choose. This ensures that the signs of stresses caused by bending correspond with the signs of stresses caused by axial actions.

Properties

(Windows Only)

Shows details about the joint or element selected.

Select Submenu

The Select menu has commands for automatically selecting parts of the structure.

All

Automatically selects all the members in the frame.

Horizontal

Automatically selects all the horizontal members (beams) in the frame.

Vertical

Automatically selects all the vertical members (columns) in the frame.

Sloping

Automatically selects all the members in the frame, which are neither vertical nor horizontal.

Section

Allows you to select all the members in the Frame window which have a given section type. You can choose the section type to be selected from a list of sections in the Sections Library.

Member

Allows you to select a member by number.

Joint

Allows you to select a joint by number.

Member Label

(Windows Only)

Selecting a member using the member label.

View Menu

The View menu contains commands for controlling the appearance of the display in the graphical window.

Zoom

(Windows: Ctrl W / Macintosh: command W)
Zoom in on part of the current display. A cross-hair will appear and the view to be viewed in close-up may be selected by pressing the mouse button and dragging a rectangle surrounding the area of interest. Release the button to draw the zoomed view.

Pan

(Windows: Ctrl E / Macintosh: command R)
Pan across the structure displayed in the frontmost window. Press and drag in the window to move the frame.

Shrink

(Windows: Ctrl R / Macintosh: command E)
Reduce the size of the drawing in the frontmost window to half its current size.

Size To Fit

(Windows: Ctrl T / Macintosh: command T)
Scale the drawing in the frontmost window so that it just fits inside the window. The Size To Fit menu is a sub menu of the View menu, and is described in detail following this section on the View menu.

Clipping

See “ Clipping Submenu”

Masking

See “Masking Submenu”

Depth

Set the depth that drawing will occur in the two dimensional views (not available in 2D).

Size

Set the maximum and minimum coordinates available in the Frame window. Use this to set up the overall coordinates before you begin creating a structure.

Grid

Switch on or off the use of the grid in the Frame window and set the spacing of the grid.

Axes

Turn on or off the display of axes in the frontmost window.

Font

Set the font size and style for the text in the frontmost window.

Numbers

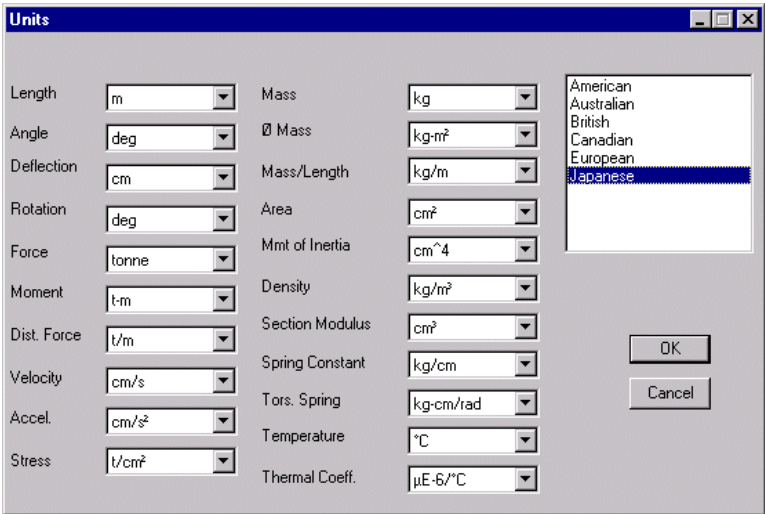
Set the type of numeric format you would like to use to display numbers in the frontmost window. You can choose to use decimal or scientific notation and specify how many digits of precision you wish to display.

Units

Multiframe allows you to work in a range of units. Related sets of units are conveniently grouped together under a country name. You can switch from one country setting to another and/or change the units used for any particular item.

To change the units used for display

- **Choose Units... from the View menu**



- **Click on the name of the set of units you wish to use or change**
- **Use the pop-up menus beside each type of unit to set its units type**
- **Click the OK button**

The units you choose will be saved with the Multiframe application for subsequent use.

Color

Specify which colors to use for the various diagrams, for the background of the window, for the clipped or masked members and for the rendering of the structure.

Status Bar

(Windows Only)
Makes the Status Bar visible or invisible.

Toolbar

(Windows Only)
See “Toolbar Submenu”

Size To Fit Submenu

The Size To Fit submenu commands allow you to change the view factor to fit the frame to the current window.

Frame

Fits the entire frame to the window.

Selection

Fit the selection to the window.

Clipping

Fit the active clipping region to the window.

Clipping Submenu

The Clipping submenu commands allow you to control the display and positioning of the clipping bars. Clipping allows you to define how much of the structure is visible at one time.

No Clipping

Turns off clipping if it is on. This hides the clipping bars and makes all members in the frame visible.

Clip Gray

Turns on clipping and makes the clipping mode gray. This means all members, which do not lie completely within the boundaries of the clipping bars, will be drawn in gray.

Clip Invisible

Turns on clipping and makes the clipping mode invisible. This means all members, which do not lie completely within the boundaries of the clipping bars, will be made invisible.

Clip To Frame

Turns on clipping if it wasn't already on and positions the clipping bars just outside the outermost limits of the frame in each direction. This means all members will be visible.

Clip To Window

Turns on clipping if it wasn't already on and positions the four clipping bars which are visible in the current view so that they lie just inside the boundaries of the window.

Clip To Selection

Turns on clipping if it wasn't already on and positions the clipping bars so that they lie just outside the maximum extents of the selected members in the window.

Masking Submenu

The Masking submenu commands are used to control the display of member in the frame. Masking allows you to define, which members are visible and which are invisible.

No Masking

Turns off masking if it is on. This makes all members in the frame visible.

Mask Gray

Turns on masking and makes the masking mode gray. This means all members, which have been masked out, will be drawn in gray.

Mask Invisible

Turns on masking and makes the masking mode invisible. This means all members, which have been masked out, will be made invisible.

Mask To Frame

Turns on masking if it wasn't already on and makes all of the members in the frame visible.

Mask To Window

Turns on masking if it wasn't already on and makes all the members, which lie completely inside the boundaries of the window visible.

Mask To Selection

Turns on masking and masks out all the members in the frame, which are not selected. This means the only members in the frame which will be visible are those which were selected.

Mask Out Selection

Turns on masking and masks out all the members in the frame, which are selected. This has the effect of hiding the selected members and leaving all remaining visible members visible.

Toolbar Submenu

(Windows Only)

The Toolbar submenu commands allow you to make selected toolbars visible or invisible.

File Toolbar

Makes the File toolbar visible or invisible. The File toolbar contains common file functions.

View Toolbar

Makes the View toolbar visible or invisible. The View Toolbar provides common view functions such as Zoom and Pan.

Generate Toolbar

Makes the Generate toolbar visible or invisible in the Frame Window. The Generate Toolbar allows you generate frames.

Joint Toolbar

Makes the Joint toolbar visible or invisible in the Frame Window. The Joint Toolbar allows you restrain joints.

Member Toolbar

Makes the Member toolbar visible or invisible in the Frame Window. The Member Toolbar allows you perform functions such as Add Member, Delete Member, and Sub-divide Member.

Actions Toolbar

Makes the Actions toolbar visible or invisible in the Plot Window. The Actions Toolbar allows you to view results of an analysis such as the Mz' and Deflection.

Load Case Toolbar

Makes the Load Case toolbar visible or invisible. The Loadcase toolbar provides a simple way to transverse the different loadcases.

Load Toolbar

Makes the Load Toolbar toolbar visible or invisible. The Load Toolbar provides common loading options.

Frame Menu

The Frame menu provides functions for creating and editing the frame.

Add Member

(Windows: Ctrl A / Macintosh: command A)

Add a member to the structure in the Frame window. Press the mouse button to position the first joint of the member and drag to the position of the second joint and release the button. Windows users have the option of just clicking on the first joint, then clicking on the second joint position to draw a member.

Delete Member

(Windows: Ctrl B)
Delete the selected members in the Frame window from the structure.

Subdivide Member

(Macintosh: command B)
Divide all selected members in the Frame window into a number of equally sized smaller members.

Joint Restraint

Restrain all the joints selected in the Frame window. A dialog box will appear allowing you to specify which type of restraint to apply.

Joint Mass

Additional mass added to a structure to add inertia for dynamic analysis.

Joint Spring

Prescribe a spring at all the joints selected in the Frame window. A dialog box will appear which allows you to specify the direction and stiffness of the spring.

Joint Displacement

Prescribe a displacement at all the joints selected in the Frame window. A dialog box will appear which allows you to specify the direction and magnitude of the displacement.

Joint Type

Set the selected joint type as rigid or pinned. Rigid joints transmit moment and pinned joints do not.

Joint Linking

Links a group of joints together so that they move together in response to static or dynamic loads.

Section Type

Set the section type for all the members selected in the Frame window. A dialog box will appear which allows you to choose the section to use for the members selected.

Member Type

Allows you to choose whether a member is standard, tension only or compression only.

Member Releases

(Windows Only)
Set the type of the selected members in the Frame window - members can be fixed or pinned at either or both ends.

Member Shear Area

Allows you to turn on or off the use of the optional calculation of deflection due to shear deformation.

Member Orientation

Set the orientation of the section for all the members selected in the Frame window. A dialog box will appear which allows you to rotate the section to the desired orientation.

Member Masses

Specify whether the mass of the selected members should be included when a dynamic analysis is performed. This does not affect the inclusion of a member's weight when a static analysis is carried out using self weight. That is controlled by the Member Self Weight item from the Load menu.

Member Labels

(Windows Only)
Allows you to edit the user defined label associated with each member.

Duplicate

(Windows: Ctrl D / Macintosh: command D)
Duplicates all the selected members in the frame a given number of times in a specified direction. A dialog allows you to enter the spacing in each direction and whether the duplicated members should be connected to the existing frame.

Rotate

Rotates all the selected joints in the frame a specified number of degrees about a specified axis. A dialog allows you to enter the number of degrees and the center of rotation.

Rescale

Multiplies the coordinates of all the selected joints in the frame by a specified scaling factor. This has the effect of rescaling the structure by the specified amount.

Extrude

Creates new members from all the selected joints in the frame in a specified direction. Usually used to generate columns from a drawn floor plan or to generate beams out from an existing column line.

Move

(Windows: Ctrl M / Macintosh: command M)
Allows you to either move the selected joints in the frame a specified distance or to move the origin to a new location.

Generate

Allows you to automatically generate a continuous beam, portal frame, bay and story frame or curved beam.

Renumber

Allows you to automatically renumber the joints and/or members in the structure. This is convenient for sorting joint and member numbers by direction after you have been making modifications to the frame's geometry.

Load Menu

All commands available in the Load menu act on the current load case.

Unload Joint

Remove all loads from the selected joints in the Load window.

Joint Load

Add a point load to each of the selected joints in the Load window. A dialog box will appear allowing you to specify a direction and magnitude for the load.

Joint Moment

Add a point moment to each of the selected joints in the Load window. A dialog box will appear allowing you to specify a direction and magnitude for the moment.

Unload Member

Remove all loads from the selected members in the Load window.

Global Dist'd Load

Add a distributed load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A global distributed load acts parallel to one of the global x, y or z axes.

Global Point Load

Add a point loads to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A global point load acts parallel to one of the global x, y or z axes.

Global Moment

Add a point moment to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load for the members selected. A global moment acts about one of the global x, y or z axes.

Local Dist'd Load

Add a local distributed load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local distributed load acts parallel to one of the local x', y' or z' member axes.

Local Point Load

Add a local point load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local point load acts parallel to one of the local x', y' or z' member axes.

Local Moment

Add a local point moment to each of the selected members in the Load window. A dialog box will appear which allows you to specify the magnitude, direction and position of the load. A local moment acts about one of the local x', y' or z' member axes.

Thermal Load

Add a thermal load to each of the selected members in the Load window. A dialog box will appear which allows you to specify the temperature, thermal coefficient, depth and direction of the load. A thermal load may either be uniform or a linear variation of temperature through the depth of the member.

Member Self Weight

Specify whether the self weight of the selected members should be included or ignored during a static analysis, which includes a self weight load case. Note that this does not affect the inclusion of member masses in a dynamic analysis, which is controlled by the Member Mass command from the Frame menu.

Dynamic Load

(Macintosh Only)
Allows you to apply a dynamic time varying load to selected joints in the Load window.

Display Menu

The Display menu contains commands for controlling the data, which is displayed in the graphics window.

Symbols

This command brings up a dialog box, which allows you to specify which symbols will be displayed in the Frame, Load and Plot windows. You can turn on or off the display of joint and member numbers, loads, restraints, member axes, masses and names of sections.

Data

See the "Data Submenu"

Results

See the "Results Submenu"

Actions

See the "Actions Submenu"

Stresses

See the "Stresses Submenu"

Deflection

Display the deflection for the current load case in the Plot window. If the current load case is a dynamic case, the deflection will represent a mode shape of the structure.

Animate

Animate the diagram in the frontmost window. In the Frame and Load windows, this command only operates in the 3D view and will display views of the structure at a range of viewing angles. Once the diagrams have been displayed you can animate the range of views by moving the mouse back and forth. In the Plot window this command will compute a series of diagrams showing the change as loading increases from zero to its prescribed level.

