

# Steel Designer

Windows and Macintosh Version 5.1

## User Manual



# License & Copyright

---

## Steel Designer 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.

## Steel Designer User Manual

© 1990-99 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 reserves 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 Steel Designer.....	3
Chapter 2 Steel Designer.....	21
Chapter 3 Steel Designer Reference.....	35
Windows.....	36
Toolbars.....	37
Menus.....	38
Chapter 4 Steel Calculations.....	57
Index.....	63



# About this manual

---

This manual is about Steel Designer, a structural steel design application for the Macintosh and Windows operating systems. Steel Designer is part of the Multiframe range of structural design software.

Chapter 1 "Learning Steel Designer" gets you started reading in a Multiframe structure, assigning its design properties and checking its strength in accordance with the steel design code of practice. It is designed to help you learn the basics of Steel Designer and become confident in using it to design your structures.

Once you are familiar with the basic concepts used, you may refer to the step-by-step instructions in Chapter two.

Chapter 2 "Using Steel Designer" This will explain some of the less common commands you may wish to use while designing a steel frame.

Chapter 3 "Steel Designer Reference" gives an overview of the windows and menus of Steel Designer and a summary of the commands used.

Chapter 4 "Steel Calculations" discusses the methods used by Steel Designer. It is important for you to understand these methods and their limitations before using Steel Designer for structural design calculations. The chapter ends with a summary of the capabilities and limitations of Steel Designer.





# Chapter 1

## Learning Steel Designer

---

This chapter gets you started using Steel Designer with step by step instructions on the basics of using the program.

### Getting Started

Before you get started using Steel Designer, you will need to learn some of the basic concepts of using your computer. If terms like windows, menus and scroll bars are not familiar to you, now is a good time to read your owners' guide which will explain the basic concepts. If you are familiar with the standard operations continue reading.

Steel Designer is a program for checking or designing a steel frame in accordance with various codes of practice. The current release is for the American ASD (Allowable Stress Design), Japanese AIJ (Architectural Institute of Japan) and the Windows version also supports the Australian AS4100 code. Other codes will be supported in future releases.

**A word of caution:** Steel Designer is a very useful aid to the design of steel structures. It is NOT an automatic design tool and it should be used in conjunction with professional engineering judgment to produce well-designed frames.

If you are running Steel Designer on a Windows computer, be careful when using Steel Designer with Multiframe or Section Maker. If you have both Steel Designer and Multiframe open at the same time, any changes you make to the Section Library with Steel Designer will not be reflected in the sections available to Multiframe and vice versa. To make the new sections available to Multiframe you will have to quit Multiframe and restart the program.

On Macintosh computers, Steel Designer and Multiframe will automatically exchange information to keep the library up to date.

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

### Installing Steel Designer on Windows

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, then follow the instructions on screen.

After installation, Steel Designer should be accessible through the Start Menu. Simply select Steel Designer from the Multiframe menu under the Start menu. Steel Designer is automatically installed in

the Multiframe folder in the Program Files directory unless you specify otherwise.

## Installing Steel Designer on Macintosh

Insert the Steel Designer disk and double click on the Installer program. This will prompt you to specify a location on your hard disk where Steel Designer will be installed. You should install Steel Designer into the same folder as the Multiframe program on your hard disk.

## Introduction

After starting up Steel Designer you will open a frame you have previously created and analysed with Multiframe and then check the members in the structure for compliance with the code. You can also use Steel Designer to choose the lightest weight sections, which satisfy the design criteria. Steel Designer may only be used in conjunction with Multiframe 3D version 1.5 or later. If you are using an earlier version of Multiframe, you will have to upgrade to the most recent version.



Start up Steel Designer and after the copyright window has been displayed, a number of windows will appear on screen with the Frame window in front. The window layout and naming in Steel Designer is very similar to that used in Multiframe.

## Design Procedure

Steel Designer can check or design frames, which have been analysed using Multiframe 2D, 3D or 4D. The basic procedure you will follow is as follows;

In Multiframe

- **Set up the structure and loading**
- **Carry out the analysis**
- **Check the results to ensure your structural model is correct**
- **Save the frame to disk**

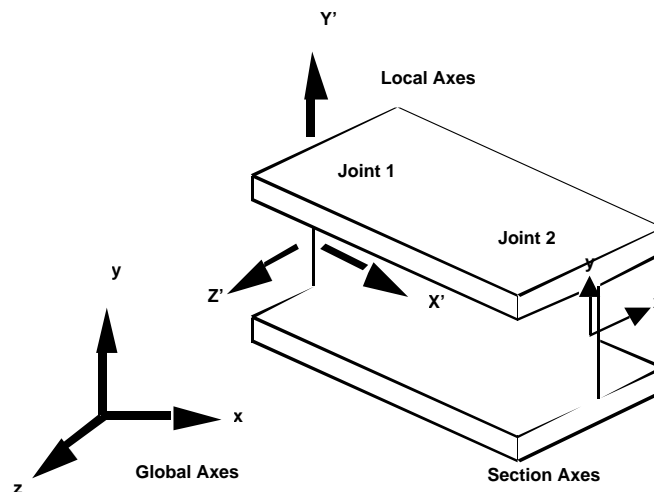
In Steel Designer

- **Open the frame from disk**

- Enter the design information (such as effective lengths, steel grades etc.)
- Carry out the design checks or search for the optimum sections

## Coordinate Systems

Much of the design information and many of the design variables are described relative to the major and minor axes of the section used for each member. This corresponds to the same terminology used to describe the properties of a section e.g.  $I_{xx}$  for moment of inertia about the major (or strong) axis and  $I_{yy}$  about the minor (or weak) axis.



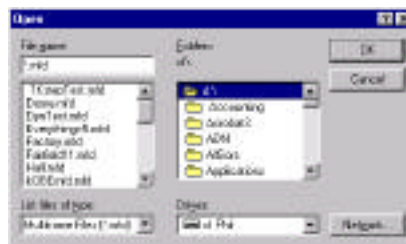
The coordinate systems corresponding to the naming conventions for the various results of analysis, section properties and design values are shown in the diagram above. Structure coordinates and global loads are defined relative to the Global Axes, member actions, deflections and stresses resulting from the Multiframe analysis are defined relative to the local member axes and design values are defined relative to the section axes. Whenever a design variable carries a subscript this indicates that it applies to the corresponding section axis. (E.g.  $f_{bx}$  refers to the design bending stress about the x-axis)

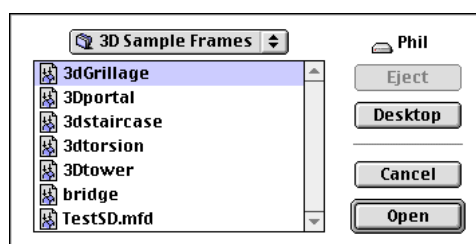
## Reading a Structure

To open a frame for checking in Steel Designer

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

The standard file dialog will appear allowing you to open the frame.





- Click on a frame in the list and click the OK/Open button

The frame will be read in and displayed in the Frame window.

## Setting Properties

Before doing the checks, it is necessary to enter basic design data such as effective length, grade of steel etc. This information can either be entered in the Frame window by selecting members and using the commands under the Design menu, or it can be entered in tabular form in the Data window. All of the windows and commands, which are common to Multiframe, work the same way in Steel Designer. You have all the display options of Multiframe and facilities to help you select the required members using clipping, masking etc. In general you can not change the frame or its loading in Steel Designer, the only change you can make is to change the section for a member. If you do change a section, you will need to re-analyse using the Analyse command.

Although most of the design variables are pre-set to the most commonly used values, you will probably want to enter the design information for at least some of the members in the frame that you wish to check. You set design variables by selecting the members you wish to change and then choosing the appropriate command from the Design menu.

There are a number of design variables, which are used when doing checking to the code. A summary of all of the design variables is as follows;

Variable Name	Description	Default ASD, AII	Default AS4100
Fy	Yield strength of the section's steel	36ksi	250Mpa
Fu	Ultimate Tensile Strength of the section's steel	58ksi	410Mpa
Kx	Effective length factor for buckling about the section's strong axis	1.0	1.0
Ky	Effective length factor for buckling about the section's weak axis	1.0	1.0
Lbx	Unbraced length for bracing preventing buckling about the section's strong axis	member's length	member's length
Lby	Unbraced length for bracing preventing buckling about the section's weak axis	member's length	member's length
Lateral restraints	The lateral restraints acting on the member.		Each end of the member is fully restrained at both flanges.

Load Height	The position of the loading on beam (shear center or top flange).		Shear Center
a or s	Spacing of web stiffeners. This is the spacing of any stiffeners along the web of a beam	0.0 (i.e. no stiffeners)	0.0 (i.e. no stiffeners)
No. of Flange Holes	The number of holes in the flanges of the section.		0
Diameter of Flange Holes	Diameter of holes in the flanges of the section.		0.0
Total Height of Flange Holes	Total height of any bolt holes in the flanges of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.		0.0
No. of Web Holes	The number of holes in the webs of the section.		0
Diameter of Web Holes	Diameter of holes in the webs of the section.		0.0
kt	Correction factor for the distribution of forces.		1.0
Total Height of Web Holes	Total height of any bolt holes in the webs of the section. This value may be input directly or computed automatically when the number and diameter of flange holes are specified.		0.0
Flange Hole Area	Area of any bolt holes in the flanges of the section. This area will be deducted from the cross sectional area when computing tensile stress	0.0	
Web Hole Area	Area of any bolt holes in the web of the section. This area will be deducted from the cross sectional area when computing tensile stress	0.0	
U	Area Reduction factor. This factor is applied to the sectional area (after bolt holes have been deducted) when calculated tensile stress. You can use it to reduce the effective area by a defined amount. It must have a value between 0 and 1.0	1.0	
Cb	Moment modification factor used to determine allowable compressive stresses in bending. (See ASD code for details)	1.0	
Cmx	Moment reduction coefficient for bending about the section's strong axis (see ASD code)	1.0	
Cmy	Moment reduction coefficient for bending about the section's weak axis (see ASD code)	1.0	
Max Depth	The maximum depth of section which may be chosen when using the Design command	depth of the initial section	depth of the initial section
Min Depth	The minimum depth of section which may be chosen when using the Design command	depth of the initial section	depth of the initial section
Max Width	The maximum width of section which may be chosen when using the Design command	width of the initial section	width of the initial section
Min Width	The minimum width of section which may be chosen when using the Design command	width of the initial section	width of the initial section

Fabrication	The method by which the section was manufactured. This describes the residual stresses in the section.	Rolled	Rolled
-------------	--------------------------------------------------------------------------------------------------------	--------	--------

It is not necessary to enter all of the above information for all members. Usually you will want to check some members for bending, others for compression and so on. The items under the Design menu help you enter just the required information depending on what type of check you are doing.

#### Bending

When performing a bending check, you may need to specify a number of properties relating to the unbraced length, lateral restraints and stiffener spacing on the member.

