

# **SODA**

## **Structural Optimization Design and Analysis**

**Version 4.0**

*March 1<sup>st</sup>, 2005*

## **User Manual**

# Table of Contents

Copyright Notice.....	4
Managing the SODA Windows Environment.....	5
User Support.....	7
SODA Description.....	8
Overview and Limits.....	9
Moving Around the Screen.....	12
Modelling the Structure.....	13
Modelling the Loads.....	16
SODA - Menu Commands.....	18
SODA File Commands.....	18
SODA General Commands.....	20
SODA Structure Commands.....	24
Nodes.....	24
Groups.....	26
Members.....	34
Regular Framework.....	38
Irregular Framework.....	43
Diagonal Bracing.....	45
Move Nodes.....	46
SODA Loads Commands.....	48
Nodal Loads.....	48
Member Loads.....	50
Temperature Loads.....	55
Support Settlements.....	58
Area Loads.....	60
Load Combinations.....	61
Displacement Limits.....	62
SODA Run Commands.....	64
SODA Output Commands.....	67
Design history.....	67
SODA Structural Topology Menu Commands.....	69
SODA View Commands.....	69
SODA Graphics Commands.....	70
SODA Mode Commands.....	72
SODA Design Constraints.....	74
Introduction.....	74
Strength and Displacement Constraints.....	74
AISC-ASD-89 Steel Design Code.....	75
AISC-LRFD-86 Steel Design Code.....	77
AISC-LRFD-93 Steel Design Code.....	79
CSA-S16.1-M89 LSD of Steel Structures.....	83
CSA-S16.1-94 LSD of Steel Structures.....	86
SODA Databases.....	90

SODA Database Editor .....	90
American Section Database .....	90
Notation: American Section Database .....	90
Shapes: American Section Database.....	93
Canadian Section Database .....	94
Notation: Canadian Section Database .....	94
Shapes: Canadian Section Database .....	96
User-Created Section Database.....	97
Database Guidelines.....	97
Customizing a Section Shape File .....	97

# Copyright Notice

Neither the SODA software nor any part of this manual may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the written permission of Acronym Software Inc.

Acronym Software Inc. makes no warranties, express or implied, with respect to the **SODA** software system described in this manual. In no event will Acronym Software Inc. be liable for any consequential damages arising from the use of the **SODA** software system.

The information contained in this manual is subject to change without notice and does not represent a commitment on the part of Acronym Software Inc.

**SODA** (c) Copyright: 1985-2005 Acronym Software Inc.

22 King Street South

Suite 302

Waterloo, ON

N2J 1N8

Canada

Portions Copyright Microsoft Corp. 1985-2005 All rights reserved.

# Managing the SODA Windows Environment

All windows in the SODA interface are linked to the Structure Topology window. Any Node, Member, Fabrication Group or Load Type that is highlighted in its parent window is also highlighted on the view of the structure displayed in the Structure Topology window.

## Default Windows Configurations

The SODA interface windows are arranged into a number of default configurations on the computer screen, as follows:

- \* The Structure Topology window appears in all configurations in the lower right of the screen.
- \* The Nodes, Members or Fabrication Groups window appears in the left side of the screen in configuration with the Structure Topology window (i.e., three individual default window configurations).
- \* The Nodal Loads, Member Loads, Temperature Loads or Settlement Loads window and associated Load Graphic window respectively appear in the left side and upper right of the screen in configuration with the Structure Topology window (i.e., four individual default window configurations).

For each default configuration, the window appearing in the left side of the screen will be truncated such that all its fields are not visible. Click on the **up-arrow** in the top right of the screen (or, alternatively, click on the **file-drawer** in the top left of the screen and select **Maximize** from the drop-down list) to maximize the window such that all its fields are visible. Then, click on the **double-arrow** in the top right of the maximized screen (or, alternatively, click on the **file-drawer** in the top-left of the screen and select **Restore** from the drop-down list) to restore the window to its default size and location.

### Creating and Saving Windows Configurations

The SODA interface windows may be arranged together in any desired configuration on the computer screen:

- \* Move each window by clicking on its **title bar** and dragging the window to its new location on the screen.
- \* Size each window by moving the mouse pointer slowly over the border of the window until the pointer changes to a double-arrow pointer and then clicking on the **border** and dragging it until the window is the desired size (or the allowable lower-bound size is reached).

To save a windows configuration, select **/File/Save Workspace/** from the SODA menu bar. Thereafter, each window in the configuration will open to its saved location and size.

## Basics you should know about using Help

### Highlighted Text

Click once on any underlined highlighted text to access more information about a topic.

### Important Toolbar Buttons

Click the **Help Topics** button on the above toolbar to return to the **Help Topics** dialog. This dialog contains three tabbed windows:

1. The **Contents** tab displays the table of contents for the On-line Help.
2. The **Index** tab is used to perform a keyword search. Every topic in the help file has one or more keywords associated with it. Use keyword search to jump directly to a help topic. A listbox indicates which keywords are used in the help file. You can pick a keyword from the list or type it in. As you enter each letter of the keyword the list will change to show those words that most closely match what you type. Press the **Enter** key and the associated help topic(s) will be displayed (there could be more than one). Press **Enter** again to see the contents of the highlighted help topic.
3. Click the **Back** button on the above toolbar to return to the previous help topic (i.e., jump back). Press this button repeatedly to re-trace your steps through the help file.

More detailed information about using Help can be found by selecting *Help* and then *How to Use Help* from

the menu bar of this window.

# User Support

Telephone, e-mail or fax support is available during normal business hours to all SODA users. This support is available to users who are having trouble installing their software, whose software appears to be behaving strangely and to users who have questions about how SODA works.

In order to assist the technical support staff for SODA, please have the following information available:

1. A complete description of the operating hardware, including the type of computer, graphics card, Windows version, printer, etc.
2. A description of the problem including any appropriate input and output.
3. The steps taken that resulted in the problem (if known). This should include information as to whether the file was saved prior to the occurrence of the problem, if any of the input was altered, if the mouse or keyboard was used, etc.

The most efficient way to report a problem is by sending an e-mail containing a copy of the SODA project file as an attachment.

If the problem description is to be sent by fax, be sure to send a sketch of the structure, a copy of the Input Echo and any other output report that you think is applicable.

If the problem description is to be conveyed by mail, send a copy of the data file for which the error occurred. Place this file on disk, put the disk in a stiff package and mail it along with the error description. On the exterior of the package write "Do Not Fold or X-ray".

Acronym Software Inc.

22 King Street South

Suite 302

Waterloo, ON

N2J 1N8

Canada

web site: <http://www.acronym.ca>

email: [soda@acronym.ca](mailto:soda@acronym.ca)

voice: (519) 885-2454

fax: (519) 885-1407

# SODA Description

## Introduction

SODA is a Structural Optimization, Design and Analysis computer program for use by structural engineers. The program has been developed over many years by a research group in the Civil Engineering Department of the University of Waterloo, Canada, in conjunction with a broad base of practising engineers throughout North America to ensure that the program meets the practical needs of the structural engineering community.

Besides having the capabilities to perform verification (Code checking) and analysis, SODA has the capability to automatically design least-weight steel frameworks under static loads. The user need only input data concerning the structural geometry, member properties and loading conditions. SODA will automatically select the members from a database of standard sections so as to minimize the weight of the entire structure, while satisfying all design code requirements for strength and displacement. In other words, a steel design generated by SODA is feasible, of least-weight and completed in a single computer run.

In addition to its unique design capability, SODA features a windowing user interface that speeds both data entry and editing. The user interface has been developed in the Microsoft Windows environment. Information is quickly entered in a spreadsheet format using both the keyboard and a mouse. All data fields are clearly labelled, with all options visible to the user. Movement through the data screens is expedited by drop-down menus. An on-line help facility can be instantly accessed with a single keystroke.

SODA provides a printable graphics display view of the structure on the screen. A zoom and pan capability allows the user to magnify the display of any part of the structure. A rotate-display capability permits the structure to be viewed from any angle. The loading on the structure may be viewed graphically as well as the deflected shape and member shear force, bending moment and deformation diagrams.

## Underlying Theory

The SODA design process involves the co-ordinated use of elastic structural analysis, first-order sensitivity analysis and a continuous/discrete optimization technique. For the initial "trial" design of the structure, which is automatically selected by SODA, structural and sensitivity analyses are conducted and the strength and displacement design conditions are formulated explicitly in terms of member cross-section sizes through the use of first-order Taylor's series. The structure weight function is formulated in terms of member sizes. The optimization technique is applied to minimize the weight function, subject to the constraints imposed by the design conditions, so as to achieve an improved (lower weight) design of the structure. The weight optimization problem is reformulated for the new design and the process is repeated until the weight of the structure converges to a minimum after a number of design stages.

The iterative design process has two phases. The first phase involves a few design stages in which member sizes are taken as continuous variables to the weight optimization so as to quickly establish a reasonably well proportioned structure. The second phase involves taking standard section sizes as discrete variables to the weight optimization. The results of the first phase are used to select an "initial" standard section for each member of the structure to commence the second design phase. Selected at the same time from the section database for each member is an "initial" set of standard sections from which the design of the member is to be chosen during the first weight optimization of the second phase. The discrete-variable optimization phase is conducted for a number of design stages until, while satisfying all strength and displacement conditions, the standard section sizes found for the members correspond to minimum structure weight.

The SODA design process is remarkably efficient. The number of iterations required to achieve the optimized design is generally small and almost totally independent of the complexity of the structure.

The structural and sensitivity analysis routines employed by SODA are based upon the conventional Displacement Method of analysis for elastic material behaviour.



# Overview and Limits

## **SODA has the following major functions and features:**

### **Easy creation of input data**

- Mouse or keyboard interaction
- Microsoft Windows™ environment (Windows XP, 2000, WindowsNT, and 98)
- Simple screen prompts, all options visible
- On-line help messages
- Imperial or Metric units (kip, lb, in, ft, N, kN, mm, m)
- Useful default values (Young's modulus, yield stress, etc.)
- Identification of members and nodes using alphanumeric names
- Fast generation of structure topologies
- Fast generation of diagonal bracing
- Fast generation of applied loads
- Fast generation of design load combinations
- Practical construction considerations

### **Graphics capability**

- Display structure topology
- Display member names and/or node names
- Display fabrication group names
- Display applied loads
- Display structure deflected shape
- Display member shear force, bending moment and deformation diagrams
- Isometric, Projection and Plan views
- Tilt and Rotate features
- Pan and Zoom features

### **Design, verification (Code check) or analysis of 2D and 3D frames and trusses**

- Up to 2,000 members and 1,500 nodes
- Up to 100 load cases
- Fixed, pinned, roller or spring supports
- Pin releases for frame members
- Account for compression-only and tension-only members.
- Net area calculation for bolted tension members
- Various gusset-plate thicknesses for double angles

## **Account for first-order and P-delta behaviour**

Small displacements (first-order behaviour)

Second-order displacements (P-delta behaviour)

## **Account for a variety of applied loads and external effects**

Nodal loads (point loads; couples) Member loads (point loads; full and partial uniformly distributed loads; full and partial trapezoidal/triangular distributed loads)

Temperature loads

Support settlements

## **Design or verification in accordance with five different steel codes**

AISC-ASD-89: American Institute of Steel Construction, "Specification for Structural Steel Buildings: Allowable Stress Design and Plastic Design", June 1, 1989

AISC-LRFD-86: American Institute of Steel Construction, "Load and Resistance Factor Design Specification for Structural Steel Buildings", September 1, 1986

AISC-LRFD-93: American Institute of Steel Construction, "Load and Resistance Factor Design Specification for Structural Steel Buildings", December 1, 1993

CSA-S16.1-M89: Canadian Standards Association, "Limit States Design of Steel Structures", December, 1989

CSA-S16.1-94: Canadian Standards Association, "Limit States Design of Steel Structures", December, 1994

## **Optimized design for proper strength and stiffness performance**

Optimization automatically determines least-weight steel structure

Simultaneous satisfaction of strength and displacement conditions

Conformance with all strength provisions of specified steel code

Displacement conditions ensure acceptable drift, deflection, etc.

## **Member properties automatically selected from section databases**

American standard section database

Canadian standard section database

User-created custom databases

## **Automatic calculation of local buckling section classifications**

American sections (Compact, Noncompact, Slender)

Canadian sections (Class 1, 2, 3, 4)

## **Automatic calculation of effective length factors $K_x$ and $K_y$**

As per Alignment Charts in "Guide to Stability Design Criteria for Metal Structures", Third Edition, 1976 (Editor: B.G. Johnston)

Side sway prevented ( $0.5 < K_x < 1.0$ )

Side sway permitted ( $K_x > 1.0$ )

## **Account for external bracing**

Compression flange bracing

Automatic calculation of unbraced compression flange length

## **Practical construction considerations for design**

Specify groups of members to have common section properties

Specify lower and upper bounds on section depths

Specify section properties to be "frozen" at fixed values

## **Convenient presentation of output data**

Tabulated least-weight section designations for members

Individual member lengths and total lengths per section type

Structure volume and weight

Evaluation of member strengths as per provisions of steel code

Evaluation of structure stiffness as per displacement conditions

Analysis results for each load case

Three levels of report detail (normal, medium, detailed)

## **Files**

Once SODA is installed, the following files will be found in the SODA directory:

SODA Help files

SODA.HLP

SODA User Interface files:

SODA.EXE

SODA Engine files:

SODAENG.EXE

SODA Database files

AISC.SDA, \*.USA

CISC.SDC, \*.CDN

## Moving Around the Screen

When using SODA it is necessary to move to specific areas or fields in the screen and make selections from them. The following are some of the basic actions:

To move the cursor to a field in a screen (or window) using the mouse, point at the field and click the button.

To move the cursor to a field using the keyboard, press the Tab key and cursor (arrow) keys as often as necessary. The cursor will move to successive fields until it is positioned in the correct field. Press Shift and Tab (i.e. Shift-Tab) at the same time to cause the cursor to move in the reverse direction.

If the screen has two distinct parts to it, it may be necessary to press the Ctrl and Tab keys at the same time to move the cursor from one half to the other. The Tab key will then move the cursor through the fields in the part of the screen that the cursor is in.

To select an object using the keyboard, repeatedly press Tab (and possibly the Ctrl and Tab keys and the cursor keys) until the cursor is positioned on the object, then press the Space Bar to cause the object to be selected.

To select an item from the menu bar using the mouse, point to the menu title in the menu bar. Click the mouse button to cause the menu to drop down over the screen. Move the pointer down to the menu item to be selected and click the mouse button.

To select a menu bar item using the keyboard, type Alt then letter to drop the menu down (where letter is the first letter in the menu title shown in the menu bar). Then, either type the underlined letter of the menu item desired, or repeatedly press the Down Arrow key until the appropriate menu item is highlighted, and then press the Enter (or Return) key to select the item.

To highlight the information in a field using the mouse, drag the pointer over the field.

To highlight the information in a field using the keyboard, just tab the desired field with the Tab key (or Shift-Tab or Ctrl-Tab keys).

# Modelling the Structure

The issues that are of concern when modelling a structure for a SODA application are listed in the following.

1. Determine whether the structure is 2D (planar) or 3D (space): 2D: all loads are applied in the plane of the structure
2. Determine whether the structure is a Truss or a Frame:  
Truss: all nodes are pinned; loads are applied only at the nodes and not in the span of members.  
Frame: nodes may be pinned or fixed; loads may be applied both at the nodes and in the span of members.
3. Establish whether to Design, Verify or Analyze the structure:  
Design: determination of a least-weight structure in conformance with the strength (stability) and stiffness (displacement) provisions of a specified steel code (AISC-ASD-89, AISC-LRFD-86, AISC-LRFD-93, CSA-SI6.1-M89, or CSA-SI6.1-94).  
Verify: comparison of the member capacities and nodal displacements for a given structure with the strength and stiffness provisions of a specified steel design code (AISC-ASD-89, AISC-LRFD-86, AISC-LRFD-93, CSA-SI6.1-M89, or CSA-SI6.1-94).  
Analysis: determination of member forces and nodal displacements for a given structure and loading.
4. Determine whether First-Order or P-Delta behaviour is of concern:  
Either the effects of the interaction between the applied loads and the structure displacements are small and can be ignored (First-Order behaviour), or they are significant and must be accounted for (P-Delta behaviour).  
Low-rise structures with light loading and structures with external bracing may be reasonably modelled using First-Order behaviour.  
Unbraced structures in general should be modelled using P-Delta behaviour.  
P-Delta behaviour should generally be selected for Design or Verification.
5. For a Frame structure, determine whether side sway is prevented or permitted:  
Side sway prevented: sway effects are taken by bracing external to the frame (braced frame).  
Side sway permitted: sway effects are taken directly by the frame itself (unbraced frame).
6. Select the Section Database for the members of the structure:  
SODA supports two databases of standard steel sections  
- American: 15 different shapes (W, HP, etc.)  
- Canadian: 17 different shapes (WWF, W, etc.)  
For each standard shape, SODA supports most of the different section designations that are available from American and Canadian steel mills (see Database files supplied on diskette with SODA).  
The user has the option to provide a custom section database.  
Determine whether "Foreign" sections in the database are to be considered.
7. Select the Units of measure for Force and Length:  
SODA allows for four different sets of Imperial Units (kip;in, kip;ft, lb;in, lb;ft), and for four different sets of Metric Units (N;mm, N;m, kN;mm, kN;m).  
The choice of units is conditioned somewhat by the specified steel design code and section database. Although, SODA does allow Metric units to be employed in conjunction with an American code and database, and Imperial units to be employed with a Canadian code and database.  
Once specified, consistent units must then be used when defining data through the User Interface (for the units chosen, SODA automatically provides the appropriate Unit Labels for all data fields).