Macintosh users: If you turn on the QuickTime check box in the Animate dialog, the animation will be saved into a QuickTime movie file. The movie is displayed in the window with the standard QuickTime controller and you can play it if you wish. The movie will also be placed on the clipboard and may then be pasted into the scrapbook or any QuickTime compatible application program. You will need to have the QuickTime extension installed in your System Folder for these options to be available. Contact your Apple dealer for further details on QuickTime.

Windows users: Saving a QuickTime movie file will be implemented once QuickTime is implemented into the Windows operating system in late 1997.

Render

Display the frame complete with web and flange details. This display will help you visualize the orientation and section types for the frame. The rendering is not an exact display of the actual shape of the members in the structure but more a visual guide to the relative size and orientation of the sections in the frame.

You can interrupt the drawing of a rendered view of the structure by typing Command-Period (\cdot) on the Macintosh, or pressing the escape key on Windows.

Plot

Specify the precision to be used in the display of deflection diagrams in the Plot window, which action, if any, is to be overlaid onto the deflection diagram and what scaling factor should be applied to the current diagram.

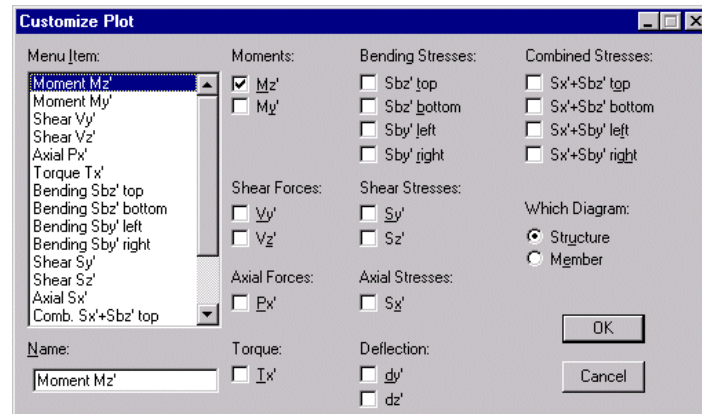
The overlaid action will be displayed as a color on the deflected shape. Red indicates a high value of the action relative to the rest of the structure while blue indicates a relatively low value.

Customise Plot

You can customise the display of diagrams in the Plot window to allow the display of one or more diagrams simultaneously. This applies to both global and local diagrams.

To choose which diagrams are displayed in the Plot window

- **Choose Customise Plot... from the Display menu**



A dialog will appear listing the menu items controlling the plot display and the actions, stresses and deflections which can be displayed.

- **Click on the name of the menu item you wish to change**
- **Click on the Member button**
- **Set the check boxes of the actions you wish to display in the local member diagrams**
- **Click on the Structure button**

- **Set the check boxes of the actions you wish to display on the diagram of the whole structure**
- **Click the OK button**

Data Submenu

The Data submenu controls, which table of data, will be displayed in the Data window. One table can be displayed and edited at a time.

Joints

Display a table of joint coordinates.

Members

Display a table of member data including joint numbers at the ends, section type, member type, length, slope and orientation.

Joint Loads

Display a table of joint load positions and magnitudes.

Member Loads

Display a table of member load positions and magnitudes.

Thermal Loads

Display a table of thermal load magnitudes.

Restraints

Display a table of joint restraints and prescribed displacements.

Linked Joints

Links a group of joints so that they move together in response to static or dynamic loads.

Springs

Display a table of joint spring stiffnesses.

Joint Masses

Display a table of joint masses.

Sections

Display a table of sections summarizing the use of sections in the frame. The table includes the number, length and mass of each type of section used.

Step Loads

Display a table of dynamic joint load positions and associated time series.

Results Submenu

The items in the Results sub-menu allow you to specify which table of results should be displayed in the Results window.

Displacements

Display the computed joint displacements for the current load case in the Result window.

Reactions

Display the computed joint reactions for the current load case in the Result window.

Member Actions

Display the computed member actions for the current load case in the Result window.

Member Details

Display the member actions at the number of points along a selected member.

Dynamics

Display the computed frequencies and periods of vibration from a dynamic analysis.

Actions Submenu

The items in the Actions submenu may be used to control which type of force or moment is displayed in the Plot window.

Moment M_z'

Display the computed bending moments about the local z' axis for the current load case in the Plot window.

Moment M_y'

Display the computed bending moments about the local y' axis for the current load case in the Plot window.

Shear V_y'

Display the computed shear forces in the local y' direction for the current load case in the Plot window.

Shear V_z'

Display the computed shear forces in the local z' direction for the current load case in the Plot window.

Axial P_x'

Display the computed axial forces for the current load case in the Plot window.

Torque T_x'

Display the computed torque about the local x' axis for the current load case in the Plot window.

Stresses Submenu

The items in the Stresses submenu may be used to control which type of stress is displayed in the Plot window.

Bending S_{bz}' top

Display the computed bending stress about the local z' axis at the top of each member for the current load case in the Plot window.

Bending S_{bz}' bottom

Display the computed bending stress about the local z' axis at the bottom of each member for the current load case in the Plot window.

Bending S_{by}' left

Display the computed bending stress about the local y' axis at the left of each member for the current load case in the Plot window.

Bending S_{by}' right

Display the computed bending stress about the local y' axis at the right of each member for the current load case in the Plot window.

Shear S_y'	Display the computed shear stress in the local y' direction for the current load case in the Plot window.
Shear S_z'	Display the computed shear stress in the local z' direction for the current load case in the Plot window.
Axial S_x'	Display the computed axial stress for the current load case in the Plot window.
Comb $S_x' + S_{bz}'$ top	Display the combined axial stress and bending stress about the local z' axis at the top of each member.
Comb $S_x' + S_{bz}'$ bottom	Display the combined axial stress and bending stress about the local z' axis at the bottom of each member.
Comb $S_x' + S_{by}'$ left	Display the combined axial stress and bending stress about the local y' axis at the left of each member.
Comb $S_x' + S_{by}'$ right	Display the combined axial stress and bending stress about the local y' axis at the right of each member.
Case Menu	The items in the Case menu control analysis and working with load cases.
Analyse	Carry out the analysis to compute all the deflections, moments and forces for the structure for all load cases. In Multiframe4D, a dialog will be displayed allowing you to specify the settings for the analysis.
Calculate	(Windows: Alt + = / Macintosh: command =) Calculate the value of the expressions in the CalcSheet.
Add Case	Add a new load case - enter load factors if desired. The Add Case menu is a sub menu of the Case menu, and is described in detail following this section on the Case menu.
Edit Case	The Edit Case command allows you to modify the attributes of the current load case.
Delete Case	The Delete Case command allows you to delete one or more loadcases.

Load Case 1 (or equivalent)

Display and allow editing of loads associated with load case 1 in the Load and Data windows. Display results for load case 1 in the Plot and Result windows.

Similarly for subsequent load cases.

Mode Shape 1

Display results associated with mode shape 1 from a dynamic analysis in Multiframe4D. Affects the display of deflections in the Plot and Result windows.

Load Case

(Windows: Ctrl L)
Dialog box, which allows you to choose a load case. Similarly for subsequent mode shapes.

Add Case Submenu

The Add Case submenu allows you to enter load factors if desired.

Self Weight

Adds a Self Weight Load Case

Static

Adds a Static Load Case

Static Combined

Combines and factors several load cases

Dynamic

Adds a Dynamic Load Case

Seismic

Adds a Seismic Load Case

Time Menu

The items in the Time menu control, which increment is selected in a time history analysis.

Maximum Envelope

Display's a Maximum Envelope of all increments in the current time history analysis case.

Minimum Envelope

Display's a Minimum Envelope of all increments in the current time history analysis case.

Absolute Envelope

Display's an Absolute Envelope of all increments in the current time history analysis case.

Time History

(Windows: Ctrl H)
(Windows Only)
Set the currently displayed time history increment.

Window Menu

The Windows menu provides window specific functionality.

Cascade

(Windows Only)
Displays all the Windows behind the active Windows.

Tile Horizontal

(Windows Only)
Layout all visible windows across the screen.

Tile Vertical

(Windows Only)
Layout all visible windows down the screen.

Arrange Icons

(Windows Only)
Rearranges the icons of any iconised window so that they are collected together at the bottom of the Multiframe program window.

Editing Layout

Makes the Frame, Data and Load windows all visible at once. This is useful when you are defining the structure and its loads and restraints.

Result Layout

Makes the Plot, Result and Load windows all visible. This is useful when you are examining the results of the analysis.

Calcs Layout

Makes the Plot, Load and CalcSheet windows visible. This is useful when you are doing calculations for a given member.

Frame

Makes the Frame window visible and brings it to the front.

Data

Makes the Data window visible and brings it to the front.

Load

Makes the Load window visible and brings it to the front.

Result

Makes the Result window visible and brings it to the front.

Plot

Makes the Plot window visible and brings it to the front.

CalcSheet

Makes the CalcSheet window visible and brings it to the front.

Tile

(Macintosh Only)
Lays out all of Multiframe's windows across the screen so that all of each window is visible at once.

Stack

(Macintosh Only)

Stacks up all of Multiframe's windows down the screen from top left to bottom right so that each window is visible and close to its maximum size.

Show All

(Macintosh Only)
Makes all of Multiframe's windows visible.

Hide All

(Macintosh Only)
Hides all of Multiframe's windows.

Help Menu

(Windows Only)
Provides access to an on-line help system.

Table of Contents

This command allows you to launch the table of contents of the Multiframe help file.

About Multiframe

Tells which version of Multiframe you are using and how many joints, members and forces etc. there are in the structure.

Chapter 4

Multiframe Analysis

This chapter describes the analysis methods used in Multiframe

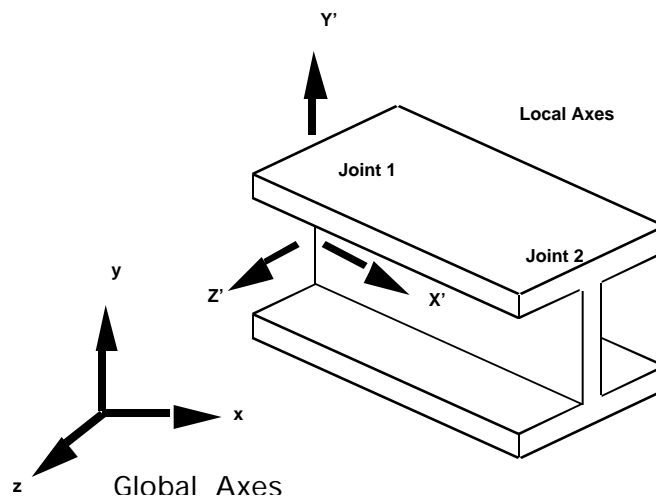
Method of Analysis

Multiframe uses the matrix stiffness method of solution for solving a system of simultaneous equations to determine the forces and deflections in a structure. Multiframe carries out a first order, linear elastic analysis to determine these forces and deflections. You should be familiar with the concepts and application of the matrix stiffness method before using this software.

The matrix stiffness method forms a stiffness matrix for each member of the structure and given a list of applied loading, solves a system of linear simultaneous equations to compute the deflections in the structure. The internal forces and reactions are then computed from these deflections. Multiframe does not take into account deformations due to shear action in deep beams or warping deformation due to torsion.

Axes and Sign Convention

Multiframe uses two coordinate systems for defining geometry and loading. The global coordinate system is a right handed x, y, z system with y always running vertically and x and z running horizontally. Gravity loads due to self weight are always applied in the negative y direction. To distinguish between local and global axes, Multiframe uses the ' suffix to indicate a local axis.



Each member in the structure is defined by the two joints at its ends. The local coordinate system is a right handed x, y, z system with the x axis running along the member from joint 1 to joint 2. The direction of the y' axis is specified by your setting of the section orientation. The orientation is the angle between the y' axis and a vertical plane passing through the ends of the member measured

from the vertical plane towards the y' axis as viewed from joint 2 looking towards joint 1.

Multiframe uses six degrees of freedom at each joint when performing its calculations (see the matrix on the next page). These comprise three displacements along the axes and three rotations about the axes at each joint.

