

Introduction

The STAAD.Pro Steel Designer is an integrated and interactive tool that works from within the STAAD.Pro environment similar to the STAAD.Pro Reinforced Concrete Designer (RC Designer). In the past, all steel and concrete design were included as part of the original STAAD run which meant that if there were changes in the steel design parameters, the entire analysis and design would have to be rerun. The new Steel Designer is interactive allowing the user to change parameters interactively after the analysis to perform code checks or member selections. The batch mode is still available in STAAD.Pro 2005. The data passed into the Steel Designer includes the geometry, section and material properties, loads and combination information and analysis results.

The advantage of using the Steel Designer over the GROUP command or the batch mode include the following:

- 1) When using the GROUP command with a series of individual beam components, you need to use the MEMBER SELECT, GROUP and CODE CHECK commands successively. The Steel Designer replaces the need for all of these commands. It also selects a new property based on the information (forces and deflection) of each individual component.
- 2) The force and deflection diagram can be seen for the entire physical member rather than individual components.
- 3) When using the GROUP command, you have to assign the bracing conditions separately for each beam element. With the steel designer, you only have to do it once.

This manual is divided into three sections for quicker reference.



Section 1

How to use Steel Designer



Section 2

Engineering Information



Section 3

Reference



Section 1. How to Use Steel Designer

Starting STAAD.Pro Steel Designer

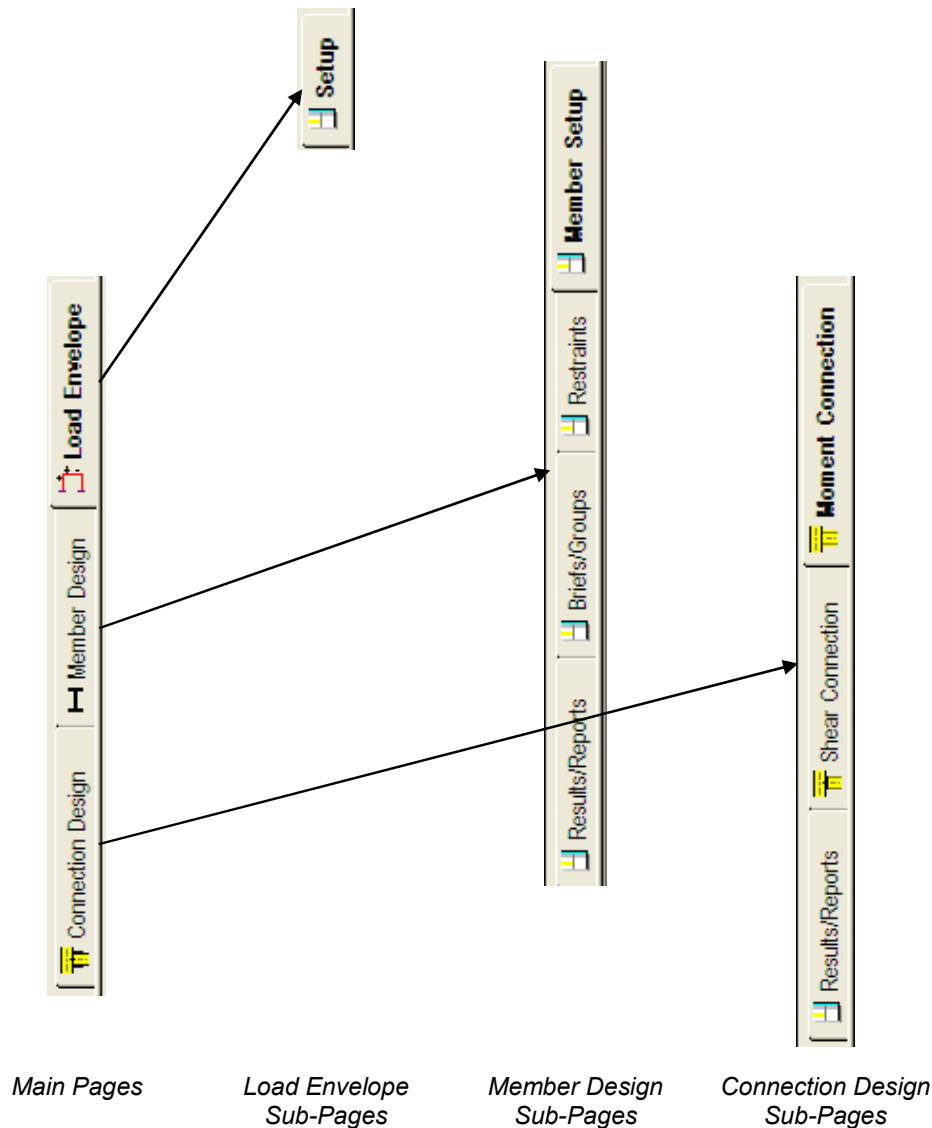
To work with STAAD.Pro Steel Designer, it is essential to be ready with all possible analysis results. After completion of a successful analysis of a STAAD file, launch the Steel Designer from **Mode | Interactive Design Modes | Steel Design** in the main menu.



Page Control Layout

Once the steel design mode is active, a set of vertical tabs will be visible on the left-hand side. The tabs on the furthest left are the main pages while the tabs to the right of the main pages are the sub-pages. The list of pages and their corresponding sub-pages are given below:

Pages	Sub-Pages
1. Load Envelope	1a. Setup
2. Member Design	2a. Member Setup 2b. Restraints 2c. Briefs/ Groups 2d. Results/ Reports
3. Connection Design	3a. Moment Connection 3b. Shear Connection 3c. Results/ Reports



Similar to STAAD.Pro, a good rule of thumb is to follow the pages and sub-pages from top to bottom to ensure you are providing STAAD.Pro with a complete set of information necessary for a successful design of physical steel members and available connections.

There are three stages in completing the entire procedure in the Steel Designer. They have been grouped inside three pages. These pages are:

- 1) Load Envelope
This is used to create set(s) from available Primary and Combination Loads in the current STAAD.Pro file for which analysis results are already available.
- 2) Member Design
Different sub-pages of this page will allow to set the overall parameters of the design including which design code is to be implemented and which members are to be grouped and designed together. Once a Design Group is defined and assigned a

Design Brief, then the Design can be done and a report with full step-by-step calculations will be reported.

3) Connection Design

This page will help to perform different connection designs for different country specific codes and specifications along with a detailed set of step-by-step calculations.

It is also possible to switch off Page Control from the Mode Menu if the additional space on the left side of the screen is required.

Load Envelope Page

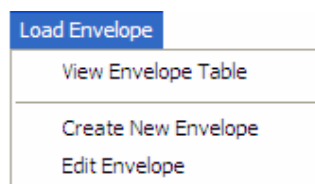
There is only one sub-page under the Load Envelope page. Envelopes are required for Design Groups to find out the design moments and shear forces for different sections of the member. The Setup sub-page is the only page under the Load Envelope page.

Setup (Sub-page # 1)

The Setup sub-page under the Load Envelope page allows the user to define envelopes of basic load cases and combination load cases that have been defined in the analysis model. Envelopes are required for defining Design Groups later and indicate to the program which of the load cases/combinations should the design be checked against.