8. Determine if there are Bolted Connections for the structure:

SODA allows for Bolted Connections of Axial Tensile members. Specify the Bolt Hole Diameter to be slightly larger than that for the bolt itself (SODA provides useful default values). Bolted connections are of no concern for an Analysis application.

SODA internally accounts for the effective Net Area of bolted axial tensile members.

9. For a Verify or Design application involving the AISC-ASD-89, AISC-LRFD-86, AISC-LRFD-93 Steel Design Code, determine the ratio of effective net area  $A_e$  to net area  $A_n$  for tension members:

The ratio  $A_e/A_n$  specifies the reduction in the effective net cross-section area at connections.

SODA provides a default value of 1.0 and allows for user-specified values between 0.75 and 1.0.

10. Establish if there are Double Angle Sections for the structure:

Determine the Gusset Plate Thickness defining the separation distance between back-to-back Double Angles (SODA allows for the standard distances specified in American and Canadian steel design codes). The separation distance for double angles is of no concern for an Analysis application.

11. Establish if there are Hollow Structural Sections for the structure:

For a Design or Verify application under the Canadian steel design code CSA-S16.1-M89 and CSA-S16.1-94, SODA allows Hollow Structural Sections to be specified as CLASS H.

12. Determine whether to input the node and member data for the structure using the SODA batch facilities:

A batch facility is applicable for either irregular frameworks or regular rectangular frameworks, and automatically establishes node, member and fabrication group data.

If necessary, individual batch-created data may be subsequently modified in the Nodes, Members and Groups worksheets under the Structure Menu.

13. If not using a batch-input facility (point 12), establish the Global Co-ordinate Axes relative to the structure:

The global X-axis and Z-axis act horizontally and the global Y-axis acts vertically.

The origin of the axis system may be located at any point on the structure.

NOTE: The origin of the axis system is often located at the bottom left-hand corner of the front face of the structure, with the positive X-axis acting horizontally to the right, the positive Y-axis acting vertically upwards and the positive Z-axis acting horizontally outwards (the X and Y co-ordinates for the nodes of the structure are then generally all positive values while the Z co-ordinates of the nodes are generally all negative values); this is the origin location adopted by the batch-input facility (point 12).

NOTE: It is sometimes convenient to position the axis system such that one or more of the axes are coincident with an axis of symmetry for the structure.

14. Establish the Nodes for the structure:

Foundation connection points (fixed, pinned, roller and spring support nodes):

Member connection points (rigid and pinned "free" nodes.)

Other points where there is a concern for stresses and displacements (e.g., at the mid-span of a girder under heavy loads).

NOTE: For a Design or Verify application, SODA internally checks the stress level at the mid-span of each member even when a node is not specified there.

NOTE: For a Design or Verify application, it is necessary to specify a node at a point in the span of a member when the displacement there is to be controlled or checked.

Node names may be alphanumeric (e.g., A or 1 or AI, etc.), and they may be assigned without concern for the bandwidth of the stiffness matrix for the structure (SODA internally re-indexes the nodes to achieve the minimum bandwidth).

NOTE: The batch-input facilities automatically establish default names for the nodes.

15. Establish the fabrication Groups of members for the structure:

For ease and convenience of data input, as well as to satisfy fabrication requirements, group together all individual members that are required, or known, to have the same cross-section properties.

16. Establish the Group Member Section Properties:

Establish values for elastic Moduli (Young's/shear) and, for Design and Verify applications, the Yield Stress and Ultimate Stress (SODA provides useful default values).

Determine the section Shape for each Group of members. SODA supports seventeen different standard steel section shapes (W, WWF, C, WT, etc.). A section shape need not be specified for an Analysis application.

For a Verify or Analysis application, determine the section Designation for each Group of members. For each of seventeen different shapes, SODA supports most section designations available from American and Canadian steel mills (refer to the section database files supplied on diskette with SODA). A section designation need not be specified for an Analysis application.

For a Design or Verify application, determine the maximum allowable Slenderness Ratios in compression and tension for the members of each Group (SODA provides useful default values).

17. Establish the individual Member Properties:

Determine the Start and End nodes for each member.

Determine the Joint Type defining the end-fixity conditions at the Start-End nodes of each member (SODA allows for "Pinned-Pinned" or "Fixed-Fixed" or "Fixed-Pinned" or "Pinned-Fixed").

NOTE: The batch-input facilities automatically establish the end nodes and joint types for the members (point 12).

Indicate if a member is "compression-only" or "tension-only" (such members are automatically assigned a Pinned-Pinned end-fixity condition).

18. For a Design or Verify application, determine whether the structure has Displacement Limits:

If Displacement Limits are specified, establish the particular nodal displacements that are to be constrained and determine their allowable upper bound values. Displacement Limits generally need be specified for only a select few of the nodes for the structure. For example, for a 2D structure:

- maximum story drift is controlled by constraining the X-axis displacement at only one column-end node for each story of the structure.
- maximum mid-span deflection of all similarly loaded beam members belonging to the same Group is controlled by constraining the Y-axis deflection at a node located at mid-span of only one of the members.

# Modelling the Loads

The issues that are of concern when modelling the loads on a structure for a SODA application are listed in the following.

1. Determine the magnitudes, signs and locations of any Nodal Loads:

Nodal Loads are concentrated point forces and couples, acting at the nodes of the structure, that are applied and signed with reference to the Global Co-ordinate Axes.

An inclined point force is represented by its projections in the global axis directions.

Nodal Loads are specified at their Service-Load Level (load factors are subsequently specified when determining the Load Combinations for the structure).

2. Determine the magnitudes, signs and locations of any Member Loads:

Member Loads are concentrated point forces and distributed loading, acting in the span of the members for the structure, that are applied and signed with reference to the Global Co-ordinate Axes. (An exception to this occurs for loads oriented along the longitudinal axis of the member.)

SODA allows for the Basic Member Load types of Point Load, Full and Partial Uniformly Distributed Load, Full and Partial Trapezoidal/Triangular Distributed Load.

NOTE: Any combination of the Basic Member Loads may be applied to a member (simply by repeatedly applying the individual basic loads to the member one-at-a-time in the Member Loads Worksheet of the User Interface).

NOTE: Do not apply member loads with a global orientation which is coincident with the longitudinal axial of the member (i.e. 'Y' member loads applied to vertical columns).

Member Loads are specified at their Service-Load Level (load factors are subsequently specified when determining the Load Combinations for the structure).

3. Determine the magnitudes, signs and locations of any Temperature Loads:

Temperature Loads are applied to the members of the structure.

A Temperature Load that is applied to a member of length L is defined by a change in temperature of  $\Delta T$  degrees that causes a change in length of the member  $\Delta L = T \alpha L$ , where  $\alpha$  is the coefficient of thermal expansion for the material (SODA provides useful default  $\alpha$  values for steel).

NOTE: A positive  $\Delta T$  temperature increment causes the member to elongate, while a negative value causes the member to shorten.

A particular Temperature Load may be applied to any or all members of the structure.

A variety of different Temperature Loads may be applied to different members of the structure.

4. Determine the magnitudes, signs and locations of any Support Settlements:

A Support Settlement is a Displacement and/or Rotation of fixed magnitude that is imposed at a support node for the structure.

NOTE: It is not possible to impose a settlement in the same sense that a support is itself free to move; e.g., a X-axis displacement cannot be imposed at a support node having a horizontal X-axis roller.

Support Settlement displacements and rotations are imposed and signed with reference to the Global Co-ordinate Axes.

A particular Support Settlement may be imposed at any or all support nodes for the structure.

A variety of different Support Settlements may be imposed at different support nodes for the structure.

5. Determine the Load Combinations for the structure:

Each Load Combination is the factored summation of one or more of the (previously) specified Nodal Loads and/or Member Loads and/or Temperature Loads and/or Support Settlements.

SODA provides the unique capability to generate any particular Load Combination for the structure by



simply assigning load factors to the individual loads (Nodal, Member, etc.) that contribute to the combination.

- A different load factor may be assigned to each of the individual loads (Nodal, Member, etc.) that contribute to a particular Load Combination

# SODA - Menu Commands

File: Commands for loading and saving SODA files

General: Commands for describing the project

Structure: Commands for creating structures

Loads: Commands for creating and modifying loads

Run: Commands for running the engine

Output: Commands for viewing output files

Help: Commands to access SODA help

The following describes items contained in the Structure Topology Menu Bar:

View: Commands for viewing structure topology

Graphics: Structure topology viewing options

Help: Commands to access SODA help

## SODA File Commands

The following describes items under the File option of the SODA Menu Bar:

### File New

This command creates a new empty file. If the current file has not been saved since the last changes, SODA will allow it to be saved before creating the new sheet.

### File Open

This command opens an existing worksheet. If the current file has not been saved since the last changes, SODA will allow it to be saved before creating the new sheet.

If you press **Cancel** instead of selecting a file to open, SODA will create a new empty file instead.

### File Save

The Save option is used to re-save an existing file (that is, one that has been previously saved). The filename that is displayed on the Title Bar is automatically used. This option is greyed out (not available) if the data is unchanged since the last save was made.

The shortcut key for doing a Save is Ctrl+S.

To save a file that has not been previously saved or to save an existing file with a new name, select the Save As option.

### File Save As

When the File Save As option is selected, a dialog box will appear on the screen prompting the user for a filename. Clicking on OK will write the file data to the filename and return you to the interface. If the file already exists you will be prompted if you wish to over write it. Click on Cancel to return to the interface without performing the save.

## File Import

When the File Import option is selected, a dialog box will appear on the screen prompting the user to select a Comma Separated Values (CSV) ASCII file. Click on OK to import the CSV file into SODA as a POP file. Click on Cancel to return to the interface without performing the import.

## File Export

When the File Export option is selected, a dialog box will appear on the screen prompting the user to select the name of the SODA file that is to be saved as a Comma Separated Values (CSV) ASCII file. Click on OK to export the file out of SODA as a (CSV) ASCII file. Click on Cancel to return to the interface without performing the export.

## Print Topology

Clicking this item will send the currently displayed structure to the printer.

## Print Output

Selecting **/File/Print Output/** will display a list of input and output files for the current SODA application. Clicking on a file name in the list will send the file to the printer.

Input echo:

Analysis results:

Final design/verify:

Summary results:

Normal report:

Medium report:

Detailed report:

Effective lengths:

Design history:

Warnings:

Errors:

Batch: This option allows users to select multiple reports for printing.

## Printer Set-up

Select this item to change printers or print options. The dialog box that appears is the familiar Windows dialog for printer set-up. When printing the topology, landscape orientation is suggested for structures that are wider than they are tall.

If no printers appear in the Specific Printer field then you must exit SODA and install a printer via the Windows control panel (refer to the Windows manual).

## Output Font

Select the Font, Font Style and Size to be used for the Output reports.

## **Exit**

Click this item to exit SODA and return to Windows. If any changes have occurred to the currently opened file the user will be prompted if they wish to save the data.

# **SODA General Commands**

## **Title Description**

### **Project Title**

A user-specified title of the SODA application that may have up to 66 alphanumeric characters.

### **Description**

A user-specified description of the structural project. Use this area to enter any lengthy descriptions that do not fit in the title field.

## **Project Definition**

The Project Definition describes the overall parameters for the current SODA application. Selections made in this dialog box affect the available choices and behaviour of SODA throughout the system. The settings may be revised in existing files, but care must be taken not to introduce inconsistencies into the structure.

### **Dimension**

A two dimensional (2D) or a three dimensional (3D) structure. Two-dimensional structures may later be converted to three-dimensional.

### **Structure type**

The type of structure may be either a Frame or a Truss.

### **Problem Type**

The type of problem may be Design, Verify, or Analysis.

### **Design Code**

A governing steel Design Code is specified whenever the Problem Type is Design or Verify, but not when the Problem Type is Analysis.

For a specified Design or Verify problem, SODA automatically accounts for the relevant strength-related provisions of the governing Design Code. Displacement-related provisions are independently specified by the user later in the input.

The user may select one of the following governing Steel Design Codes:

AISC-ASD-89, AISC-LRFD-86, AISC-LRFD-93,  
CSA-S16.1-M89, or CSA-S16.1-94.

### **Behaviour Type**

The type of displacement theory for Design or Verify problems may be either First-Order or P-Delta (2nd-order).

## **Section Database**

The section database describes the section shapes available for Design or Verify. SODA automatically selects the standard database (AISC.SDA for U.S. design codes and CISC.SDC for Canadian design codes), but a different or custom database may also be selected. Section databases are not used for Analysis.

NOTE: The name of the section database to be used for the SODA application is specified in Section Database dialog box, while the particular Section Shape to be assigned to each individual member of the structure is specified later in the Group Information.

## Database file

The database file to be used. Each of the databases contains a list of filenames corresponding to the different section shapes. The Standard Shapes are the same for both U.S. and Canadian databases, except for the WWF and WWT sections.

**AISC.SDA** (American Institute of Steel Construction) is available only for the AISC design codes, and uses imperial units.

**CISC.SDC** (Canadian Institute of Steel Construction) is available only for the Canadian design codes, and uses metric units.

**NOTE:** The AISC Design Codes cannot be used with the CISC Database, and the CISC Design Codes cannot be used with the AISC Database.

## Other

Any custom database may be specified so long as it conforms to the SODA User-Created Section Database.

## None

If the problem type is Analysis, a database need not be specified. If a database is not specified, member cross-sectional areas and moments of inertia need to be specified in Group Information in addition to cross-section designations

## Include Foreign Sections

The CISC section database includes both Canadian and foreign section shapes (custom databases may also include foreign sections). Foreign sections will only be considered for design if the Include Foreign Sections box is checked.

## Toolbars

This menu command activates the toolbars shown on the SODA desktop.

## Units

This dialog box defines the units to be used for the SODA application.

## System

Select Imperial or Metric. Selecting a system of units will cause the Force and Length options to change accordingly.

## Force

For imperial units, select either kip (kilo-pounds force) or lb (pounds force). For metric units, select either N (Newtons) or kN (kiloNewtons).

## Length

For imperial units, select either in (inches) or ft (feet). For metric units, select either mm (millimetres) or m (meters).

Based on these selections, Soda will create unit labels as follows:

Force	Length	Moment	Area	I*	Stress**
kip	in	kip-in	in <sup>2</sup>	in <sup>4</sup>	ksi

kip	ft	kip-ft	in <sup>2</sup>	in <sup>4</sup>	ksi
lb	in	lb-in	in <sup>2</sup>	in <sup>4</sup>	ksi
lb	ft	lb-ft	in <sup>2</sup>	in <sup>4</sup>	ksi
N	mm	N-mm	mm <sup>2</sup>	mm <sup>4</sup>	MPa
N	m	N-m	mm <sup>2</sup>	mm <sup>4</sup>	MPa
kN	mm	N-m	mm <sup>2</sup>	mm <sup>4</sup>	MPa
kN	m	kN-m	mm <sup>2</sup>	mm <sup>4</sup>	MPa

\* I = moment of inertia

\*\* ksi = kips/square inch

Mpa = megapascal = Newtons/square millimeter

## OK Button

If the user changes any of the above items and data currently exists, SODA will prompt as to whether both labels and values should be converted or just labels. The second choice allows for the case where the user has entered data under the wrong units and does not want the data converted.

## Fabrication Conditions

This dialog box defines the basic fabrication conditions for the SODA application.

### Bolted Connections

Bolted connections may be specified for axial members of the structure so that their tensile capacities are calculated based on the cross-section net area (i.e., gross area - bolt hole area).

Check this box if all axial members in the structure are to be considered bolted. Note that bolts are assumed to pass through the web of the member cross-section. SODA internally identifies the tensile members of the structure, including bending members in axial tension.

### Bolt Hole Diameter

This field contains the diameter of the bolt hole to be used (assuming Bolted Connections was specified) for the calculation of the cross-section area for all members of the structure. Any positive numeric value is acceptable. In order to have more than one bolt at each connection, it is necessary to specify a multiple of the actual bolt hole diameter. (E.g. for two 0.8125 in bolts, specify  $2 \times 0.8125 = 1.625$  in.)

### Hole Diameter > Bolt Diameter

Note that design codes require that the bolt hole be larger than the bolt diameter by 1/16 inches or 2 mm. The user must make this allowance when specifying the size of the bolt hole because SODA does not do it automatically.

### Units

Millimetres are used for Canadian design codes regardless of the units the user has selected.

Inches are used for American design codes regardless of the units the user has selected.

### Double Angle Gusset Plate

This number specifies the thickness of the gusset plate for back-to-back double angles. Depending on the current design code, several different values are available:

U.S. codes                      0.0, 0.375 or 0.75 in

Canadian codes            0.0, 8.0, 10.0, 12.0, 16.0 or 20.0 mm

The gusset plate thickness is used by SODA to calculate the Y-axis radius of gyration of all double-angle sections in the structure.

**Default**

0.375 in (U.S. codes) or 10.0 mm (Canadian codes).

**Units**

Millimetres are used for Canadian design codes regardless of the units the user has selected.

Inches are used for American design codes regardless of the units the user has selected.

**$A_e/A_n$**

This field corresponds to the U reduction coefficient (the ratio of effective net area  $A_e$  to net area  $A_n$ ) for tension members. This specification is only available for the AISC design codes. The specified  $A_e/A_n$  ratio is used to calculate the effective net area for **all** axial tensile members of the structure.

**Class H Sections**

Checking this box will cause SODA to use Clause 13.2.2 of CSA-S16.1-M89, or CAN/CSA-S16.1-94 for calculating the axial capacities of all hollow structural sections in the structure.

**Restore Messages**

This command restores the messages that were previously turned off by checking the *Do not ask again* box when Windows messages appear.