The local element stiffness matrix K used by Multiframe is as follows:

$\frac{AE}{L}$	0	0	0	0	0	$-\frac{AE}{L}$	0	0	0	0	0
0	$\frac{12EI_z'}{L^3}$	0	0	0	$\frac{6EI_z'}{L^2}$	0	$\frac{12EI_z'}{L^3}$	0	0	0	$\frac{6EI_z'}{L^2}$
0	0	$\frac{12EI_y'}{L^3}$	0	$\frac{-6EI_y'}{L^2}$	0	0	0	$\frac{12EI_y'}{L^3}$	0	$\frac{6EI_y'}{L^2}$	0
0	0	0	$\frac{GJ}{L}$	0	0	0	0	0	$\frac{-GJ}{L}$	0	0
0	0	$\frac{-6EI_y'}{L^2}$	0	$\frac{4EI_y'}{L}$	0	0	0	$\frac{6EI_y'}{L^2}$	0	$\frac{2EI_y'}{L}$	0
0	$\frac{6EI_z'}{L^2}$	0	0	0	$\frac{4EI_z'}{L}$	0	$\frac{-6EI_z'}{L^2}$	0	0	0	$\frac{2EI_z'}{L}$
$-\frac{AE}{L}$	0	0	0	0	0	$\frac{AE}{L}$	0	0	0	0	0
0	$\frac{12EI_z'}{L^3}$	0	0	0	$\frac{-6EI_z'}{L^2}$	0	$\frac{12EI_z'}{L^3}$	0	0	0	$\frac{6EI_z'}{L^2}$
0	0	$\frac{12EI_y'}{L^3}$	0	$\frac{6EI_y'}{L^2}$	0	0	0	$\frac{12EI_y'}{L^3}$	0	$\frac{6EI_y'}{L^2}$	0
0	0	0	$\frac{-GJ}{L}$	0	0	0	0	0	$\frac{GJ}{L}$	0	0
0	0	$\frac{-6EI_y'}{L^2}$	0	$\frac{2EI_y'}{L}$	0	0	0	$\frac{6EI_y'}{L^2}$	0	$\frac{4EI_y'}{L}$	0
0	$\frac{6EI_z'}{L^2}$	0	0	0	$\frac{2EI_z'}{L}$	0	$\frac{-6EI_z'}{L^2}$	0	0	0	$\frac{4EI_z'}{L}$

A = Area, E = Young's Modulus, G= Shear Modulus, L = Length, J = Torsion Constant, I = Moment of Inertia

Where each element behaves according to the equation

$$F=Kx$$

Where F is the vector of applied loads, K is the stiffness matrix above and x is a vector of calculated displacements

$$F = \{P_{x1}, P_{y1}, P_{z1}, M_{x1}, M_{y1}, M_{z1}, P_{x2}, P_{y2}, P_{z2}, M_{x2}, M_{y2}, M_{z2}\}$$

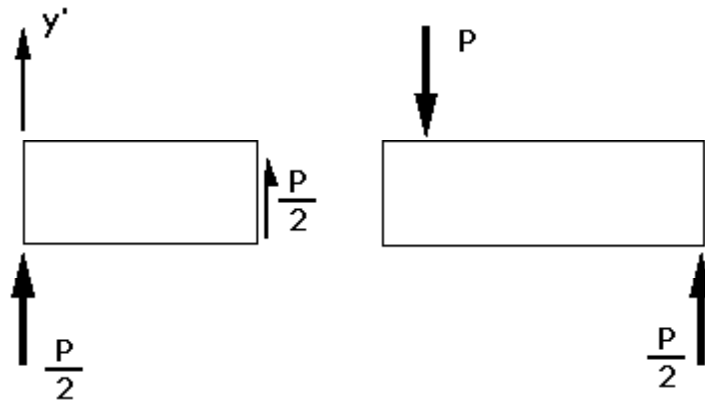
$$x = \{dx1, dy1, dz1, \theta_{x1}, \theta_{y1}, \theta_{z1}, dx2, dy2, dz2, \theta_{x2}, \theta_{y2}, \theta_{z2}\}$$

All relative to the local member coordinate system.

Note that the $I_{z'}$ and $I_{y'}$ above refer to the moment of inertia with respect to the local coordinate system. Since common engineering convention is to use I_x and I_y for the major and minor moments of inertia of a section, for a member with a section orientation of zero, $I_{z'}$ corresponds to the conventional I_x while $I_{y'}$ corresponds to the conventional I_y . The Multiframe Sections Library uses the conventional I_x, I_y notation for the two moments of inertia of the sections stored in the library.

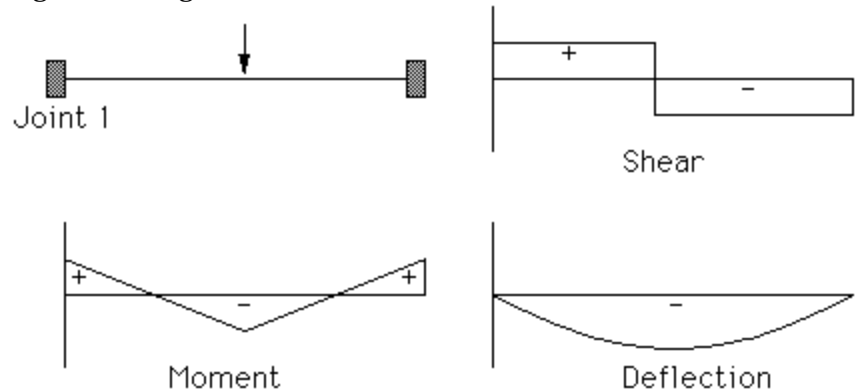
Member Actions

Multiframe computes member actions relative to the local member coordinate system. When calculating an action at an intermediate point along a member, Multiframe checks the free body diagram of the member to the left of the point of interest and uses the balance of forces at this point for the sign of the computed action. As an example, consider the shear force at a point on a simple beam subject to a central point load.



Following the above approach at the left hand portion of a beam, the sum of the shear forces is therefore positive as shown in the diagram above.

For the common case of a beam with joint 1 at the left hand end and joint 2 at the right hand end, a load acting downwards will be negative in magnitude and the forces will be as shown below.



When you specify the loads applied to a structure it will not be necessary to enter the signs of the load magnitudes since Multiframe will offer you a number of icons to choose from to specify a direction.

Dynamic Analysis

Multiframe4D allows you to perform a dynamic analysis of a frame. This analysis will determine the natural frequencies and mode shapes of the frame. These frequencies and mode shapes will reflect the interaction between the stiffness of the frame and the inertial effects of its mass and any joint masses you have applied to it.

Multiframe4D determines the natural dynamic response of the frame by using the subspace iteration method. This method solves the equation

$$[m]\{\ddot{u}\} + [k]\{u\} = \{0\}$$

Where $[m]$ is the mass matrix, $\{\ddot{u}\}$ is the vector of joint accelerations, $[k]$ is the stiffness matrix and $\{u\}$ is the vector of joint displacements. The solutions of this equation of undamped free vibration, of which there are a number, represent the natural responses of the frame. Multiframe will calculate the solutions corresponding to the highest periods of vibration (i.e. the lowest frequencies).

The local member mass matrix used by Multiframe4D varies according to whether a lumped or distributed mass model is chosen in the Analysis dialog.

The mass matrix for a lumped model is as follows

$\frac{mL}{2}$	0	0	0	0	0	$\underline{0}$	0	0	0	0	0
0	$\frac{mL}{2}$	0	0	0	$\underline{0}$	0	$\underline{0}$	0	0	0	$\underline{0}$
0	0	$\frac{mL}{2}$	0	$\underline{0}$	0	0	0	$\underline{0}$	0	$\underline{0}$	0
0	0	0	$\frac{mLI_0}{2A}$	0	0	0	0	0	$\underline{0}$	0	0
0	0	$\underline{0}$	0	$\underline{0}$	0	0	0	$\underline{0}$	0	$\underline{0}$	0
0	$\underline{0}$	0	0	0	$\underline{0}$	0	$\underline{0}$	0	0	0	$\underline{0}$
$\underline{0}$	0	0	0	0	0	$\frac{mL}{2}$	0	0	0	0	0
0	$\underline{0}$	0	0	0	$\underline{0}$	0	$\frac{mL}{2}$	0	0	0	$\underline{0}$
0	0	$\underline{0}$	0	$\underline{0}$	0	0	0	$\frac{mL}{2}$	0	$\underline{0}$	0
0	0	0	$\underline{0}$	0	0	0	0	0	$\frac{mLI_0}{2A}$	0	0
0	0	$\underline{0}$	0	$\underline{0}$	0	0	0	$\underline{0}$	0	$\underline{0}$	0

Chapter Four Multiframe Analysis

$$\left[\begin{array}{cccccc|cccccc} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{array} \right]$$

A = Area, m = Mass per unit length, Io= polar moment of inertia=Ix + Iy, L = Length

The mass matrix for a distributed mass model (also known as the consistent mass matrix in some texts) is as follows

$\frac{140mL}{420}$	0	0	0	0	0	$\frac{70mL}{420}$	0	0	0	0	0
0	$\frac{156mL}{420}$	0	0	0	$\frac{22mL^2}{420}$	0	$\frac{54mL}{420}$	0	0	0	$-\frac{13mL^2}{420}$
0	0	$\frac{156mL}{420}$	0	$-\frac{22mL^2}{420}$	0	0	0	$\frac{54mL}{420}$	0	$\frac{13mL^2}{420}$	0
0	0	0	$\frac{140mLI}{420A}$	0	0	0	0	0	$\frac{70mLI}{420A}$	0	0
0	0	$-\frac{22mL^2}{420}$	0	$\frac{4mL^3}{420}$	0	0	0	$-\frac{13mL^2}{420}$	0	$-\frac{3mL^3}{420}$	0
0	$\frac{22mL^2}{420}$	0	0	0	$\frac{4mL^3}{420}$	0	$\frac{13mL^2}{420}$	0	0	0	$-\frac{3mL^3}{420}$
$\frac{70mL}{420}$	0	0	0	0	0	$\frac{140mL}{420}$	0	0	0	0	0
0	$\frac{54mL}{420}$	0	0	0	$\frac{13mL^2}{420}$	0	$\frac{156mL}{420}$	0	0	0	$-\frac{22mL^2}{420}$
0	0	$\frac{54mL}{420}$	0	$-\frac{13mL^2}{420}$	0	0	0	$\frac{156mL}{420}$	0	$\frac{22mL^2}{420}$	0
0	0	0	$\frac{70mLIO}{420A}$	0	0	0	0	0	$\frac{140mL}{420A}$	0	0
0	0	$\frac{13mL^2}{420}$	0	$-\frac{3mL^3}{420}$	0	0	0	$\frac{22mL^2}{420}$	0	$\frac{4mL^3}{420}$	0
0	$-\frac{13mL^2}{420}$	0	0	0	$-\frac{3mL^3}{420}$	0	$-\frac{22mL^2}{420}$	0	0	0	$\frac{4mL^3}{420}$

A = Area, m = Mass per unit length, Io= polar moment of inertia=Ix + Iy, L = Length

In general, a distributed mass matrix will give a more accurate result. However, in some circumstances the distributed mass approach may not converge. In this case use the lumped mass approach. If in doubt, use both methods and compare the results between the two.

Capacity

The absolute maximum capacities for Multiframe are as follows

Number of joints	No limit
Number of members	No limit
Number of restraints and prescribed displacements	No limit
Number of springs	No limit
Number of load cases	300
Number of joint loads	No limit
Number of member loads	No limit
Number of thermal loads	No limit
Number of members connected at one joint	18

In practice, most of the above limits will be reduced by the amount of memory available at the time the program is running.

The amount of memory required is independent of the order in which the joints are numbered. Multiframe will automatically optimize the internal numbering of the joints to best use the memory available.

The actual size of the structure you will be able to solve will depend on the number of load cases and the geometric configuration of the structure. The more load cases you use the smaller structure you will be able to analyse. If you do not have enough memory to analyse a structure, try setting up the different load cases as separate files and analysing each load file separately.

References

You may find the following books useful to refer to if you need information on the matrix stiffness method of structural analysis.

- **Matrix Structural Analysis**
R L Sack, PWS-Kent, Boston 1989
- **Computer Methods of Structural Analysis**
F W Beaufait, Prentice Hall, New Jersey, 1970
- **Matrix Methods of Structural Analysis**
R K Livesley, The MacMillan Co, New York, 1964
- **Matrix Analysis for Structural Engineers**
N Willems & W M Lucas, Prentice Hall, New Jersey, 1968
- **Structural Dynamics, Theory and Computation**
Mario Paz, Van Nostrand, New York, 1991
- **Structural Dynamics, An introduction to computer methods**
R R Craig, J Wiley & Sons, New York, 1981
- **Meek J L, Matrix Structural Analysis**
McGraw Hill, New York, 1971
- **Przemieniecki, Theory of Matrix Structural Analysis**
Dover Publication, New York, 1968

Appendix A

Troubleshooting

This appendix describes some solutions to commonly encountered problems, which may occur with Multiframe.

Troubleshooting

Most Multiframe users, at some stage, experience an error message after analysis saying, "The solution does not make sense, please check the structure, restraints and section properties." Why does this happen and what should you do if it does?

During analysis, Multiframe checks to see if there is a zero on the diagonal of the stiffness matrix. If so, it will display a error message saying either "The structure has unrestrained degrees of freedom" or "Suspected unrestrained degrees of freedom". The program will select the suspect joints in the Frame window after displaying this message. Sections 2 and 3 below describe how to deal with this problem.

After carrying out the analysis Multiframe checks the results of analysis to verify that the solution makes sense. It does this by examining the deflections and seeing if any of them are infinite. If any deflections are bad it will display the "Solution doesn't make sense" message. There are four main causes for this condition and you should check each of them if you receive the above warning message.