Keyboard Shortcut ALT+L

List of the sub-menu items under the Load Envelope menu item (shown below) are described in the following sections.



View Envelope Table

The View Envelope Table is used to view a table containing all the load envelopes created by the user for the current model. This table will also come up even if there is no load envelope created yet. You can click on the New Envelope button to create a new load envelope containing any or all of the primary load cases as well as combined load cases.

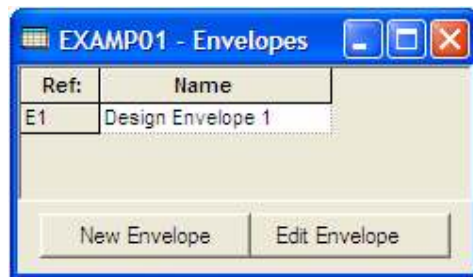
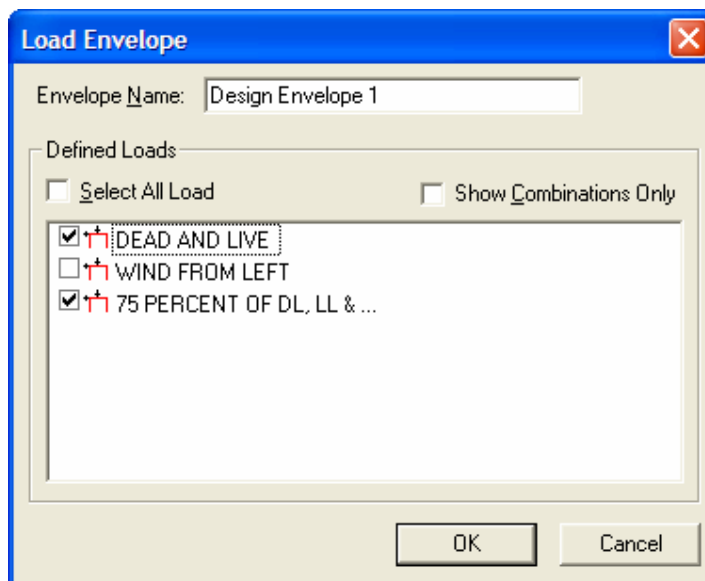


Table Tabs

New Envelope: Create a new Envelope of Basic and Combined Load Cases

Edit Envelope: Edit an existing Load Envelope that is selected by highlighting the row from the Envelopes table

Create New Envelope (after clicking on the New Envelope button): The Create New Envelope is used to define an envelope of primary and combined load cases used in the analysis that will now be used by the Steel Designer for performing code checks or member selections.



Dialog Box Options

Name:
Provide a name for the Design Load Envelope

Load:
All the available primary load cases and combinations

Select All Loads:
Check box to select or unselect the entire displayed load

Show Combination Only:

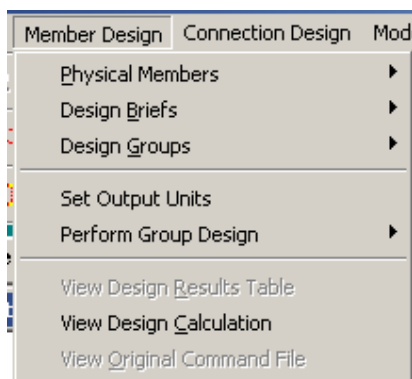
By default this check box is unselected. If checked, only Load Combinations will be shown.

Member Design Page

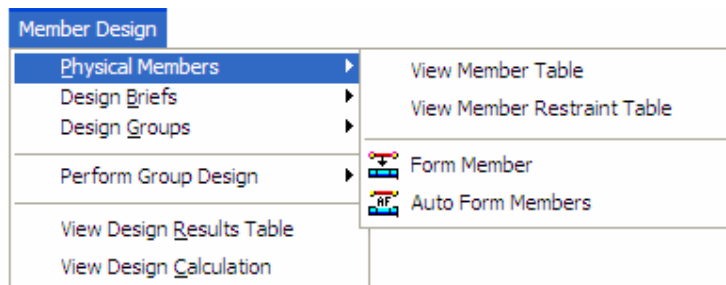
Once the load cases and combinations under which the steel design is to be checked against are provided (Design Load Envelopes), the actual physical members need to be created along with their restraint and specification attributes. The Member Design page consists of four sub-pages to help facilitate this input. As of STAAD.Pro 2003 (US Version), only the AISC ASD and IS800 standards are supported. The four sub-pages under the Member Design page are the following:

Member Setup
Restraints
Briefs/ Groups
Results/ Reports

Member (Sub-page #1)

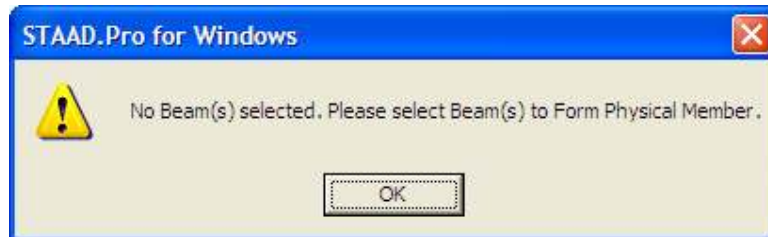



The Member sub-page allows the user to combine together a number of individual beam elements into one physical member. In STAAD.Pro or any other structural software, beams must be broken down into smaller beams if there are other columns or beams framing into it. The reason for this is so that any applied loading can flow from through these connection or break points. Without a connection point (a node or joint) at the intersection of two or more framing beams, there would be an improper load transfer which could lead to susceptible forces and moments. The problem with this is when it comes to designing these individual beam elements one must assign the restraints and design parameters individually for each component. With the new physical member, the individual beam elements are designed as one monolithic physical member. STAAD.Pro can either form these physical members from individual beams automatically or the user can do it manually.

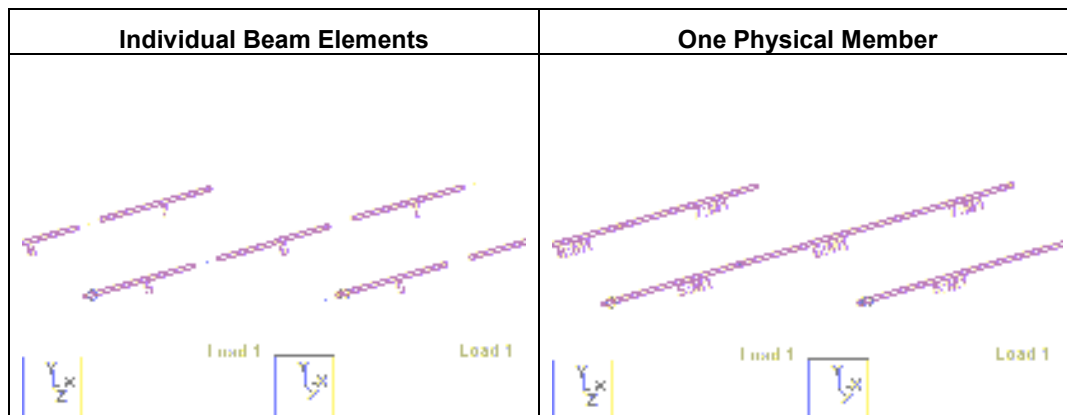


Form Member

The Form Member option is used to manually form a physical structural member from a selection of one or more connected beam elements. The formed member is known as a Physical Member. Beam Elements must be selected before this command will run successfully. Otherwise a dialog box shall pop-up that is shown below.