#### Unbraced Length (ASD, AIJ)

To determine the critical buckling condition of a member, it is necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements, which are not modeled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section,  $L_{bx}$  corresponding to the spacing of restraints preventing buckling about the x-x axis and  $L_{by}$  corresponding to the spacing of restraints preventing buckling about the y-y axis.

The initial values of  $L_{bx}$  and  $L_{by}$  are the length of the member.

#### Lateral Restraints (AS4100)

When performing a bending check, you may need to specify the location and type of lateral restraints acting on the member.

To determine the moment member capacity of a member, it is necessary to know the spacing of any lateral restraints (if any) along the member. The restraints could be provided by purlins, girts or other structural elements, which are not modeled in Multiframe. Steel Designer uses this information to determine the length of segments used in the design calculations. The lateral restraints acting at a particular section on a member are dependent upon which flange is the critical flange. For a member/segment restrained at both ends the critical flange is the flange under compression. For a cantilever or a segment with an unrestrained end, the critical flange is the tension flange. For each restraint on the member, the user must specify the type of restraint. As this depends upon which flange is the critical flange, the user must specify the type of lateral restraint that would be present at a section if

- i) the top flange were the critical flange, and
- ii) the bottom flange was the critical flange.

Lateral restraints must always be specified at the ends of the beam and so the minimum number of lateral restraints is two. If no restraint exists at the end of a member then it should be specified as unrestrained. The initial lateral restraints applied to the member

are full restraints at each end for either of the flanges being the critical flange.

The different restraints acting on the member are entered into the grid using the following codes;

F	Fully restrained
P	Partially restrained
L	Laterally Restrained
U	Unrestrained
C	Continuous restraint

Fully or partially restrained sections may also be specified as lateral rotational restraints using;

FR	Fully restrained and Rotationally restrained
PR	Partial restrained and Rotationally restrained

The initial position of the loads is at the shear center. If there are no transverse stiffeners, leave the stiffener spacing set to zero.

#### Bending Coefficient (ASD)

The ASD code requires a bending coefficient  $C_b$  which is either calculated by the program according to the rules in the code, or may be specified by the user. If you leave  $C_b$  unchanged, Steel Designer will select a value for you, which will be displayed in italics in the Design Details table in the Data window. This value is most commonly 1.0. If you type in a value, Steel Designer will always use this value and display it in non-italic (i.e. standard) text in the Design Details table.

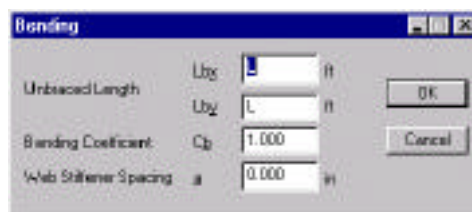
#### Web Stiffener Spacing (ASD, AIJ, AS4100)

When checking or designing a member for bending, you may need to specify the spacing of any stiffeners along the web of the member. This affects the member's susceptibility to buckling due to bending. If there are no transverse stiffeners, you should leave the stiffener spacing set to zero.

#### Bending Dialog

To set the properties for bending

- **Select the required members in the Frame window**
- **Choose Bending from the Design menu**



ASD, AIJ

- Type in values for Lbx and Lby
- If necessary enter a value for the bending coefficient Cb



AS4100

- **Type in the number of lateral restraints.**
- **Enter the position and type of lateral restraints**
- **Choose the position of the load from popup menu**
- **If there are transverse stiffeners on the web, type in values for the stiffener spacing (s)**
- **Click OK**

## Tension

A number of modification factors may be entered to change the section properties used for checking tension.

Bolt Holes (ASD, AIJ, AS4100)

When checking or designing a member for tension, you need to specify any reduction in area due to bolt holes or other reductions. If the members contain significant areas of bolt holes, which need to be taken into account when determining the cross-sectional area of the section, you will need to enter the amount of cross-sectional area to be deducted to allow for these holes. The initial value for the area of bolt holes is zero.

The net area of the section is the gross area minus the combined area of bolt holes in the flange and web.

In AS4100, the reduction in area due to bolt holes can be entered by entering values for the number and diameter of any holes.

Area Reduction (ASD, AIJ)

The net area is multiplied by the area reduction coefficient, U, to give the effective net area of the section. The default value of U is 1.0, i.e. no reduction in area.

Correction Factor (AS4100)

When checking or designing a member for tension in AS4100, you need to specify the correction factor for the distribution of forces at



the ends of the member. The correction factor  $k_t$  has a default value of 1.0

### Tension Dialog

To enter the properties for tension

- **Select the required members in the Frame window**
- **Choose Tension... from the Design menu**



ASD, AIJ

- **Type in the area of holes in the web and flanges**
- **Type in a value for the area reduction coefficient (U) if required**



AS4100

- **Type in the number and diameter of holes in the webs and flanges (and the total height of holes will be computed automatically) or...**
- **Type the total height of holes in the webs and flanges directly**
- **Choose a value for the correction factor ( $k_t$ ) if required**
- **Click OK**

### Compression

When checking or designing members for compression, it is necessary to specify the effective length and unbraced length of the member.

To determine the critical buckling load for a member, it is necessary to enter an effective length to indicate the type of restraint on the ends of the member. The effective length is given by an effective length factor multiplied by the length of the member. The effective length may be different for buckling in the major and minor axis directions. The effective lengths are given by

$$L_x = K_x \cdot L \text{ and } L_y = K_y \cdot L$$

Where  $L$  is the length of the member and  $K_x$  and  $K_y$  are the two effective length factors for the major and minor axes respectively.

The initial values of  $K_x$  and  $K_y$  are 1.0.

#### Unbraced Length

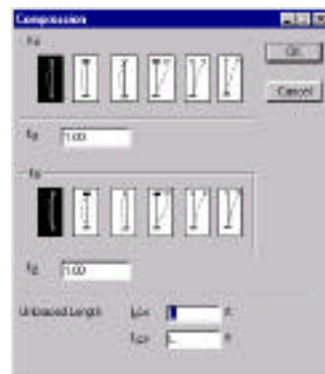
To determine the critical buckling condition of a member, it is also necessary to know the spacing of any bracing (if any) along the member. This bracing could be provided by purlins, girts or other structural elements, which are not modeled in Multiframe. Some bracing may only restrain lateral deflection in one direction, therefore it is necessary to enter unbraced lengths for both axes of the section,  $L_{cx}$  corresponding to the spacing of restraints preventing compression buckling about the x-x axis and  $L_{cy}$  corresponding to the spacing of restraints preventing compression buckling about the y-y axis.

The initial values of  $L_{cx}$  and  $L_{cy}$  are the length of the member.

#### Compression Dialog

To set the properties for compression

- **Select the required members in the Frame window**
- **Choose Compression... from the Design menu**



Either

- **Click on the icons for the end conditions in each direction or...**
- **Type in values for  $K_x$  and  $K_y$**
- **Type in values for  $L_{cx}$  and  $L_{cy}$**
- **Click OK**

If you choose a standard end condition, the recommended  $K_x$  and  $K_y$  values will be automatically entered for you.

#### Combined Actions

No information is required when checking or designing members for combined actions using AS4100.

When checking or designing members for combined bending and compression actions under the ASD code, you may wish to enter coefficients as prescribed by the code. If you leave the  $C_m$  unchanged, Steel Designer will select a value for you, which will be

displayed in italics in the Design Details table in the Data window. This value is most commonly 1.0.

To set the coefficients for combined checks

- **Choose Combined... from the Design menu**



- **Enter the values for Cmx and Cmy**
- **Click OK**

## Design Properties

Sometimes you may wish to set all of the design properties for a member or group of members at once. This may be quicker than setting each of the design values in turn using the commands above.

To set all of the design variables

- **Select the required members in the Frame window**
- **Choose Design Details from the Design menu**



- **Enter the design values**
- **Click OK**

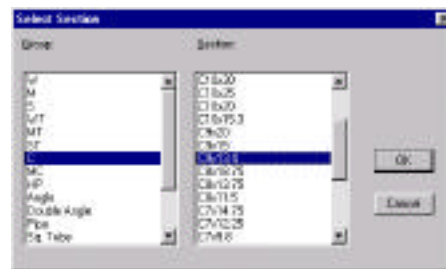
As a short-cut, you can examine and change the design details for a single member by double clicking on it in the Frame window.

## Section Type

If necessary you can change the section type of a member manually in Steel Designer. Note however, that if you do so, you will need to re-analyse the structure using the Analyse command from the Case menu.

To set the section type for a member or group of members

- **Select the required members in the Frame window**
- **Choose Section Type... from the Design menu**



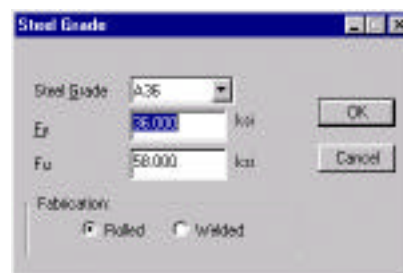
- Choose the section from the list
- Click OK

## Steel Grade

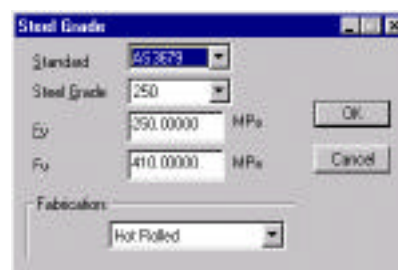
To determine the allowable stresses for a member, it is necessary to know the grade of steel to be used for the section. This grade determines the yield strength ( $F_y$ ) and ultimate tensile strength ( $F_u$ ) of the material of the section.

To set the Steel Grade

- Select the required members in the Frame window
- Choose Steel Grade from the Design menu



ASD, AIJ



AS4100

Either

- Choose a standard and/or steel grade from the pop-up menu or...
- Type in values for  $F_y$  and  $F_u$
- Choose the fabrication type for the section
- Click OK

If you choose a standard grade of steel, the  $F_y$  and  $F_u$  values will be automatically entered for you.

The initial value for the steel grade for all members is

Code	Grade	Fy	Fu
ASD	A36	36ksi	58ksi
AJ	SS42	2.4t/cm <sup>2</sup>	4.0t/cm <sup>2</sup>
AS4100	AS3679 grade 250	250MPa	410MPa
User(US)	-	36ksi	58ksi
User(Japan)	-	2.4t/cm <sup>2</sup>	4.0t/cm <sup>2</sup>
User(Australia)	-	250MPa	410MPa

## Constraints

When designing a member to determine the lightest weight section, which may be used, you may wish to apply some constraints to the way the sections are selected. For example, you may wish to limit the section's depth or width or you may wish to ensure that a group of members all use the same section.

To constrain the selection of a member's section

- Select the required members in the Frame window
- Choose Constraints... from the Design menu



- Check the boxes corresponding to the sizes you wish to constrain
- Type in the limits for the sizes you wish to constrain
- If you wish to make the sections the same, check the "Make sections the same" check box
- Click OK

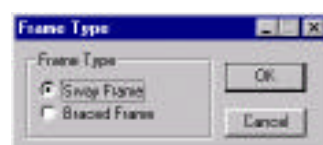
The initial value of constraints is for no limits on the sizes of sections and all members are free to be designed using a different section.