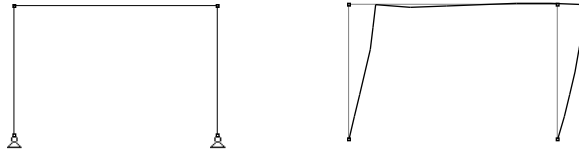
1. Check for bad section's properties

When Multiframe computes the stiffness matrix, it uses the properties in the Sections Library to determine the resistance of the sections to bending, axial and torsional deformation. If any of the key properties are zero, this will result in a zero in the matrix and consequently a bad solution. For a successful analysis, the following properties must be greater than zero:

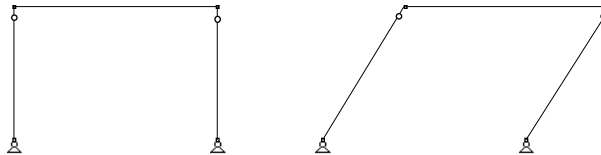
Area	Cross-sectional area of the section
I _x	Moment of inertia of the section about its strong axis
I _y	Moment of inertia of the section about its weak axis
J	St Venant torsion constant
E	Elastic or Youngs Modulus
G	Shear Modulus

2. Check for mechanisms

If you use member releases (or pinned members) or pinned joints in your structure, there is a possibility that you can create a mechanism, which allows the structure to deflect using a rigid-body mode of deformation. The simplest example is a portal frame with pinned restraint at the base of the columns under a lateral load. If all the members are rigid (no member release) analysis can proceed with no problems and the frame deforms by flexure of the columns.



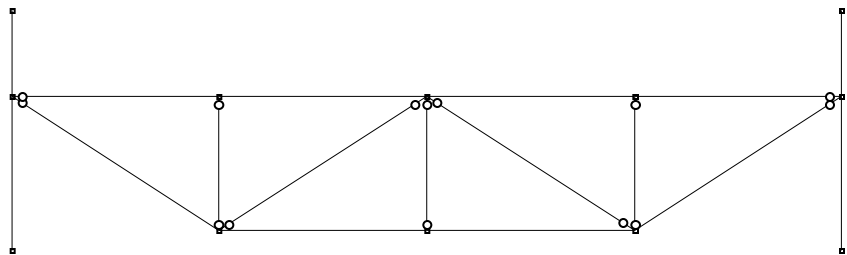
However, if we put member releases at the top of the columns, the frame is free to deflect by a mechanism allowing infinite lateral movement by rotation at the released ends of the columns. In this case, Multiframe will report the error.



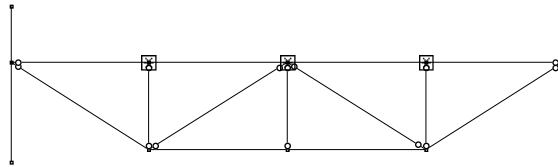
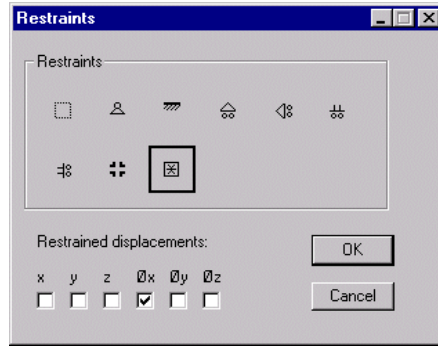
The solution is either to restrain the tops of the columns laterally or to use rigid members for the columns.

More subtle rigid body modes of deformation can develop if you release torsion (T_x) as well as bending moments (M_y and M_z) when you specify the member type. This means that it is possible to develop a torsional rigid mode of deformation in a structure. The simplest case of this is a member with torsional releases at both ends and a moment applied in the middle. In this case, there is no torsional restraint on the member and so the applied moment will cause an infinite torsional rotation.

A more complex case can develop where a group of members are connected to the rest of the structure by pin ended members and so the whole group is free to rotate to an infinite angle of rotation. A common example of this would be the top chord of a truss where the top chord is considered to be rigid but the rest of the members are pinned together.



In this case, because the top chord of the truss is pinned at its left hand end, pinned to the intermediate members along the truss, and also, pinned at its right hand end, the whole of the top chord is free to rotate about its longitudinal axis. This results in an infinite rotation of the top joints of the truss. There are three solutions to this problem. Either restrain these top joints against rotation by using a custom restraint to restrain θ_x to zero



Or use a member type that does not release the torsional moment, T_x or use the pinned joint type available in Multiframe3D version 1.6 and later. The pinned joint type sets all of the rotations of the joint to zero.

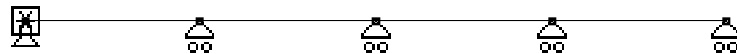
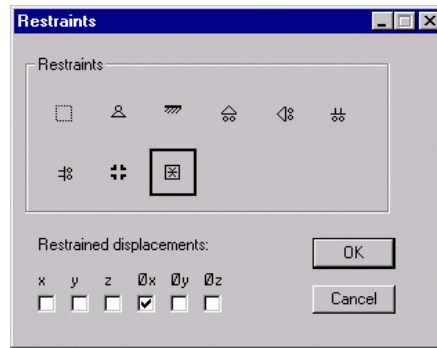
This will prevent these infinite torsional rotations without affecting the rest of the analysis.

3. Check for bad restraints

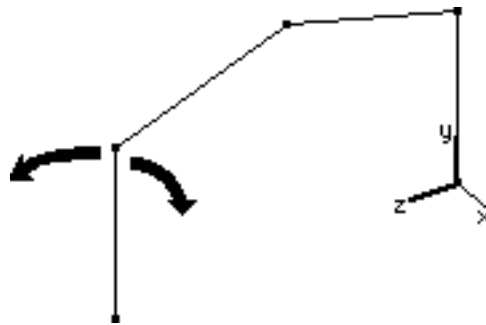
When analysing a structure, it is necessary to ensure that the whole structure cannot move in a single rigid body motion. For example, if you try to analyse a two dimensional frame using Multiframe 3D, you may find the "Solution doesn't make sense" warning occurring. (This will only be a problem if the structure lies in a plane other than the Front plane. In Multiframe3D version 1.6 and later, 2D structures with 2D loading in the Front plane will be automatically recognized and analysed correctly). This is probably because you have not restrained the structure sufficiently to stop the whole frame rotating about the global x axis passing through its supports. For example, a simple or continuous beam with pinned restraints will not analyse because it is free to twist about its longitudinal axis.



The solution to these problems is to add a rotational restraint to one or more of the restraints on the structure.



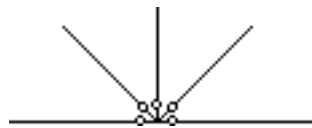
The same problem can occur with more complex 2D structures, in fact, a torsional rigid body mode of rotation will occur on any structure where the only restraints are pinned and all the restrained joints are co-linear.



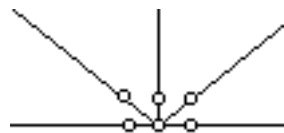
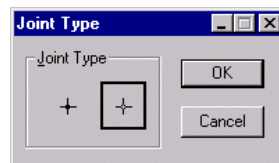
2D portal with two pinned restraints is free to rotate about the global z axis

4. Check for unrestrained joints

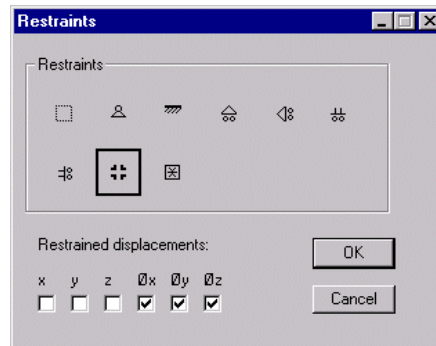
If you have a large number of members in your structure with member releases you may develop a situation where all of the members meeting at a joint have a pin at their ends. In this case there is no rotational restraint for the joint and it will result in an infinite rotation displacement of the joint.



The solution to this is to use the pinned joint type.



Alternatively, you can apply a joint restraint with only the three rotations restrained.



This will prevent the joint rotating but because the members are connected to the joint via a pinned end, they are free to deflect and rotate as usual.

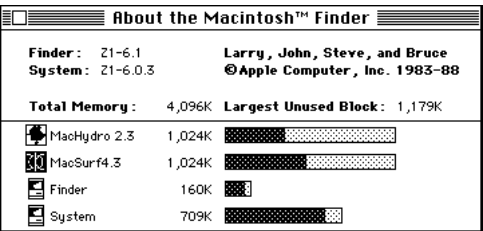
Appendix B

Macintosh Memory

This appendix describes how to optimise memory usage on Macintosh computers.

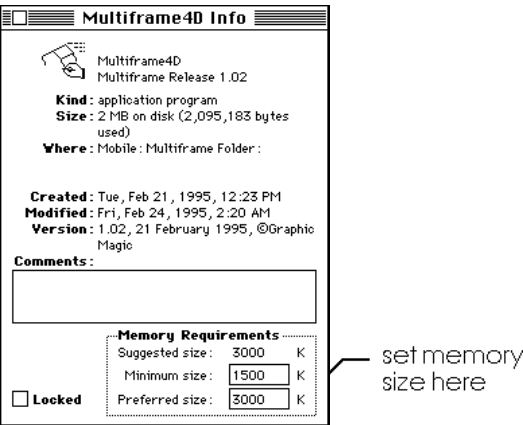
Macintosh Memory

With version 6 or later of the Macintosh system software, a capability was introduced to allow you to run more than one application at a time. This allows you to allocate a portion of the available memory to an application, and to keep starting up applications until no more memory is available.



For example, suppose you have a Macintosh with 16 megabytes of memory. Approximately 8 megabytes will be used by the System and Finder, and of the remaining 8 megabytes 4 could be used by Multiframe. This means you could be analysing with Multiframe, and at the same time be using a spreadsheet to do design calcs and a word processor to collate the data into a report.

If you want to increase the amount of memory available for an application, click on the application's icon when in the Finder and choose Get Info from the File menu. The Finder will display a dialog with a number at the bottom indicating the amount of memory to allocate for the application.



Multiframe normally requires 4000 k of space however if you are using Multiframe with a large structure or a large number of load cases, you will probably want to allocate 6000k or more of space.

The Multiframe suite tries to ensure that as much calculation as possible is done when the applications are running in the background. When any of the Multiframe applications are running a time consuming task, one of two cursors will be displayed. The

watch cursor is displayed when a task, which is running, requires all of the Macintosh's processing power, and in this case you cannot switch to another application until the task finishes.

For tasks that may take a very long time, the beach ball cursor is displayed, and you can switch to another application at any time and have the task continue to calculate in the background. Tasks of this type can normally be interrupted by typing Command Period.

To switch to another application, click on the desktop or in the window of another application. The most useful aspect of this is that while you are analysing a large structure, you can switch to another application and continue working with your Macintosh while Multiframe carries out the analysis in the background.

Virtual Memory

With the introduction of System 7 a new memory feature known as virtual memory is available to users with a Macintosh with a 68030 or better processor, or a Power Mac. Virtual memory allows you to use hard disk space to make more memory available to your computer. You turn on virtual memory using the Memory control panel. After you have allocated more memory to your machine using VM, you will need to resize the Multiframe partition using the Get Info option in the Finder as described above.

Appendix C

Analysing Trusses

This appendix describes how to analyse truss structures using Multiframe.

Analysing Trusses

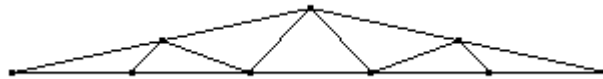
When you create a frame for analysis by Multiframe, it initially assumes that all of the connections between the members in the structure are fully rigid, that is, there is full moment transfer across all joints. To allow you to include joints which do not transmit moment, Multiframe allows you either to use the Member Type command to release the moments at one or both ends of a member or to use the Joint Type command to pin the ends of all of the members connected at a joint. If you are using Multiframe3D or Multiframe4D, you should use the Joint Type option. However, if you are using Multiframe2D, you will need to follow the procedures listed below.

Although you can release the end moments for as many members in a frame as you like, a problem arises if you release the moments at the ends of all of the members connected at a common joint. In this case, the joint is unrestrained against rotation, infinite rotations will be computed and the "Solution doesn't make sense" error message will appear. If you are trying to analyse a truss structure it is necessary to make all of the members in the structure pinned at both ends and consequently this problem arises for all the joints in the frame.

The solution to this problem is to restrain the joint against all rotations, this sets the rotations of the joint to zero, but the pins on the members allow them to deflect freely with no moment restraint.

If you are analysing a truss structure, you will probably find it easiest to follow these steps.

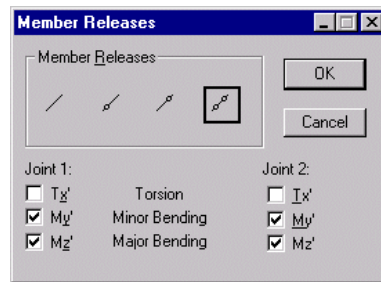
1. Draw the frame as usual



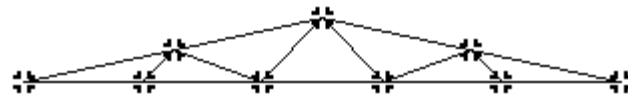
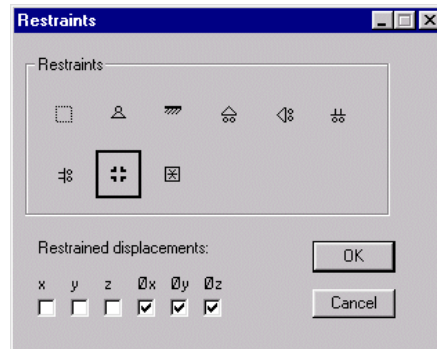
2. Use the Select All command from the Select menu to select all the members in the frame



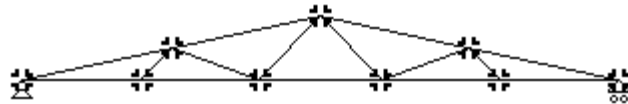
3. Use Member Type to set all the members to Pin-Pin ends



4. Use the Joint restraint command to restrain the rotations of all of the joints in the frame to zero.



5. Use pinned restraints as usual on the foundations of the truss.



Multiframe's analysis will accurately compute the deformations of the structure due to axial forces only and the resulting rotations of all the joints will be zero.