To manually create a physical member, select the individual beams that will constitute a physical member and either right click the mouse button and choose  (Form Member) or select the same icon from the Steel Design toolbar. If a successful physical member can be created, the physical member label will be shown (i.e "M1") underneath each of the individual beam elements. If the selection of elements is not connected, the form member command uses the first connected part of the selection only. A warning regarding this appears.



To label the physical members in the structure, use the Member Number option in the Labels tab (in the Physical Member section), which is available by right-clicking the mouse.

To form a member, the following rules must be obeyed.

- 1) All elements must form a single continuous line. But they do not have to form a straight line. Thus curved members may be formed.
- 2) There must be a free end. Whilst curved members are allowed, they cannot form a closed loop.
- 3) All elements should have the same beta angle.
- 4) All elements must point in the same direction. Check with the orientation labels if necessary. Use the reverse element command on elements that point the wrong way.
- 5) None of the elements can be part of another member.

6) The section properties must be consistent at each element end. Elements can taper along their length, but where one element ends and the next starts, they must have the same section reference.

7) All elements must be made from the same material.

Auto Form Members



The Auto Form Members option forms a number of physical members from a selected range of beam elements. By selecting a group of beams (using any selection tool or criteria), STAAD.Pro will automatically form physical members. Each member will obey the general rules for generating a member specified above, but uses the following rules in addition:

- 1) Vertical elements are converted into columns first.
- 2) Where four elements cross and share a single node, if the elements can form two members, the deeper section will form a continuous member, but the other elements will form parts of two separate members.

View Member Table

The View Member Table option brings up the table containing all relevant data for all the physical members present in that structure.

EXAMP01 - Member Table							
Member No	Beam No	Node A	Node B	Property	Beta	Length m	O. Length m
M1	5	3	4	3	0.000	3.000	9.000
	6	4	5	3	0.000	3.000	
	7	5	6	3	0.000	3.000	
M2	20	7	9	5	0.000	2.250	9.000
	21	9	11	5	0.000	2.250	
	22	10	11	5	0.000	2.250	
	23	8	10	5	0.000	2.250	
M3	3	2	6	1	90.0	6.000	6.000



Member Table Tabs

Member No:

Shows physical member designation. Currently, physical member cannot be renumbered.

Beam No:

Shows number of beam(s) and their numbers used to form that physical member.

Node A:

Start Joint of the particular beam.

Node B:

End Joint of the particular beam.

Property:

Reference number for the section property assigned to that beam

Beta:

Value of Beta Angle for that particular Beam element.

Length:

Span length for the particular Beam.

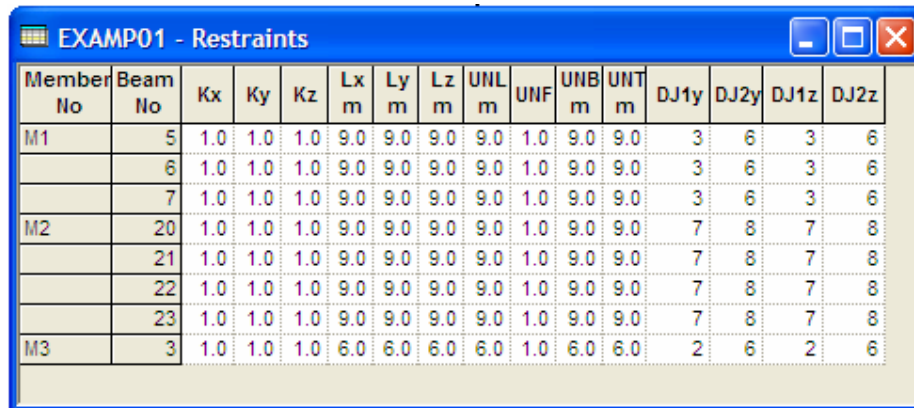
O Length:

Overall Length of the Physical Member.

Restraints (Sub-page #2)

The Restraints sub-page allows the user to edit, redefine or change the restrained condition in both major and minor axis, by defining the start and end joint numbers for which that member will be designed in that particular axis.

The View Member Restraint Table option brings up the following table containing all relevant restraint info for all the physical members present in the current model.



Member No	Beam No	Kx	Ky	Kz	Lx m	Ly m	Lz m	UNL m	UNF	UNB m	UNT m	DJ1y	DJ2y	DJ1z	DJ2z
M1	5	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	3	6	3	6
	6	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	3	6	3	6
	7	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	3	6	3	6
M2	20	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	7	8	7	8
	21	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	7	8	7	8
	22	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	7	8	7	8
	23	1.0	1.0	1.0	9.0	9.0	9.0	9.0	1.0	9.0	9.0	7	8	7	8
M3	3	1.0	1.0	1.0	6.0	6.0	6.0	6.0	1.0	6.0	6.0	2	6	2	6

Restraints Table Columns**Member No:**

Shows Physical Member designation. Currently, physical member cannot be renumbered. By default, all the restraint conditions are based on the entire length of the physical member.

Beam No:

Shows number of beam(s) and their numbers used to form that Physical Member.

Kx:

K (slenderness factor) value in local-x axis. Default value is 1.00.

Ky:

K (slenderness factor) value in local-y axis. Default value is 1.00.

Kz:

K (slenderness factor) value in local-z axis. Default value is 1.00.

Lx:

Effective Length in local-x axis. $Lx = Kx * \text{Overall Length of the Physical Member}$

Ly:

Effective Length in local-y axis. $Ly = Ky * \text{Overall Length of the Physical Member}$

Lz:

Effective Length in local-z axis. $Lz = Kz * \text{Overall Length of the Physical Member}$

UNL:

Unsupported length for calculating bending stress, provided as length.

UNF:

Unsupported length for calculating bending stress, provided as fraction of length of the Physical Member.

UNB:

Unsupported length of the bottom flange for calculating allowable bending compression stress will be used only if flexural compression is on the bottom flange. Bottom represents the negative side of the local Y-axis (local Z-axis if SET Z UP is used).

UNT:

Unsupported length of the top flange for calculating allowable bending compression stress will be used only if flexural compression is on the top flange. Top represents the positive side of the local Y-axis (local Z-axis if SET Z UP is used).

DJ1y:

Joint number denotes starting point for calculation of "Deflection Length"¹ about local y-axis. It is the same for all the span(s) of a physical member.

DJ2y:

Joint number denotes end point for calculation of "Deflection Length"¹ about local y-axis. It is the same for all the span(s) of a physical member.

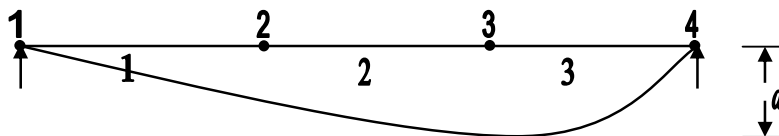
DJ1z:

Joint number denotes starting point for calculation of "Deflection Length"¹ about local z-axis. It is the same for all the span(s) of a physical member.