# SODA Structure Commands

## Nodes

Nodes are used to define the start and end locations of members in the structure. This worksheet allows the user to define, modify or delete the name and position of each node in the structure. It also allows the user to define which nodes are supported and by what type of support. A Regular Framework feature can be used to quickly generate nodes and members.

## Nodes Name

Each node of the structure is identified by a unique alphanumeric name that may have up to 9 characters (e.g., 2, or Two, or NODE2, etc.)

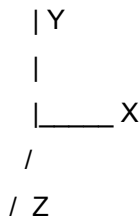
Do not use blanks between characters.

SODA does not permit node names that differ from one another only by upper and lower case (e.g. Node and NODE are not considered to be unique names).

Default names are generated when the Regular Framework feature is invoked.

## X, Y, Z - Co-ordinates

The X,Y,Z co-ordinates for each node of the structure are specified in accordance with the user-defined Global X-Y-Z Axis System. The Z co-ordinate is zero for 2D structures.



## X-COORD

The horizontal distance (in a direction orthogonal to the horizontal z-axis) that the node is from the origin of the axes system (a positive or negative value).

## Y-COORD

The vertical distance that the node is from the origin of the axes system (a positive or negative number).

## Z-COORD

The horizontal distance (in a direction orthogonal to the horizontal x-axis) that the node is from the origin of the axes system (a positive or negative value).

## Support Type

Support Type defines the support condition for a node.

### None

A free node that can rotate and translate in the X-Y plane (2D structure) or in the X-Y, Z-Y and X-Z planes (3D structure). This is the default support condition.

### Fixed

A support node that cannot rotate or translate in the X-Y plane (2D structure) or in the X-Y, Z-Y and X-Z planes (3D structure).



### **Pinned**

A support node that can rotate but cannot translate in the X-Y plane (2D structure) or in the X-Y, Z-Y and X-Z planes (3D structure).

### **Pinned [except X]**

Same as a Pinned support except cannot rotate about the global X-X axis.

### **Pinned [except Y]**

Same as a Pinned support except cannot rotate about the global Y-Y axis.

### **Pinned [except Z]**

Same as a Pinned support except cannot rotate about the global Z-Z axis.

### **Roller [along X]**

A Pinned support that is free to translate along the global X-X axis.

### **Roller [along Y]**

A Pinned support that is free to translate along the global Y-Y axis.

### **Roller [along Z]**

A Pinned support that is free to translate along the global Z-Z axis.

### **Other Nodal Support Conditions**

#### **Fixed X,Y,Z**

A support node that cannot rotate or translate for the corresponding X, Y, or Z-axis direction.

#### **Spring Value X,Y,Z**

The magnitude of the stiffness of a translational or rotational spring support associated with the corresponding X, Y or Z-axis direction.

### **Graphic Representation**

Fixed and Pinned supports use typical symbols.

A Roller support is represented by two black circles, one atop the other.

Other supports are represented by a hollow circle with an X through it.

### **Graphical Selection**

With the Nodes window open, select a node by clicking the mouse left button while pointing at the desired node in the Structure Topology window; the selected node will be highlighted in the Nodes window. Several Nodes may be selected by clicking on each desired node while holding down the **Ctrl** key, or by creating a selection box around the desired nodes by clicking and dragging the mouse.

### **Add, Insert, Modify and Delete Nodes**

#### **Add Node**

Clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new node to the list box of nodes displayed on the screen.

If any of the data entered for the new node is incorrect, a warning message will appear noting what the problem is.

#### **Insert Node**

Clicking the Insert button with the mouse or tabbing to the Insert button and pressing Enter will cause SODA to attempt to add a new node to the list box of nodes preceding of the node which is currently highlighted. (This differs from the Add button in that the Add button always appends the node information.)

If any of the data entered for the new node is incorrect, a warning message will appear noting what the problem is.

### **Modify Node**

Clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the node that is currently highlighted in the list box of nodes displayed on the screen.

NOTE: If none of the data associated with the node to be modified has been changed, then this action will have no effect.

If any of the data modified for the node is incorrect, a warning message will appear noting what the problem is.

### **Delete Node**

Clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the node which is currently highlighted in the list box of nodes displayed on the screen.

NOTE: If the node is in use in a nodal load, for example, SODA will not allow the deletion to occur until the corresponding nodal load has been modified or deleted.

### **Nodes List Box**

This displays all the nodes that have currently been entered into SODA. Although none of the displayed items are directly editable, clicking an item in the list box will place its contents in the editable fields above the list box. The user can move up and down the list by using the scroll bar on the right hand side of the list box.

### **Extended Selection - Selecting Multiple Nodes**

It is possible to modify or delete more than one item at a time by selecting multiple items from the list box. Holding down the shift key when clicking will extend the selection from the previously selected line to the line being clicked. Holding down the control key (Ctrl) will add the current line to the selection. Multiple lines can also be selected by dragging the mouse over lines in the list box.

## **Groups**

A Group is a collection of individual members that all have the same cross-section properties. SODA produces a design for each group and assigns that design to all members of each group.

### **Group**

Each Group of members is identified by a unique alphanumeric name that may have up to 9 characters (e.g., 4 or FOUR or Group4, etc.)

Default names are generated when the Regular Framework feature is invoked.

A Group Name is given even for the case of an individual member that has cross-section properties that are unique for the structure (i.e., a Group having only one member).

For Problem Type Design, Verify or Analysis, all individual members belonging to a particular Group are automatically assigned the cross-section properties specified for the Group.

For Problem Type Design, the SODA design process automatically determines the same cross-section designation (e.g. W12x24) for all individual members that belong to the same Group.

For Problem Type Design, the SODA design process automatically updates the cross-section designation (e.g. W12x24) for each Group after each Run Engine activity.

## **Shape**

SODA designs for eighteen different Section Shapes, as described in the following table:

SHAPE SECTION DESCRIPTION

WWF*	= Welded Wide Flange
W	= Wide Flange
WFOR*	= Foreign W-shapes
HP	= Bearing Piles
M	= Miscellaneous Beams and Columns
S	= Standard Beams
C	= Standard Channels
MC	= Miscellaneous Channels
RHS	= Rectangular Hollow Sections (Tubing)
SHS	= Square Hollow Sections (Tubing)
CHS	= Circular Hollow Sections (Pipe)
WWT*	= Tees cut from WWF sections
WT	= Tees cut from W Sections
EL1L**	= Equal Leg Angles
UL1L**	= Unequal Leg Angles
EL2L**	= Two Equal Leg Angles back to back
LL2L**	= Two Unequal Leg Angles with Long Legs back to back
SL2L**	= Two Unequal Leg Angles with Short Legs back to back
IE1L*	= Equal Leg Angles (Imperial series)
IU1L*	= Unequal Leg Angles (Imperial series)
IE2L*	= Two Equal Leg Angles back to back (Imperial series)
IL2L*	= Two Unequal Leg Angles with Long Legs back to back (Imperial series)
IS2L*	= Two Unequal Leg Angles with Short Legs back to back (Imperial series)
ME1L*	= Equal Leg Angles (Metric series)
MU1L*	= Unequal Leg Angles (Metric series)
ME2L*	= Two Equal Leg Angles back to back (Metric series)
ML2L*	= Two Unequal Leg Angles with Long Legs back to back (Metric series)
MS2L*	= Two Unequal Leg Angles with Short Legs back to back (Metric series)

\* CISC Database only

\*\* AISC Database only

For Problem Type Design or Verify, a Section Shape is specified by selecting one of the acronym descriptors appearing in the first column of the foregoing table (e.g., W, or WWF, etc.)

NOTE: The user may specify any 4 letter acronym for a shape as long as that same acronym appears in the '.SDC' or '.SDA' file associated with the section database selected in the Title screen. For example, WCOM, might be used to specify a custom database containing only compact W shapes.

For Problem Type Design, the SODA design process may be constrained to only consider sections for a Group that have the same specified nominal depth, as follows:

1. Specify the Section Shape that is desired for the members of the Group.

2. Specify the Minimum and Maximum Section Depths for the Group to be both equal to the same nominal section depth.

(For example, if Section Shape = W and Min. Depth = Max. Depth = 16, the SODA design process will only consider W Shapes having a nominal section depth = 16 for the members of the group.)

The Section Shape may or may not be specified for the Group if the Problem Type is Analysis; if specified, the Section Designation common to all members of the Group need alone be given in the Group Member Information; if not specified, the Section Area and Moments of Inertia common to all members of the Group need instead be given in the Group Member Information.

NOTE: If the Problem Type is Analysis, some Groups may be assigned a Section Shape and Designation, while other Groups may simple be assigned Area and Moment of Inertia values (in the latter case, click on <unknown> in the drop-down Designation list to activate the User-Defined fields in the Group worksheet).

## Designation

A section Designation uniquely identifies an individual section in the specified section Shape file of the Section Database.

If the Problem Type is Design, the section with the largest area or moment of inertia in the specified section Shape file, depending on whether the member is functioning axially or flexurally, is assigned to the Group by default to commence the SODA application.

A section Designation is specified for the Group if the Problem Type is Verify (i.e., the Designation existing for the Group for the structure to be verified.)

A section Designation is specified for the Group if the Problem Type is Analysis and a Section Database has been specified under the General Menu.

NOTE: The default section Designations for the CISC Database refer to Canadian sections, with the exception that those for the WFOR, M and MC Section Shapes refer to a Foreign section (since there are no corresponding section shapes available from Canadian mills).

The section Designation for each of the different section Shapes has a form similar to that given in the AISC or CISC Steel Design Manuals, except that it contains no letter characters other than the times symbol "X" (In fact, the usual letter characters are contained in the section Shape name). The user is encouraged to consult the Database section Shape files on disk to become familiar with the conventions employed by SODA for section Designations (also refer to the following NOTES).

NOTE: The section Designations for Single and Double Angle Section Shapes (EL1L, UL1L, EL2L, LL2L and SL2L) in the AISC Section Database are given in decimal form rather than fraction form, using the following decimal equivalents:

Fraction	Decimal
1/8	= .125
3/16	= .1875
1/4	= .25
5/16	= .3125
3/8	= .375
7/16	= .4375
1/2	= .5
9/16	= .5625
5/8	= .625
3/4	= .75
7/8	= .875

-----  
(For example, the Angle Section L3 1/2X2 1/2X7/16 in the AISC Manual of Steel Construction, Eighth Edition, has the Section Designation 3.5X2.5X.4375 in the AISC Section Database)

NOTE: The section Designations for Structural Tubing Section Shapes (RHS and SHS) in the AISC Section Database are given as:

**NOMINAL SIZE X WALL THICKNESS (decimal)**

(For example, the Square Structural Tubing having NOMINAL SIZE = 3.5X3.5 and WALL THICKNESS = .3125 in the AISC Manual of Steel Construction, Eighth Edition, has the Section Designation 3.5X3.5X.3125 in the AISC Section Database).

NOTE: The section Designations for Pipe Section Shapes (CHS) in the AISC **Section Database are given as:**

**NOMINAL DIAMETER X WALL THICKNESS (decimal)**

(For example, the Pipe having NOMINAL DIAMETER = 3 1/2 and WALL THICKNESS = .226 in the AISC Manual of Steel Construction, Eighth Edition, has the Section Designation 3.5X.226 in the AISC Section Database).

NOTE: The section Designations for Single and Double Angle Section Shapes (IE1L, IU1L, IE2L, IL2L, IS2L, ME1L, MU1L, ME2L, ML2L and MS2L) in the CISC Section Database are given as:

**SIZE X THICKNESS**

(For example, the Angle Section having SIZE = 125X90 and THICKNESS = 13 in the CISC Handbook of Steel Construction, Fourth Edition, has the section Designation 125X90X13 in the CISC Section Database).

NOTE: The section Designations for Hollow Structural Sections (RHS, SHS and CHS) in the CISC Section Database are given in terms of dimensions that have been "soft converted" from those specified in the CISC Handbook of Steel Construction, Fourth Edition; i.e., dimensions have been rounded to the nearest whole integer (with the exception that the two wall thicknesses 2.54 mm and 2.79 mm have been rounded down to 2 mm in all instances to achieve Section Designations that are uniquely different from all others).

NOTE: The section Designations for Rectangular and Square Hollow Section Shapes (RHS and SHS) in the CISC Section Database are given as:

**OUTSIDE DIMENSIONS X WALL THICKNESS**

(For example, the Rectangular Hollow Section having OUTSIDE DIMENSIONS = 101.6X76.2 and WALL THICKNESS = 9.53 in the CISC Handbook of Steel Construction, Fourth Edition, has the section Designation 102X76X10 in the CISC Section Database)

NOTE: The section Designations for Round Hollow Section Shapes (CHS) in the CISC Section Database are given as:

**OUTSIDE DIAMETER X WALL THICKNESS**

(For example, the Round Hollow Section having OUTSIDE DIAMETER = 323.9 and WALL THICKNESS =

6.35 in the CISC Handbook of Steel Construction, Fourth Edition, has the Section Designation 324X6 in the CISC Section Database).

For the specified section Shape, the corresponding default section Designation (e.g., as shown in the foregoing table) is assigned to the Group to commence the SODA design process when the Problem Type is DESIGN. While possible, it is NOT necessary to change the default section Designation (in fact, the SODA design process is quite insensitive to its initial starting point and will generally find the same final minimum-weight section for the members of the Group regardless of the initial default section Designation given).

For Problem Type Design, the section Designation for a Group may be "frozen" at a user-specified designation that remains unchanged by the SODA design process, as follows:

1. Specify the section Designation that is desired for the members of the Group for the final design of the structure (e.g., W16X89)
2. Specify the Minimum and Maximum Section Depths for the Group to be both equal to zero (0).

## Modulus

### Young's Modulus

The default value for the Young's Modulus for Steel depends on whether the specified Default Units are Imperial or Metric:

UNITS	YOUNG'S MODULUS
Imperial	29000 ksi
Metric	200000 MPa

While possible, it is not advisable to change the foregoing default values for Young's Modulus if the Problem Type is Design or Verify (i.e., since other values are not consistent with the Steel Design Codes supported by SODA).

Depending on the specified Default Units, any Young's Modulus value having the units ksi (Imperial) or MPa (metric) may be specified if the Problem Type is Analysis.

### Shear Modulus

The default value for the Shear Modulus for steel depends on whether the specified Default Units are Imperial or Metric:

UNITS	SHEAR MODULUS
Imperial	11200 ksi
Metric	77000 MPa

While possible, it is not advisable to change the foregoing default values for the Shear Modulus if the Problem Type is Design or Verify (i.e., since other values are not consistent with the steel Design Codes supported by SODA).

The Shear Modulus is not specified for a 2D structure if the Problem Type is Analysis.

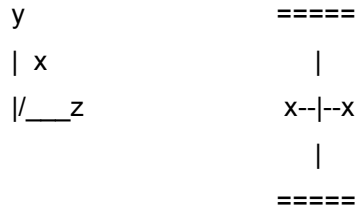
## Section Area

The cross-section Area is specified if the Problem Type is Analysis and a section Designation has not been specified for the Group.

NOTE: The Area is specified in units of INCHES\*\*2 (Imperial) or MILLIMETRES\*\*2 (metric).

## Moment of Inertia

The Moment of Inertia is specified if the Problem Type is Analysis and a Section Designation has not been specified for the Group. The Moment of Inertia for each local member axis is defined below:



$I_x$  = flexural moment of inertia about local x-axis of the cross-section

$I_y$  = flexural moment of inertia about local y-axis of the cross-section

$J_z$  = torsional moment of inertia about local longitudinal axis of the member

NOTE: The Moment of Inertia is specified in units of INCHES\*\*4 (Imperial) or MILLIMETRES\*\*4 (Metric).

## Stress

### Yield Stress

The default value for the Yield Stress depends on the specified Section Shape, the Design Code and whether the specified Default Units are Imperial or Metric.

If one of the AISC design codes has been specified, then the default values are as follows:

SECTION SHAPE	UNITS	YIELD STRESS
W,HP,M,S,C,MC, WT,EL1L,UL1L, EL2L,LL2L,SL2L	Imperial	36 ksi
	Metric	245 MPa
RHS,SHS,CHS	Imperial	50 ksi
	Metric	340 MPa

If a Canadian Design Code has been specified, then the default values are as follows:

SECTION SHAPE	UNITS	YIELD STRESS
W,WFOR,HP,M,S, C,MC,WT,EL1L,	Metric	300 MPa

UL1L,EL2L,LL2L, SL2L,WWF,WWT	Imperial	44 ksi
---------------------------------	----------	--------

---

RHS,SHS,CHS	Metric	350 MPa
	Imperial	50 ksi

---

Depending on the specified Default Units, any Yield Stress value having the units ksi (Imperial) or MPa (Metric) may be specified if the Problem Type is Design or Verify.

**Ultimate Stress**

The default value for the Ultimate Stress depends on the specified Section Shape and whether the specified Default Units are Imperial or Metric.

If one of the AISC design codes has been specified, then the default values are as follows:

SECTION SHAPE	UNITS	ULTIMATE STRESS
=====		
W,HP,M,S,C,MC	Imperial	58 ksi
WT,EL1L,UL1L, EL2L,LL2L,SL2L	Metric	395 MPa
-----		
RHS,SHS	Imperial	62 ksi
	Metric	425 MPa
-----		
CHS	Imperial	70 ksi
	Metric	480 MPa
-----		

If a Canadian Design Code has been specified, then the default values are as follows:

SECTION SHAPE	UNITS	ULTIMATE STRESS
=====		
W,WFOR,HP,M,S, C,MC,WT,EL1L, UL1L,EL2L,LL2L, SL2L,RHS,SHS, CHS,WWF,WWT	Metric	450 MPa
	Imperial	66 ksi
-----		

Depending on the specified Default Units, any Ultimate Stress value having the units ksi (Imperial) or MPa (Metric) may be specified if the Problem Type is Design or Verify.