If you find it inconvenient having so many restraints displayed in the Frame window, you can always turn them off by using the Symbols... command from the Display menu.

Appendix D

Importing And Exporting Data

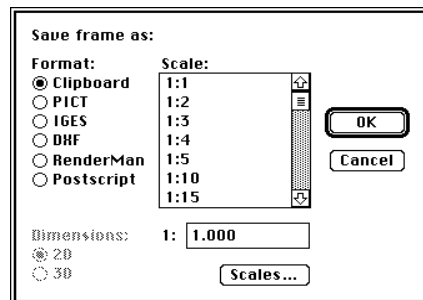
This appendix describes how to exchange data between Multiframe and other programs.

Exporting Data

Multiframe allows you to export numbers, drawings or animations to other programs. For example you may wish to take a table of numbers to a spreadsheet for further calculations or you might want to take graphics to a CAD or visualization package. The following paragraphs outline how to use the various options.

1. Clipboard/Pict File (Macintosh Only)

You can copy the current contents of the Frame, Load or Plot windows to the clipboard by using the Copy command from the Edit menu with the window in front. If you hold down the option key while choosing the Copy command, a dialog will appear allowing you to choose to send the graphics to the clipboard or a Pict file.



You can also choose what scale the drawing will be copied at. This is particularly useful for matching the output from Multiframe with other data you might be importing in another application. The main disadvantage of this option is that the picture is stored in 72 dot per inch precision, which may produce slightly jagged lines when output on higher resolution printers. If you need higher resolution you may want to consider using the Postscript option (see below).

2. DXF

Multiframe can export 2D or 3D DXF files, which can be read by AutoCAD and other CAD systems. Depending on what is displayed on screen, the file will contain either lines representing the members in the frame, or polygons representing the shapes of the sections which make up the members. Normally, lines only will be output. However, if you save a DXF file while rendering is turned on in the front window, the more complex polygon option will be used. The polygon option saves the detailed models of the members as 3DFACE entities. This is particularly useful for exporting data to rendering programs, which need a polygonal format to do a good job of rendering the frame.

3. PostScript

When you need very high resolution output for presentation purposes, the PostScript option will be most suitable. Hold down the Option key while choosing the Copy command from the Edit menu. Choose the PostScript option in the Copy dialog, and a scale

for your drawing and the contents of the front window will be saved into a PostScript file, you can then place this file into a high resolution drawing program such as Adobe Illustrator or Aldus Freehand or place it into a page layout program.

4. Renderman

If you wish to create very high quality rendered images of a frame using a rendering system such as Renderman, you will find the RIB (Renderman Interface Bytestream) file format useful. Hold down the Option key while choosing the Copy command from the Edit menu. Choose the Renderman option in the Copy dialog, and a scale for your drawing and the contents of the front window will be saved into a RIB file. You can then read this file into a rendering program such as Renderman, StrataVision, ShowPlace etc.

5. Copying and Pasting Tables of Numbers

If you wish to use the results from Multiframe in another program, you can copy and paste the tables, which are displayed in the Result window. To copy the table, click in the box at the top left corner of the table to select all the cells and then use the Copy command from the Edit menu. The table of numbers will be in "tab delimited format". You can then switch to your spreadsheet or word processor and paste in the data. You can also copy and paste any of the tables displayed in the Data window in a similar way.

6. Saving Text Files

Multiframe provides a capability for saving all of the data and results for a structure in a text file, which can be read in by another program. This is useful either for having a complete summary of the structure in one file, or for linking up with a post-processing program which could be written to receive the results from Multiframe and do further calculations. To save a text file, choose Save As... from the File menu, choose the Fortran Text option from the pop-up menu of file formats, and click Save.

7. Saving QuickTime Movies on the Macintosh

Multiframe provides a capability for creating a QuickTime movie from any animation, which can then be pasted into any QuickTime-aware program. This is useful for keeping a record of an analysis or for linking up with a post-processing program, which could be used for visualization. To save a QuickTime movie, hold down the Option key while choosing Animate from the Display menu. Multiframe will create and play the animation as usual, but will also place a QuickTime movie on the clipboard which can then be pasted into the ScrapBook or any other QuickTime aware application. You must have the QuickTime system extension turned on your computer to use this capability. If you also have the Wild Magic system extension installed, you will be able to view and play the movie in any Macintosh application, which can display a picture. This includes the Steel Designer report, which means you can paste an animation from Multiframe into the Steel Designer report.

Importing Data

1. DXF

Multiframe can read in 2D and 3D DXF files (Multiframe2D can only read 2D files). This means you can create the geometry for a frame in a CAD program and then save it in DXF format to be read into Multiframe. When reading a DXF file, Multiframe will extract all of the LINE, LINE3D and POLYLINE entities from the file and treat each line segment as a Multiframe member. Lines, which are within 0.2 inches (5mm) of each other, will be connected together.

2. Tables of numbers

You can paste tables of numbers into Multiframe in the data window by preparing a table of data in a spreadsheet and then pasting it into the selected cells in the Data window. When pasting in numbers, make sure that the numbers to be pasted have the same number of rows and columns as the selected area in the Data window.

3. Reading Text Files

Multiframe can read in a structure from a text file in the same format saved using the Save As command (see 6 above). This file could be created using a pre-processor program to create the geometry and other attributes for the structure.

Appendix E

Text File Format

This appendix describes a neutral text file format, which can be used to send data from Multiframe to a post processor or to prepare data for Multiframe, either manually or using a pre-processor.

Text File Format

Multiframe has the capability of saving and reading files in a text format. This facility is designed to allow pre and post processing programs to transfer information to and from Multiframe. The file may also be used as a convenient summary of the data in a human readable format.

The format of a Multiframe text file is as follows.

All numbers in the file are in Fortran style format. Integers are in I5 while real numbers are in F12.4 format. Data within the file is saved in groups, each group is preceded by the name of the group in upper case letters and followed by a number of lines with the relevant information.

The text file uses the units currently selected in Multiframe.

Job Title	Up to 80 characters describing the file and the date
METRIC or IMPERIAL Units	Number of joints
JOINTS	Start of data describing joints
I5	Number of joints
15,F12.4,F12.4,F12.4	Joint number, x coord, y coord, zcoord
	...repeat line for each joint
MEMBERS	Start of data describing members
I5	Number of members
15,15,15,15,15,15,F12.4	Member no, Node 1, Node 2, Section Group, Section Number, Member Orientation
	...repeat line for each member
SECTIONS	Start of section data
I5	Number of different sections used in frame
A15,15,15	Section Name, Group Number, Section Number
F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4	Mass, Area, Ixx, Iyy, J, Youngs Modulus, Shear Modulus
	...repeat line for each different section used in frame
RESTRAINTS	Start of restraints data
I5	Number of restraints
I5,I5,I5,I5,I5,I5,I5,I5,F12.4,F12.4,F12.4,F12.4,F12.4	Restraint #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz flag, x value, y value, z value, Øx value, Øy value, Øz value
	...repeat line above for each restraint
SPRINGS	Start of data for springs
I5	No of springs
15,15,15,15,15,15,15,15,F12.4,F12.4,F12.4,F12.4,F12.4	

Spring #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz flag,
x value, y value, z value, Øx value, Øy value, Øz value
... repeat line above for each spring

LOADS
15 Start of loads
Number of load cases

LOAD CASE
A15 Start of load case
Name of load case

JOINT LOAD
15 Start of joint loads for this load case
15,15,15,15,15,15,15,15,F12.4,F12.4,F12.4,F12.4 Number of joint loads in this load case
Load #, Joint #, x flag, y flag, z flag, Øx flag, Øy flag, Øz flag, x
value, y value, z value, Øx value, Øy value, Øz value
... repeat line above for each joint load in this case

MEMBER LOAD
15 Start of member loads for this load case
15,15,15,F12.4,F12.4,F12.4,F12.4 Number of member loads in this load case
Load #, Member #, Load Type, Left dist, right dist, left
magnitude, right magnitude
... repeat line above for each member load in this case

THERMAL LOAD
15 Start of thermal load for this load case
15,15,F12.4,F12.4,F12.4,F12.4 Number of thermal loads in this load case
Load #, Member #, top temp, bottom temp, thermal coeff,
depth
... repeat line above for each thermal load in this case
...repeat for each thermal load in this case

RESULTS
Start of results of analysis

LOAD CASE
A15 Start of results for load case
Load case name

DISPLACEMENTS &
REACTIONS Start of displacements and reactions
15,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4
Joint 3, x disp, ydisp, z disp, Øx, Øy, Øz, Pxreact, Pyreact,
Pzreact, Mx, Myreact, Mzreact
...repeat line for each joint

MEMBER ACTIONS
15,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4,F12.4
Start of local member actions
Member #, Fx1, Fy1, Fz1, Mx1, My1, Mz1, Fx2, Fy2, Fz2, Mx2,
My2, Mz2
...repeat each line for each member
...repeat for results for each load case




















END
End of file

Following is a summary of the various flags and codes used in the
text file

RESTRAINTS and SPRINGS xflag, yflag, zflag, Øxflag, Øyflag,
Øzflag

One of these should be 1 and the others zero to indicate which degree of freedom is being restrained. Only 1 degree of freedom can be set at a time for non-zero restraints.

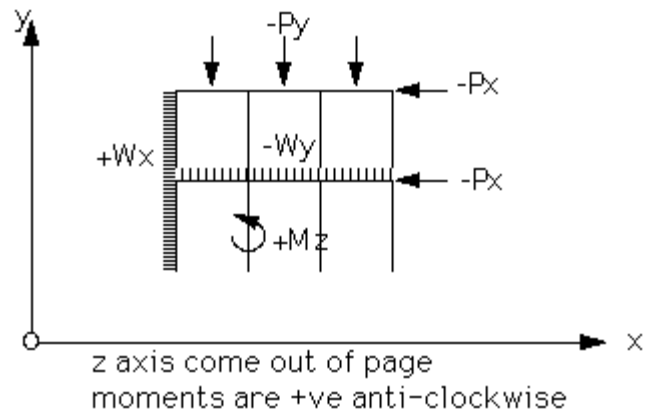
As an example, the following tables indicate the flags appropriate to the icons in the Front view.

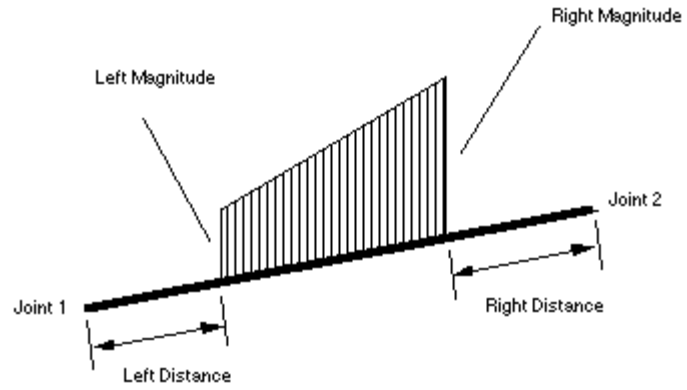
	x	y	z	θ_x	θ_y	θ_z			x	y	z	θ_x	θ_y	θ_z
	1	1	1	0	0	0			0	1	0	0	0	0
	1	1	1	1	1	1			1	0	0	0	0	0
	0	1	0	0	0	0			0	0	0	0	0	1
	1	0	0	0	0	0								
	x	y	z	θ_x	θ_y	θ_z			x	y	z	θ_x	θ_y	θ_z
	0	1	0	1	1	1			0	1	0	0	0	0
	1	0	0	1	1	1			1	0	0	0	0	0
	0	0	0	1	1	1			0	0	0	0	0	1

Multiframe load codes

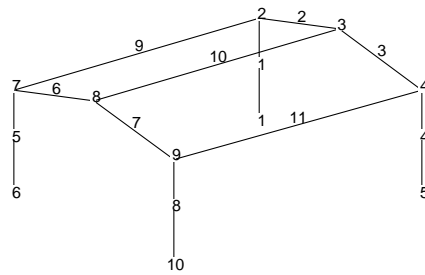
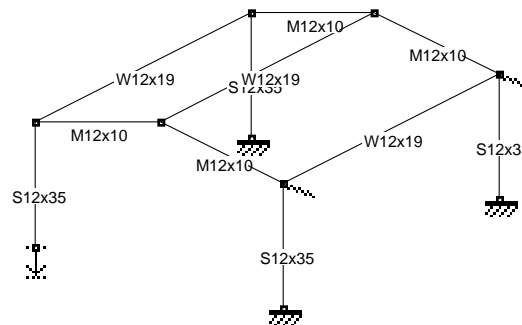
{global load codes}	{local load codes}
{Point loads}	{Point loads}
Px=3;	Px'=-3;
Py=13;	Py'=-13;
Pz=23;	Pz'=-23;
{Distributed loads}	{Distributed loads}
Wx=4;	Wx'=-4;
Wy=14;	Wy'=-14;
Wz=24;	Wz'=-24;
{Moments}	{Moments}
Mx=5;	Tx'=-5;
My=15;	My'=-15;
Mz=25;	Mz'=-25;