DJ2z:

Joint number denotes end point for calculation of "Deflection Length"¹ about local z-axis. It is the same for all the span(s) of a physical member.

¹ "Deflection Length" is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different. For example, refer to the figure below where a beam has been modelled using four joints and three members. Note that the "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Also the straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured. Thus, for all three members here, DJ1 should be "1" and DJ2 should be "4".



d = Maximum Local Deflection for Member 1,2 and 3

Briefs/ Group (Sub-page #3)

Design Briefs

The Design Briefs menu provides the ways to view, create and edit a set of design parameters, which allows the Steel Design checks to be done. Each Design Brief is set for a particular Design Code.

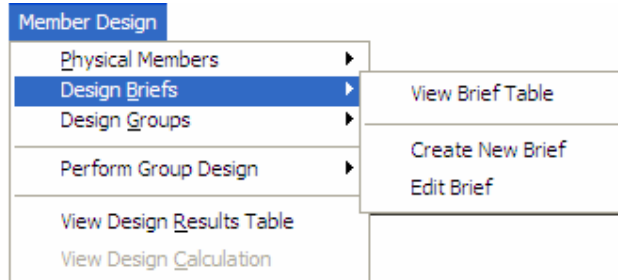
Briefs are independent of Design Groups so that the same brief can be re-used for several design groups. For example, you may have several groups of members but only one brief for them. Alternatively, you may tailor each brief to each member group.

Available Codes:

AISC ASD

IS800

A list of different options in Design Briefs is described in the following section.



View Brief Table

The View Brief Table lists the design briefs already defined and the codes of practice used for each brief.

To edit an existing brief, select the entry with the cursor and then press the Edit Brief button.

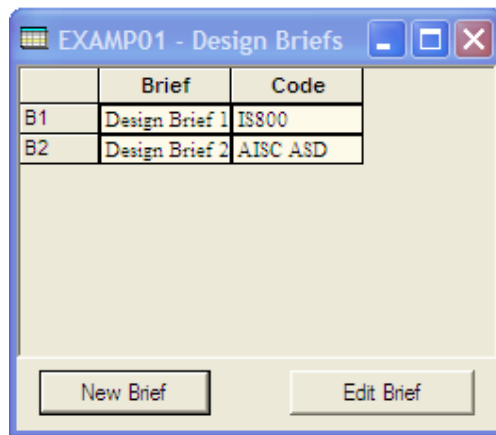


Table Tabs

(blank):

First column has no heading. It only lists sequential brief reference numbers (B1, B2, etc.)

Brief:

Name of the Design Brief.

Code:

The particular Design Code for which the Brief is defined.

Create New Brief

The Create New Brief option is used to provide the steel parameters for a particular design brief. Users can select a particular Design Code and select the options as well as change or edit values for other parameters. The entire list of design parameters for the AISC-ASD code is given below.

Dialog Box Options

Brief Name:

Type a Name of Design Brief

Design Code:

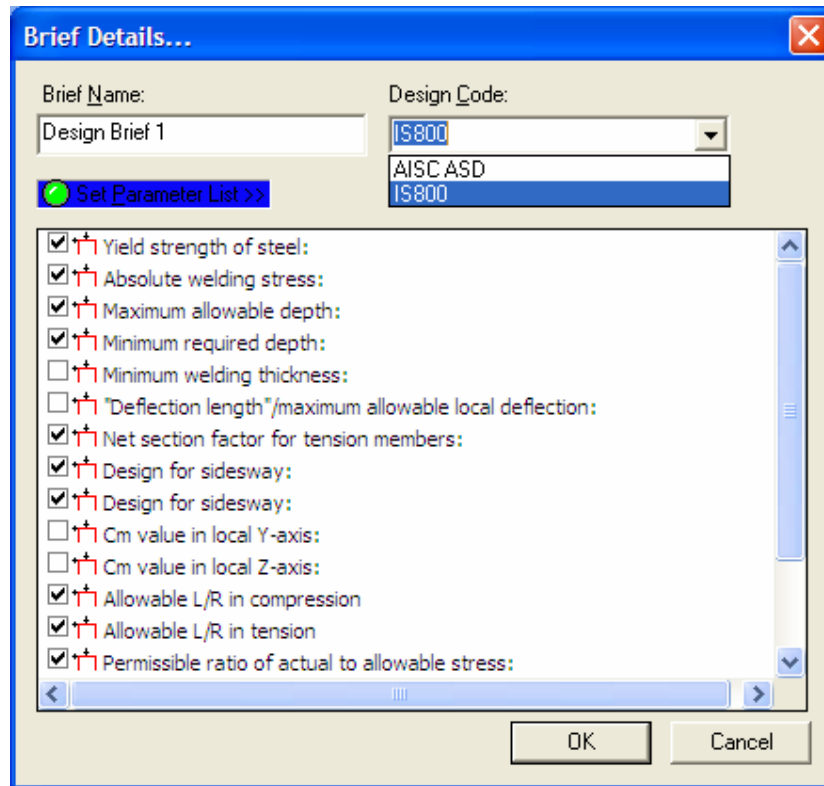
Select a Design Code from Drop Down list.

Set Parameter List:

This option allows the user to select additional design parameters for which to alter or modify the default parameter for. As so not to overwhelm the user with a myriad of design parameters, many of the options are kept within the Set Parameter List section. A list of the available parameters and their explanations are provided in the table below.

Parameters	Description	Default/ Options
Fyld	Yield strength of steel	36 kip/in ²
Wstr	Allowable welding stress	0.4 * Fyld
Torsion	Design for Torsion as per AISC T-114	(0) - Do not design for torsion (1) - Design for torsion
Dmax	Maximum allowable depth	39370.079 in
Dmin	Minimum required depth	0 in
Wmin	Minimum welding thickness	0.0625 in
Stiff	Spacing of stiffeners for plate girder design	Default is selected beam's length.