## Frame Type

Some design calculations depend on whether the frame is free to deflect laterally (sway) or is restrained by internal or external bracing to prevent side-sway (braced). A sway frame develops all of its horizontal stiffness due to the flexural actions of the columns in the structure. In contrast, the bracing in a braced frame absorbs the horizontal forces and horizontal deflections of the columns are reduced to a minimum.

To set the type of frame

- Choose Frame Type from the Design menu



- Click on type of the frame
- Click OK

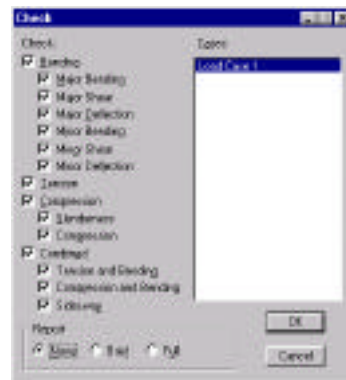
The initial setting for the frame type is a sway frame.

## Checking a Frame

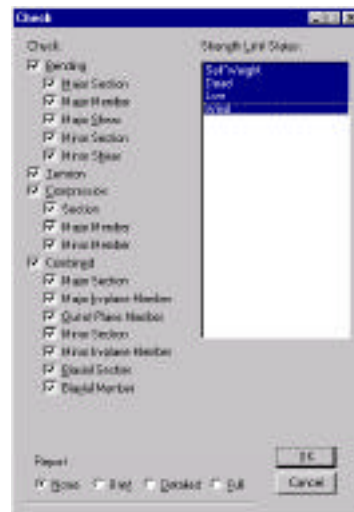
Once you have set up the structure and its design properties, you can check it for compliance with the code rules.

To check a member or group of members

- Select the required members in the Frame window
- Choose Check... from the Design menu



ASD, AIJ



AS4100

- Check the boxes of the design rules to be checked
- Shift-Click on the load case names in the list to include or remove them from the check
- If you want a summary report in the Report window, check the Brief or Full report radio buttons
- Click OK

Steel Designer will work through the selected members checking the stresses for the load cases you have chosen for compliance with the design rules you specified. The result of the check for the current load case will be displayed in the Design Efficiency table in the Result window. Each column in this table shows the member's strength as a percentage of the allowable strength according to the code. For example, an efficiency of 95% means that the member is being stressed to 95% of its allowable value. An efficiency greater than 100% indicates that the member is being stressed to a higher level than that permitted by the code. The Overall column shows the highest value of all of the design checks for the member for the current load case. The subsequent columns show the result for the individual checks, which have been carried out.

You can display the results for different load cases by choosing the appropriate item from the Case menu.

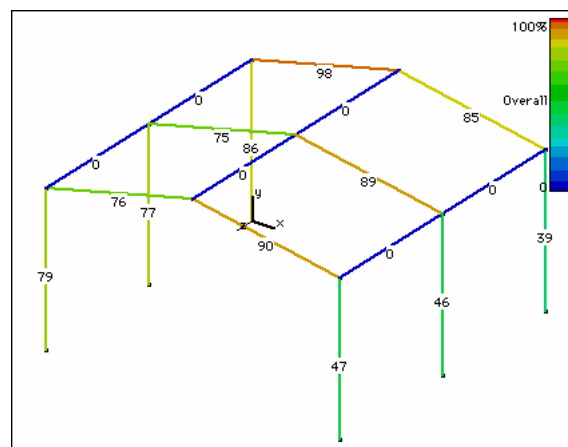
The check will be much slower if you choose to have a summary report generated, however the report will contain detailed information about all of the design checks carried out. You will probably find it best to do an overall check on the areas of interest without the report on and then check a few key members using the full report option.

#### Displaying Efficiency

As well as displaying the table of member efficiency in the Result window, you can view these values graphically in the Plot window (if you have a colour monitor. If you have a black and white monitor, you will only be able to view the efficiency values on the Plot diagram.)

To view the member efficiency

- Choose the required item from the Efficiency sub-menu under the Display menu



The members will be drawn in the Plot window with a colour code indicating the efficiencies of the members. The scale on the right hand side of the window may be used to determine the relative values of the colours. Members, which exceed the allowable capacity, will have an efficiency greater than 100% and will be drawn in red.

If you turn on the display of Plot Values in the Symbols dialog under the Display menu, the values of the efficiencies will be displayed on the members.

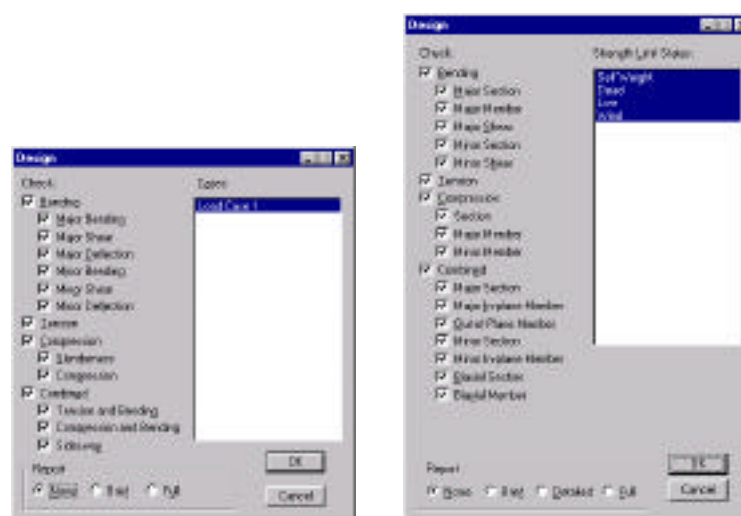
Values and colours will only be drawn for members, which have been checked. You can also use the clipping and masking commands to restrict which members have their efficiency values displayed.

## Designing a Frame

As well as helping to check a frame's compliance with the design rules, Steel Designer can also help you to select the lightest weight section which satisfies the design rules.

To design a member or group of members

- **Select the required members in the Frame window**
- **Choose Design... from the Design menu**



ASD, AIJ

AS4100

- **Check the boxes of the design rules to be used when designing**
- **Shift-Click on the load case names in the list to include or remove them from the check**
- **If you want a summary report in the Report window, check the Brief or Full report radio buttons**
- **Click OK**

Steel Designer will design each of the selected members, searching through the group of sections the member's original section comes from, to find the lightest section in this group that meets the design rule requirements. Once the design has finished, you can view the optimum section in the Best Section column in the Member Efficiency table in the Result window. If you want to automatically assign all of the optimum sections to their respective members, you can use the Use Best Sections command from the Design menu to do this. Because changing the sections will change the results of the analysis, you will have to re-analyse the structure after doing this. You may find it useful to wait until you have designed all of the members you wish to optimize before using the Use Best Sections command.



## Tips On Optimisation

When you use the Design command, Steel Designer will try to find the lightest weight section in a member's group, which will satisfy the design requirements. If there are a large number of sections in the group, this may take some time. If you use the options to constrain the width or depth of the optimum section, Steel Designer will automatically skip the check for any sections, which don't satisfy these criteria. This means you can speed up the optimisation greatly by specifying constraints for the size of the section. For example, if you are selecting an optimum section from the W sections in the United States Section Library which contains a large number of sections, specifying an upper and lower bound for the depth will let Steel Designer automatically skip most of the sections and quickly find one of the right size.

Checking for sway when using the Design command is not recommended. It is unlikely that Steel Designer will find an optimum size member because the amount of sway is likely due to the stiffness of other members (probably the columns in another part of the frame) rather than the member under consideration. These other members will not be changed while the current member is being checked.

## Saving your Work

You can save your design work at any time and then open the frame later to continue where you left off. Note however, that if you re-analyse a frame in Multiframe and save it using that program, the design information in the file will be lost and you will need to enter the design information again in Steel Designer.

To save the frame and its design information to disk

- **Choose Save from the File menu**

The frame will be saved to disk complete with the design information you added to it.

Hint: You can copy and paste the contents of the Design Details table in the Data window into the scrapbook for later use. This means, if you know you are going to make some changes to the frame and open it in Steel Designer once again and the number of members remains the same, you can paste the data back into the Design Details table from the scrapbook.



## Chapter 2

# Steel Designer

---

This chapter describes all of the commands, which are available for use in Steel Designer. If you have read Chapter 1 you are now familiar with some of the features of designing a steel frame using Steel Designer. This chapter presents some additional design procedures you will want to know about. This chapter contains the following sections:

- Working with windows
- Checking a Frame
- User Code
- Printing
- Saving your Work
- Saving the report

### Working with windows

There are six windows in Steel Designer; they are titled Frame, Data, Load, Result, Plot and Report.

The first five windows work the same as their counterparts in Multiframe. The only difference is that you can not change a frame's geometry or loading in Steel Designer as you can in Multiframe. There is also an additional window titled Report. This window can be used to display a summary of the design checks made by Steel Designer. You can also paste text and graphics into the report to help document your calculations.

The following sections document the differences between the contents of the windows in Steel Designer and the windows in Multiframe.

#### Frame Window

The Frame window operates the same as the Frame window in Multiframe except that you can not change the geometry or restraints of the frame. You can however change the section type for a member in this window. You also use the Frame window to set up the design properties for the members in the frame. You can do this by selecting members and then using the items in the Design menu to set the various design values. You can also change the design properties of a member by double-clicking on it in the Frame window. This will produce the Design Details dialog, the same dialog which appears if you choose Design Details from the Design menu.

#### Data Window

The Data window displays the same information as that displayed in Multiframe with the addition of a new table named Design Details. You can display this table by choosing Design Details from the Data sub-menu under the Display menu.

This table displays all of the design information required for each member so that Steel Designer can carry out the design checks. You can change this data by clicking on the value you wish to change, typing in the new value, and typing Enter. You may also copy and paste data to and from the table in the same way you can in Multiframe. (See "Working With Tables" below for a fuller description)

Numbers in this table which are displayed in italics (in the Cb, Cmx and Cmy columns) will be calculated by Steel Designer, you do not have to enter them. If you wish however, you can override the calculation of these values by typing in a value to be used. Any values you enter will be displayed in normal type. To revert to the automatic calculation of any value, type in a value of zero.

#### Load Window

The Load Window displays the same information you see in the Load Window in Multiframe however you can not change, add or delete loads in Steel Designer. The only exception to this is if you change the section types of any members in the frame (either manually or by using the Design command) Steel Designer will automatically update the self weight load case before carrying out the analysis of the structure.

#### Result Window

In addition to the tables of results displayed in Multiframe, the Result Window in Steel Designer contains a table named Design Efficiency. This table displays the result of the design checks for the various load cases, one case at a time. The values displayed in each row reflect the design stresses or deflections of the member as a percentage of its allowable capacity. An efficiency of 100% indicates that the member is at the limit of its design capacity, anything over 100% indicates that the member is overstressed (or is deflecting too much in the case of deflection) and anything under 100% indicates that the member is understressed.

You can copy data from the Results window to the report or other Macintosh programs (see "Working with tables" below).