## Max Allow KL/r

The Maximum Allowable Slenderness Ratios in Compression and Tension depend on the specified Steel Design Code:

MAX. ALLOW. SLENDERNESS RATIOS		
CODE	TENSION	COMPRESSION
AISC	240	200
CISC	300	200

For Problem Type Design or Verify, any change in the above given Maximum Allowable Slenderness Ratios should be in conformance with the specified Steel Design Code.

NOTE: Since SODA treats horizontal members as beam-columns even when they experience small axial load, it may be necessary to increase the Maximum Allowable Slenderness Ratio in Compression in order that beam-type sections can be selected for such members.

## Depth

When the Problem Type is Design, the section Depth for the members of the Group can be constrained by the SODA design process so as to satisfy certain fabrication and design requirements:

### Depth Min Max

Adequate headroom, utility space, etc. may be ensured by specifying the section Depth to be greater than a Minimum value and/or less than a Maximum value. The minimum and maximum default values are 0 and 10000 respectively to allow as much design latitude as possible.

### Setting a Fixed Depth

The design sections considered for the members of the Group may be constrained to have the same nominal depth by specifying the Minimum and Maximum section Depths to be equal to the required nominal depth value for the specified section Shape.

## "Freezing" a Designation

When the Shape and Designation are together specified, the design section for the members of the Group may be "frozen" to be the specified section by specifying the Min=0 and Max=0 section Depths to both be equal to zero (refer to section Designation).

## Units

Metric units are (mm) used for Canadian design codes regardless of the units the user has selected.

Imperial units (inches) are used for American design codes regardless of the units the user has selected.

## Nominal vs. Actual Depth

Sometimes the nominal depth may differ significantly from the actual depth of the section; therefore the designer should either customize the database to include only those sections that are of interest or be especially careful in specifying the depth limits.

## Graphical Selection

With the Groups window open, select a group by clicking the mouse left button while pointing at the desired member of the group in the Structure Topology window; the selected group will be highlighted in the Groups window.

## Add, Insert, Modify and Delete

## **Add Group**

Clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Group to the list box of Groups displayed on the screen.

If any of the data entered for the new Group is incorrect, a warning message will appear noting what the problem is.

## **Insert Group**

Clicking the Insert button with the mouse or tabbing to the Insert button and pressing Enter will cause SODA to attempt to add a new Group to the list box of Groups preceding of the Group which is currently highlighted. (This differs from the Add button in that the Add button always appends the Group information.)

If any of the data entered for the new node is incorrect, a warning message will appear noting what the problem is.

## **Modify Group**

Clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Group that is currently highlighted in the list box of Groups displayed on the screen.

NOTE: If none of the data associated with the Group to be modified has been changed, then this action will have no effect.

If any of the data modified for the Group is incorrect, a warning message will appear noting what the problem is.

## **Delete Group**

Clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Group which is currently highlighted in the list box of Groups displayed on the screen.

NOTE: If the Group is in use for a Member, for example, SODA will inform of this fact before proceeding with the deletion. If you such a member is deleted, it will also be removed from all of the load cases it is a part of.

## **Group List Box**

\*This displays all the groups that have currently been entered into SODA. Although none of the displayed items are directly editable, clicking an item in the list box will place its contents in the editable fields above the list box. The user can move up and down the list by using the scroll bar on the right hand side of the list box.

## **Extended Selection (for Analysis only)**

It is possible to modify or delete more than one item at a time by selecting multiple items from the list box. Holding down the shift key when clicking will extend the selection from the previously selected line to the line being clicked. Holding down the control key (Ctrl) will add the current line to the selection. Multiple lines can also be selected by dragging the mouse over lines in the list box.

## **Members**

### **Member Name**

Each Individual Member of the structure is identified by a unique alphanumeric name that may have up to 9 characters (e.g., 3, or THREE, or Beam-3, etc.).

Default names are generated when the Regular Framework feature is invoked.

### **Start Node**

Each Start Node is identified by a unique alphanumeric name that may have up to 9 characters (e.g., 2, or TWO, or NODE2, etc.). All nodes have been previously defined in the Nodes worksheet.

The Start Node can be assigned to any one of the two ends of the member, and the End Node is assigned to

the other end of the member.

Two members cannot have the same Start and End Nodes.

**Selecting from the List Box**

It is not necessary to type the name of the node. Instead click the arrow button beside the node name box and select a node from the drop-down list box that appears.

**End Node**

Each End Node is identified by a unique alphanumeric name that may have up to 9 characters (e.g., 2, or TWO, or NODE2, etc.)

The Start Node can be assigned to any one of the two ends of the member, and the End Node is assigned to the other end of the member.

The set of Start and End Nodes for a member must be unique (i.e., two members cannot have the same Start and End Nodes.)

**Selecting from the List Box**

It is not necessary to type the name of the node. Instead click the arrow button beside the node name box and select a node from the drop-down list box that appears.

**Group Name**

A Group is a collection of individual members that all have the same cross- section properties.

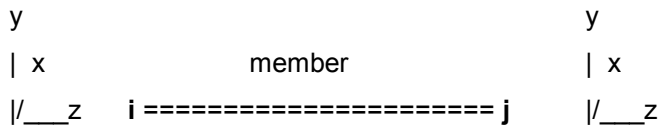
The Group name assigned to each individual Member must be previously specified in the Group worksheet.

**Selecting from the List Box**

It is not necessary to type the name of the Group. Instead click the arrow button beside the Group name box and select a Group from the drop-down list box that appears.

**Beta Angle**

The Beta Angle defines the rotation (twisting) of a member about its local longitudinal z-axis. It is recommended that the Beta Angle be specified as either 0 or a multiple of 90 degrees.



The Beta Angle is measured from a vertical plane containing the local longitudinal z-axis of the member to the local (weak) y-axis for the cross-section (e.g., the y-axis for a wide flange cross-section is coincident with the web).

The positive sense of a Beta Angle is defined by the right-hand screw rule (i.e., upon pointing the thumb of the right hand along the local longitudinal z-axis of the member from the start node to the end node, the direction of curl of the fingers defines the positive sense of the Beta Angle).

For horizontal girders, the default Beta Angle = 0 means that the local x-axis and local y-axis of the cross-section are horizontal and vertical, respectively (e.g., for a member subject to gravity loading, this means that bending takes place about the local (strong) x-axis of the cross- section).

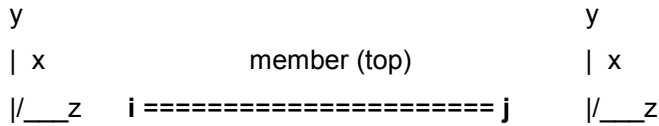
For vertical columns, the default Beta Angle = 0 means that the local minor y-axis of the cross-section lies in the global X-Y plane for the structure (i.e., bending of the column in the global X-Y plane takes place about

the local (strong) x-axis of the cross- section).

Local member axes are indicated in the Structural Topology when the corresponding check box is filled on the Graphics / View Options Menu in the Structural Topology window.

## Effective Length Factors

The factors  $K_x$ ,  $K_y$ ,  $B_t$  and  $B_b$  are applicable when the Problem Type is Design or Verify.



### **K<sub>x</sub>**

Effective Length Factor for axial buckling of the member about the local x-axis of the cross-section. Axial slenderness =  $KL/r$  ; where  $K = K_x$ ,  $L$  = actual member length,  $r = r_x$ .

### **K<sub>y</sub>**

Effective Length Factor for axial buckling of the member about the local y-axis of the cross-section. Axial slenderness =  $KL/r$  ; where  $K = K_y$ ,  $L$  = actual member length,  $r = r_y$ .

### **B<sub>t</sub>**

Effective Length Factor defining the unbraced length  $L_u$  of the compression flange for the top of a member in bending about the local x-axis (i.e., strong axis) of the cross-section. Note that the top of a member is the surface where the local positive y-axis projects from (refer to above figure).  $L_u = B_t * L$  ; where  $L$  = actual member length.

Use this factor to define the physical bracing of the top flange. During the design/verify process the physical braced length ( $B_t * L$ ) is compared to the unbraced length produced by the bending moment diagram. Except for certain conditions ( $B_t > 1$ ) the smaller of the two values will be used as  $L_u$ .

### **B<sub>b</sub>**

Effective Length Factor defining the unbraced length  $L_u$  of the compression flange for the bottom of a member in bending about the local x-axis (i.e., strong axis) of the cross-section.  $L_u = B_b * L$  ; where  $L$  = actual member length.

Use this factor to define the physical bracing of the bottom flange. During the design/verify process the physical braced length ( $B_b * L$ ) is compared to the unbraced length produced by the bending moment diagram. Except for certain conditions ( $B_b > 1$ ) the smaller of the two values will be used as  $L_u$ .

Note:  $B_b$  and  $B_t$  factors corresponding to bending about the local y-axis (i.e., weak axis) of the cross-section are not required because weak-axis bending capacity is not dependent upon the unbraced member length (for the design codes supported by SODA).

### **K<sub>x</sub> = 0, K<sub>y</sub> = 0**

Indicates full bracing against axial buckling of the member about the local x or y-axis of the cross-section.  $K_x = 0$  is rare but  $K_y = 0$  is possible for members whose flanges are continuously secured to wall, floor or roofing assemblies.

### **B<sub>t</sub> = 0, B<sub>b</sub> = 0**

Indicates full bracing of the top or bottom flange against lateral buckling.  $B_t = 0$  or  $B_b = 0$  is possible for members whose flanges are continuously secured to wall, floor or roofing assemblies.

### **K<sub>x</sub> = 1, K<sub>y</sub> = 1**

Indicates no bracing between the start and end nodes of the member. The full length of the member is used as the unbraced length. This situation is typical for trusses.

### **B<sub>t</sub> = 1, B<sub>b</sub> = 1**

Indicates no bracing between the start and end nodes of the member. The full length of the member is used as the unbraced length.

### **K<sub>x</sub> = Calc., K<sub>y</sub> = Calc.**

Indicates SODA will calculate  $K_x$  or  $K_y$  based on the relative stiffnesses of members framing into the start and end nodes of the member. The method used is that outlined in the appendices of both Canadian and American handbooks of steel design. Calculated values appear in the SODA output file of Effective Lengths.

### **$B_t > 1$ , $B_b > 1$**

During the design/verify process the physical bracing ( $B_t \cdot L$  and  $B_b \cdot L$ ) is compared to the unbraced lengths produced by the bending moment diagram. The smaller of the two values will be used for each flange. However, if a  $B_t$  or  $B_b$  value is specified as greater than unity then  $B_t \cdot L$  or  $B_b \cdot L$  will be used instead of the lengths from the moment diagram (refer to the Special Situations note below).

### **Special Situations and Recommendations**

If a member is divided into two (or more) segments by placing node(s) in span, the user must directly provide the value of  $K_x$ ,  $K_y$ ,  $B_t$  and  $B_b$  for each of the segments so as to reflect the total length of the member. For example, set  $K_x = K_y = B_t = B_b = 2.0$  for each of the two segments of a member with a mid-span node and no lateral bracing.

## **Joint Type (Compression-only / Tension-only)**

The Joint Types define the end-fixity conditions at the member Start Node -->End Node:

- o---o = PINNED-PINNED
- +---+ = FIXED-FIXED
- +---o = FIXED-PINNED
- o---+ = PINNED-FIXED

Only the PINNED-PINNED condition may be specified for a Truss member, whereas any one of the four end-fixity conditions may be specified for a Frame member.

The Joint Type field is also used to define a member as being Compression-only or Tension-only:

- C---C = COMPRESSION-ONLY
- T---T = TENSION-ONLY

A member defined as C---C or T---T is automatically specified to have PINNED-PINNED end-fixity condition.

## **Add, Insert, Modify and Delete Member**

### **Add Member**

Clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Member to the list box of Members displayed on the screen.

If any of the data entered for the new Member is incorrect, a warning message will appear noting what the problem is.

### **Insert Member**

Clicking the Insert button with the mouse or tabbing to the Insert button and pressing Enter will cause SODA to attempt to add a new Member to the list box of Members preceding of the Member which is currently highlighted. (This differs from the Add button in that the Add button always appends the Member information.)

If any of the data entered for the new node is incorrect, a warning message will appear noting what the problem is.

## **Graphical Selection**

With the Members window open, select a member by clicking the mouse left button while pointing at the desired member in the Structure Topology window; the selected member will be highlighted in the Members

window. Several Members may be selected by clicking on each desired member while holding down the **Ctrl** key, or by creating a selection box around the desired members by clicking and dragging the mouse.

## **Add, Insert, Modify and Delete Member**

### **Modify Member**

Clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Member that is currently highlighted in the list box of Members displayed on the screen.

NOTE: If none of the data associated with the Member to be modified has been changed, then this action will have no effect.

If any of the data modified for the Member is incorrect, a warning message will appear noting what the problem is.

### **Delete Member**

Clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Member which is currently highlighted in the list box of Members displayed on the screen.

NOTE: If the Member is in use in a Member Load, for example, SODA will inform of this fact, and ask whether to proceed with the deletion. Deleting a member will delete that member from all the Loads that it is associated with.

### **Member List Box**

This displays all the members that have currently been entered into SODA. The user can move up and down the list by using the scroll bar on the right hand side of the list box.

### **Extended Selection**

It is possible to modify or delete more than one item at a time by selecting multiple items from the list box. Holding down the shift key when clicking will extend the selection from the previously selected line to the line being clicked. Holding down the control key (Ctrl) will add the current line to the selection. Multiple lines can also be selected by dragging the mouse over lines in the list box.

## **Regular Framework**

This dialog box allows the user to specify a rectangular framework in a single batch operation. Specifically, SODA automatically assigns:

- default names to all nodes, members and fabrication groups
- default values for material modulus, design stresses, section depth limits, maximum allowable slenderness ratios, and effective length factors
- default fabrication groupings, end-fixity conditions, section shape designations and support type

The default names, assignments and values for nodes, members, groups, supports material, etc. may be later modified in the Nodes, Members and Groups worksheets under the Structure Menu.

Bracing members may be subsequently added to either a truss or a flexural frame with the Diagonal Bracing feature.

X, Y, Z Directions:

Bays/Storey

Dimension:

Storeys Per Group:

Roof/Exterior Groups:

Pinned/Fixed Joints:

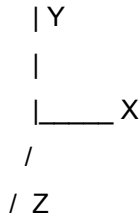
Section Shape:

Beta Angle:

Pinned/Fixed Supports:

Default Naming Conventions:

## X, Y, Z, Directions



The horizontal bays and vertical storeys of a rectangular framework are specified in the global X and Z-axis directions (horizontal) and the global Y-axis direction (vertical).

## Bays/Storeys

The number of bays in the global X-axis or Z-axis direction and the number of storeys in the global Y-axis direction for a rectangular framework.

## Dimension

The common width of all bays in the global X-axis or Z-axis direction and story heights in the global Y-axis direction for a rectangular framework.

## Storeys Per Group

### X-Direction (comments also apply to the Z-Direction)

Enter the number of storeys of a rectangular framework over which the horizontal members running parallel to the global X-axis are grouped together as having common cross-section properties (i.e., belonging to the same fabrication group). The intent here is to allow the user to specify how often a new beam design group is required. The smallest value is 1, indicating that a new group is created for each floor. The largest value is the number of storeys, which indicates that only one design group will apply over all floors.

#### Default = 1

The members are grouped together into a single fabrication group for each story.

#### Trusses

Members cannot be grouped when creating a regular framework for a truss.

### Y-Direction

Enter the number of storeys of a rectangular framework over which the vertical members running parallel to the global Y-axis are grouped together as having common cross-section properties (i.e., belonging to the same fabrication group).

#### Default = 20

Beginning from the base of the structure, columns are by default grouped together over two storeys (i.e., a prismatic column extends over storeys 1 and 2, etc.).

#### Trusses

Members cannot be grouped when creating a regular framework for a truss.

## Roof/Exterior Groups

This option permits the user to specify that members on the exterior of a framework are to be considered as a

separate group from that for interior members. The intent here is to recognize that roof beams experience different loads than interior floor beams and exterior columns experience different loads than interior columns.

**Default**

Exterior columns are grouped separate from interior columns at each story level.

**Pinned/Fixed Joints**

This option permits the user to specify either Fixed-Fixed or Pinned- Pinned as the common Joint Type prevailing at the end-sections of all of the global X-direction or Z-direction members (or Y-direction columns) for a rectangular (moment-resistant) framework.

**Trusses**

Only Pinned-Pinned members may be specified for a truss.

**Section Shape**

The common Section Shape for the global X-direction or Z-direction members (or Y-direction columns) for a rectangular framework.

This field is not available if the problem type is Analysis and no Section Database has been specified under the General Menu.

**Pinned/Fixed Supports**

The common support types at the base of all first- story columns of a rectangular (moment-resistant) framework. Less common support conditions may be specified in the Nodes screen once the framework has been generated.

**Fixed** = a support node that cannot rotate or translate (in any plane).

**Pinned** = a support node that can rotate but cannot translate in the X-Y plane (2D structure) or in the X-Y, Z-Y and X-Z planes (3D structure).

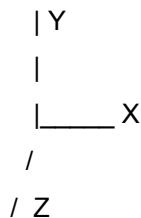
**Trusses**

Only Pinned supports may be specified for a truss.

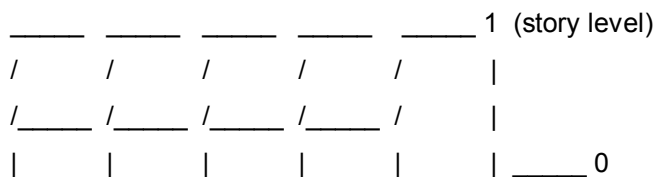
A truss will be created with supports at each corner only.