Dff	"Deflection length"/maximum allowable local deflection	Default is 0 indicating that deflection check is not performed
Nsf	Net section factor for tension members	Default is 1.0
Cb	Cb value as specified in Chapter F of AISC-ASD	Default is 1.0
Ssy	Design for sidesway	(0) - Sidesway in local Y-axis (1) - No sidesway
Ssz	Design for sidesway	(0) - Sidesway in local Z-axis (1) - No sidesway
Cmy	Cm value in local Y-axis	0.85 for sidesway and calculated for no sidesway
Cmz	Cm value in local Z-axis	0.85 for sidesway and calculated for no sidesway
Main	Design for slenderness	(0) - Check for slenderness (1) - Do not check for slenderness.
Track	Track parameter	Print the design output at: (0) - the minimum detail level (1) - the intermediate detail level (2) - maximum detail level.
Ratio	Permissible ratio of actual to allowable stress	Used in altering the RHS of critical interaction equations
Weld	Design for weld	(1) - Closed sections (2) - Open sections. Weld design per American Welding code
Beam	Beam parameter	(0) - Perform design at ends and those locations specified in the section command (1) - Perform design at ends and 1/12th section locations along member length
Profile	Enter profile for design	Choices are available to select: W4X /W5X /W6X /W8X /W10 /W12 /W14 /W16 /W18 /W21 /W24 /W27 /W30 /W33 /W36 /W40 /M4X /M5X /M6X /M8X /M10 /M12 /M14 /S3X /S4X /S5X /S6X /S7X /S8X /S10 /S12 /S15 /S18 /S20 /S24 /HP8 /HP1 /B45 /B49 /B53 /B57 /B61 /B68 /B74 /B80 /B92 /C3X /C4X /C5X /C6X /C7X /C8X /C9X /C10 /C12 /C15 /MC6 /MC7 /MC8 /MC9 /MC1 /L20 /L25 /L30 /L35 /L40 /L50 /L60 /L70 /L80 /L90
Fu	Ultimate tensile strength of steel	60 kip/in ²
Taper	Design for Tapered member	(0) - Design Tapered I-Section based on rules of Chapter F and Appendix B (1) - Design Tapered Section Based on Appendix F.



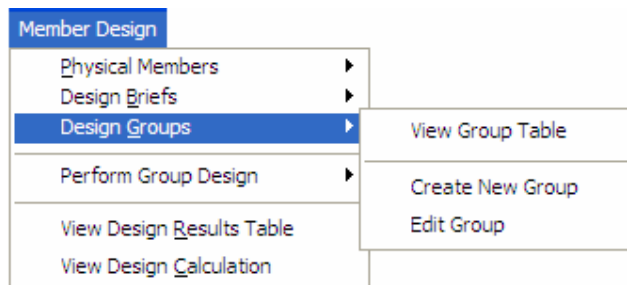
Edit Brief

The Edit Brief option is used to edit a selected brief, which has already been created.

Design Groups

The Design Groups menu provides a way to view, create and edit a set of physical members defined in the same group. The user can create a design group and add in as many physical members in each group. Each design group will follow same design code, same sets of parameter (i.e. same design brief) and same load envelope.

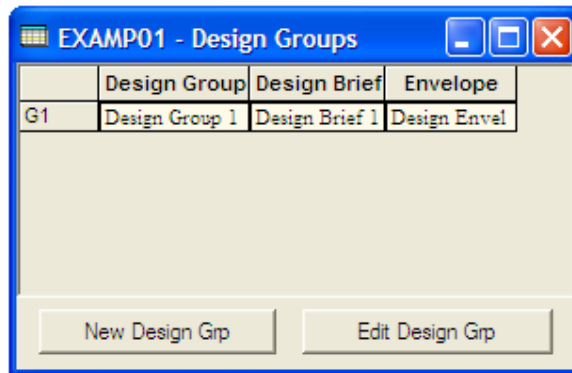
A list of different options in the Design Groups menu is described below.



View Group Table

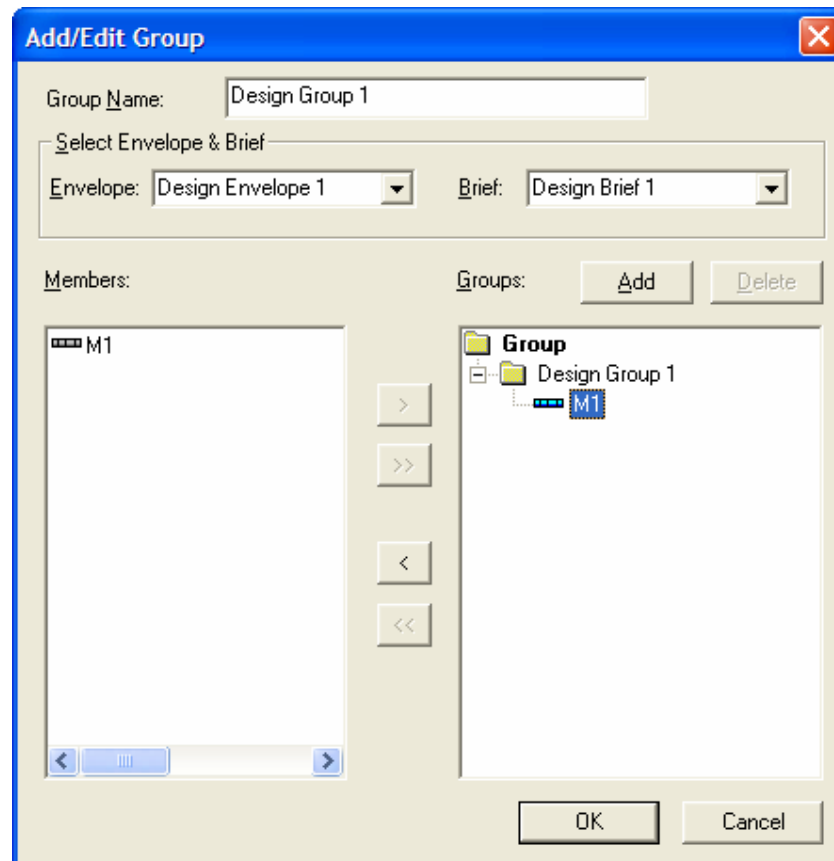
The View Group Table lists the design groups already defined along with the associated design brief and load envelope.

To edit an existing group, select the entry with the cursor and then press the Edit Design Group button.



Create New Group

The Create New Group option is used to create a design group, which encompasses a list of physical members, associated with a certain design brief and load envelope.



Dialog Box Options

Group Name:
Provide a Design Group name

Envelope:

Select a Load Envelope from the dropdown list to attach to the group

Brief:

Select a Design Brief from the dropdown list to attach to the group

Members:

List of all the physical members that have been created in the model.

Group:

List of Design Groups and the physical members under each group. Currently, a physical member can only belong to one design group. If the member color under the Members list (left column) has a gray color (as opposed to aqua blue), it has already been assigned to another design group and cannot be used unless it is unassigned.

Add

Add a new design group

Delete

Delete a selected design group (first highlight the Design Group to delete)

>

Press to include the selected physical member in the current design group.

<

Press to unselect the selected physical member in the current design group.

>>

Press to select all the physical member(s) in the current design group.

<<

Press to deselect all physical member(s) in the current design group.

Edit Group

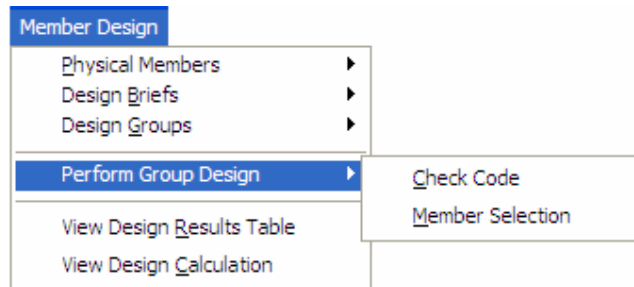
The Edit Group option is used to edit a selected group that has already been created.

Results/ Reports (Sub-page #4)

The Results/ Reports sub-page allows the user to view detailed design results (including step-by-step calculations) of all the physical members incorporated in design groups. The results can be incorporated into the main STAAD.Pro report for printing.