#### Plot Window

The Plot window in Steel Designer operates in the same way as the Plot Window in Multiframe. There is also an additional display function in the Plot window which lets you display a graphical representation of the efficiency of the members relative to the design code requirements.

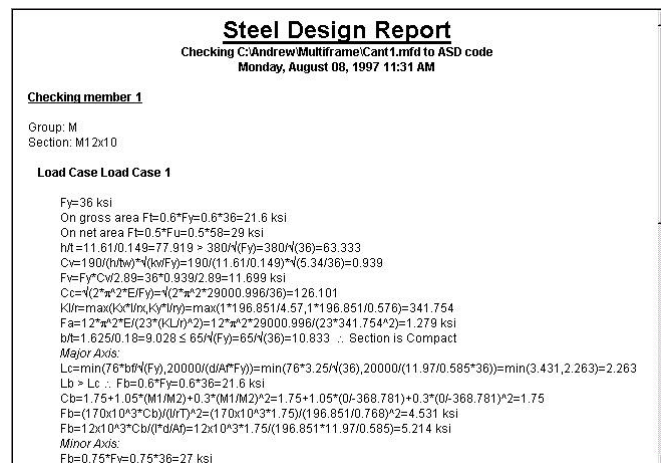
You can display efficiency by choosing the required item from the Efficiency sub-menu under the Display menu. The efficiency is a measure of the member's stress or deflection as a percentage of the allowable value as calculated using the design rules. That is, an ideal member is stressed to 100% of its allowable design capacity (or slightly less) and a member labeled as being 50% efficient is twice as strong as it needs to be. This displays the same information that is displayed numerically in the Efficiency table in the Result window. Steel Designer uses a colour display to show the stress or deflection level in the member relative to its allowable value. The scale on the right hand side of the window indicates the

relationship between the colours and the level of efficiency. Members which are more highly stressed or deflected than the level allowed by the code are shown in red.

You can use the Symbols command from the Display menu to turn on the display of Plot values. When this option is on, the values of the efficiency will also be displayed on each member which has been checked.

## Report Window

Steel Designer contains a new window titled Report. This window is used to create a progressive summary of the design that has been carried out.



This report can be edited via Cut, Copy, Paste and Clear, printed, or saved to and recalled from a disk file. You can not however type directly into the report or edit the text in the report.

You can copy and paste data from the report into a word processor such as Microsoft Word. You can also delete paragraphs, tables and pictures from the report. To edit paragraphs in the report, click and drag or shift-click to select the paragraphs and then copy them to the clipboard. To delete a paragraph, select it and choose Clear from the Edit menu or press the Delete (or Backspace) key.

You can change the format of paragraphs in the report by selecting them and choosing formatting options from the ruler at the top of the Report window. To change a paragraph's format, click on the paragraph(s) to select it and then choose the desired item in the ruler.

Windows

Macintosh

You can only select information in the report on a paragraph by paragraph basis, you can't select the text within a paragraph and you can't type directly into the report. If you wish to add pictures to the report, you can copy diagrams from the other windows in Steel Designer and paste them in. If you wish to add text to the report, you may find it useful to use a word processor or the Note Pad desk accessory to type the text and then copy and paste it into the report.

To paste into the report, click at the left hand edge of a paragraph and the blinking cursor will be displayed at that point to show you where the pasted data will appear, then select Paste from the Edit menu.

The report can be saved to a file or read in from a file using the Save and Open Menu command. This is useful if you wish to append design checks to a report that had been created at some time in the past.

## Working with Tables

The design data for the structure is displayed in tables in the Data window. You can change this data by typing in new numbers. If there is more data than will fit in a window, you can use the scroll bars to scroll through the table to view rows or columns that are not visible.

## Column Widths

If you want to change the width of a column in the table

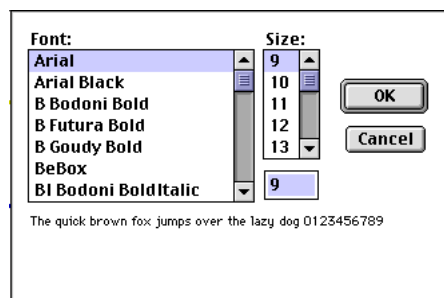
- Press on the line dividing the column's title from the next column
- Drag the column divider to make the column wider
- Release the mouse button

## Text Styles

If you wish you can change the font and text size used to display the numbers in the table by using the Font command from the View menu.

To change the font and/or size of the text in a table

- Choose Font from the View menu



- Click on the name of the font you require
- Click on a font size or type in a size

## Changing Numbers or Names

- Click the OK button

To change a number in a table

- Click on the number to be changed
- Type in a new value
- Press the Enter key

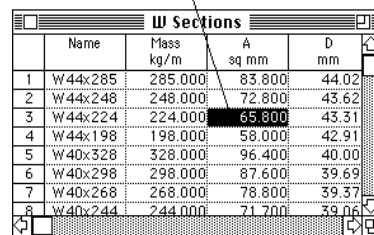
## Selecting Numbers in Tables

You can also copy and paste data in tables. The copy or paste will act on the current selection in the window. You can select a number or a range of numbers, a row or range of rows, or a column or range of columns.

To select a number

- Click on the number

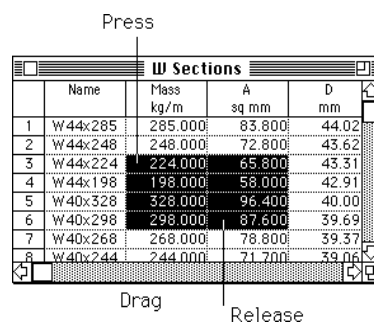
Click on a number to select it



	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

To select a range of numbers

- Press on the number at the top left of the range to be selected
- Drag the mouse until the last number is selected
- Release the mouse button.



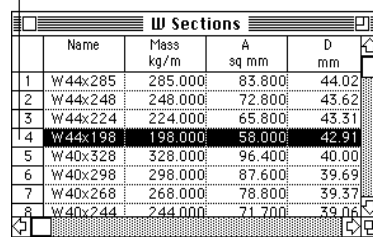
	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

If the last number you want to select is outside the window, drag the mouse outside the window and the table will automatically scroll to bring the number into view.

To select a row

- Click on the number at the left of the row

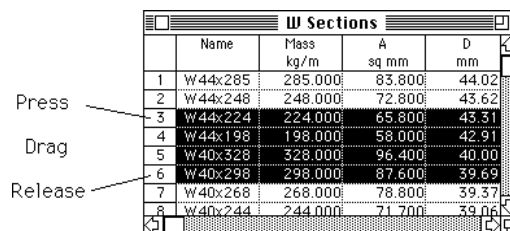
Click here to select the row



	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

To select a range of rows

- Press in the number at the left of the first row
- Drag down until the last row is selected
- Release the mouse button



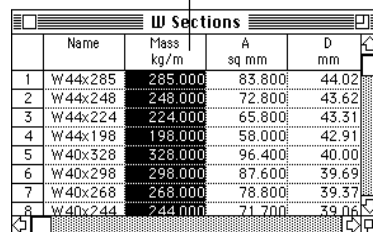
	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

If the last row is outside the window, drag outside the window and the table will scroll to bring the row into view.

To select a column

- Click on the title at the top of the column

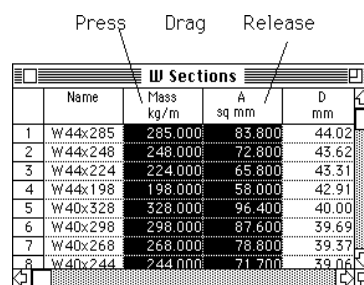
Click here to select the column



	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

To select a range of columns

- Press in the title of the left hand column in the range
- Drag the mouse across until the right hand column is selected
- Release the mouse button



	Name	Mass kg/m	A sq mm	D mm
1	W44x285	285.000	83.800	44.02
2	W44x248	248.000	72.800	43.62
3	W44x224	224.000	65.800	43.31
4	W44x198	198.000	58.000	42.91
5	W40x328	328.000	96.400	40.00
6	W40x298	298.000	87.600	39.69
7	W40x268	268.000	78.800	39.37
8	W40x244	244.000	71.700	39.06

If the right hand column is outside the window, drag outside the window and the table will scroll to bring the column into view.



## Copying and Pasting Numbers

To select the whole table

- **Click in the box at the top left corner**

To copy or paste numbers in Steel Designer, first select the numbers then choose the command you require.

To copy a number or range of numbers

- **Select the numbers to be copied**
- **Choose Copy from the Edit menu**

The numbers will be placed on the clipboard. Each row of numbers will have a return character at the end and the numbers in the row will be separated by a Tab character. You can transfer data copied from Steel Designer into other Macintosh applications or you can paste them into other tables in Steel Designer.

To paste the numbers on the clipboard into a table

- **Select the numbers in the table that will be replaced by the numbers on the clipboard**
- **Choose Paste from the Edit menu**

## Allowable Stresses

Some steel design codes permit you to increase the allowable stresses by a set amount (usually 33 or 50%) for load cases which only involve temporary loading. Steel Designer allows you to utilize this option by using the Allowable Stresses option from the Design menu. This allows you to enter a factor for the allowable stress increase for each load case.

To set the allowable stress increase factors

- **Choose Allowable Stresses from the Design menu**
- **Type in the stress increase factors for the required cases**
- **Use the tab key to move from row to row if you wish**
- **Click OK**

The initial value of the allowable stress increase factor is 1.0 for all load cases. If, for example, you wanted the stresses for a load case to be allowed to increase by 33%, you would enter a value of 1.33.

## Optimum Sections

Once you have selected an optimum weight section for a member using the Design command, the best section will be displayed in the third column of the Design Efficiency table in the Result window. You can refer to this table to compare the selected section with the original section. If you decide that you want to permanently replace the original section with the best section you should use the Use Best Sections command from the Design menu. If you have selected members in the front window you can choose to only update the selected members or you can update the entire frame. In any case, only members, which have been designed, will be updated.

To change sections to the optimum sections designed

- **Choose Use Best Sections from the Design menu**



- **Click the radio button to change just the selected members or the entire frame**

- **Click OK**

The sections of the member's chosen will be changed to the lightest weight sections. After using this command you will have to re-analyse the frame to determine the effect of your change on the structure.

## Finding Design Values

The Find command from the Design menu can be used to automatically search through the structure to find members which have design values exceeding a specified value for the current load case. You can search for actions, deflections, stresses or efficiencies.

To search for a category of members

- **Choose Find from the Design menu**



- **Click on the pop-up menu to choose the category to search for**
- **Click on the radio buttons to set the criteria for the search**
- **Click OK**

After searching through the frame, Steel Designer will select all of the members, which meet the specified criteria.

## Checking a Frame

When you use the Check command you have the option of specifying which design checks will be carried out. The types of checks are grouped into four categories; Bending, Tension, Compression and Combined.