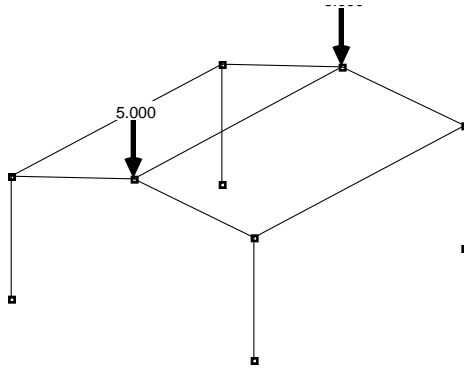
Load conventions and coordinate systems



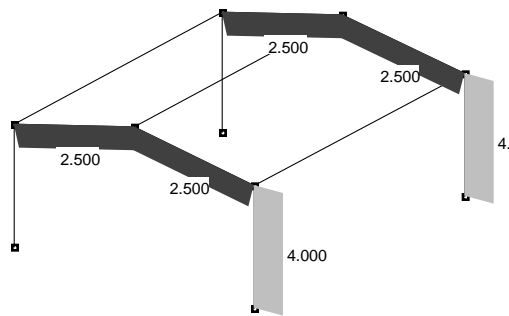


Multiframe Text Example File





TextFileFrame Load Case 1



TextFileFrame Load Case 2

TextFileFrame.text Fri, Apr 24, 1992

IMPERIAL

JOINTS

10

1	0.0000	0.0000	0.0000
2	0.0000	9.0000	0.0000
3	10.0000	11.1256	0.0000
4	20.0000	9.0000	0.0000
5	20.0000	0.0000	0.0000
6	0.0000	0.0000	25.0000
7	0.0000	9.0000	25.0000
8	10.0000	11.1256	25.0000
9	20.0000	9.0000	25.0000

10

MEMBERS

11

1	1	2	3	16	0.0000
2	2	3	2	4	0.0000
3	3	4	2	4	0.0000
4	5	4	3	16	0.0000
5	6	7	3	16	0.0000
6	7	8	2	4	0.0000
7	8	9	2	4	0.0000
8	10	9	3	16	0.0000
9	7	2	1	253	0.0000
10	8	3	1	253	0.0000
11	9	4	1	253	0.0000

SECTIONS

3

Section	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
S12x35	114.8156	10.3007	228.9832	9.8693	1.0799	29000.6935	11150.2663								
M12x10	32.8044	2.9402	61.5955	0.9939	0.0300	29000.6935	11150.2663								
W12x19	62.3285	5.5704	129.9905	3.7597	0.1800	29000.6935	11150.2663								

RESTRAINTS

4

1	1	1	1	1	1	1	1	0.0000	0.0000	0.0000	0.0000	0.0000
2	5	1	1	1	1	1	1	0.0000	0.0000	0.0000	0.0000	0.0000

Appendix E Text File Format

```

      3 10 1 1 1 1 1 1 0.0000 0.0000 0.0000 0.0000 0.0000
0.0000
      4 6 0 1 0 0 0 0 0.0000 -1.0000 0.0000 0.0000 0.0000
0.0000
SPRINGS
      2
      1 4 1 0 0 0 0 0 1.0000 0.0000 0.0000 0.0000 0.0000
0.0000

      2 9 1 0 0 0 0 0 1.0000 0.0000 0.0000 0.0000 0.0000
0.0000
LOADS
      2
LOAD CASE
Load Case 1
JOINT LOAD
      2
      1 3 0 1 0 0 0 0 0.0000 -5.0000 0.0000 0.0000 0.0000
0.0000
      2 8 0 1 0 0 0 0 0.0000 -5.0000 0.0000 0.0000 0.0000
0.0000
MEMBER LOAD
      0
THERMAL LOAD
      0
LOAD CASE

Load Case 2
JOINT LOAD
      0
MEMBER LOAD
      6
      1 2 -14 0.0000 0.0000 2.5000 2.5000
      2 3 -14 0.0000 0.0000 2.5000 2.5000
      3 6 -14 0.0000 0.0000 2.5000 2.5000
      4 7 -14 0.0000 0.0000 2.5000 2.5000
      5 4 4 0.0000 0.0000 -4.0000 -4.0000
      6 8 4 0.0000 0.0000 -4.0000 -4.0000
THERMAL LOAD
      0

RESULTS
LOAD CASE
Load Case 1
DISPLACEMENTS AND REACTIONS
      1 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 2.2274 2.5446
0.0426 -0.4825 0.0035 -9.8863
      2 -0.0338 -0.0009 0.1492 0.1749 -0.0213 -0.0015 0.0000 0.0000
0.0000 0.0000 0.0000 0.0000
      3 -0.0014 -0.1716 0.1358 0.1272 -0.0086 -0.0005 -0.0000 -0.0000
-0.0000 0.0000 0.0000 0.0000

      4 0.0309 -0.0009 0.0089 0.0011 -0.0062 0.0035 -0.0309 -0.0000
-0.0000 0.0000 -0.0000 0.0000
      5 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 -2.1622 2.4827
-0.0213 -0.1002 0.0010 9.4208
      6 -0.4120 -1.0000 -0.2251 0.1986 -0.0202 -0.0411 0.0000 1.4778
0.0000 -0.0000 0.0000 -0.0000
      7 -0.3345 -1.0005 0.1492 0.1986 -0.0202 -0.0411 0.0000 -0.0000
-0.0000 0.0000 0.0000 0.0000

      8 -0.3696 -0.8375 0.1358 0.1272 -0.0079 0.3159 -0.0000 -0.0000
0.0000 0.0000 -0.0000 0.0000
      9 -0.1930 -0.0013 0.0089 0.0011 -0.0064 0.2085 0.1930 0.0000
0.0000 -0.0000 0.0000 0.0000
      10 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 -0.2273 3.4949
-0.0213 -0.1002 0.0010 -17.6279
MEMBER ACTIONS
      1 2.5446 -2.2274 0.0426 0.0035 0.4825 -9.8863 -2.5446 2.2274
-0.0426 -0.0035 -0.8659 -10.1601

      2 2.6910 1.9920 0.0210 0.0002 -0.1149 10.1598 -2.6910 -1.9920
-0.0210 -0.0002 -0.0996 10.2054
      3 2.6723 -1.9797 0.0213 0.0005 -0.1010 -10.2023 -2.6723 1.9797
-0.0213 -0.0005 -0.1173 -10.0368
      4 2.4827 2.1622 -0.0213 0.0010 0.1002 9.4208 -2.4827 -2.1622
0.0213 -0.0010 0.0915 10.0388
      5 1.4778 -0.0000 0.0000 0.0000 0.0000 -0.0000 -1.4778 0.0000
-0.0000 -0.0000 -0.0000 0.0000

      6 0.3240 1.4794 0.0216 0.0003 -0.1196 0.0004 -0.3240 -1.4794
-0.0216 -0.0003 -0.1013 15.1241
      7 0.7492 -3.4042 0.0212 0.0005 -0.1001 -15.1272 -0.7492 3.4042
-0.0212 -0.0005 -0.1170 -19.6751
      8 3.4949 0.2273 -0.0213 0.0010 0.1002 -17.6279 -3.4949 -0.2273
0.0213 -0.0010 0.0915 19.6731
      9 0.0216 -0.0366 -0.0093 0.0004 0.1169 -0.0251 -0.0216 0.0366
0.0093 -0.0004 0.1158 -0.8899

```

Appendix E Text File Format

10	-0.0004	-0.0000	-0.0157	-0.0031	0.1971	-0.0005	0.0004	0.0000
0.0157	0.0031	0.1964	-0.0000					
11	0.0001	0.0093	-0.0092	-0.0020	0.1154	0.1163	-0.0001	-0.0093
0.0092	0.0020	0.1156	0.1163					
LOAD CASE								
Load Case 2								
DISPLACEMENTS AND REACTIONS								
1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	-8.1758	-23.9131
-0.0503	-0.9305	-0.0332	13.4469					
2	-0.1168	0.0086	0.1907	0.1827	0.2050	0.2610	-0.0000	0.0000
0.0000	-0.0000	0.0000	0.0000					
3	-0.2978	0.9501	-0.0073	-0.2147	0.0999	-0.0842	0.0000	-0.0000
0.0000	-0.0000	-0.0000	0.0000					
4	-0.4781	0.0094	-0.0106	-0.0015	0.0782	0.0770	0.4781	-0.0000
0.0000	0.0000	0.0000	0.0000					
5	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	43.6556	-26.0598
0.0251	0.1189	-0.0127	-149.3328					
6	6.0137	-1.0000	-0.1786	0.1959	0.2213	2.5272	0.0000	-20.0706
0.0000	0.0000	-0.0000	-0.0000					
7	1.2501	-0.9927	0.1907	0.1959	0.2213	2.5272	0.0000	-0.0000
-0.0000	-0.0000	-0.0000	-0.0000					
8	0.6043	2.0743	-0.0073	-0.2146	0.0974	-0.1494	0.0000	0.0000
0.0000	-0.0000	0.0000	-0.0000					
9	0.1750	0.0108	-0.0106	-0.0015	0.0789	-0.4912	-0.1750	-0.0000
-0.0000	-0.0000	0.0000	-0.0000					
10	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	36.2179	-29.9577
0.0251	0.1189	-0.0128	-65.0523					
MEMBER ACTIONS								
1	-23.9131	8.1758	-0.0503	-0.0332	0.9305	13.4469	23.9131	-8.1758
0.0503	0.0332	-0.4782	60.1356					
2	-12.9606	-21.7098	-0.0241	0.0016	0.1299	-60.1136	12.9606	-3.8490
0.0241	-0.0016	0.1161	-31.1856					
3	-13.3874	-1.7513	-0.0253	-0.0008	0.1215	31.1849	13.3874	-23.8075
0.0253	0.0008	0.1373	81.5601					
4	-26.0598	-43.6556	0.0251	-0.0127	-0.1189	-149.3328	26.0598	7.6552
-0.0251	0.0127	-0.1072	-81.5656					
5	-20.0706	-0.0000	0.0000	-0.0000	0.0000	-0.0000	20.0706	0.0000
-0.0000	0.0000	-0.0000	-0.0000					
6	-4.1813	-19.6129	-0.0262	0.0017	0.1462	-0.0220	4.1813	-5.9459
0.0262	-0.0017	0.1217	-69.8395					
7	-6.2564	3.7271	-0.0250	-0.0008	0.1186	69.8402	6.2564	-29.2859
0.0250	0.0008	0.1365	98.9126					
8	-29.9577	-36.2179	0.0251	-0.0128	-0.1189	-65.0523	29.9577	0.2175
-0.0251	0.0128	-0.1073	-98.9071					
9	-0.0262	-0.0170	0.0122	-0.0220	-0.1434	0.0287	0.0262	0.0170
-0.0122	0.0220	-0.1606	-0.4528					
10	0.0013	0.0002	0.0187	0.0006	-0.2349	0.0031	-0.0013	-0.0002
-0.0187	-0.0006	-0.2322	0.0013					
11	-0.0002	-0.0109	0.0117	0.0055	-0.1461	-0.1365	0.0002	0.0109
-0.0117	-0.0055	-0.1468	-0.1366					
END								

Appendix F

Using Spreadsheets With Multiframe

This appendix explains how you can effectively use spreadsheets with Multiframe to simplify the generation of frames and interpretation of the results of a Multiframe analysis.

Spreadsheets with Multiframe

You are probably aware that you can choose Copy from the Edit menu to copy a Multiframe picture into a word processor or drawing program. However, you may not be aware that you can also copy tables of data from Multiframe into a spreadsheet or copy and paste information from a spreadsheet into Multiframe.

The basics of copying data in Multiframe are to select the area of the table to be copied (by dragging the mouse or shift-clicking in the usual way, see the Multiframe manual for more details) and then copy the data to the clipboard using the Copy command from the Edit menu. You can then switch to or start up your spreadsheet, click at the location where you want to paste the data, and then use the Paste command to place the data into the spreadsheet. You can copy data from any of the tables in the Data and Result windows.

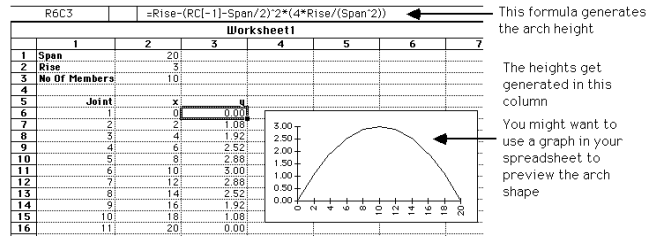
To paste data into Multiframe you reverse the procedure, selecting the data to be copied in the spreadsheet, choosing the Copy command in the Edit menu, and then switching to Multiframe, selecting the area to paste it into, and then using the Paste command. You can paste data into the Joint, Member, Joint Load and Member Load tables in the Data window.