Perform Group Design

Under the Perform Group Design option, there are two options to design the members: 1) Check Code and 2) Member Selection.

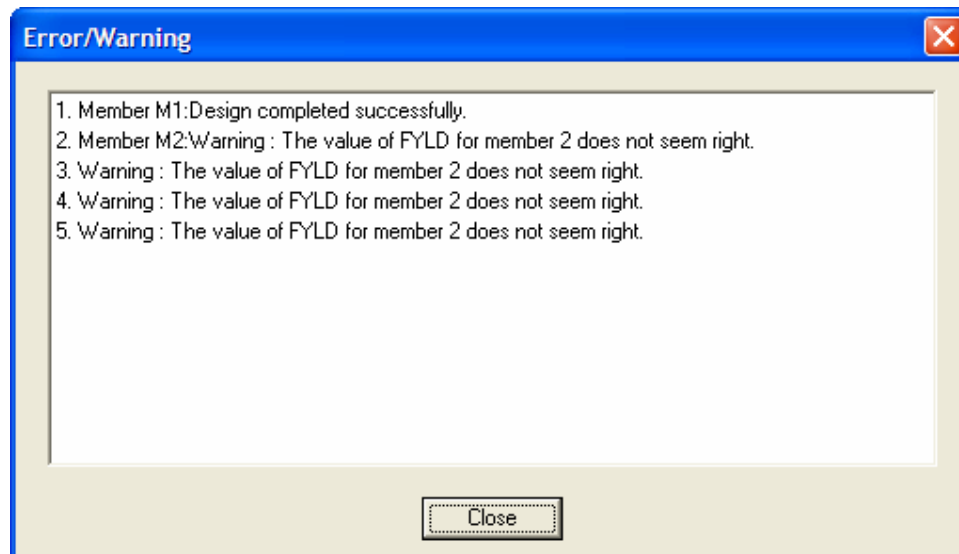


Check Code

The Check Code command performs the checks for the physical members formed and assigned into different design groups. After successful completion of the design, confirmation dialogs will popup.



The results are now available to view in the output table. If the steel design could not be performed for a particular reason, the dialog box shown below will be displayed with corresponding error and/or warning messages.



Member Selection

The Member Selection command performs optimum profile selection for the formed physical members. The steel designer is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section (lightest section), which satisfies the applicable code

requirements. The section selected must match with the type (I-Section, Channel etc.) as originally specified by the user. Selection of members, whose properties are originally provided from user specified table, will be limited to sections in the user provided table. Member selection can't be performed on members whose cross sectional properties are specified as PRISMATIC.

The process of MEMBER SELECTION may be controlled using the parameters listed in the Brief Details input above. It may be noted that the parameters DMAX and DMIN may be used to specify member depth constraints for selection. The following dialog box will appear when the Member Selection command is executed successfully.



Similar to a Code Check, if the steel design could not be performed for a particular reason, the dialog box shown below will be displayed with corresponding error and/or warning messages.



View Design Results Table

The View Design Results Table sub-menu displays the summarized Design Results for all the Physical Members under different groups. The design result may be from either code checking or member selection, whichever is performed last.

EXAMP01 - Design Results Table											
Group No	Group Name	Member No	Member Spec	Slenderness	Axial Chk	Comb. Axial & Bend Chk	Shear Along Y	Shear Along Z	Defln Along Y	Defln Along Z	
G1	Design	M1	BEAM	PASS	N/A	PASS	PASS	PASS	IGNORE	IGNORE	
				0.799	N/A	0.834	0.172	0.000			
G2	Design	M2	TRUSS	FAIL	FAIL	N/A	FAIL	FAIL	IGNORE	IGNORE	
				2.308	0.000	N/A	0.000	0.000			

Table Columns

Group No:

Reference of the group number created in the Design Groups

Group Name:

Name given to the Design Group

Member No:

Reference of the physical member

Member Spec:

Specification defined (if any) for the physical member (i.e. beam, truss, tension-only, etc.). By default, all member specifications will be displayed as "BEAM".

Slenderness Check:

Result of Slenderness Ratio Check. If a Slenderness Check is performed, either a PASS or FAIL will appear with the corresponding ratio (Actual/Allowable). IGNORED will appear when certain design parameters are defined to bypass the slenderness ratio check.

Axial Check:

Result of check for Axial Tension or Compression.

Combined Axial & Bending Check:

Result of Combined Axial as well as Bending stress check as per Interaction Equation.

Shear Along Y Check:

Check for adequate resisting area for Shear Force along Y-axis.

Shear Along Z Check:

Check for adequate resisting area for Shear Force along Z-axis.

Defln Along Y Chk:

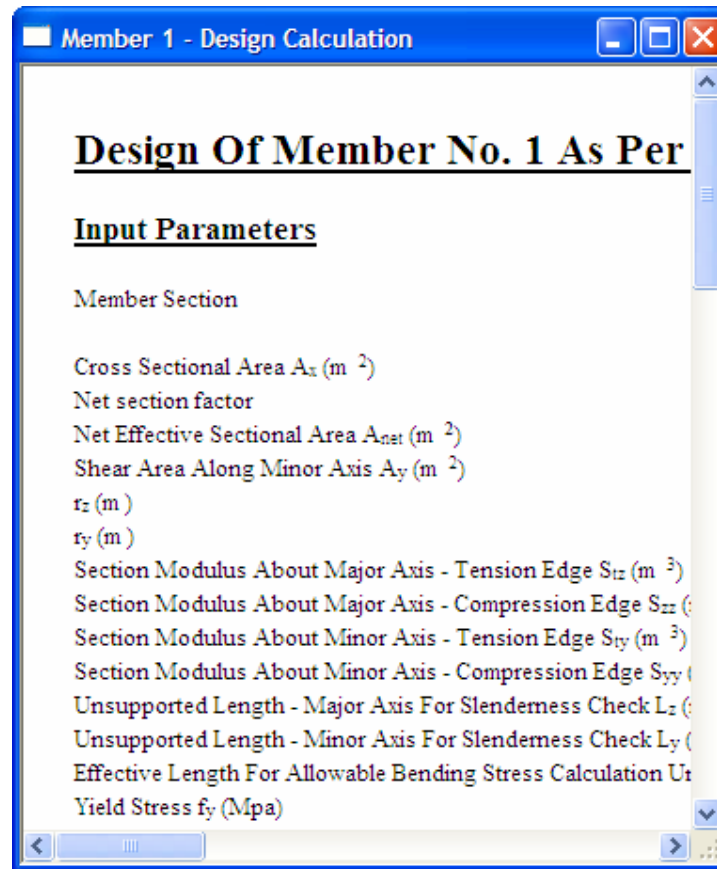
Check for Deflection along Y-axis against its allowable limit.

Defln Along Z Chk:

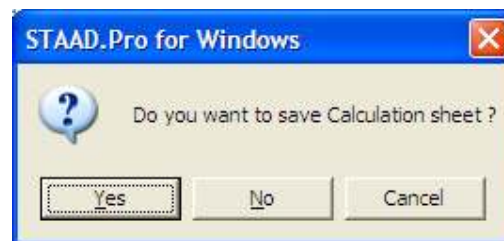
Check for Deflection along Z-axis against its allowable limit.