### Bending Checks

- ☒ **Bending**
  - ☒ **Major Bending**
  - ☒ **Major Shear**
  - ☒ **Major Deflection**
  - ☒ **Minor Bending**
  - ☒ **Minor Shear**
  - ☒ **Minor Deflection**

Bending checks are usually used on members which resist the applied loads by flexural and shear action. Typically the horizontal members in a frame will support the live and gravity loads in this way. A member may be subject to flexure and shear in either the major or minor axis directions (or both) depending the orientation of the section and the direction of the loading. You may specify which of these checks should be performed.

#### Tension Checks

##### ☒ Tension

Tension checks are performed on members which are subject to axial tension. This would include members such as bracing and members in trusses, which are under tension.

#### Compression Checks

##### ☒ Compression ☒ Slenderness ☒ Compression

Compression checks are used on members, which support axial compression. Bracing in frames and compression members in trusses are some of the types of members which are likely to be checked using this option. You may choose to check the slenderness of a member and/or its compressive stress. The slenderness is measured as

$$\text{Slenderness} = \text{Maximum of } \left\{ \begin{array}{l} K_x L / r_x \\ K_y L / r_y \end{array} \right.$$

#### Combined Checks

##### ☒ Combined ☒ Tension & Bending ☒ Compression & Bending ☒ Sidesway

When a member is subject to bi-axial bending or a combination of axial tension or compression and bending, it is likely to be necessary to carry out a combined check on the member's performance as a beam-column. This combined check usually takes the form of a comparison of the sum of the ratios of the actual stress to the allowable stress for each of the considered actions. As columns are frequently subject to these type of actions, there is also an option to check the sidesway of a beam-column. The sidesway check usually takes the form of a comparison of the horizontal deflection at the top of the member with a proportion of its height above ground level.

#### Design Codes

Steel Designer supports checking and designing of your structure in accordance with a range of design codes. At present, Steel Designer allows you to use

- **AII (Architectural Institute of Japan 198)**
- **ASD (American Institute of Steel Construction Allowable Stress Design, 8th Ed 1989)**

- **AS4100 (Australian Steel Design Code Standards Australia 1990)**
- **User codes**

## User Code

At times, you may find you want to carry out design checks, which are different from those prescribed in the standard codes. To facilitate this, Steel Designer has an additional code named User, which lets you enter design rules and check members according to these rules.

To activate the User code, choose User from the Code menu. Now whenever you do any checking or designing, Steel Designer will use the User code rules to determine a member's efficiency. You can view and edit the design rules in the User code by choosing the Edit User Code item from the Code menu. The rules in the User code are grouped into the four groups which appear in the Check and Design dialogs, that is Beams, Ties (or tension) Struts (or compression) and Beam-Columns (or combined).

To edit the User code

- **Choose Edit User Code... from the Code menu**



- **Click on the button of the part of the code you wish to change**
- **Type in new rules or modify the existing design rules**

The syntax of the design rules is the same as that of the Calculation sheet in Multiframe. This is very similar to the format used in most programming languages and spreadsheets. The following variables are available to help you construct your design rules. These variables are evaluated for each member as the member is checked.

Variable	Value
L	Length of member*
Kx	Effective length factor in major plane
Ky	Effective length factor in minor plane
Lbx	Unbraced length for buckling about the major axis*
Lby	Unbraced length for buckling about the minor axis*
rx	radius of gyration about major axis*
ry	radius of gyration about minor axis*
E	Young's modulus of steel
ft	maximum tensile stress
fc	maximum compressive stress
fbx	maximum bending stress about major axis
fby	maximum bending stress about minor axis
fy	yield stress of the steel
fu	ultimate tensile strength of the steel

y	height of the highest end of the member above $y=0^*$
a	web stiffener spacing*
$C_b$	bending coefficient
$C_{mx}$	major interaction coefficient
$C_{my}$	minor interaction coefficient

Note that all length variables (marked with an asterisk \* above) are given values in the same units as the units for deflection as specified in the Units dialog. This ensures that the dimensions of the resulting calculations will be consistent. All stresses and strengths have units as set for the Stresses option in the Units dialog.

The four different parts of the User code correspond to the four groups of checks available when using the Check and Design commands.

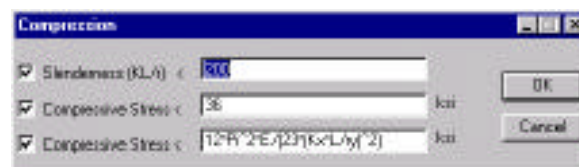
The bending checks can be used to check bending stresses, shear stresses and deflections. These formulas will be applied to both the major and minor axis beam calculations.



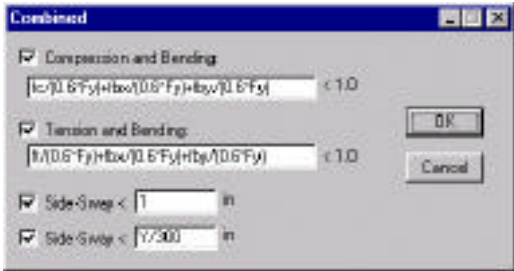
The tension checks will be used to evaluate the tensile stress on the member.



The compression checks will be used for the Slenderness and Compression check options when using the Check and Design commands.



The combined checks will be used for the Combined check options when using the Check and Design commands. The combined stress checks check the user formula against a combined stress ratio (CSR) of 1.0.



Only the calculations which have their check box checked will be used when you use the Check or Design commands.

Printing

The contents of any of the windows in Steel Designer can be printed. When printing tables they will be printed over multiple pages as there will generally not be enough room on one page for all the rows and columns in the table.

Setting up the Printer

Before printing you should set up the printer with the page size and orientation that you wish to use.

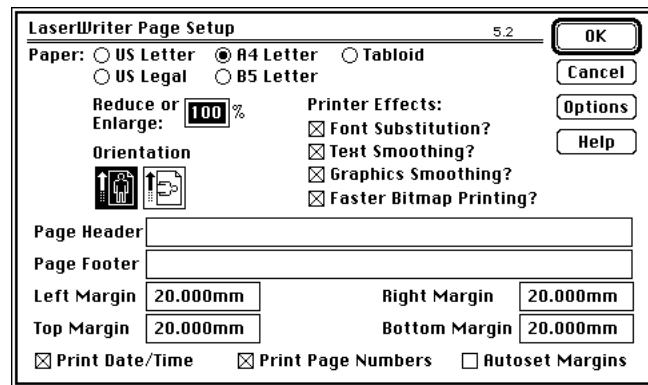
To set up the printer

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

The standard page setup dialog will appear allowing you to enter the appropriate information. On the Macintosh you may also enter text to be displayed at the head and foot of each page and set the width of the margins on the page. On Windows you can do this in the Print dialog (see below).



Windows

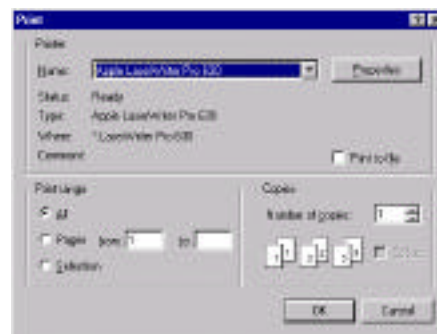


Macintosh

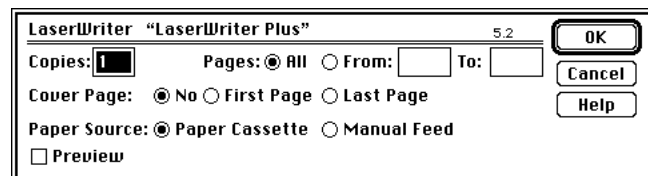
## Printing a Window

To print the contents of the frontmost window

- **Choose Print Window from the File menu**



Windows



Macintosh

On Macintosh, you can check the Preview button to preview your printed output on screen, on Windows this happens automatically.

On Windows, you may also enter text to be displayed at the head and foot of each page and set the width of the margins on the page. On Macintosh, you can do this in the Page Setup dialog (see above).

## Saving your Work

You can save your design work at any time and then open the frame later to continue where you left off. Note however, that if you re-analyse a frame in Multiframe and save it using that program, the design information in the file will be lost and you will need to enter the design information again in Steel Designer.

To save the frame and its design information to disk

- **Choose Save from the File menu**

The frame will be saved to disk complete with the design information you added to it.

**Hint:** You can copy and paste the contents of the Design Details table in the Data window into the scrapbook for later use. This means, if you know you are going to make some changes to the frame and open it in Steel Designer once again and the number of members remains the same, you can paste the data back into the Design Details table from the scrapbook.

## **Saving the report**

You can also save the report to disk and recall it at a later date.

To save the report to disk

- **Ensure the Report window is in front**
- **Choose Save from the File menu**

The report will be saved to disk. Use the Open command to read the report in again. If you need to transfer the data in the report to another program like Microsoft Word, use the Select All and Copy and Paste command to paste the data into the other program. Steel designer places the report data on the clipboard in the RTF (Rich Text) format.



## **Chapter 3**

# **Steel Designer Reference**

---

This chapter summarises the overall structure, windows, toolbars and menu commands of Steel Designer. (The Toolbars are only available in the Windows version of Steel Designer.)

## Windows

---

### **Frame Window**

Steel Designer uses a range of windows.

This window is used for specifying the sections and design properties of the members in a frame.

### **Data Window**

This window is used for viewing the data describing the geometry and loading of the frame and for displaying and editing the design properties of the structure.

### **Load Window**

This window is used for viewing the loading applied to the frame. One load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu.

### **Result Window**

This window is used for viewing the results of the analysis and design calculations carried out on the frame. The results for one load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu.

### **Plot Window**

This window is used for viewing diagrams of the results of the analysis carried out on the frame. The results for one load case at a time may be viewed in this window. You can choose which load case is displayed by choosing the appropriate item from the bottom of the Case menu.

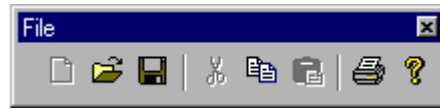
### **Report Window**

This window is used for viewing a summary report of the design checks carried out on the frame. You can turn on or off the option to create a summary report when you use the Check or Design commands.

## Toolbars

Users of the Windows version of Steel Designer 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.

### File



New - Open - Save - Cut - Copy - Paste - Print Preview - About

### Formatting



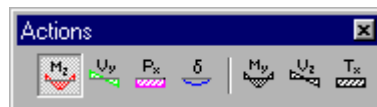
Font - Font Size - Bold - Italic - Underlined - Colour - Align Left - Align Center - Align Right - Bullets

### View



Zoom - Shrink - Pan - Size To Fit - Toggle Clipping - Toggle Masking

### Actions



$M_z'$  -  $V_y'$  -  $P_x'$  - Deflection -  $M_y'$  -  $V_z'$  -  $T_x'$

### Load Case



Select Load Case - Previous Load Case - Next Load Case

## Menus

---

Steel Designer uses the standard set of Macintosh or Windows menu commands for File, Edit and Windows operations. It also has a range of menus for:

### Apple Menu

(Macintosh Only) The Apple menu provides access to the standard desk accessories and control panels.

About Steel Designer

Tells which version of Steel Designer you are using.