**Default Naming Conventions: Regular Frameworks**

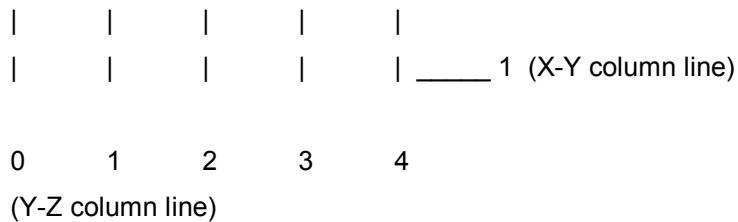
The default names assigned for nodes, members and groups may be later viewed in the Nodes, Members and Groups worksheets under the Structure Menu.



**Numbering of Column Lines and Story Levels**







### Y-Z Column Lines

The column line that lies in the Y-Z plane of the global co-ordinate axis system (i.e., corresponding to an exterior face of the structure) is the initial column line in the global X-axis direction and is identified as column line 0 (zero) in that direction.

### X-Y Column Lines

The column line that lies in the X-Y plane of the global co-ordinate axis system (i.e., corresponding to an exterior face of the structure) is the initial column line in the global Z-axis direction and is identified as column line 0 (zero) in that direction.

### X-Z Story Levels

The story level that lies in the X-Z plane of the global co-ordinate axis system (i.e., corresponding to the support foundation level for the structure) is the initial story level in the global Y-axis direction and is identified as story level 0 (zero).

### Node Naming Conventions

Each default node name is assigned by SODA as:

**xijzk**

**i** = column line no. in global X-axis direction

**j** = story level no. in global Y-axis direction

**k** = column line no. in global Z-axis direction

Example

x3y1z2 = node at intersection of members corresponding to column line 3 in that X-axis direction, story level 1 in the Y-axis direction and column line 2 in the Z-axis direction.

x2y0z1 = support node at the base (i.e., story level 0) of the column member in column line 2 in the X-axis direction and column line 1 in Z-axis direction.

### Member Naming Conventions

The member name is expected to be of the form xijzk where (i, j, k) denotes the x-column line, the y-story level and z-column line, respectively. One of 'x', 'y' and 'z' is replaced by an 'm' to indicate a member parallel to that axis.

### X-Direction

The default member name for each horizontal member in the global X-axis direction is assigned by SODA as:

**miyjk**

**i** = bay no. in global X-axis direction over which the member spans (i.e., from column line i-1 to i).

**j** = story level no. in global Y-axis direction

**k** = column line no. in global Z-axis direction

### Example

m3y2z4 = horizontal member that spans bay 3 in the X-axis direction at story level 2 in the Y-axis direction

and in column line 4 in the Z-axis direction.

### **Z-Direction**

The default member name for each horizontal member in the global Z-axis direction is assigned by SODA as:

**xijm<sub>k</sub>**

**i** = column line no. in global X-axis direction

**j** = story level no. in global Y-axis direction

**k** = bay no. in global Z-axis direction over which the member spans (i.e., from column line k-1 to k).

### **Example**

x2y2m2 = horizontal member that spans bay 2 in the Z-axis direction at story level 2 in the Y-axis direction and in column line 2 in the X-axis direction.

### **Y-Direction**

The default member name for each vertical member in the global Y-axis direction is assigned by SODA as:

**ximjz<sub>k</sub>**

**i** = column line no. in global X-axis direction

**j** = story level no. in global Y-axis direction over which the column spans (i.e., from story j-1 to j).

**k** = column line no. in global Z-axis direction

### **Example**

x3m4z1 = vertical column that spans story 4 in the Y-axis direction at the intersection of column line 3 in the X-axis direction and column line 1 in the Z-axis direction.

## **Group Naming Conventions**

The group name is expected to be of the form GX<sub>i-j</sub>, GY<sub>i-j</sub> and GZ<sub>i-j</sub> where G denotes a Group name, X, Y and Z denote members parallel to each axis respectively and i-j is the story range over which a fabrication group applies.

### **X-Direction**

The default group name for each fabrication group of members in the global X-axis direction is assigned by SODA as:

**GX<sub>i-j</sub>**

**i-j** = story range in the global Y-axis direction over which all X-axis members are in the fabrication group

### **Example**

GX3-7 = a fabrication group consisting of all X-axis members over storeys 3 to 7 inclusive.

### **Z-Direction**

The default group name for each fabrication group of members in the global Z-axis direction is assigned by SODA as:

**GZ<sub>i-j</sub>**

**i-j** = story range in the global Y-axis direction over which all Z-axis member are in the fabrication group

### **Y-Direction**

If vertical exterior and interior members are not separated into different fabrication groups, the default group name for each fabrication group of vertical members is assigned by SODA as:

**GY<sub>i-j</sub>**

**i-j** = story range in the global Y-axis direction over which all Y-axis columns are in the fabrication group

## **Y-Direction - Exterior Columns**

The default group NAME for each fabrication group of Exterior Columns is assigned by SODA as:

ECi-j

i-j = story range in the global Y-axis direction over which all Y-axis exterior members are in the fabrication group

### Example

EC7-7 = a fabrication group consisting of all exterior members for story 7 of the structure.

### Y-Direction - Interior Columns

The default group name for each fabrication group of vertical interior members is assigned by SODA as:

ICi-j

i-j = story range in the global Y-axis direction over which all Y-axis interior columns are in the fabrication group

### Irregular Framework

This series of dialog boxes allows the user to specify an irregular framework in one batch operation.

Initially, the user creates a single 2D or 3D module of the structure containing the irregular geometry features that are to be automatically copied in an expanded structure. Thereafter:

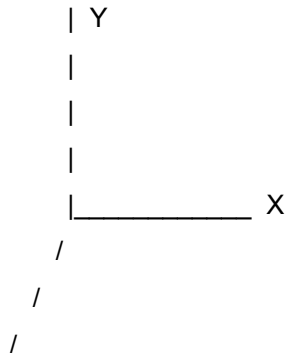
The dialog box IRREGULAR FRAMEWORK EXPANSION identifies the plane of an exterior face of the current structure that is to be copied in the expanded structure;

The dialog box IRREGULAR FRAMEWORK FACE VERIFICATION highlights the face of the structure to be copied and provides an opportunity to save the file before proceeding;

The dialog box IRREGULAR FRAMEWORK DEFINITION identifies the number of times the face is to be copied, the spacing between copied faces, the fabrication group for the out-of-plane members connecting the copied faces, and the loads that are also to be copied for the expanded structure.

The nodes and members in the expanded structure are assigned default names, which may be later viewed in the Nodes and Members worksheets under the Structure Menu.

### X-Y and Y-Z and X-Z Planes



Each pair of axes X-Y, Y-Z and X-Z defines a principal plane of the structure that may be copied using the Irregular Framework command.

### **Front(nearest) / Back(furthest)**

Copy either the front(nearest) or back(furthest) exterior face of the structure that lies in the selected principal plane.

### **Out-Of-Plane Member Group**

Select the fabrication Group for the connecting members between copied faces from the drop-down list of Group names (click on the down arrow beside the Out-Of-Plane Member Group field). The Group must be previously specified in the Group Worksheet under the Structure Menu.

### **Number of copies**

Specify the number of times the selected exterior face of the structure is to be copied ( i.e., 1, 2, etc.).

### **Copy offset**

Specify a uniform offset distance between the copied faces of the structure.

### **Copy loads / settlements**

Specify the Nodal, Member, Temperature Loads and/or the Support Settlements that are to be copied for the expanded structure.

### **Exclude nodes and members**

Click on, and thereby grey out, those nodes and members that are not to appear in the expanded structure.

### **Default Naming Conventions: Irregular Frameworks**

The Irregular Framework option automatically names the nodes and members created in the expanded part of the structure, as follows:

**Node Name:** The first two indices define the axes of the plane of the copied face (i.e., x-y, x-z or y-z), the third index defines the number of the copied face (i.e., 1, 2, etc.), the fourth index is the letter 'n' (i.e., for 'node'), the final index is the number of the node in the copied face (i.e., 0, 1, 2, etc.).

**Member Name:** The first two indices define the axes of the plane of the copied face (i.e., x-y, x-z or y-z), the third index defines the number of the copied face (i.e., 1, 2, etc.), the fourth index is the letter 'm' (i.e., for 'member'), the final index is the number of the member in the copied face (i.e., 0, 1, 2, etc.).

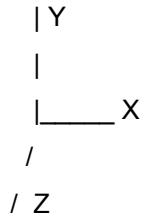
**Name of Connecting Member Between Copied Faces:** The first and third indices are the axis orientation of the member (i.e., x, y or z), the second index is the number of the face at which the member starts (i.e., 0, 1, 2, etc.), the fourth index is the number of the face at which the member ends (i.e., 1, 2, etc.), the fifth index is the letter 'm' (i.e., for 'member'), the final index is the number of the member between copied faces (i.e., 1, 2, etc.).

The default names assigned to nodes and members may be later viewed in the Nodes and Members worksheets under the Structure Menu.

## Diagonal Bracing

This feature generates single or x bracing patterns in regular frame and truss structures. It should only be used on structures generated by the Regular Framework feature as it depends upon the default node naming convention used by Regular Framework.

### Surface



### Plan X,Y,Z

Select a surface on which to place the bracing. Plan X, Y or Z refers to the surface of the structure that is orthogonal to the global X, Y or Z-axis directions (e.g., Plan X surfaces are viewed by looking along the X-axis).

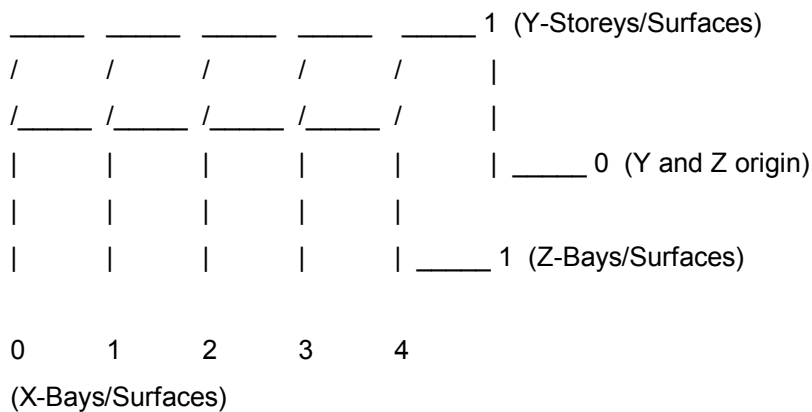
### Partial

Leave this check box blank to place bracing on all surfaces of the selected plan.

Checking this box will permit the user to specify a range of bays and storeys.

## Surface Ranges

### Numbering of Surfaces, Bays and Storeys



The above figure indicates the numbering to use when specifying a range of bays or storeys on which to place diagonal bracing.

### Note the following definitions

*Surfaces* : bracing is placed on a surface

*Bays* : a surface spans between two bays

*Storeys* : a surface spans between two storeys

### Surface Range

When specifying a surface range the start and end values may have the same number indicating that only one surface will be braced. If no surface range is entered a range is 0-0 is assumed which will limit bracing to the first surface (labeled as 0 in the above figure).

### Bay and Storey Ranges

When specifying a bay or storey range the start and end values must be different or else there is no distance for the brace to span.

**SODA will not warn the user if a zero range is entered and no bracing will be generated.**

### **X-Surface Bracing**

Bracing may be limited to a range of *X-Surfaces*, *Y-Storeys* and *Z-Bays*.

### **Y-Surface Bracing**

Bracing may be limited to a range of *X-Bays*, *Y-Surfaces* and *Z-Bays*.

### **Z-Surface Bracing**

Bracing may be limited to a range of *X-Bays*, *Y-Storeys* and *Z-Surfaces*.

### **Example**

An *X-Surface* range of 0-1 limits X surface bracing to the zero and 1st X-Bays.

A *Y-Story* range of 0-1 limits bracing to the first story (i.e., from the ground or zero story to the 1st story).

A *Z-Bay* range of 0-1 limits bracing to the first Z bay (i.e., from the zero bay to the 1st bay)

### **Group**

The generated bracing members are associated with the highlighted fabrication Group in the list box of current Groups. (New groups cannot be added here; this must be done in the Groups worksheet).

### **Excluded Surface**

For the selected Plan X, Y or Z, the user can specify that diagonal bracing is excluded from either the exterior or interior surface of the structure.

### **Bracing Type**

X = cross bracing

/ = right-leaning single brace

\ = left-leaning single brace

### **Add, Replace and Cancel**

#### **Add Button**

Adds the new diagonal braces to the selected Group, which may or may not already have members in it.

#### **Replace Button**

Replaces all existing members of the selected Group with the new diagonal bracing members. To avoid deleting other structural members, an independent group should be used for braces.

#### **Cancel**

Exits the dialog box.

### **Move Nodes**

This worksheet allows the user to move one or more nodes of a structure.

Note: Example # 3 describes in **/General/Title/** the procedure to create a peaked roof from a flat roof.

### **Node Selection**

The node name is expected to be of the form <Prefix><Value><Suffix> .The case (upper or lower) of the letters in the Prefix and Suffix is significant (e.g., a lowercase suffix is considered to be different than an uppercase SUFFIX).

Nodes to be moved are selected by specifying the <Prefix> and <Suffix> and central <value> of the node names. The node names may be those generated by the Regular Framework command, or they may have been manually entered into SODA.

**Prefix**

The string <Prefix> which begins the names of all the nodes to be moved

(e.g.,.Moving x1y1 through x9y1,.the prefix is x)

**Suffix**

The string <Suffix> which ends the names of all the nodes to be moved

(e.g.,.Moving x1y1 through x9y1,.the suffix is y1)

**From**

The central <value> of the first node to be moved. If no value is entered, 0 is assumed.

**To**

The central <value> of the last node to be moved. If no value is entered, 9999 is assumed.

Note: The From <value> and To <value> must respectively correspond to the beginning and end of a sequence of nodes having central <values> that progressively increment by one (1).

To move a single node, specify From <value> = To <value>.

**Movement**

The Base and Increment values determine the distance by which each selected node is moved. The movement units are the current distance units.

The selected nodes may be moved in any global-axis direction; each of X, Y and Z directions have Base and Increment movement values. The actual distance a node is moved is determined by the following formula:

$\text{Movement} = \text{base} + \text{increment} * (\text{num} - \text{from})$

where 'num' is the node number as determined by the selection, and 'from' is the value entered in the FROM field. Both Base and Increment values may be negative, and any combination of X, Y and Z movement is permitted simultaneously.

**Move/Cancel**

Click Move to execute the specified movement operations. It is a good idea to save the current file prior to executing a Move command because it cannot be easily undone if you make a mistake. To undo a Move you must reverse the effect of the original move by reversing the signs of the Base and Increment values.

Click Cancel to exit the dialog box and return to the main worksheet.

**View Structure**

This selection displays the structure on the screen in accordance with the options selected under the Graphics item of the Structure Topology menu bar.

# SODA Loads Commands

## Nodal Loads

### Load Name

Each Load Name defines a set of individual Nodal, Member, Temperature, or Support Settlement loads that are applied to one or more nodes or members of the structure at their service load level.

Each Load Name is specified by a unique alphanumeric string that may have up to 9 characters (e.g. DL1, or DeadL1, etc.)

It is desirable to specify a descriptive Load Name that is easily recognized when later specifying the Load Combinations for the structure.

When later specifying the Load Combinations for the structure, a common Load factor is applied to all individual Nodal, Member or Temperature loads in each set of loads defined by a Load Name.

### Add, Modify, Delete a Load Name

#### Add

For the left-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Load Name to the list box.

If there are blanks in the Load Name, or if no name is given, a warning message will appear noting what the problem is.

#### Modify

For the left-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name to be modified has not been changed, then this action will have no effect.

If there are blanks in the new Load Name, or if no name is given, a warning message will appear noting what the problem is.

#### Delete

For the left-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name is part of a previously specified Load Combination, SODA will not allow the deletion to occur until the Load Combination has been modified to exclude the Load Name.

### Node

The Node name identifies a particular node of the structure to which individual Nodal Force(s) and/or Couple(s) are applied.

#### Selecting from the List Box

It is not necessary to type the name of the node. Instead click the arrow button beside the Node box and select a node name from the drop-down list box which appears.

### Forces and Couples

#### Force X,Y,Z



A positive or negative valued service-level Nodal Force that is applied to the node in the positive or negative global X,Y,Z-axis direction, respectively.

### **Couple X,Y,Z**

A positive or negative valued service-level Nodal Couple that is applied to the node about the global X,Y,Z-axis direction in a positive or negative sense in accordance with the Right-hand Screw Rule.

A Nodal Couple may be applied at a node to reflect the effect of any significant eccentricity arising from the manner in which the members are connected together at the node.

### **Loads at Supports**

A Nodal Force or Couple should not be applied to a support node in a direction or sense for which there is no corresponding degree of freedom to translate or rotate (if this is done, SODA will inform that such loading will have no affect on the structure response).

### **Adding Nodal Loads Graphically**

With the Nodal Loads window open, first add the Load Name and Forces desired and then select a node to load by clicking the mouse right button while pointing at the desired node in the Structure Topology window; the selected node will be added in the Nodes List box of the Nodal Loads window. Continue to point and click until all desired nodal loads are added to the Nodes List box.

### **Add, Modify, or Delete a Loaded Node**

#### **Add**

For the right-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Node (and associated loads) to the list box.

NOTE: The loaded Nodes in the right-hand list box are associated with the highlighted Load name in the left-hand list box.

If any of the data entered for the new Node is incorrect, a warning message will appear noting the problem.

#### **Modify**

For the right-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Nodal Forces and/or Couples associated with the Node that is currently highlighted in the list box.

NOTE: The loaded Nodes in the right-hand list box are associated with the highlighted Load name in the left-hand list box.