Example 1 - Generating An Arch

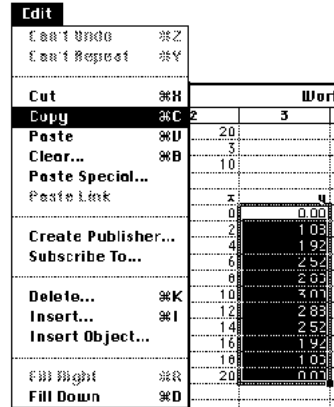
As one example of generating data in a spreadsheet and pasting it into Multiframe, consider the problem of analysing a parabolic arch. To draw or type in the geometry would be very time-consuming and difficult to modify if you wanted to consider several options for the shape of the arch. You can, however, use a spreadsheet to generate the geometry of the arch and then paste the coordinates into the Data table in Multiframe.

First, you would generate a continuous beam in Multiframe with the appropriate number of spans and then copy and paste the x column into the table in the spreadsheet. Next, write a formula for the y coordinate (or height) of the arch in the column adjacent to the data you have pasted in. You can then produce the y coordinate for each joint automatically.

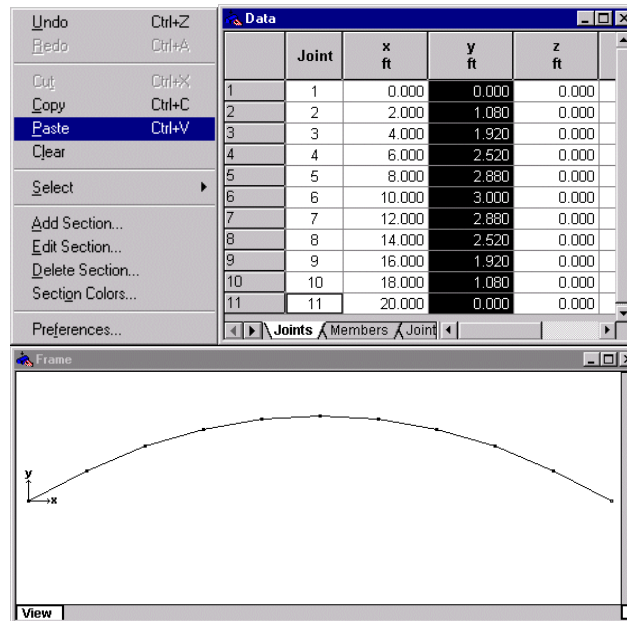
Appendix F Using Spreadsheets With Multiframe



Once you have generated the coordinates you can then copy the column of y coordinates from the spreadsheet and paste them into the y column in Multiframe.



In your spreadsheet



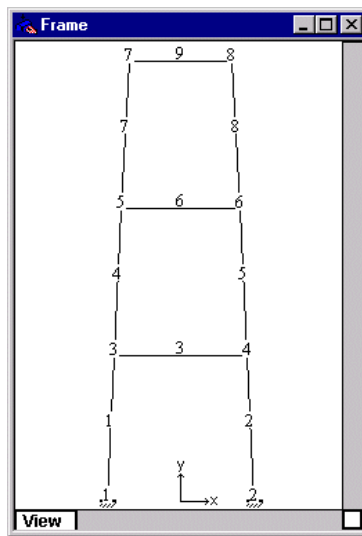
This will automatically generate the correct geometry for the arch which will be displayed in the other Multiframe windows.

If you have applied the section types, restraints and loads to the structure, these will remain intact when you paste in the new geometry, and you can immediately re-analyse to determine the results of the new shape. You can then investigate different alternatives by generating different shapes in the spreadsheet and pasting them into Multiframe. One important point to note is that

pasting the coordinates in requires that the joints in Multiframe are listed in the same order as the joints in the spreadsheet. You will find the Renummer... command in Multiframe useful for ensuring that this is the case. A second point to note is that you can select just a part of a column or range of columns in Multiframe, you do not necessarily have to generate coordinates for the whole structure.

Example 2 - Generating Loads

Generating the loads for a structure is often the most time-consuming part of setting up a structural model. For example, if you have a structure subject to wind loads which depend on the height of the members above the ground, there may be hundreds of calculations required to determine the magnitudes of these loads. However, you can use a spreadsheet, in particular a spreadsheet's ability to look up values from tables, to automate the generation of loads.



If a load depends on the height of a member, you will first need to calculate the height of each member. You can do this by using the Member table from the Data window to find out which joints are at the ends of each member and the Joint table to find out the coordinates of these joints. Start by pasting these two tables into different areas of your spreadsheet and defining arrays, which refer to these two tables.

<i>JointData</i>			
Joint	x	y	
1	-2.5	0	
2	2.5	0	
3	-2.25	5	
4	2.25	5	
5	-2	10	
6	2	10	
7	-1.75	15	
8	1.75	15	

<i>MemberData</i>			
Member	Joint1	Joint2	
1	1	3	
2	4	2	
3	3	4	
4	3	5	
5	6	4	
6	5	6	
7	5	7	
8	8	6	
9	7	8	

Next, add a column to the Member array and put a formula in it to look up the y coordinates of the joints and use them to find out the height of the mid-point of the member. Then add another column and put a formula in it to calculate the load magnitude from the member midpoints (calculated values are shown in the illustrations below in italics).

Appendix F Using Spreadsheets With Multiframe

R14C5 $= (VLOOKUP(RC[-2], JointData, 3) + VLOOKUP(RC[-1], JointData, 3)) / 2$ ← Formula to look up joint coordinates from the JointData array and calculate the member mid-point

LoadCalcs										
1	2	3	4	5	6	7	8	9	10	11
JointData					Height 15					
Joint					MaxLoad 25					
		x	y		LoadData					
1	1	-2.5	0		Member	Type	L	R	L Mag	R Mag
2	2	2.5	0		1	Wy	0	0	2.5	2.5
3	3	-2.25	5		4	Wy	0	0	7.5	7.5
4	4	2.25	5		7	Wy	0	0	12.5	12.5
5	5	-2	10							
6	6	2	10							
7	7	-1.75	15							
8	8	1.75	15							
MemberData										
Member	Joint1	Joint2	Mid Point	Load Value						
1	1	3	2.5	4.17						
2	4	2	2.5	4.17						
3	3	4	5	8.33						
4	3	5	7.5	12.50						
5	6	4	7.5	12.50						
6	5	6	10	16.67						
7	5	7	12.5	20.83						
8	8	6	12.5	20.83						
9	7	8	15	25.00						

This last column of the Member Data array now contains the required load magnitude for each member in the structure. The next step is to apply the distributed load to all of the appropriate members in Multiframe and then copy and paste the Member Load table from the Data window in Multiframe into a new array in your spreadsheet.

LoadData

Loads						
Member	Type	L	R	L Mag	R Mag	
1	Wy	0	0	0	0	
4	Wy	0	0	0	0	
7	Wy	0	0	0	0	

You can now put a formula for the load magnitude into the last two columns of the load array by looking up the required load magnitude that you have calculated for each member.

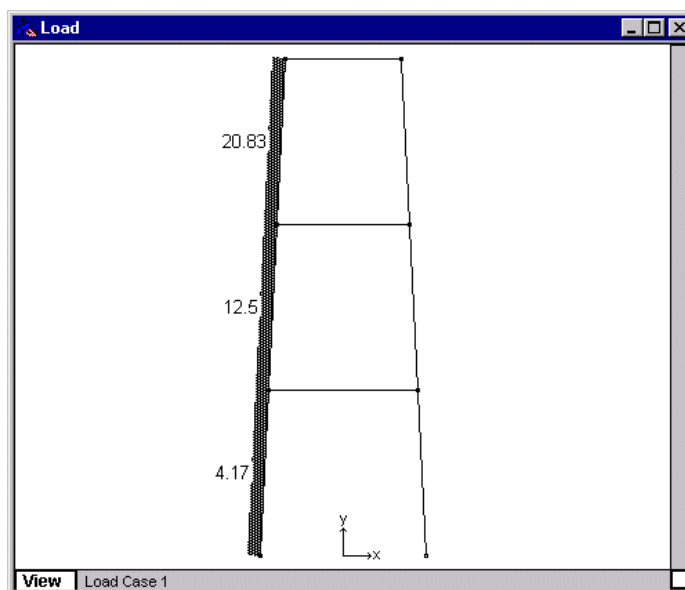
$=VLOOKUP(RC[-4], MemberData, 5)$ ← This formula looks up the required load for each member from the Load Value column in the MemberData array

LoadCalcs										
1	2	3	4	5	6	7	8	9	10	11
JointData					Height 15					
Joint					MaxLoad 25					
		x	y		Member	Type	L	R	L Mag	R Mag
1	1	-2.5	0		1	Wy	0	0	-4.17	-4.17
2	2	2.5	0		4	Wy	0	0	-12.50	-12.50
3	3	-2.25	5		7	Wy	0	0	-20.83	-20.83
4	4	2.25	5							
5	5	-2	10							
6	6	2	10							
7	7	-1.75	15							
8	8	1.75	15							
MemberData										
Member	Joint1	Joint2	Mid Point	Load Value						
1	1	3	2.5	4.17						
2	4	2	2.5	4.17						
3	3	4	5	8.33						
4	3	5	7.5	12.50						
5	6	4	7.5	12.50						
6	5	6	10	16.67						
7	5	7	12.5	20.83						
8	8	6	12.5	20.83						
9	7	8	15	25.00						

Once you have done that, you can copy and paste the load magnitudes from the load array back into the magnitude columns of the Member Load table in Multiframe.

Undo	Ctrl+Z
Redo	Ctrl+Y
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Clear	
Select	▶
Add Section...	
Edit Section...	
Delete Section...	
Section Colors...	
Preferences...	

This will update the member loads you have applied to their required values. Keep in mind the direction of these loads, you may need to use a negative sign in your spreadsheet to ensure that the magnitudes pasted into Multiframe point in the right direction.



This may sound like a rather long process but once you have set it up once you can adapt the spreadsheet for use with other structures. Also, once you get used to using the VLOOKUP command in your spreadsheet you will find it useful for many other operations (see quantities example below).

Example 3 - Calculating Quantities & Costs

When you generate a structure in Multiframe it displays a table of the sections you have used in the Sections table in the Data window.

	Section	Group	Length m	No. Used	Total Length m	Mass/L kg/m	Total Mass kg
1	TS8x8x1/2	Sq. Tube	5.006	6	30.037	72.689	2183.388
2	C8x11.5	C	4.500	1	4.500	17.112	77.004
3	C8x11.5	C	4.000	1	4.000	17.112	68.448
4	C8x11.5	C	3.500	1	3.500	17.112	59.892
5	L4x4x5/8	Angle	6.897	1	6.897	23.362	161.115
6	L4x4x5/8	Angle	6.562	1	6.562	23.362	153.304
7	L4x4x5/8	Angle	6.250	1	6.250	23.362	146.010

You can copy this table to a spreadsheet for calculations of quantities and/or costs. If you copy the whole table you can easily calculate the total weight of the structure by summing the values in the Total Weight Column.

SectionData							
Section	Group	Length	No Used	Total Length	Mass	Total Mass	
TS8x8x1/2	Sq. Tube	5.0	6.0	30.0	72.7	2183.4	
C8x11.5	C	4.5	1.0	4.5	17.1	77.0	
C8x11.5	C	4.0	1.0	4.0	17.1	68.4	
C8x11.5	C	3.5	1.0	3.5	17.1	59.9	
L4x4x5/8	Angle	6.9	1.0	6.9	23.4	161.1	
L4x4x5/8	Angle	6.6	1.0	6.6	23.4	153.3	
L4x4x5/8	Angle	6.3	1.0	6.3	23.4	146.0	
Structure Weight						2849.2	

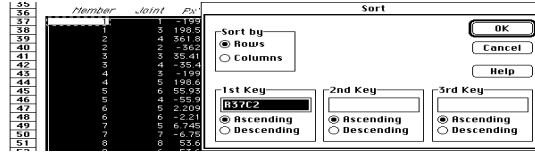
If you want to calculate the cost of structural painting you might use the values in the Length, D, B, tf and tw columns to find the total surface area and use this as a basis for your costing.

SectionData						
Section	Length	D	B	tf	tw	Surface Area
TS8x8x1/2	30.0	203.2	203.2	12.7	12.7	12.9
C8x11.5	4.5	203.2	57.404	9.906	5.588	1.0
C8x11.5	4.0	203.2	57.404	9.906	5.588	0.9
C8x11.5	3.5	203.2	57.404	9.906	5.588	0.9
L4x4x5/8	6.9	101.6	101.6	15.875	15.875	1.7
L4x4x5/8	6.6	101.6	101.6	15.875	15.875	1.6
L4x4x5/8	6.3	101.6	101.6	15.875	15.875	1.6
						20.7

Example 4 - Finding Maximum Forces

After you have analysed a structure, Multiframe displays a table of the member actions in the Result window. You can copy this table to

a spreadsheet and then sort it to find for example the members with the greatest axial force.



If you wish you could also add additional calculations based on the results to compute other design parameters and then sort based on the calculated values.

Summary

Spreadsheets are an invaluable tool for all design engineers. Used by themselves they can provide quick answers to difficult design problems but used in conjunction with Multiframe they can provide you with a greatly expanded range of design tools. Next time you have a design requirement that Multiframe does not provide, consider using a spreadsheet to help provide the facilities you need to solve your design problem.

Appendix G

Quality Assurance

This appendix describes the quality assurance processes used to ensure Multiframe gives reliable and accurate results.

Quality Assurance

Many Multiframe users ask us how we know that Multiframe produces the correct results. This appendix explains how Formation Design Systems has verified that Multiframe gives accurate results and what steps we take to make sure that each version of the software we ship is as reliable as possible.