View Design Calculation

The View Design Calculation sub-menu displays the detailed Calculation Sheet for a selected physical member (the member must be highlighted first) for which a design has already been performed (Code Check or Member Selection).





While closing the calculation sheet window, an option to save the calculation sheet will be prompted. By saving the calculation sheet, the steel designer will not have to recalculate the intermediate values again.



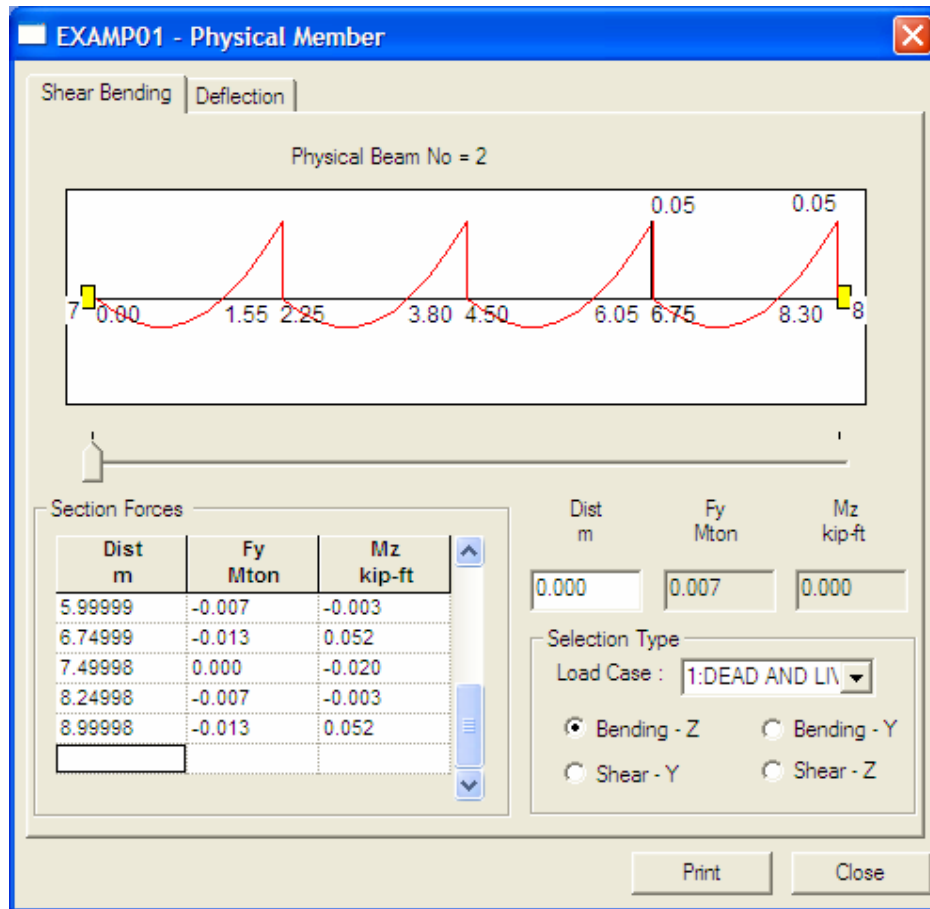
Member Query Dialog Box

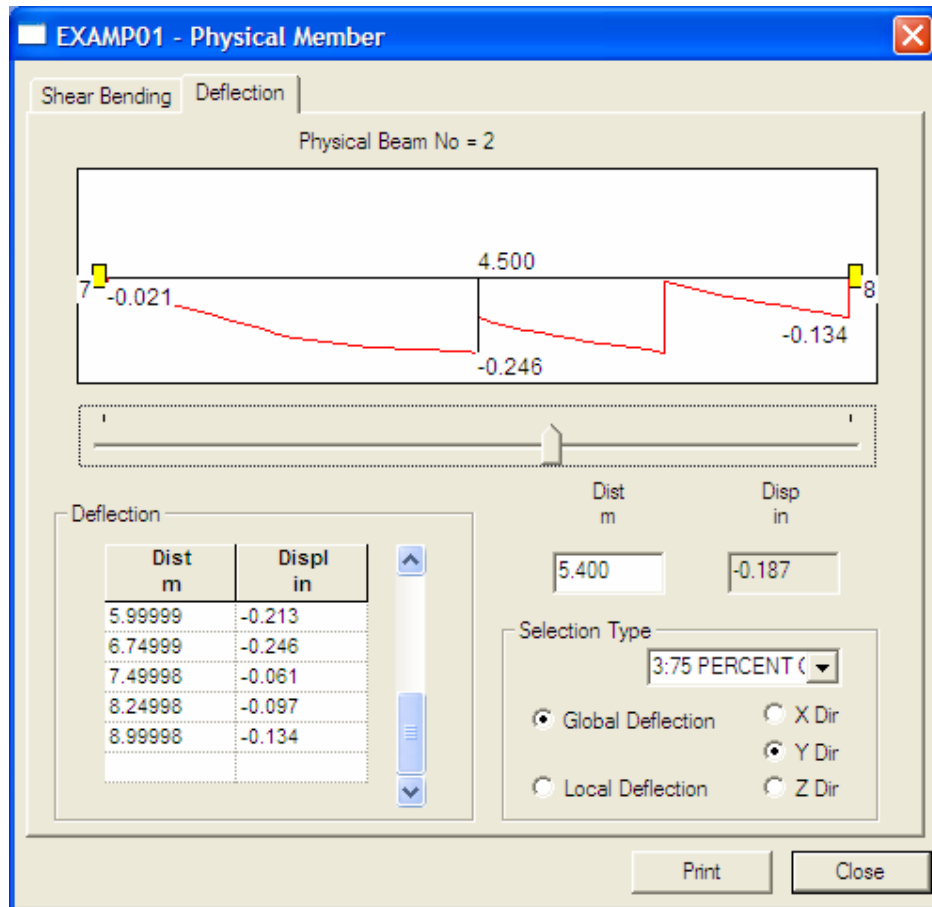
A quick way to view the intermediate force and deflection results and diagrams for the entire physical member or each of the individual beam elements that comprise the member is to simply double-click on the entity with the appropriate selection cursor. If

the Beam Cursor  is selected, an individual beam element can be double-clicked on

to view the output for that beam. If the Physical Member Cursor  is selected, the design results and intermediate forces and section displacements for the entire physical member can be viewed.

A sample physical member query box consisting of four individual beam elements is shown below.