If any of the data entered for the new Node is incorrect, a warning message will appear noting the problem.

#### **Delete**

For the right-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Nodal Forces and/or Couples associated with the Node that is currently highlighted in the list box.

NOTE: The loaded Nodes in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

### **Nodal Loads List Box**

The right-hand list box displays all the loaded nodes that have currently been entered into SODA under the specified load name. The user can move up and down the list by using the scroll bar on the right hand side of the list box. Above the list box is a pick list with all the node names for the structure.

### **View**

Clicking the VIEW button or double-clicking will cause SODA to display the Nodal Load graphic for the loaded node or

member which is currently highlighted.

The Load Name, Node Name, the load magnitude, and the nodal co-ordinates are displayed on the Load graphic.

Single and double-headed arrows are used to indicate Forces and Moments, respectively, in the Nodal Load graphic. The sign convention follows the Right Hand Screw rule.

## **Batch Loads**

The Batch Load feature allows the user to quickly specify an entire range of nodes to load. Prior to clicking the Batch button the user must enter the nodal/member load information. Upon clicking the Batch button the following options are presented:

### **Load Surface**

Select the surface to be loaded. Plan Y is the plan view as seen along the global Y axis; similarly for Plan X and Plan Z. Selecting Partial will allow the specification of a range of floor or wall surfaces for loading.

### **Excluded Nodes**

Allows the user to specify that the exterior or interior nodes lying in the selected Plan X, Y or Z surface(s). This only works for 3D structures using the node naming convention of the Regular Framework feature.

### **Surface Ranges**

Specify the range of floors or column lines to be loaded. The default is all floors or column lines. If Partial has been selected, the loaded portion of the surface should also be entered.

### **Named**

When the Named option is selected the Named Nodes part of the dialog box becomes available to allow the user to specify a range of loaded members or nodes that have been named differently than the default naming convention of the Regular Framework feature.

Note: with the exception of the Named option, the Batch feature may only be used for structures that were previously created using Regular Framework.

## **Add, Replace, Cancel**

### **Add Button**

Adds the new loads to the currently specified Load Name (i.e., that highlighted in the parent window), which may or may not be already associated with loaded nodes or members.

### **Replace Button**

Replaces existing loads associated with the currently specified Load Name with the new loads. SODA first asks the user if the existing loads are to be overwritten.

### **Cancel Button**

Exits the dialog box.

## **Member Loads**

### **Load Name**

If there are blanks in the Load Name, or if no name is given, a warning message will appear noting the problem.

Each Load Name is a unique alphanumeric string that may have up to 9 characters (e.g. DL1, or DeadL1, etc.) and defines a set of individual Member loads that are applied to one or more members of the structure.

It is desirable to specify a descriptive Load Name that is easily recognized when later specifying the Load Combinations for the structure.

When later specifying the Load Combinations for the structure, a common Load factor is applied to all individual Member loads in each set of loads defined by a Load Name.

## **Add, Modify, Delete a Load Name**

### **Add**

For the left-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Load Name to the list box.

### **Modify**

For the left-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name to be modified has not been changed, then this action will have no effect.

### **Delete**

For the left-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name is part of a previously specified Load Combination, SODA will not allow the deletion to occur until the Load Combination has been modified to exclude the Load Name.

## **Member**

The Member name identifies a particular member of the structure to which an individual Point Load or Distributed Load is applied.

### **Selecting from the List Box**

It is not necessary to type the name of the member. Instead click the arrow button beside the Member box and select a member name from the drop-down list box.

## **Load Type**

SODA allows for the different Member Load Types listed in the following:

UDL = Uniformly Distributed Load.

TDL = Trapezoidal or Triangular Distributed Load.

Full = a distributed load that extends over the full length of the member.

Partial = a distributed load that extends over only part of the length of the member.

Point Load = a concentrated load applied at a point in the span of the member.

Other DL = Other Distributed Loads such as a Partial UDL or a Full or Partial TDL.

# SODA Loads Commands (cont'd)

## Member Loads (cont'd)

Orient. ( ) Type	Y axis ( Y )	X or Z axis ( X or Z )	Axial ( A )	Perpendic. ( P )
Point Load				
Full UDL				
Partial UDL				
Full TDL				
Partial TDL				

UDL = Uniformly Distributed Load  
 TDL = Trapezoidal (Triangular) Distributed Load

## Load Information

### Load Intensity at Start of Load

$W@a$  = the load Intensity at the location on the member where a Distributed Load begins (or the load Magnitude at the point of application of a concentrated Point Load).

NOTE: The load quantities  $w@a$  and  $w@b$  are specified as positive or negative in accordance with the Global X-Y-Z Axes System. An exception to this occurs when the member load is specified as acting in the direction of the local x, y or axial axes of the member (see the Orientation field.)

### Distance to Start of Load

$a/L$  = the Decimal Fraction of the member length measured from the Start Node at which a Distributed Load begins (or at which a concentrated Point Load is applied).

### Load Intensity at End of Load

$w@b$  = the load Intensity at the location on the member where a Distributed Load ends.

NOTE: The load quantities  $w@a$  and  $w@b$  are specified as positive or negative in accordance with the Global X-Y-Z Axes System. An exception to this occurs when the member load is specified as acting in the direction of the local x, y or axial axes of the member (see the Orientation field.)

### Distance to End of Load

$b/L$  = the Decimal Fraction of the member length measured from the Start Node at which a Distributed Load ends.

### Inclined members

The fractions  $a/L$  and  $b/L$  refer to distances measured along the local longitudinal axis of the member.

### Orientation

From the drop-down list for this input field to select a load orientation for the member. Click the arrow button to produce the following selections:

NOTE: Do not apply member loads with a global orientation which is coincident with the longitudinal axial of the member (i.e. 'Y' member loads applied to vertical columns).

### 2D Structures only

**Perpendic.** = a distributed load or a point load acting perpendicular to the length of the member. The signed-direction of a Perpendic.-type load is dependent on the global axes and is independent of the local axes of the member. For horizontal or inclined members the signed-direction is based on the global Y-axis ( the load is negative if any component of the load points down). For vertical members the global X-axis is used ( the load is negative if the load points to the left).

### 3D Structures only

**y** = a distributed load or a point load acting perpendicular to the local longitudinal axis of the member in a direction parallel to the local y-axis. A y-type load has the same positive and negative sense as the local y-axis.

**x** = a distributed load or a point load acting perpendicular to the local longitudinal axis of the member in a direction parallel to the local x-axis. A x-type load has the same positive and negative sense as the local y-axis.

**Z** = a distributed load or a point load acting in the global Z-axis direction.

### 2D or 3D Structures

**Axial** = a distributed or point load acting along the local longitudinal axis of the member. An Axial-type load is

positive when directed from the Start Node to the End Node of the member.

**Y** = a distributed load or a point load acting in the global Y-axis direction.

**X** = a distributed load or a point load acting in the global X-axis direction.

## **Adding Member Loads Graphically**

With the Member Loads window open, first add the Load Name and Member Forces desired and then select a member to load by clicking the mouse right button while pointing at the desired member in the Structure Topology window; the selected member will be added in the Members List box of the Member Loads window. Continue to point and click until all desired member loads are added to the Members List box.

## **View**

Clicking the VIEW button or double-click will cause SODA to display the Member Load graphic for the loaded member which is currently highlighted.

The Load Name, Member Name, load magnitude, and member-end nodes, are displayed on the Load graphic.

The Member Load graphic displays load magnitude and direction with respect to the local axes of the member. Thus, a global Y-axis load applied to an inclined member will be displayed with both local x and y axes components. The member is always displayed with the local x-axis running from left to right (i.e., from i-node to j-node as entered in the Members worksheet).

## **Batch Loads**

The Batch Load feature allows the user to quickly specify an entire range of members to load. Prior to clicking the Batch button the user must enter the member load information. Upon clicking the Batch button the following options are presented:

### **Named**

When the Named option is selected the Named Members/Nodes part of the dialog box becomes available to allow the user to specify a range of loaded members or nodes that have been named differently than the default naming convention of the Regular Framework feature.

Note: with the exception of the Named option, the Batch feature may only be used for structures that were previously created using Regular Framework.

### **Group Name**

When the Group Name option is selected the Group Name field of the dialog box becomes available. Click on the down arrow to see the list of all member groups. Select the name of the group for which all its members are to be loaded with the batch loads.

### **Load Surface**

Select the surface to be loaded. Plan Y is the plan view as seen along the global Y axis; similarly for Plan X and Plan Z. Selecting Partial will allow the specification of a range of floor or wall surfaces for loading.

### **Excluded Members**

Allows the user to specify that the exterior or interior X and Z-axis beams and Y-axis columns lying in the selected Plan X, Y or Z surface(s) are to be excluded from being loaded. This only works for 3D structures using the member naming convention of the Regular Framework feature.

### **Surface Ranges**

Specify the range of floors or column lines to be loaded. The default is all floors or column lines. If Partial has been selected, the loaded portion of the surface should also be entered.

## **Add, Replace, Cancel**

### **Add Button**

Adds the new loads to the currently specified Load Name (i.e., that highlighted in the parent window), which may or may not be already associated with loaded members.

### **Replace Button**

Replaces existing loads associated with the currently specified Load Name with the new loads. SODA first asks the user if the existing loads are to be overwritten.

### **Cancel Button**

Exits the dialog box.

## **Add, Modify and Delete Member Load Data**

NOTE: The loaded Members in the right-hand list box are associated with the highlighted Load name in the left-hand list box. If any of the data entered for the new Member is incorrect, a warning message will appear.

### **Add Member Load Data**

For the right-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Member (and associated loads) to the list box.

### **Modify Member Load Data**

For the right-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the loads associated with the Member that is currently highlighted in the list box.

### **Delete Member Load Data**

For the right-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the loads associated with the Member that is currently highlighted in the list box.

### **Loaded Member List Box**

The right-hand list box displays all the loaded members that have currently been entered into SODA under the specified load name. The user can move up and down the list by using the scroll bar on the right hand side of the list box. Above the list box is a pick list with all the member names for the structure.

## **Temperature Loads**

### **Load Name**

Each Load Name defines a set of individual Temperature loads that are applied to one or more members of the structure at their service load level.

Each Load Name is a unique alphanumeric string that may have up to 9 characters (e.g. DL1, or DeadL1, etc.)

It is desirable to specify a descriptive Load Name that is easily recognized when later specifying the Load Combinations for the structure.

When later specifying the Load Combinations for the structure, a common Load factor is applied to all individual Temperature loads in each set of loads defined by a Load Name.

### **Add, Modify, Delete a Load Name**

#### **Add**

For the left-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing

Enter will cause SODA to attempt to add a new Load Name to the list box.

If there are blanks in the Load Name, or if no name is given, a warning message will appear noting what the problem is.

### **Modify**

For the left-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name to be modified has not been changed, then this action will have no effect.

If there are blanks in the new Load Name, or if no name is given, a warning message will appear noting what the problem is.

### **Delete**

For the left-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name is part of a previously specified Load Combination, SODA will not allow the deletion to occur until the Load Combination has been modified to exclude the Load Name.

### **Member**

The Member name identifies a particular member of the structure for which a Temperature Change occurs.

### **Temperature Change**

Temperature Change = the temperature increment in degrees that is applied to the member.

NOTE: Depending on the specified Default Units, the Temperature Change is specified in units of DEGREES FAHRENHEIT (Imperial) or DEGREES CELSIUS (Metric).

### **Thermal Expansion Coefficient**

The default value for the Thermal Expansion Coefficient depends on whether or not the Default Units are Imperial or Metric:

UNITS	THERMAL EXPANSION COEFFICIENT
Imperial	0.650x10 <sup>-5</sup> per degree Fahrenheit
Metric	0.117x10 <sup>-4</sup> per degree Celsius

### **Add, Modify and Delete Temperature Data**

#### **Add Temperature Load Data**

For the right-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new MEMBER with its associated Temperature Load to the list box.

NOTE: The loaded Members in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

If any of the data entered for the new Member is incorrect, a warning message will appear noting what the problem is.



## **Modify Temperature Load Data**

For the right-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Temperature Load associated with the Member that is currently highlighted in the list box.

NOTE: The loaded Members in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

If any of the data entered for the new Member is incorrect, a warning message will appear noting what the problem is.

## **Delete Temperature Load Data**

For the right-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Temperature Load associated with the Member that is currently highlighted in the list box.

NOTE: The loaded Members in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

## **Batch Loads**

The Batch Load feature allows the user to quickly specify an entire range of members to load. Prior to clicking the Batch button the user must enter the member load information. Upon clicking the Batch button the following options are presented:

### **Load Surface**

Select the surface to be loaded. Plan Y is the plan view as seen along the global Y axis; similarly for Plan X and Plan Z. Selecting Partial will allow the specification of a range of floor or wall surfaces for loading.

### **Excluded Members**

Allows the user to specify that the exterior or interior X and Z-axis beams and Y-axis columns lying in the selected Plan X, Y or Z surface(s) are to be excluded from being loaded. Similarly for nodes lying in the selected Plan X, Y or Z surface(s). This only works for 3D structures using the member naming convention of the Regular Framework feature.

### **Surface Ranges**

Specify the range of floors or column lines to be loaded. The default is all floors or column lines. If Partial has been selected, the loaded portion of the surface should also be entered.

## **Adding Temperature Loads Graphically**

With the Temperature Loads window open, first add the Load Name and Temperature Change desired and then select a member to load by clicking the mouse right button while pointing at the desired member in the Structure Topology window; the selected member will be added in the Members List box of the Temperature Loads window. Continue to point and click until all desired temperature loads are added to the Members List box.

### **Named**

When the Named option is selected the Named Members part of the dialog box becomes available to allow the user to specify a range of loaded members that have been named differently than the default naming convention of the Regular Framework feature. Note: with the exception of the Named option, the Batch feature may only be used for structures that were previously created using Regular Framework.

### **Group Name**

When the Group Name option is selected the Group Name field of the dialog box becomes available. Click on the down arrow to see the list of all member groups. Select the name of the group for which all its members are to be loaded with the batch loads.

## Temperature Loaded Member List Box

The right-hand list box displays all the loaded members that have currently been entered into SODA for the highlighted Load Name in the left-hand list box. The user can move up and down the list by using the scroll bar on the right hand side of the list box. Above the list box is a pick list with all the member names for the structure.

### View

Clicking the VIEW button or double clicking on a member name will cause SODA to display the Temperature Load graphic for the loaded member that is currently highlighted. The Load Name, Member Name, the temperature magnitude, and member-end nodes, are displayed on the Load graphic. The Temperature Load graphic depicts an increase in temperature as red in colour and a decrease in temperature as negative and blue in colour. The coefficient of thermal expansion is also shown in the graphic.

## Support Settlements

### Load Name

Each Load Name defines a set of individual Support Settlement loads that are applied to one or more nodes of the structure at their service load level.

Each Load Name is a unique alphanumeric string that may have up to 9 characters (e.g. DL1, or DeadL1, etc.)

It is desirable to specify a descriptive Load Name that is easily recognized when later specifying the Load Combinations for the structure.

When later specifying the Load Combinations for the structure, a common Load factor is applied to all individual Nodal, Member or Temperature loads in each set of loads defined by a Load Name.

### Add, Modify, Delete a Load Name

#### Add

For the left-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Load Name to the list box.

If there are blanks in the Load Name, or if no name is given, a warning message will appear noting what the problem is.

#### Modify

For the left-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name to be modified has not been changed, then this action will have no effect.

If there are blanks in the new Load Name, or if no name is given, a warning message will appear noting what the problem is.

#### Delete

For the left-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Load Name which is currently highlighted in the list box.

NOTE: If the Load Name is part of a previously specified Load Combination, SODA will not allow the deletion to occur until the Load Combination has been modified to exclude the Load Name.

### View

Clicking the VIEW button will cause SODA to display the Support Settlement Load graphic for the support node which is currently highlighted.

The Load Name, Node, the settlement magnitude, and the nodal co-ordinates are displayed on the Load graphic.

Single and double-headed arrows are used to indicate Translation and Rotation, respectively, in the Support Settlement graphic. The sign convention follows the Right Hand Screw rule.

## **Node**

The Node name identifies a particular support node of the structure at which (Displacement and/or Rotation) Settlement is imposed.

## **Support Settlements**

### **DISP-X**

A positive or negative valued Displacement that is imposed at the support node in the positive or negative global X-axis direction.

### **ROTN-X**

A positive or negative valued Rotation that is imposed at the support node about the global X-axis in a positive or negative sense in accordance with the Right-hand Screw Rule.

### **DISP-Y**

A positive or negative valued Displacement that is imposed at the support node in the positive or negative global Y-axis direction.

### **ROTN-Y**

A positive or negative valued Rotation that is imposed at the support node about the global X-axis direction in a positive or negative sense in accordance with the Right-hand Screw Rule.

### **DISP-Z**

A positive or negative valued Displacement that is imposed at the support node in the positive or negative global Z-axis direction.

### **ROTN-Z**

A positive or negative valued Rotation that is imposed at the support node about the global Z-axis direction in a positive or negative sense in accordance with the Right-hand Screw Rule.

## **Adding Support Settlements Graphically**

With the Support Settlements window open, first add the Load Name and Settlements desired and then select a support node to settle by clicking the mouse right button while pointing at the desired support node in the Structure Topology window; the selected support node will be added in the Nodes List box of the Support Settlements window. Continue to point and click until all desired support settlements are added to the Nodes List box.

## **Add, Modify, Delete Settlements**

### **Add Support Settlement Data**

For the right-hand list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new support Node with its associated Support Settlements to the list box.

NOTE: The settled support Nodes in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

If any of the data entered for the new support Node is incorrect, a warning message will appear noting what

the problem is.