Desk Accessories

The usual list of desk accessories will appear under the Apple menu

### File Menu

Only available when the Report window is in front. Choose this command to start work on a new report. If you have any changes to the current report unsaved, Steel Designer will prompt you to save any changes before starting the new work.

New

(Windows: Ctrl N / Macintosh: command N)  
The New command is used to open a new Steel Designer report.

Open

(Windows: Ctrl O / Macintosh: command O)  
When the Report window is in front  
Opens a report which has previously been saved on disk.  
Otherwise  
Opens a frame which has previously been saved to disk.

If you have any changes to the current frame or report unsaved, Steel Designer will prompt you to save any changes before opening the new file. Steel Designer can read Multiframe 2D, 3D and 4D files.

Open Sections Library

(Windows Only)  
The Open Library command opens a Multiframe Sections Library.

Close

When the Report window is in front  
Close the report which is open in this window.  
Otherwise  
Closes the frame which is currently open.

Save

(Windows: Ctrl S / Macintosh: command S)  
When the Report window is in front  
Save the current report with the same name you saved with last time or...  
the same name the file had when you opened it.  
Otherwise  
Save the current frame with the same name you saved with last time or...

the same name the file had when you opened it.

#### Save As

When the Report window is in front  
Save the current report with a new name.  
Otherwise  
Save the current frame with a new name.

#### 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.

#### 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

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.

#### Exit/Quit

Leave Steel Designer and go back to the Finder. If you have any work unsaved, Steel Designer will prompt you to save any changes before quitting.

### Edit Menu

The Edit menu provides commands for 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.

#### Redo

(Windows Only)  
(Windows: Ctrl Y)  
Return to the last action you carried out before selecting Undo from the Edit menu.

#### Cut

(Windows: Ctrl C / Macintosh: command C)  
Remove the current selection and place it on the clipboard

#### Copy

(Windows: Ctrl V / Macintosh: command V)  
Copy the current selection to the clipboard

#### Paste

Paste the contents of the clipboard into the current selection or at the current insertion point.

#### Clear

Remove the current selection without placing it on the clipboard.

Show/Hide Clipboard	Make the clipboard visible/invisible.
Select	See “Select Submenu”
Find	<p>(Windows Only)</p> <p>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.</p> <p>The equivalent Macintosh command can be found under the Design Menu.</p>
Fill Down	Copy the top cell of the selected cells in the Data window and repeatedly paste it down the selection.
Fill Right	Copy the left cell of the selected cells in the Data window and repeatedly paste it right across the selection.
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 Colours	Allows you to set the colours to be used for display of section types in the Frame window.
Preferences	Allows you to set how moment diagrams are drawn, what type of moment symbols to use, how member details are computed and what fonts and sizes should be used for printing.
<b>Select Submenu</b>	<p>The Select menu has commands for automatically selecting parts of the structure in the Frame window. Steel Designer includes a new command to select a member by number.</p>
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 given member in the front window. You can type in the number of the member in the dialog which appears.
Joint	Allows you to select a joint by number.
Member Label	(Windows Only) Selecting a member using the member label.
<b>View Menu</b>	The View menu provides commands for working in the graphics windows.
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 E) 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 R) 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.
Clipping	See "Clipping Submenu"
Masking	See "Masking Submenu"
Axes	Turn on or off the display of axes in the front window.

Ruler	Turn on or off the display of the ruler in the Report window.
Font	Set the font size and style for the text in the front window.
Numbers	Set the type of numeric format you would like to use to display numbers in the front window. You can choose to use decimal or scientific notation and specify how many digits of precision you wish to display.
Units	Choose to work in a range of metric or English units.
Colours	Specify which colours 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) Makes the toolbar visible or invisible. The Toolbar menu is a sub menu of the View menu, and is described in detail following this section on the View menu. See “Toolbar Section”
<b>Clipping Submenu</b>	The clipping menu is used 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 menu is 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.

**UnMask All**

(Windows Only)  
The UnMask All command removes all masking of members and joints.

**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.

**Design Menu**

The Design menu provides commands for checking and optimizing the members in your structure.

**Check**

Check the selected members in the Frame window for their compliance with the current code. You may use the Check dialog to

	choose which design calculations should be carried out and which load cases should be checked.
Design	Select the lightest weight sections for the selected members in the Frame window which will satisfy the design criteria. You may use the Design dialog to choose which design calculations should be carried out and which load cases should be examined.
Bending	Specify the design parameters controlling bending checks. Enter the unbraced lengths for the selected members in the Frame window and specify any web stiffener spacing.
Tension	Specify the design parameters used for tension checks. Specify the area of any bolt holes which must be subtracted from the cross-sectional area of the section when doing design calculations.
Compression	Specify the design parameters controlling compression checks. Allows you to select the effective lengths and the unbraced lengths for the selected members in the Frame window.
Combined	Specify the design parameters controlling combined bending and compression checks.
Serviceability	(Windows Only) The Serviceability command allows you to set design information regarding serviceability of the frame this is currently only used for the AS4100 design code.
Design Details	This command allows you to set all of the design information for the members selected in the Frame window. As a short-cut, you can double click on a member to bring up this design dialog for that member.
Section Type	Use this command to manually change the section type for the selected members in the Frame window. You will need to re-analyse after choosing this command.
Steel Grade	Specify the grade of steel for the selected members in the Frame window. You can choose from a list of standard grades or enter custom values for the yield and ultimate tensile strength.
Constraints	Specify whether there are any constraints on the size of section which may be chosen for the selected members. You can also specify if you require all of the selected members to be of the same section type.

Member Labels	<p>(Windows Only)</p> <p>Allows you to edit the user defined label associated with each member.</p>
Frame Type	<p>Specify whether the current frame is able to sway or is braced against horizontal movement.</p>
Allowable Stresses	<p>This command allows you to specify the allowable stress increase for each load case on the structure. The allowable increase is entered as a factor (usually 1.33 or 1.5).</p>
Capacity Factors	<p>(Windows Only)</p> <p>The Capacity Factors command allows you to modify the capacity factors for the frame, this is only used with limit state design codes.</p>
Use Best Sections	<p>Automatically replace the section type of each member with its lightest weight section as chosen using the design command.</p>
Find	<p>(Macintosh Only)</p> <p>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.</p> <p>The equivalent Windows command can be found under the Edit Menu.</p>
<b>Code Menu</b>	<p>The code menu allows you to select the design code you wish to use for checking.</p>
AS1250	<p>Choose any of these items to make it the current code. The current code is indicated with a check mark beside the item. This determines which code is used when you do design calculations.</p>
AS 4100	<p>Current Australian steel design code.</p>
BS 449	<p>Not currently implemented</p>
NZS 3404	<p>Not currently implemented</p>
BS5 950	<p>Not currently implemented</p>
CISC	<p>Not currently implemented</p>

Eurocode	Not currently implemented
AIJ	Current Japanese steel design code.
ASD	American ASD steel design code.
LRFD	Not currently implemented
User	Allows the user to set their own design criteria and checks.
Edit User Code	This command lets you edit the design calculations which will be used when you choose to check or design a frame using the User code. You can choose which checks should be performed and what calculations should be used for each check. You can type in your own equations
<b>Display Menu</b>	The Display menu lets you control what s displayed in each of the windows.
Symbols	This command brings up a dialog box which allows you to specify which symbols will be displayed on the graphics. You can turn on or off the display of joint and member numbers, loads, restraints, symbols of sections, member axes, section axes and names of sections.
Data	See “Data Submenu”
Results	See “Results Submenu”
Actions	See “Actions Submenu”
Stresses	See “Stresses Submenu”
Efficiency	See “Efficiency Submenu”
Deflection	Display the deflection for the current load case in the Plot window.
Animate	Animate the diagram in the front 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 Only)

If you turn on the QuickTime check box in the Animate dialog, the animation will be saved into a QuickTime movie file. QuickTime is a system extension developed by Apple which allows you to create animations, save them to disk and edit and display them in other programs. The movie will also be 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.

#### 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 on Macintosh or Esc. 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 colour 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.

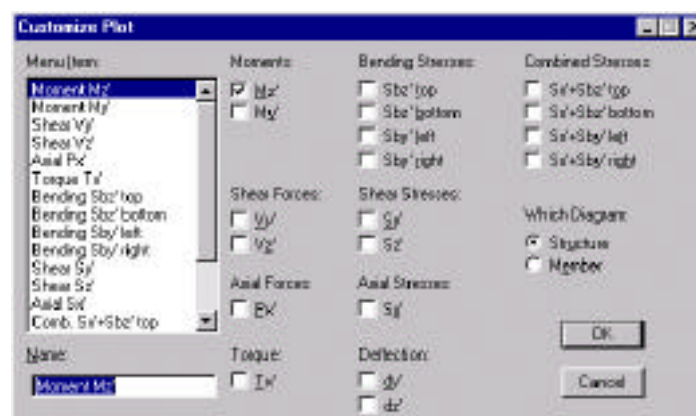
#### Customise Plot

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

This menu 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.

**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.

**Load Case Factors**

The Load Case Factors table allows the user to view and edit the factors used in the combined static load case.

**Design Details**

Display a table of design information for each of the members in the frame. The table includes steel grade, effective and unbraced lengths and limits on the size of the section for the member.

**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 computed member actions and stresses for the current load case for the selected member in the Plot window. The values are displayed at a number of evenly spaced locations along the member.

**Member Efficiency**

Display the computed efficiency of the members in the frame as compared with their required design strengths according to the current code.

**Actions Submenu**

The items in this menu 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.

Display the computed shear forces in the local  $y'$  direction 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.
<b>Stresses Submenu</b>	The items in this menu 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.
<b>Efficiency Submenu</b>	The items in this menu may be used to control which type of efficiency diagram is displayed in the Plot window.
AS 4100	The following items are available in the Efficiency submenu when using the Australian / International version of Steel Designer.
Overall	Display the Overall efficiency as a colour on each member for the current load case in the Plot window.
Bending (Major Section)	Display the Major Bending/Major Section Bending efficiency as a colour on each member for the current load case in the Plot window.
Bending (Major Member)	Display the Major Member Bending efficiency as a colour on each member for the current load case in the Plot window.
Bending (Major Shear)	Display the Major Shear efficiency as a colour on each member for the current load case in the Plot window.
Bending (Minor Section)	Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.
Bending (Minor Shear)	Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.
Tension	Display the Tension efficiency as a colour on each member for the current load case in the Plot window.
Compression (Section)	Display the Compression/Section Compression efficiency as a colour on each member for the current load case in the Plot window.

Compression (Major Member)

Display the Major Member Compression efficiency as a colour on each member for the current load case in the Plot window.

Compression (Minor Member)

Display the Minor Member Compression efficiency as a colour on each member for the current load case in the Plot window.

Combined (Major Section)

Display the Combined (Major Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Minor Section)

Display the Combined (Minor Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Major In-Plane)

Display the Combined (Major In-Plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Minor In-Plane)

Display the Combined (Minor In-Plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Out-of-plane)

Display the Combined (Out-of-plane) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Biaxial Section)

Display the Combined (Biaxial Section) efficiency as a colour on each member for the current load case in the Plot window.