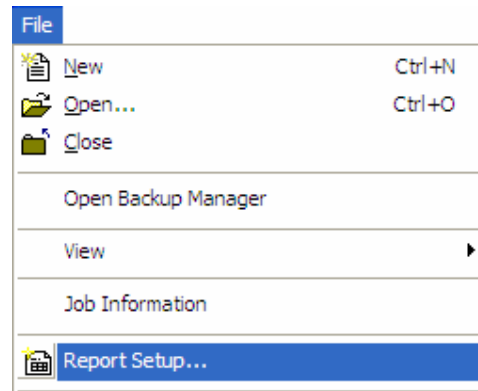



Results/ Reports (Sub-page #3)

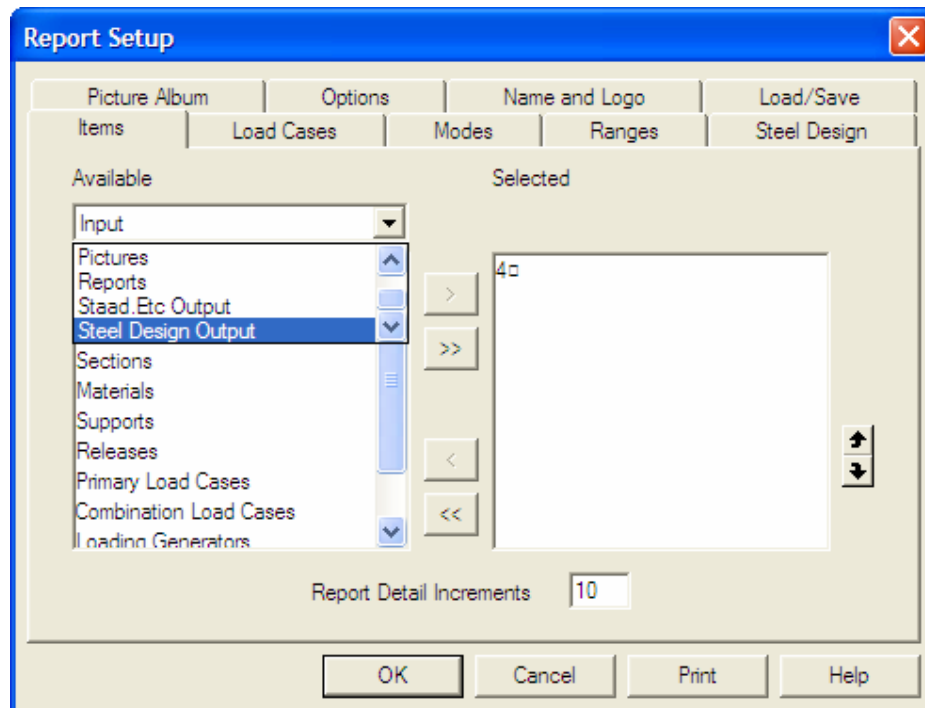
The 'Results/ Reports' sub-page is to allow the user to view detailed design results of already defined connection design for moment as well as shear. This result can be incorporated into main STAAD.Pro report to documentation purpose just as a part of STAAD.Pro main report.


Report Setup

To include the output from the steel designer with the rest of the STAAD.Pro report, use the Report Setup option from the File menu.



Alternatively, the Report Setup icon  can be selected from the toolbar. The Report Setup dialog box will appear on the screen. Under the Items tab, select the Steel Design Output from the drop-down list to include the steel design results.



Once all the necessary report items are selected, use the Print Preview icon  from the toolbar (or access it from File | Print Preview in the main menu) to view or print the report. The report can also be exported to MS-Word for further editing.

Section 2. Engineering Information

Basic Principles

Introduction

Selecting the **Steel Design** option under the **Interactive Design Modes** menu under the **Mode** main menu will take you to the **STAAD.Pro Steel Designer** page. When this is selected, STAAD.Pro will update the Page Control area (vertical buttons on the left) with the options mentioned in Section 1. Additional data that is created during the use of the program, such as the members, envelopes, design groups and design briefs, is stored in a persistent file. This means that if the program is closed and re-opened at a later date, the data remains available and does not need to be re-entered.

Suitable Member Properties

As of the first release of STAAD.Pro 2003, the steel designer cannot perform designs on beam elements that are assigned a PRISMATIC or GENERAL property. Plate girder design is also not supported.

Filenames

These are the external files that are used or created by the STAAD.Pro Steel Designer:

<filename>.std	Structure Input file.
<filename>.ben	Analysis result file.
<filename>.bmd	Analysis result file.
<filename>.dbs	Analysis result file.
<filename>.dsp	Analysis result file.
<filename>.rea	Analysis result file.
<filename>.scn	Analysis result file.
<filename>.sec	Analysis result file.
<filename>. REI_SPRO_Auxiliary_Data	Auxiliary Data File
<filename>.REI_STEELDESIGNER_DATA	Persistent File

All these files are stored in the same folder as the STAAD (.std) file. The program also makes use of the files SPRORC.INI and QSE.INI that are stored in the user's Windows folder. SPRORC.INI contains the user settings for fonts, colors and the current units. QSE INI contains the conversion factors for the various unit settings.

AISC-ASD Steel Member Design Principles

Introduction

The AISC-ASD Steel Member Design Brief is for single or multi span steel member sections. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.

The member sections must be assigned from the User section database. Currently the new interactive STAAD steel designer supports the steel design of wide flange, S, M, HP shapes, angle, double angle, channel, double channel, beams with cover plate, composite beams and code checking of prismatic properties. Design is being done as per AISC-ASD. Beams with cover plates, composite sections and GENERAL and PRISMATIC sections can be checked using the non-interactive mode of steel design still supported by STAAD.Pro 2003.

Members are designed for combined axial force and biaxial moment, shear force and checked for slenderness and deflection criteria.

To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter TORSION with a value of 1.0. See chapter 2.3.7 of the STAAD Technical Reference Manual for more details.

Code Checking

The purpose of code checking is to check whether the provided section properties of the physical members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections (use BEAM 1.0 parameter) of those physical members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the physical member) of forces in the member where the critical condition occurs.

Member Selection

STAAD.Pro is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e. the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3)

profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN, which limit the maximum and minimum depth of the members. Note that if the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections except the ones listed under the Suitable Member Properties section above.



Section 3. Reference

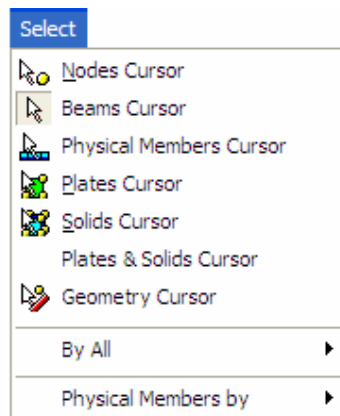
Commands

The following commands can only be found in the interactive steel designer mode. All other menu items have the same functionality in other modes.

Select Menu Alt+S

The Select menu provides various means of selecting beam elements, plate/solid elements, nodes and physical members.

Keyboard Shortcut ALT+S



Physical Members Cursor

The Physical Members is used to graphically select single span or multi span physical members of the structure defined in the Steel Designer mode. Choosing the Physical

Member Cursor option  from the Select menu activates the Physical Member cursor.



. Physical Member Cursor

To select physical members using the Physical Members Cursor, simply click on the any of the formed members. To select multiple physical members, hold down the Control key while selecting. You may also select members graphically by creating a rubber band window on the screen using this cursor.

Physical Members by...

The Physical Member by... selection option is used to select Physical Members present in the structure by certain specifications such as Physical Members by List, by Design Group, by Sectional Property, by Design Brief and by Design Load Envelope.