### **Modify Support Settlement Data**

For the right-hand list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the settlements associated with the Support Node that is currently highlighted in the list box.

NOTE: The settled support Nodes in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

If any of the data entered for the new support Node is incorrect, a warning message will appear noting what the problem is.

### **Delete Support Settlement Data**

For the right-hand list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Settlements associated with the Support Node that is currently highlighted in the list box.

NOTE: The settled support Nodes in the right-hand list box are associated with the highlighted Load Name in the left-hand list box.

### **Settled Support Node List Box**

The right-hand list box displays all the support nodes that have currently been entered into SODA for the highlighted Load Name in the left-hand list box. The user can move up and down the list by using the scroll bar on the right hand side of the list box. Above the list box is a pick list with all the node names for the structure.

## **Area Loads**

The Area Loads dialog box allows the quick specification of a load to be applied on an area. This load is then automatically converted by SODA to a member load of appropriate magnitude and direction. Area loads are only available for 3D structures created with the Regular Framework feature.

### **Load Surface**

Select the surface to be loaded. Plan Y is the plan view as seen along the global Y axis; similarly for Plan X and Plan Z. Selecting Partial will allow the specification of a range of floor or wall surfaces for loading.

### **Surface Ranges**

Specify the range of floors or column lines to be loaded. The default is all floors or column lines. If Partial has been selected, the loaded portion of the surface should also be entered.

### **Distributed To**

Select the direction of the beams the Area Load is to be distributed to. For Plan Y, the available selections are X-beams and Z-beams, for Plan X the selections are Y-columns and Z-beams, and for Plan Z the selections are Y-columns and X-beams.

### **Specified Load**

Specify the Load Name associated with the Area Load. Clicking the arrow at the right end of the field will pop up a list of existing specified loads, or the name of a new specified load may be entered. If the Load Name specified does not yet exist, SODA will ask for confirmation that the new Load Name is to be created.

### **Load Intensity**

This edit field specifies the Intensity of the area Load. The Intensity (specified either in psf or kPa) is then converted to the appropriate member load magnitude by SODA.

### **Add, Replace, Cancel**

## **Add**

Pressing this button will add the resulting member loads to the specified load. If any loaded members were already loaded, the two loads will be added together (if appropriate) to give a new combined load magnitude.

## **Replace**

Pressing this button will replace the specified load with the resulting member loads. If the specified load was non-empty, SODA will request confirmation prior to deleting the existing member loads.

## **Cancel**

Pressing this button will exit the dialog box while ignoring the current contents of the Area Loads edit fields.

## **Load Combinations**

SODA will only analyze and design after Load Combinations have been specified.

The Load Combinations worksheet allows the user to define combinations of loads for the structure. Each Load Combination is some combination of the previously defined Load Names, each of which refers to a set of service-level Nodal, Member, Temperature or Settlement loads.

A Load Factor is specified for each Load Name within each Load Combination. A zero Load Factor excludes a Load Name from a Load Combination.

### **Load Combination Name**

Each specified Load Combination is identified by a unique alphanumeric name that may have up to 59 characters (e.g., "LOAD COMBINATION 2", or "1.5\*DEAD1 + 1.0\*SNOW3 + 1.7\*LIVE2", etc.)

NOTE: A Load Combination is the summed combination of one or more individual Load Names that are each multiplied by a Load factor.

### **Load Combination List Box**

This displays all the Load Combinations that have currently been entered into SODA. Although none of the displayed items are directly editable, clicking an item in the list box will place its contents in the editable field above the list box. The user can move up and down the list by using the scroll bar on the right hand side of the list box.

## **Add, Insert, Modify, Delete Load Combination**

### **Add Load Combination**

For the top list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Load Combination to the list box.

### **Insert Load Combination**

For the top list box, clicking the Insert button with the mouse or tabbing to the Insert button and pressing Enter will cause SODA to attempt to add a new Load Combination to the list box preceding of the Load Combination which is currently highlighted. (This differs from the Add button in that the Add button always appends the Load Combination information.)

### **Modify Load Combination**

For the top list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load Combination name currently highlighted in the list box.

### **Delete Load Combination**

For the top list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Load Combination currently highlighted in the list box.

## **Load Name List Box**

Lists all specified loads by Name, Type and Load Factor. The Name and Type fields will not change but the Load Factor may vary for each Load Combination.

**Load Name**

Names of all previously specified sets of service-level Nodal, Member, Temperature and Settlement Loads (e.g., Dead1, FullSnow).

**Load Type**

Identifies the nature of the load (Nodal, Member, Temperature, Settlement)

**Load Factor**

A non-zero Load Factor indicates that the specified load is part of the currently selected Load Combination.

**Load Factor**

A non-zero Load Factor is specified for each Load Name that is part of the Load Combination.

NOTE: The Load Factor = 0.0 for all Load names that are not part of the Load Combination

NOTE: Specify the Load factor as a Negative value if the load is to participate in the Load Combination in a sense opposite to that originally specified.

**Modify Load Factor**

For the bottom list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Load factor of the Load Name that is currently highlighted in the list box.

## Displacement Limits

**Load Combination**

Select Load Combination(s) for which Displacement Limits are to be imposed at one or more Nodes of the structure.

NOTE: Typically, Displacement Limits need be specified for only a select few of the Load Combinations (e.g., for those that cause maximum story drift and/or maximum beam deflection, etc.)

**Constrained Node**

Select name of node(s) at which the magnitude of displacement and/or rotation is to be constrained to be no greater than a specified upper-bound value.

NOTE: Typically, Displacement Limits need be specified at only a few nodes for each Load Combination (e.g., maximum story drift in the global X-axis direction is controlled by constraining the X-axis displacement at the top-end node of one column line for each story of the structure; or, the maximum mid-span deflection of all beam members belonging to the same Group that are similarly loaded is controlled by constraining the Y-axis displacement at a node located at mid-span of only one of the beam members).

## Displacement Limits

**X-TRAN**

The maximum Translation allowed at the node in the global X-axis direction.

**X-ROTN**

The maximum Rotation allowed at the node about the global X-axis direction.

**Y-TRAN**

The maximum Translation allowed at the node in the global Y-axis direction.

**Y-ROTN**

The maximum Rotation allowed at the node about the global Y-axis direction.

**Z-TRAN**

The maximum Translation allowed at the node in the global Z-axis direction.

**Z-ROTN**

The maximum Rotation allowed at the node about the global Z-axis direction.

**Notes**

Each Displacement Limit is specified as a Positive value that is uniformly applied by SODA to constrain both senses of possible action (e.g., left or right, up or down, clockwise or counter-clockwise).

Each Displacement Limit is specified as a Factored multiple of its service-level value, consistent with the design Load Factor(s) on the service loads (e.g., if the X-axis translation at a node is to be less than 0.75 in. under service loads, and the design load factor on the service loads is 1.5, then specify X-TRAN =  $0.75 \times 1.5 = 1.125$  in.).

A rotation limit cannot be specified for a pinned node (it will have no effect on structure response).

A displacement or rotation limit cannot be specified as zero(0) value, and should not be specified as a restrictively small value (an infeasible design situation will likely result).

## **Add, Modify, Delete Constrained Node**

### **Add Constrained Node**

For the bottom list box, clicking the Add button with the mouse or tabbing to the Add button and pressing Enter will cause SODA to attempt to add a new Node with its associated Displacement Limits to the list box.

NOTE: The Nodes in the bottom list box are associated with the highlighted Load Combination in the top list box.

If any of the data entered for the new Node is incorrect, a warning message will appear noting what the problem is.

### **Modify Constrained Node**

For the bottom list box, clicking the Modify button with the mouse or tabbing to the Modify button and pressing Enter will cause SODA to attempt to modify the Displacement Limits associated with the Node that is currently highlighted in the list box.

NOTE: The Nodes in the bottom list box are associated with the highlighted Load Combination in the top list box.

### **Delete Constrained Node**

For the bottom list box, clicking the Delete button with the mouse or tabbing to the Delete button and pressing Enter will cause SODA to attempt to delete the Displacement Limits associated with the Node that is currently highlighted in the list box.

NOTE: The Nodes in the bottom list box are associated with the highlighted Load Combination in the top list box.

## **Constrained Node List Box**

This displays all the nodes for which Displacement Limits have been specified. The user can move up and down the list by using the scroll bar on the right hand side of the list box.

# SODA Run Commands

(SODA will only analyze and design once Load Combinations have been entered)

The following describes items under the Run option of the SODA Menu Bar:

## Save Data

From the Menu Bar, select **/Run/** and place a check beside **Save Data before Running Engine**, and the input data will be automatically saved to the \*.POP file before the *sodaengine* is executed.

## Review Input

Select this option to generate and display a file containing an echo of the input data. The input echo provides a means of viewing all data in a tabular format. SODA automatically generates a new input echo file prior to running the engine so as to provide a record of the input that generated the current output.

## Run Engine

SODA will only analyze and design after Load Combinations have been specified.

## Displacement Constrained

The Displacement Constrained field allows the user to specify whether or not Displacement Limits are applied for the current run of the SODA Engine.

NOTE: This runtime option allows the user to readily run SODA both with and without Displacement Limits in order to determine their influence on the Design or Verify results for a structure.

## Constraint Factor

A design constraint is a design code requirement of the form  $\text{Actual} < \text{Allowable}$ .

This may also be expressed in the form  $\text{Actual}/\text{Allowable} < \text{Limit}$

The Constraint Factor provides a means to Relax or Tighten the Limit of all the Design Constraints for the structure in one convenient operation. The SODA engine output lists this factor as the Allowable Response Ratio.

Relax: specify a Constraint Factor  $> 1.0$

Tighten: specify a Constraint Factor  $< 1.0$

For example, suppose the design of a structure has been completed using SODA, and it is subsequently required to redesign the structure for loads that are 10% greater in magnitude. This objective is achieved by specifying the Constraint Factor = 0.9 so as to tighten all design constraints by 10%, and then running SODA directly (i.e., without having to change the load data in any way).

Or, as another example, suppose it is desired to relax the constraints for a design by 5%. This objective is directly achieved by specifying the Constraint Factor = 1.05.

NOTE: The Constraint Factor is a Global factor that is uniformly applied to all existing Strength and Displacement design constraints for the structure (e.g., if Displacement Limits exist for the latter example above, all of the specified upper-bound displacement and/or rotation values are increased by 5%).

## Output Format

This runtime option allows the user to specify the output format for the numerical data to be either exponent-free or exponential.

NOTE: Output units are restricted to those commonly used in practice and are:

Imperial

Metric



```
=====
```

Force	kips	kN
Moments	kip-ft	kN-m
Length	ft	m
Displacement	in	mm

## Report Detail

This runtime option allows the user to specify the level of detail needed in the output. Normal, Medium and Detailed are the three options available for Verify and Design tasks.

### Normal

Minimum output.

Recommended during preliminary design.

Lists the design codes responses of the critical member in each design group.

### Medium

Lists the design code responses for all members.

Recommended during secondary design stage when user wants to redefine member grouping so as to have more members in each group perform at a higher level of efficiency.

### Detailed

Maximum output.

Lists design code equations and values for all members.

Recommended during the final stage of design to provide a design record.

## Optimization

This runtime option allows the user to turn the SODA Optimization capability On or Off for a Design run.

When the On option is selected, the SODA Optimization capability is applied for the Design process to determine a least-weight structure.

When the Off option is selected, the SODA Optimization capability is de-activated and the Design of the structure is found using only a member-by-member selection routine.

NOTE: Both options produce feasible designs, with that produced by the Optimization On option having the least weight.

NOTE: The Optimization Off option requires the least amount of computer time to find the Design of the structure.

## Behavior Type

The First-Order and P-Delta behaviour types are as previously defined in the Definition worksheet under the General Menu.

This runtime option allows the user to change the underlying behaviour of the structure for analysis (i.e., First-Order or P-Delta) without changing any associated data.

NOTE: Refer to the description of the Design Process under RUN SODA where it is noted that this runtime option for Behaviour Type can be used to sometimes decrease overall computational effort for a Design run.

## Bending Coefficients

### Calculate

This option specifies that the value of the Bending Coefficients ( $C_m$  and  $C_b$  for AISC codes;  $\omega$  for Canadian codes) is to be calculated by SODA according to the moment gradient, presence of span loads, side-sway

permitted/prevented, and type of analysis (First-Order or P-Delta).

### **Unity**

This option conservatively sets the value of the Bending Coefficients to one (1.0).

## **Design Process**

### **Continuous**

This option specifies that the iterative Design Process is to be conducted continuously without user intervention at any design stage until the final design of the structure is found.

### **Stage-by-Stage**

This option specifies that the iterative Design Process is to be interrupted after each design stage to allow the user to specify different runtime options if required.

### **Runtime Options**

When the Stage-by-Stage option is selected, all of the runtime options noted for the RUN SODA worksheet become available to the user after completion of each design stage (i.e., during the actual execution of the SODA run). For example, with a view to decreasing the overall computational effort for a Design run, First-Order behaviour may be adopted for the majority of design stages, with P-Delta behaviour being adopted only for the final design stages.

### **Review Input**

Select this option to generate and display a file containing an echo of the input data. The input echo provides a means of viewing all data in a tabular format. SODA automatically generates a new input echo file prior to running the engine so as to provide a record of the input that generated the current output.

### **Save Data**

From the Menu Bar, select **/Run/** and place a check beside **Save Data before Running Engine**, and the input data will be automatically saved to the **\*.POP** file before the *sodaengine* is executed

## SODA Output Commands

The following describes items under the Output option of the SODA Menu Bar:

### Input echo

Displays all input data in a tabular format.

### Analysis results

Displays forces, displacements and reactions for each load combination. As an aid to connection design, the maximum forces at each member end are listed at the end of this file.

### Final design/verify

Only available for Verify or Design. Lists the cross-section designations for the final design of the structure, along with the total length, volume and weight of member required for each fabrication group. Also listed is the performance of the displacement constraints(if any), the total weight of the structure, and the governing code clause, fabrication group, member and load combination.

### Summary results

Only available for Verify or Design. Lists the governing load combination and code clause for each fabrication group, the support reactions for each load combination, and the maximum member-end forces from among all load combinations.

### Normal report

Lists analysis results for each load combination. For Design or Verify, critical member performance is assessed clause-by-clause for the prevailing Steel Code, and the governing load combination is identified for each fabrication group. For Design, the cross-section designations for the final design of the structure are given, along with the total length, volume and weight of member required for each fabrication group, and the total weight of the structure.

### Medium report

Lists all of the Normal report detail. In addition for Design or Verify, the performance of every member is assessed clause-by-clause for the prevailing Steel Code, and the governing load case for each member is identified.

### Detailed report

Lists all of the Medium report detail. In addition, a great deal of secondary data is provided to enable full tracing and verification of the SODA computer run. (WARNING: the amount of Detailed output is extensive any may make considerable demands on available disk storage space).

### Effective lengths

Lists calculated and specified  $K_x$ ,  $K_y$ ,  $K_xL$  and  $K_yL$  values for all members.

### Design history

Lists the design sections found for each design stage as well as the maximum response ratio for each design stage.

### Member Diagrams

This dialog box allows the examination of moment, shear and deformation effects of individual load cases on individual members. Moment, shear and deformation diagrams are displayed along with maximum values for each, plus the axial force for the given member. The deformation diagram is superimposed on the moment diagram as a dashed line.

The member forces and deformations are relative to the local x-y-z axes. Regardless of how a member is oriented in the structure it is displayed with the i-node (start node) at the left of the diagram and the j-node (end node) at the right. (This can lead to some confusion when viewing weak-axis diagrams or column member diagrams).

## **Member**

Selects the member whose diagrams are to be displayed.

## **Load Combination**

Selects the load combination for which the member diagrams are to be displayed.

## **Bending Axis**

Selects either the strong-axis or weak-axis for bending of the member for the moment, shear and deformation displays. This choice is not available for 2D structures

## **Maximums**

The maximum positive and negative moment (M), shear (V) and deformation (D) values are shown, along with the positions at which these maximums occur. The total length (L) of the member, from start to end node, and the axial force(P) are also shown.

## **Printing Diagrams**

Select Print to print the moment, shear and deformation diagrams for the member along with the corresponding maximum values.

## **Warnings**

Lists any SODA generated warnings that occur during a run of the SODA engine.

Typical warnings concern uplift and excessive deflection.

## **Errors**

Lists any SODA generated non-fatal errors that occur during a run of the SODA engine.

# SODA Structural Topology Menu Commands

## SODA View Commands

The following describes items under the View option of the Structure Topology Menu Bar:

### View Structure

This selection displays the structure on the screen in accordance with the options selected under the Graphics item of the Structure Topology menu bar.

### Rotating the View

The view of the structure may be rotated in adjustable degree increments about the Global Y and Z axes by pressing the Left/Right and Up/Down arrow keys on the keyboard. The rotation step can be adjusted in the Graphics View Options dialog.

Alternatively, the Graphics View Options allow you to directly specify viewing angles.

### Graphic Representation of Supports

Fixed and Pinned supports use typical symbols.

A Roller support is represented by two black circles, one atop the other.

Other supports are represented by a hollow circle with an X through it.

### View Deflections

View the deflected shape of the structure for any Load Combination. A scale factor makes it possible to increase or decrease the magnitudes of the deflections.

Only nodal X, Y and Z translations are displayed. Thus, intermediate displacements along a member are not shown

### Load Combinations

Contains a list of all Load Combinations for the structure. Select a Load Combination to display the corresponding deflected shape of the structure.

### Scale Factor

The scale factor serves to magnify the deflection profile of the structure so as to make the deflections easier to view. The default scale factor is 100.

### Cursor Keys

The cursor (arrow) keys are the quick way to rotate the structure about the global X and Y axis. Each time you press a left or right cursor key the structure rotates 15 degrees about the Y-axis. Similarly for the up and down cursor keys and the X-axis.