Combined (Biaxial Member)

Display the Combined (Biaxial Member) efficiency as a colour on each member for the current load case in the Plot window.

Primary Deflection

Display the Primary Deflection efficiency as a colour on each member for the current load case in the Plot window.

Secondary Deflection

Display the Secondary Deflection efficiency as a colour on each member for the current load case in the Plot window.

### **ASD / AIJ**

The following items are available in the Efficiency submenu when using USA and Japan versions of Steel Designer.

Overall	Display the Overall efficiency as a colour on each member for the current load case in the Plot window.
Major Bending	Display the Major Bending/Major Section Bending efficiency as a colour on each member for the current load case in the Plot window.
Minor Shear	Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.
Major Deflection	Display the Major Deflection efficiency as a colour on each member for the current load case in the Plot window.
Minor Bending	Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.
Minor Shear	Display the Minor Shear/Bending (Minor Shear) efficiency as a colour on each member for the current load case in the Plot window.
Minor Deflection	Display the Minor Bending/ Minor Section Bending efficiency as a colour on each member for the current load case in the Plot window.
Tension	Display the Tension efficiency as a colour on each member for the current load case in the Plot window.
Slenderness	Display the Slenderness efficiency as a colour on each member for the current load case in the Plot window.
Compression	Display the Compression/Section Compression efficiency as a colour on each member for the current load case in the Plot window.
Bending Tension	Display the combined Bending Tension efficiency as a colour on each member for the current load case in the Plot window.
Bending Compression	Display the combined Bending Compression efficiency as a colour on each member for the current load case in the Plot window.
Sway	Display the Sway efficiency as a colour on each member for the current load case in the Plot window.
<b>Case Menu</b>	The Case menu lets you choose which load case is displayed in the Data, Load, Plot and Result windows.

Analyse	Carry out the analysis to compute all the deflections, moments and forces for the structure for all load cases.
Load Case 1 (or equivalent)	<p>(Macintosh Only) (Macintosh: command 1) Display loads and design information associated with Load Case 1 in the Load and Data windows. Display results for load case 1 in the Plot and Result windows.</p> <p>Similarly for subsequent load cases</p>
Load Case...	<p>(Windows Only) Display's a selection list dialog with all the load cases.</p>
<b>Window Menu</b>	<p>The Window menu lets you select which window is in front on the screen.</p>
Cascade	<p>(Windows Only) Displays all the Windows behind the active Windows.</p>
Title Horizontal	<p>(Windows Only) Layout all visible windows across the screen.</p>
Tile Vertical	<p>(Windows Only) Layout all visible windows down the screen.</p>
Arrange Icons	<p>(Windows Only) Rearranges the icons of any ionized window so that they are collected together at the bottom of the Steel Designer program window.</p>
Editing Layout	Makes the Frame, Data and Load windows all visible at once. This is useful when you are defining the design information for the structure.
Results Layout	Makes the Plot, Result and Load windows all visible. This is useful when you are examining the results of the design calculations.
Report Layout	Makes the Plot, Load and Report windows visible. This is useful when you are reviewing a summary report of design calculations.
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.
Report	Makes the Report window visible and brings it to the front.
Tile	(Macintosh Only) Lays out all of Steel Designer's windows across the screen so that all of each window is visible at once.
Stack	(Macintosh Only) Stacks up all of Steel Designer'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 Steel Designer's windows visible.
Hide All	(Macintosh Only) Hides all of Steel Designer's windows.
<b>Help Menu</b>	(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 Steel Designer help file.
About Steel Designer	(Windows Only) Tells which version of Steel Designer you are using and how many joints, members and forces etc. there are in the structure.



# Chapter 4

## Steel Calculations

---

This chapter describes the calculation methods used by Steel Designer.

### Properties for Design

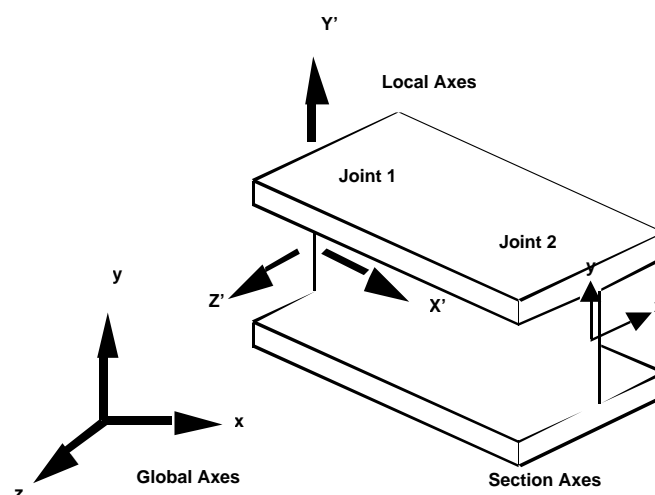
When checking or designing structures, Steel Designer uses sections properties stored in the Sections Library. The key properties used by Steel Designer are:

Name	Property
A	Cross sectional area
I <sub>x</sub>	Major moment of inertia
I <sub>y</sub>	Minor moment of inertia
E	Young's Modulus
D	Depth
B	Breadth or Width
t <sub>f</sub>	Flange thickness
t <sub>w</sub>	Web thickness
r <sub>x</sub>	Major radius of gyration
r <sub>y</sub>	Minor radius of gyration
r <sub>z</sub>	Radius of gyration about weakest axis

When you add a section to the Sections Library you must ensure that all of the properties above are correctly entered and are all non-zero.

### Axes and Sign Convention

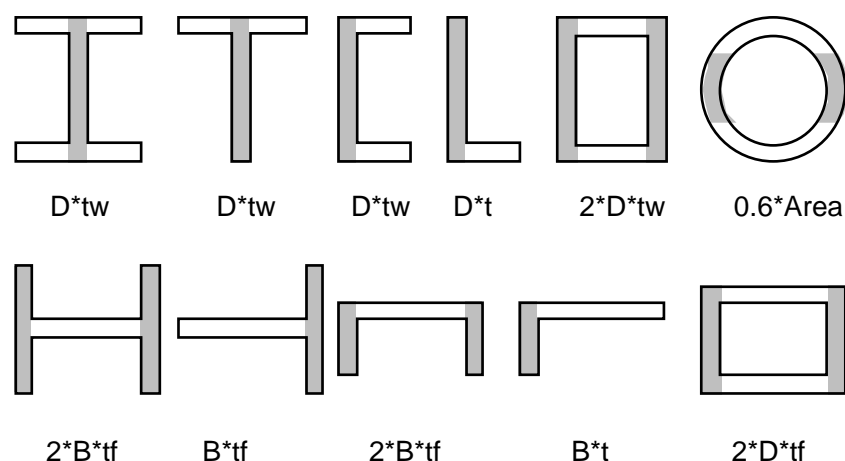
Much of the design information and many of the design variables are described relative to the major and minor axes of the section used for each member. This corresponds to the same terminology used to describe the properties of a section e.g. I<sub>xx</sub> for moment of inertia about the major (or strong) axis and I<sub>yy</sub> about the minor (or weak) axis.



The coordinate systems corresponding to the naming conventions for the various results of analysis, section properties and design values are shown in the diagram above. Structure coordinates and global loads are defined relative to the Global Axes, member actions, deflections and stresses resulting from the Multiframe analysis are defined relative to the local member axes and design values and section properties are defined relative to the section axes. Whenever a design variable carries a subscript this indicates that it applies to the corresponding section axis.

## Shear Area

When calculating shear stresses for comparison with allowable shear stresses, Steel Designer uses the following shear areas or the full sectional area for other sectional shapes.



## Code Clauses Checked

When carrying out code checks, Steel Designer uses the following clauses of the applicable codes to check your structure. No other checks are performed unless they are specifically listed below.

ASD

"Specification for Structural Steel Buildings, Allowable Stress Design and Plastic Design", American Institute of Steel



Construction, June 1, 1989 (contained in Manual of Steel Construction, Allowable Stress Design, 1989, 9th Edition).

Clauses used are A5.1, A5.2, B1, B3, B5, B7, C2, D1, E1, E2, F1, F2, F3, F4, G1, G2, G3, H1, H2

The design checking procedure is as follows;

The section is classified and tensile area and limiting slenderness ratios are determined according to section B.

For major and minor bending checks, the bending stress is checked to be less than the allowable  $F_b$  as found in sections F1, F2 and F3.

For major and minor shear, the shear stress is checked to be less than the allowable  $F_s$  found from section F4. The shear stress is computed using a shear area as shown above.

For major and minor deflection due to bending, the maximum deflection is checked to be less than  $L/300$ . No specific check is made for cantilevered members.

For tension checks, the tensile stress is checked to be less than the allowable  $F_t$  on both the gross and net areas as computed in section D1.

For slenderness checks, the slenderness ratio is computed as the maximum of  $K_x L/r_x$  and  $K_y L/r_y$ . This is checked to be less than the allowable slenderness ratio of 200 for compressive members or 300 for tensile members in accordance with clause E1.

For compression checks, the compressive stress is checked to be less than the allowable  $F_a$  as computed in section E2.

For combined compression and bending checks, the stresses are checked to be low enough to satisfy equations H1-1 to H1-3.

For combined tension and bending checks, the stresses are checked to be low enough to satisfy equation H2-1.

For sway checks, the horizontal deflection of the highest part of the member is checked to be less than  $Y/300$  where  $Y$  is the height of the highest part of the member above the plane  $y=0$ .

Checks are not carried out on hybrid members, composite members or tapered members.

## AIJ

"Design Standard for Steel Structures", Architectural Institute of Japan, March 1979.

Clauses used are 5.1, 5.6, 6.1, 6.2, 8.1, 10.1, 11.1, 11.2, 11.3

The design checking procedure is as follows;

Allowable stresses are determined from table 5.1 and according to equations 5.1, 5.2, 5.3, 5.4, 5.5, 5.6, 5.7 and 5.8 as appropriate.

For major and minor bending checks, the width-thickness ratio of the section's elements are checked in accordance with equations 8.1, 8.2, 8.3, 8.5 and 8.6 as appropriate. The bending stress is checked to be less than the allowable  $f_b$  as found in section 5.1.4.

For major and minor shear, the shear stress is checked to be less than the allowable  $f_s$  found from equation 5.2. The shear stress is computed using a shear area as shown above.

For major and minor deflection due to bending, the maximum deflection is checked to be less than  $L/300$  in accordance with clause 10.1. No specific check is made for cantilevered members.

For tension checks, the tensile stress is checked to be less than the allowable  $f_t$  as computed using equation 5.1.

For slenderness checks, the slenderness ratio is computed as the maximum of  $K_x L/r_x$  and  $K_y L/r_y$ . This is checked to be less than the allowable slenderness ratio of 200 for vertical members or 250 for non-vertical members in accordance with clause 11.2 (A vertical member is assumed to be one which is within 100mm of vertical).

For compression checks, the compressive stress is checked to be less than the allowable  $f_c$  as computed in equation 5.3 or 5.4.

For combined compression and bending checks, the stresses are checked to be low enough to satisfy equations 6.1 and 6.2.