Quality Principles

While it is impossible to ensure that any software product is completely free of bugs, we follow a series of engineering and testing principles and procedures to ensure that Multiframe will produce results which are consistent with the level of accuracy and thoroughness a professional engineer applies to design work. To this end we follow a development and testing path which includes use of structured programming techniques, verification of the underlying algorithms, testing of the computer implementation of those algorithms, testing of real world problems in-house and beta testing in the field at Multiframe user sites.

Structured Programming

The best defense against bugs in software is to use structured programming techniques, which have been proven to improve software reliability. Without going into the technical details of our software development methodology, we summarize by saying that we utilize structured code, object oriented design, data hiding and encapsulation and fault tolerant programming practices to enhance our software's reliability. Multiframe is a complex software system of over 200,000 lines of code and we believe our history of reliability reflects the effort we have put into using reliable coding practices.

Verification of Algorithms

When new design or analysis algorithms are introduced into Multiframe, we first carry out testing on the algorithms on proven test cases with known analytical solutions. These generally come from engineering texts such as Refs 1 to 3. These test cases will include samples which independently examine the various degrees of freedom (Mx bending, My bending, Axial tension etc.) followed by examples which superimpose the effects of multiple degrees of freedom. These simple test cases are performed for structures aligned with the principal axes as well as those rotated to arbitrary angles.

Testing of Implementation

Once the basic algorithms have been proven correct, testing is then carried out on more complex sample problems to which a solution has already been established using a proven analysis program. These results may either come from structural engineering texts such as Refs 1 to 3 as well as from other results carried out by Formation Design Systems or other engineers using other software products such as SAP, Nastran etc.

Testing of Upgrades

As each new version of Multiframe is released we perform a series of tests to ensure it functions correctly. Among these tests is an analysis of a frame which exercises every different feature of Multiframe i.e. every possible member type, section type and orientation, load type, combination etc. At each release the results from this frame are compared with the results from the previous release to ensure conformance with answers which have been established as being correct.

Beta Testing

Immediately prior to the release of each new version of Multiframe, we conduct a beta test of the software. This involves sending the software to practicing engineers and having them use it on design work in progress to determine its reliability for actual design use. These beta testers provide us with feedback on the reliability and accuracy of the program as well as its useability and suitability for everyday work. Once the beta test program is completed and all testers are happy with the program, we begin shipping the commercial version.

Version Control

Each new version of Multiframe displays a version number indicating the version and the date the software was first shipped. If the version is a development, alpha test or beta test release, the version number may also include a letter and number suffix indicating the type and number of the release. A development version is usually only for internal use and is a very early demonstration of a possible new product or feature. It is highly experimental and not reliable. An alpha release is a first public release of a program for initial testing and comment, it is not reliable. A beta release is a final test version of the program released for field testing prior to commercial release. It is mostly reliable but may contain some bugs. A commercial release is a completed, debugged program reliable and ready for professional use.

For example

- 1.0d1 The first development release of version 1.0
- 1.5a2 The second alpha test release of version 1.5
- 1.6b2 The second beta test release of version 1.6
- 1.64 A commercial release of version 1.64

But we're not Perfect

We make every effort to ensure that our software will meet our users' needs and perform accurately. However, as with all complex software systems, it is possible for errors to occur. If you suspect a problem with Multiframe, please contact our technical support staff by email at support@formsys.com and explain what you believe the problem to be. In the unlikely event of a problem being found, we will correct it immediately, and send you a new corrected version of the program.

To get accurate results from Multiframe, it is necessary for you to model the problem correctly and to correctly interpret the results produced. This requires structural engineering experience combined with an understanding of matrix structural analysis. It is the users' responsibility to correctly model the structure and assume responsibility for the results.

Index

2D Analysis	102	Click	41
2D DXF	129	Clip Gray	134
3D Analysis	102	Clip Invisible.....	134
3D DXF.....	129	Clip To Frame.....	47, 134
3D Tutorial	28	Clip To Selection	47, 134
About Multiframe	146	Clip To Window.....	134
Absolute Envelope	144	Clipping.....	33, 46, 132, 134
Actions	139	Clipping and Masking	46
Actions Sub Menu	142	Close.....	127
Actions Toolbar	126, 135	Color	133
Add Case	93	Color printing.....	119
Add Case Sub Menu	144	Comb Sx' + Sby' left	143
Add Case.....	143	Comb Sx' + Sby' right	143
Add Member.....	52, 135	Comb Sx' + Sbz' bottom	143
Add Section.....	78, 130	Comb Sx' + Sbz' top	143
Adding a load case.....	93	Combined Load Case	94
Adding Dynamic Load Cases.....	97	Combining load cases.....	15
Adding Seismic Load Cases	98	Command or Ctrl key	42
Analysis	16, 37, 101, 147	Connections	53
Analyze	143	continuous beam.....	58
Animate	139	Convergence	101
Animation.....	39	Coordinates.....	68
Applying Dynamic Loads.....	98	Copy.....	130
Applying Loads.....	84	Creating a Structure.....	50
Arrange Icons.....	145	Ctrl-Click (Windows only)	42
Arrow Keys	42	curved member	59
Axes.....	132	custom section	78
Axial Px'	142	Customize Plot	112, 140
Axial Sx'	143	Cut.....	129
Backspace.....	42	Cylindrical Coordinates.....	62
Bending Sby' left.....	142	Data.....	145
Bending Sby' right.....	142	Data Menu.....	139
Bending Sbz' bottom	142	Data Sub Menu.....	141
Bending Sbz' top.....	142	Data Window	124
beta angle.....	76	Day Star Text	129
Beta Testing.....	182	Deflection	139
Calcs Layout.....	145	Delete	42
CalcSheet	116, 145	Delete Joint Masses	76
CalcSheet Variables.....	116	Delete Case.....	95, 143
CalcSheet Window.....	124	Delete Member	56, 136
Calculate	143	Delete Section	79, 130
Calculating Quantities & Costs	179	Deleting a load case	95
Calculation Sheet.....	116	Depth	51, 132
Calculations.....	116	Displacements	141
cancel.....	26	Display Menu	139
cancel printing	121	Distributed.....	102
Canceling.....	103	double precision.....	102
capacities for Multiframe	151	Double-Click.....	42
Cascade	145	Drag	41
Case Menu.....	143	Drawing.....	50
Choosing the Time Step	99	Duplicate	60, 137
Clear	130	Duplicate current load case	94
		DXF	128
		Dynamic	144

Dynamic Analysis	102	Joint and Member Numbers.....	71
Dynamic Load.....	139	joint coordinates.....	64
Dynamic Load Case	97	Joint Displacement.....	20, 105, 114, 136
Dynamic Results.....	107, 115	Joint Displacement.....	72
Dynamics.....	142	Joint Linking	136
		Joint Load	13, 84, 138, 141
Edit Case.....	94, 143	Joint Mass.....	75, 136, 141
Edit Load Library	96, 130	Joint Moment.....	85, 138
Edit Menu.....	129	Joint Reactions.....	105, 114
Edit Section.....	80, 130	Joint Restraint	136
Editing a load case	94	Joint Spring	136
Editing Coordinates.....	68	Joint Spring	73
Editing Layout	145	Joint Toolbar	125, 135
Editing Load Cases	95	Joint Type	81, 136
Editing Loads Numerically.....	91	Joints	141
Editing Restraints Numerically.....	73	Joints Links	74
El Centro.....	96		
Envelope Cases.....	109	Keyboard techniques.....	42
Exaggerate sizes.....	49	Kobe	96
Export.....	127		
Export Sub menu.....	128	Linked Joints.....	141
Exporting Data.....	163	Linking Joints	74
Extrude.....	68, 137	Load	145
		Load Case.....	144
factored load case	93	Load Case Toolbar	126
File Menu.....	127	Load Library	95
Find.....	130	Load Menu.....	137
fixed joint.....	9	load units.....	84
Fixed Time Interval.....	97	Load Window.....	124
Font.....	132	Loading a joint.....	12
force diagram.....	113	Loading a member.....	13
FORTRAN	129	local and global axes.....	147
Frame	145	Local Dist'd Load	86, 138
Frame Menu	135	Local Moment.....	90, 138
Frame Window	124	Local Point Load	88, 138
		Lumped	102
Generate.....	56, 57, 137		
Generate Toolbar	125, 135	Macintosh Memory.....	159
Generating a continuous beam	58	Mask Gray.....	134
Generating a curved member.....	59	Mask Invisible	134
Generating a multi-story frame.....	57	Mask Out Selection.....	48, 135
Generating a portal frame.....	56	Mask To Frame.....	134
Generating a regular frame.....	60	Mask To Selection	135
Generating An Arch.....	175	Mask To Selection	48
Generating Loads	177	Mask To Window.....	135
Global Axes.....	38	Masking.....	48, 132
Global Dist'd Load	85, 138	Masking Sub Menu.....	134
Global Moment.....	89, 138	Mass Matrix Type	102
Global Point Load.....	87, 138	Master-Slave	74
Grid.....	50, 132	Max Iterations.....	102
Grid.....	50	Maximum Envelope	144
		Maximum Forces	179
Help Menu	146	Member	77
Hide All	146	Member Actions.....	23, 105, 142, 149
		Member Axes.....	38
Import	127	Member Details	106, 142
Import Sub Menu	128	member diagrams.....	19, 113
Importing Data	164	Member Labels.....	137
Importing Load Data	97	Member Loads.....	141
Installing Multiframe on Macintosh.....	3	Member Masses.....	76, 137
Installing Multiframe on Windows	3	Member Orientation.....	137
Invisible	51	member releases.....	81, 136

Member Self Weight	92, 139	reactions	21, 141
Member Shear Area	136	reactions	20
Member Toolbar	125, 135	Redo	129
Member Type	81, 136	References	152
Members	70, 141	regular frame	60
memory requirements	102	Remove Restraints	34
Minimum Envelope	144	Render	48, 140
Mode Shape	144	Renumber	137
Moment My'	142	Renumber	71
Moment Mz'	142	Rescale	66, 137
moment of inertia	149	Resizing a Member	66
Mouse techniques	41	Restraining a joint	8
Move	64, 137	restraint values	73
Move	51	Restraints	71, 141
Moving a joint	63	Result	145
Moving a member	65	Result Layout	22, 145
Multiframe load codes	169	Result Window	104, 124
Multiframe Text Example File	170	Results	21, 139
Multiple load cases	93	Results Sub Menu	141
		Return	42
New	127	Rotate	67, 137
No Clipping	134	Rotating a 3D View	43
No Masking	134		
No of Modes	101	Save	127
No Restraint	35	Save As	23, 127
Numbers	132	Saving Calculations	118
		Saving your work	23
Open	127	Section	77, 141
Open Library	127, 128	Section Axes	38
Orientation	76	Section Colors	130
origin	51	Section Orientation	76
Osaka	96	Section Properties	9, 35, 77, 117
		Section Type	77, 81, 136
Page Layout	120, 128	Sections Library	78
Page Setup	119, 127	Sections Table	81
Pan	44, 132	Seismic	144
Paste	130	Seismic Load Case	99
Pinned Joint	35, 81	Select	130
pinned restraint	9	Select All	131
Plot	140, 145	Select Horizontal	131
Plot Window	109, 124	Select Joint	131
portal frame	56	Select Member	131
Precision	102	Select Section	131
Preferences	107, 131	Select Sloping	131
Prescribed Displacement	72	Select Sub Menu	131
Print	128	Select Vertical	131
Print Command	121	Selecting Joints	8, 54
Print Report	128	Selecting Members	9, 29, 54
Print selected members	120	Self Weight	92, 144
Print Selection	121	Self Weight Loads	92
Printing	24, 119	Setting up the printer	24, 119
Printing Data	25	Shear Area	82
Printing Diagrams	26, 121	Shear Sy'	143
Printing Results	119	Shear Sz'	143
Properties	131	Shear Vy'	142
		Shear Vz'	142
Quality Assurance	181	shift or Ctrl keys	42
Quality Principles	181	Shift-Click	41
QuickTime Movies	164	Show All	146
Quit / Exit	128	Show/Hide Clipboard	130
		Shrink	45, 132
Rayleigh Damping Factors	100	Sign Convention	147

single precision	102	Tile Horizontal	145
Size	50, 132	Tile Vertical	145
Size To Fit	45, 132	Time History (Ctrl + H)	144
Size To Fit Sub menu	133	Time History Results	108
Spherical Coordinates	63	Time Menu	144
Spreadsheets	175	Toolbar	133
Springs	73, 141	Torque Tx'	142
Stack	145	Troubleshooting	153
Starting Multiframe	4	Trusses	161
Static	144	Undo	129
Static Combined	144	Units	133
Status Bar	133	Unload Joint	84, 137
Step Loads	141	Unload Member	138
stiffness matrix	148	Variable Time	97
stress diagram	113	View Menu	132
Stresses	113, 139	View Toolbar	125, 135
Stresses Sub Menu	142	Viewing Applied Time History Loads	98
Structure Diagrams	110	Views	43
Subdivide Member	55, 136	virtual memory	103, 160
Symbols	110, 139	Window Menu	145
Tab	42	Zero Displacement Restraint	71
Table of Contents	146	Zoom	44, 132
Text File Format	167		
Text Files	129		
Thermal Load	90, 138, 141		
Tile	145		