For combined tension and bending checks, the stresses are checked to be low enough to satisfy equations 6.3 and 6.4. The area of bolt holes as specified in the Bolt Holes dialog is deducted from the gross section area to calculate the net section area.

For sway checks, the horizontal deflection of the highest part of the member is checked to be less than  $H/300$  where  $H$  is the height of the highest part of the member.

## AS4100

"Australian Standard AS4100-1990: Steel Structures", Standards Australia, October 26, 1990 including Amendment No.1 (August 3, 1992), Amendment No.2 (June 14, 1993) and Amendment No.3 (December 5, 1995).

Clauses used are 4.4, 4.6, 5.1, 5.2, 5.3, 5.6, 5.11, 6.1, 6.2, 6.3, 7.1, 7.2, 7.3, 8.3 and 8.4

The design checking procedure is as follows;

For first order analyses, the design bending moments are amplified using the factors determined using clause 4.4.2 and 4.6.2. Amplification factors for sway frames are not considered and a second order analysis should be used for sway frames requiring moment amplification.

The section is classified as compact, non-compact or slender about its major and minor axes using clause 5.2. The effective area and form factors are determined using clause 6.2.

For major and minor bending section checks, the design bending moment is checked to be less than the nominal section moment design capacity as found using clause 5.2.

For bending member checks, the design bending moment about the major principle axis is checked to be less than the nominal member moment design capacity as found using clause 5.3. and 5.6. **Clause 5.6.3 and clause 5.6.4 are NOT considered.**

For major and minor shear checks, the design shear force is checked to be less than the nominal shear capacity found from section 5.11. The flange restraint factor ( $\phi$ ) of clause 5.11.5.2 is always set to 1.0.

For tension checks, the design axial tension force is checked to be less than the nominal section design capacity in tension as computed using clause 7.2.

For compression section checks, the design axial compressive force is checked to be less than the nominal section design capacity in compression as computed using clause 6.2.

For major and minor compression member checks, the design axial compressive force is checked to be less than the nominal member design capacity in compression as computed using clause 6.3. **Clause 6.3.4 is NOT considered.**

For all combined action section checks, the design axial force ( $N^*$ ) is the maximum axial force in the member, and the design bending moments ( $M_x^*$ , and  $M_y^*$ ) are the maximum bending moments in the member.

For major and minor combined section checks, the design bending moment is checked to be less than the nominal section moment design capacity reduced by axial force (compression or tension) as computed using clause 8.3.2 and 8.3.3.

For combined biaxial section checks, the design bending moments are checked to satisfy clause 8.3.4.

For major and minor combined in-plane member checks, the design bending moment is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.2. **Clause 8.4.3 is NOT considered.**

For combined out-of-plane member checks, the design bending moment about the major axis is checked to be less than the nominal in-plane member moment design capacity as computed using clause 8.4.4.

For combined biaxial member checks, the design bending moments are checked to satisfy clause 8.4.5.

**Clause 8.4.6 is NOT considered.**

Checks are not carried out on hybrid members, composite members or tapered members.

Checks on monosymmetric I sections are not considered.  
Checks using plastic analysis are not considered.

## References

You may find the following books useful to refer to if you need information on the methods used to check members in Steel Designer.

- **Manual of Steel Construction, Allowable Stress Design**  
American Institute of Steel Construction, New York, 1989, 9th Edition
- **Manual of Steel Construction, Load & Resistance Factor Design**  
American Institute of Steel Construction, New York, 1986, 1st Edition
- **Steel Buildings, Analysis and Design**  
S W Crawley & R M Dillon, John Wiley & Sons, New York, 1984, 3rd Edition
- **Structural Steel Design, LRFD Fundamentals**  
J C Smith, John Wiley & Sons, New York, 1988, 1st Edition
- **The Behaviour and Design of Steel Structures**  
N S Trahair and M A Bradford, Chapman and Hall, London, 1988
- **Australian Standard AS4100-1990: Steel Structures**  
Standards Australia
- **Steel Designers Handbook**  
B.Gorenc, R. Tinyou and A. Syam, UNSW Press, Sydney, 1996, 6th Edition
- **Design Capacity Tables for Structural Steel. Volume 1: Open Sections**  
Australian Institute of Steel Construction, Sydney, 1994, 2nd Edition
- **Design Capacity Tables for Structural Steel Hollow Sections**  
Australian Institute of Steel Construction, Sydney, 1992, 1st Edition

# Index

---

About Steel Designer.....	38, 55	Clip To Window.....	42
About this manual.....	1	Clipping.....	41, 42
Actions.....	37, 46, 49	Close.....	38
Actions Submenu.....	49	Code Menu.....	45
Add Section.....	40	Colours.....	42
AII.....	45, 59	Column Widths.....	24
Allowable Stresses.....	27, 45	Comb Sx' + Sby' left.....	50
Analyze.....	53	Comb Sx' + Sby' right.....	50
Animate.....	46	Comb Sx' + Sbz' bottom.....	50
Apple Menu.....	38	Comb Sx' + Sbz' top.....	50
area reduction coefficient.....	10	Combined.....	44
Arrange Icons.....	54	Combined (Biaxial Member).....	52
AS 4100.....	45, 50	Combined (Biaxial Section).....	52
AS1250 to User.....	45	Combined (Major In-Plane).....	51
ASD.....	45	Combined (Major Section).....	51
<b>ASD / AII</b> .....	52	Combined (Minor In-Plane).....	52
Axes.....	41	Combined (Minor Section).....	51
Axes and Sign Convention.....	57	Combined (Out-of-plane).....	52
Axial Px'.....	49	Combined Actions.....	12
Axial Sx'.....	50	Combined Checks.....	29
Bending.....	8, 44	Compression.....	11, 12, 44, 53
Bending (Major Member).....	51	Compression (Major Member).....	51
Bending (Major Section).....	51	Compression (Minor Member).....	51
Bending (Major Shear).....	51	Compression (Section).....	51
Bending (Minor Section).....	51	Compression Checks.....	29
Bending (Minor Shear).....	51	Constraints.....	15, 44
Bending Checks.....	28	Coordinate Systems.....	5
bending coefficient.....	10	Copy.....	39
Bending Compression.....	53	Copying and Pasting.....	27
Bending Sby' left.....	50	Customize Plot.....	47
Bending Sby' right.....	50	Cut.....	39
Bending Sbz' bottom.....	50	Data.....	46, 54
Bending Sbz' top.....	49	Data Submenu.....	48
Bending Tension.....	53	Data Window.....	21, 36
bolt holes.....	10	Deflection.....	46
BS 449.....	45	Delete Section.....	40
BS5 950.....	45	Design.....	44
Capacity Factors.....	45	Design Details.....	44, 48
Cascade.....	53	Design Menu.....	43
Case Menu.....	53	Design Properties.....	13
Check.....	43	Designing a Frame.....	18
Checking a Frame.....	16, 28	Desk Accessories.....	38
CISC.....	45	Display Menu.....	46
Clear.....	39	Edit Menu.....	39
Clip Gray.....	42	Edit Section.....	40
Clip Invisible.....	42	Edit User Code.....	46
Clip To Frame.....	42	Editing Layout.....	54
Clip To Selection.....	43	Efficiency.....	17, 46, 49

Efficiency Menu.....	50	Minor Bending.....	52
Eurocode.....	45	Minor Deflection.....	53
Exit/Quit.....	39	Minor Shear.....	52
File.....	37	Moment My'.....	49
File Menu.....	38	Moment Mz'.....	49
Fill Down.....	40	New.....	38
Fill Right.....	40	No Clipping.....	42
Find.....	40, 45	No Masking.....	43
Finding Design Values.....	28	Numbers.....	42
Font.....	42	NZS 3404.....	45
Formatting.....	37	Open.....	38
Frame.....	54	Open Sections Library.....	38
Frame Type.....	15, 45	Optimization.....	19
Frame Window.....	21, 36	Optimum Sections.....	27
Fu.....	14	Overall.....	50, 52
Fy.....	14	Page Layout.....	39
Help Menu.....	55	Page Setup.....	39
Hide All.....	55	Pan.....	41
Installing Steel Designer.....	4	Paste.....	39
Joint Displacements.....	48	Plot.....	47, 54
Joint Loads.....	48	Plot Window.....	22, 36
Joint Reactions.....	49	Preferences.....	40
Joints.....	48	Primary Deflection.....	52
Kx.....	12	Print.....	39
Ky.....	12	Print Window.....	33, 39
Lateral Restraints (AS4100).....	8	Printing.....	32
Lbx.....	8	Reading a Structure.....	5
Lby.....	8	Redo.....	39
Lcx.....	12	References.....	62
Linked Joints.....	48	Render.....	47
Load.....	54	Report.....	54
Load Case.....	37, 53	Report Layout.....	54
Load Case 1.....	53	Report Window.....	23, 36
Load Case Factors.....	48	Restraints.....	48
Load Window.....	22, 36	Result.....	54
LRFD.....	46	Result Window.....	22, 36
Major Bending.....	52	Results.....	46
Major Deflection.....	52	Results Layout.....	54
Mask Gray.....	43	Results Submenu.....	48
Mask Invisible.....	43	Ruler.....	41
Mask Out Selection.....	43	Save.....	38
Mask To Frame.....	43	Save As.....	39
Mask To Selection.....	43	Saving the report.....	34
Mask To Window.....	43	Saving your Work.....	19, 33
Masking.....	41, 43	Secondary Deflection.....	52
Member Details.....	49	Section Colors.....	40
Member Labels.....	44	Section Type.....	13, 44
Member Loads.....	48	Sections.....	48
Members.....	48	Select.....	40
Menus.....	38	Select All.....	40
		Select Horizontal.....	40
		Select Joint.....	41

Select Member.....	41	Tension.....	10, 44, 51, 53
Select Member Label.....	41	Tension Checks.....	29
Select Section.....	41	Text Styles.....	24
Select Sloping.....	41	Thermal Loads.....	48
Select Vertical.....	40	Tile.....	54
Serviceability.....	44	Tile Verticle.....	54
Setting Properties.....	6	Title Horizontal.....	54
Shear Area.....	58	Toolbar.....	42
Shear $S_y'$ .....	50	Toolbars.....	37
Shear $S_z'$ .....	50	Torque $T_x'$ .....	49
Shear $V_y'$ .....	49		
Shear $V_y'$ .....	49	ultimate tensile strength.....	14
Shear $V_z'$ .....	49	Unbraced Length.....	12
Show All.....	55	Unbraced Length (ASD,AIJ).....	8
Show/Hide Clipboard.....	40	Undo.....	39
Shrink.....	41	Units.....	42
Size To Fit.....	41	UnMask All.....	43
Slenderness.....	53	Use Best Sections.....	27, 45
Springs.....	48	User.....	46
Stack.....	54		
Status Bar.....	42	View.....	37
Steel Grade.....	14, 44	View Menu.....	41
Stresses.....	46, 49		
Stresses Submenu.....	49	Window Menu.....	53
Sway.....	53		
Symbols.....	46	yield strength.....	14
Table Of Contents.....	55	Zoom.....	41
Tables.....	24		