### View Zoom In

#### Double-Clicking

Selecting this item has the same effect as double-clicking the Left mouse button; it causes the Zoom factor to double.

#### Click and Drag

To zoom in on a specific area:

- imagine a box around the area of interest
- place the mouse pointer at one corner of this imaginary box
- click and hold down the left mouse button
- a box will appear as you drag the mouse diagonally across the area to another corner
- release the mouse button and the graphics screen will zoom to the contents of the box

Use the Zoom Reset option of the View menu to restore the full view of the structure.

## View Zoom Out

### Double-Clicking

Selecting this item has the same effect as double-clicking the Right mouse button; it causes the Zoom factor to halve.

### View Zoom Reset

Select this option to set the Zoom to 100%.

## SODA Graphics Commands

The following describes items under the Options selection of the Structure Topology Menu Bar:

### Show Member Fixity

Member-end fixity is displayed in the Structure Topology view when this corresponding menu item is checked on the Graphics Menu.

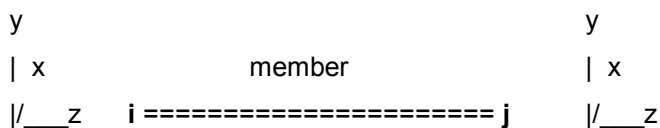
### Graphics Axes Menu Items

From the Menu Bar of the Structure Topology window, select **/Graphics/**.

A check beside *Show Structure Axes* causes the global axes to be shown in the Structure Topology window.

A check beside *Show Structure Orientation Axes* causes an icon of the global axes to be shown in the Structure Topology window.

A check beside *Show Local Member Axes* causes the local axes to be shown on each member in the Structure Topology window. (Local member axes colour coding: y and z are red and x is grey).



### Graphics View Options

This option selects the viewing mode for the graphics display.

#### Isometric View

The Isometric View corresponds to a perspective, three-dimensional representation of the structure. It allows the selection of the horizontal and vertical viewing angles.

#### Projection View

The Projection View allows the selection of the horizontal and vertical viewing angles and always has a

vertical Y axis and a horizontal X axis.

## **Plan X, Plan Y and Plan Z**

Select one of the X, Y or Z plan views to show the structure as viewed along the X, Y or Z-axis, respectively. This is a shortcut method for setting the Horizontal and Vertical fields to zero or multiples of 90 degrees.

## **Horizontal Angle**

This value specifies the Horizontal Angle at which the structure is viewed. Increasing this angle (which can also be done using the right arrow key while in the graphics display) will rotate the structure about the Y-axis in a counter- clockwise direction. Decreasing the angle (left arrow key) will rotate the structure in a clockwise direction.

## **Vertical Angle**

This value specifies the Vertical Angle at which the structure is viewed. Increasing this angle (which can also be done using the up arrow key while in the graphics display) will rotate the structure about the X-axis in a counter- clockwise direction. Decreasing the angle (down arrow key) will rotate the structure in a clockwise direction.

## **Rotation Step**

This editable field allows the user to set the angle that the structure is rotated with at every arrow keystroke. It is useful when you need to view certain elements of the structure that would normally be covered by other elements.

## **Show large nodes**

This allows for a larger symbol to be displayed for the structure's nodes, in case the support type needs to be more visible.

## **Thick highlighted elements**

This option when checked causes the currently selected members or nodes to be drawn using a larger line width than the non-selected elements. This is useful for large structures where the selected element is not easily visible in the view.

## **Save as default for new files**

This option when activated will save the currently selected options as default settings for new SODA projects and future sessions of SODA.

## **Filters**

Filters for specified ranges on the X, Y, Z-axes or by structural group are initiated when the corresponding check box is filled on the Graphics / View Options Menu in the Structural Topology window Graphics Text Options: Options for setting text on diagrams

## **Graphics Text Options**

This dialog allows the user to control the font type and size that is used to display text when the View Topology or Print Topology option is selected. The user can also control what text appears, as listed below.

### **Font**

Select a font to be used for displaying text on the graphics screen. This is done in the same way as selecting a font for a word processor. The printout of the graphics screen will also use this font.

### **Size**

Select a font size to be used for displaying text on the graphics screen and printout. This is done in the same way as selecting a font for a word processor.

### **Node Names**

This option turns on or off the displaying of the Node Names on the graphics display and printout.

### **Member Names**

This option turns on or off the displaying of the Member Names on the graphics display and printout.

### **Group Names**

This option turns on or off the displaying of the Group Names on the graphics display and printout.

### **Group X-Sections**

This option turns on or off the displaying of the Group X-Sections (shape+designation) on the graphic display and printout.

### **Align Text with Member**

This option only works with True Type fonts.

When selected, this option aligns member names along the axis of the member they refer to. This feature is optional because it may slow down screen refresh on some computers.

### **Save as default for new files**

This option when activated will save the currently selected options as default settings for new SODA projects and future sessions of SODA.

## **SODA Mode Commands**

### **Select Mode**

In order to select nodes, groups, members or graphically apply loads in the Structure Topology window the Select Mode must be checked in the Mode menu in the Structure Topology window.

Starting with version 4.0, double-click Zoom in and out and Pan capabilities are available in Select Mode too, in order to be able to perform graphical selection of elements in a zoomed view.

### **Zoom Mode**

In this mode the view can be zoomed in and out as needed.

The zoom-in can be performed by a double-click of the left button of the mouse in the desired location in the Structure Topology view, or by defining the zoom region via a click-and-drag mouse action.

The zoom-out action is performed by double-clicking the right button of the mouse in the zoomed-in view.

Alternatively, the *Zoom In* and *Zoom Out* commands in the *View* menu can be used.

Starting with version 4.0, double-click Zoom in and out and Pan capabilities are available in Select Mode too, in order to be able to perform graphical selection of elements in a zoomed view.

In order to zoom in to a rectangle created with a click-and-drag mouse action, the Zoom Mode must be checked in the Mode menu in the Structure Topology window.

## **SODA Help Commands**

The following describes items found under the Help option of both the SODA and Structure Topology Menu Bars:



Context Sensitive: Finding help information for any field of the SODA interface

Contents: Finding general help information

How to Use Help: Using the SODA help facilities

About SODA: SODA copyright and development information

The following describes items found under the Help option of the SODA main Menu Bar only:

User Manual: Opens the on-line user manual in Acrobat format

SODA Website: Takes you to the SODA Home page (<http://www.acronym.ca/sodahome.html>)

E-mail SODA Support: Launches an e-mail session to SODA support ([soda@acronym.ca](mailto:soda@acronym.ca))

NOTE: Each of the underlined items above is accessed from the Help menu.

# SODA Design Constraints

## Introduction

This Chapter details the strength and stiffness provisions that SODA explicitly accounts for when conducting a Design or Verification of a structure in conformance with each of the five Steel Design Codes:

AISC-ASD-89 : American Institute of Steel Construction," Specification for Structural Steel Buildings: Allowable Stress Design and Plastic Design", June 1, 1989

AISC-LRFD-86 : American Institute for Steel Construction," Load and Resistance Factor Design Specification for Structural Steel Buildings", September 1, 1986

AISC-LRFD-93 : American Institute for Steel Construction," Load and Resistance Factor Design Specification for Structural Steel Buildings", December 1, 1993

CSA-S16.1-M89 : Canadian Standards Association, "Limit States Design of Steel Structures", December, 1989

CSA-S16.1-94 : Canadian Standards Association, "Limit States Design of Steel Structures", December, 1994

It is **important** to note that SODA makes no engineering judgements of its own, and that it is solely the **user's responsibility** to ensure that the topology, loading and other details for the structure are correctly identified. In other words, the onus is on the user to correctly specify the design or verification problem to SODA. Moreover, it is also the responsibility of the user to confirm that the results produced by SODA are correct.

## Strength and Displacement Constraints

The strength constraints imposed by SODA refer to the strength/stability provisions of the specified steel design code for the proportioning of members to ensure that the structure has sufficient capacity to withstand the applied loading.

The displacement constraints imposed by SODA refer to the upper-bound values for nodal displacements that are specified by the user (in the SODA Interface) to ensure that the structure has sufficient stiffness for the applied loading.

The specific provisions underlying the strength and displacement constraints imposed by SODA are listed clause-by-clause in this On-Line Manual for each of the five steel design codes supported by SODA (see Related Topics below). For reasons of space, the number and title of each code clause is alone given (the user is referred to the design code itself for the written descriptions of the provisions). Any exceptions or specific interpretations made by SODA for a code provision are noted. Any code clause that is not listed is not considered by SODA.

# AISC-ASD-89 Steel Design Code

(Allowable Stress Design and Plastic Design)

The provisions of the AISC-89 steel design code that are explicitly accounted for by SODA are listed clause-by-clause in the following.

## Chapter A. General Provisions

Clause A2: Limits of Applicability

A2.2 - Types of Construction

- Type 3 construction is not accounted for.
- For Type 1 construction, design based on plastic analysis is not accounted for.
- For Type 1 and 2 construction, the user is responsible for ensuring proper connections.

Clause A3: Material

A3.1 - Structural Steel

- The user is responsible for acceptable material and product standards and specifications.

Clause A4: Loads and Forces

A4.1 - Dead Load and Live Load

- The user is responsible for the specification of dead loads.
- The user is responsible for the specification of live loads, including any snow loads.
- The user is responsible for the specification of any probable arrangement of loads resulting in the highest stresses.

A4.4 - Wind

- The user is responsible for the specification of wind loads.

Clause A5: Design Basis

A5.1 - Allowable Stresses

- Design by plastic analysis is not accounted for.
- Moment reductions at rigid connections are not accounted for.

A5.2 - Wind and Seismic Stresses

- The 1/3 increase in allowable stresses is not accounted for (the capability exists, however, to reduce loads by 25% or to allow a 33% 'overstress').

A5.4 - Design for Serviceability and Other Considerations.

- Deflection serviceability is alone accounted for, for which the user must specify the appropriate upper-bound deflection values.
- Serviceability of connections is the responsibility of the user.

## Chapter B. Design Requirements

Clause B2: Net Area

- For the net area in tension, the user is responsible for specifying a bolt hole value that incorporates the effect of a chain of holes extending across a part in any diagonal or zigzag line.

Clause B3: Effective Net Area

- The user is responsible for the specification of the reduction coefficient  $U (= A_e/A_n)$ .

Clause B4: Stability

## Clause B5: Local Buckling

### B5.1 - Classification of Steel Sections.

- Plates are not accounted for.
- Built-up sections are not accounted for.
- For rectangular hollow structural sections, the inside corner radius is not accounted for.
- Tapered flanges of rolled sections are not accounted for.

### B5.2 - Slender Compression Elements.

- Plate girders are not accounted for.

## Clause B7: Limiting Slenderness Ratios

- Rods in tension are not accounted for.

## **Chapter C. Frames and Other Structures**

### Clause C1: General

- The user is responsible to ensure overall stability.

## **Chapter D. Tension Members**

### Clause D1: Allowable Stress

- Block shear strength is not checked.

## **Chapter E. Columns and Other Compression Members**

### Clause E1: Effective Length and Slenderness Ratio

### Clause E2: Allowable Stress

## **Chapter F. Beams and Other Flexural Members**

### Clause F1:

- Hybrid or built-up beams and members are not accounted for.
- Moment reduction, where members are continuous over supports or rigidly framed to columns, is not accounted for.
- It is the responsibility of the user to specify  $C_b = 1$  for cantilever beams.

### Clause F2

### Clause F3

### Clause F4

- $K_v = 5.34$

## **Chapter G. Plate Girders**

### Clause G1: Web Slenderness Limitation

- The effect of transverse stiffeners is not accounted for.

### Clause G2: Allowable Bending Stress

- $R_e = 1.0$

## **Chapter H. Combined Stresses**

### Clause H1: Axial Compression and Bending

Clause H2: Axial Tension and Bending

## Appendix B. Design Requirements

Note the following exceptions:

Section B.5.2.b:  $v_f = v0.6 F_y Q_s$

Limiting proportions for tees are not checked

## AISC-LRFD-86 Steel Design Code

The provisions of the AISC-LRFD steel design code that are explicitly accounted for by SODA are listed clause-by-clause in the following.

### Chapter A. General Provisions

Clause A2: Limits of Applicability

A2.2 - Types of Construction

- For Type FR construction, the user is responsible for ensuring proper connections.
- Type PR construction accounts only for "simple framing", for which the user is responsible for ensuring proper connections.

Clause A3: Material

A3.1 - Structural Steel

- The user is responsible for acceptable material and product standards and specifications.

Clause A4: Loads and Load Combinations

A4.1 - Loads, Load Factors and Load Combinations.

- The user is responsible for the specification of dead loads, live loads, wind loads, snow loads and loads due to initial rainwater or ice.
- Earthquake loads and dynamic load effects in general are not accounted for.
- The user is responsible for the specification of critical combinations of factored loads.

Clause A5: Design Basis

A5.1 - Required Strength at Factored Loads.

- Design by plastic analysis is not accounted for.
- Moment reductions at rigid connections are not accounted for.

A5.2 - Limit States.

A5.3 - Design for Strength.

A5.4 - Design for Serviceability and Other Considerations.

- Deflection serviceability is alone accounted for, for which the user must specify the appropriate upper-bound deflection values.

### Chapter B. Design Requirements

Clause B2: Net Area

- For the net area in tension, the user is responsible for specifying a bolt hole value that incorporates the effect of a chain of holes extending across a part in any diagonal or zigzag line.
- The net area for shear is not accounted for.
- The user is responsible for specifying the actual bolt hole width.

#### Clause B3: Effective Net Area

- The user is responsible for the specification of the reduction coefficient  $U (= A_e./A_n)$ .

#### Clause B5: Local Buckling

##### B5.1 - Classification of Steel Sections.

- Built-up sections are not accounted for.
- For rectangular hollow structural sections, the flat width is taken as the total section width minus three times the wall thickness.
- Tapered flanges of rolled sections are not accounted for.

##### B5.3 - Slender Compression Elements.

- Plate girders are not accounted for.

#### Clause B7: Limiting Slenderness Ratios

- Rods in tension are not accounted for.

## **Chapter C. Frames and Other Structures**

#### Clause C1: Second Order Effects

#### Clause C2: Frame Stability

##### C2.1 - Braced Frames.

- SODA calculates  $K = 1$  if the user selects "SWAY PREVENTED" in the input.
- The user is responsible for the specification of the lay-out and characteristics of the vertical bracing system for the structure.

##### C2.2 - Unbraced Frames.

- SODA calculates  $K = 1$  if the user selects "SWAY PERMITTED" in the input.
- Plastic design is not accounted for.

## **Chapter D. Tension Members**

#### Clause D1: Design Tensile Strength

## **Chapter E. Columns and Other Compression Members**

#### Clause E1: Effective Length and Slenderness Limitations

##### E1.1 - Effective Length.

#### Clause E2: Design Compressive Strength

#### Clause E3: Flexural-Torsional Buckling

## **Chapter F. Beams and Other Flexural Members**

#### Clause F1: Design for Flexure

- Hybrid or built-up beams and girders are not accounted for.
- Clauses pertaining to welded shapes are not accounted for.

##### F1.5 - Tees and Double-angle Beams

- SODA conservatively assumes the stem is in compression. Thus  $B$  is a negative value in equation. F1-15

#### Clause F2: Design for Shear

##### F2.1 - Web Area Determination.

F2.2 - Design Shear Strength.

- The web plate buckling coefficient is taken as  $k=5$ .

## Chapter H. Members Under Torsion and Combined Forces

Clause H1: Symmetric Members Subject to Bending and Axial Force

H1.1 - Doubly and Singly Symmetric Members in Flexure and Tension.

H1.2 - Doubly and Singly Symmetric Members in Flexure and Compression.

H1.2.a - Determination of  $\mu$

- Design on the basis of plastic analysis is not accounted for.
- The term in Eq.(H1-2) involving the coefficient  $B_2$  is ignored.
- The value  $M_{nt}$  is the bending moment taken directly from the analysis results.
- In determining  $P_e$ ,  $K = 1$  is used unless the user has specified a value for  $K$  in the input.

H1.2.b - Determination of  $M_n$ .

## Chapter L. Serviceability Design Considerations

Clause L3: Deflections, Vibration and Drift

L3.1 - Deflections.

- The user is responsible for the specification of upper-bound deflection values under service loads.

L3.3 - Drift.

- The user is responsible for the specification of upper-bound lateral deflection values under static loads; seismic and dynamic loads in general are not accounted for.

## Appendix B. Design Requirements

Note the following exceptions:

B.5.3.b:  $\nu_f = \nu \phi_b F_y Q_s$ , (where  $\phi_b = 0.90$ )

Limiting proportions for Tees are not checked

## Appendix E. Columns and Other Compression Members

Note the following exceptions:

E3:  $K_z = \max(K_x L, K_y L)$

E3: Equation A-E3-7,  $F_e$  for unsymmetrical shapes is not accounted for.

## Appendix F. Beams and Other Flexural Members

Web Tapered Members are excluded

## AISC-LRFD-93 Steel Design Code

The provisions of the AISC-LRFD steel design code that are explicitly accounted for by SODA are listed clause-by-clause in the following.

## Chapter A. General Provisions

Clause A2: Limits of Applicability

A2.2 - Types of Construction

- For Type FR construction, the user is responsible for ensuring proper connections.
- Type PR construction accounts only for "simple framing", for which the user is responsible for ensuring proper connections.

#### Clause A3: Material

##### A3.1 - Structural Steel

- The user is responsible for acceptable material and product standards and specifications.

#### Clause A4: Loads and Load Combinations

##### A4.1 - Loads, Load Factors and Load Combinations.

- The user is responsible for the specification of dead loads, live loads, wind loads, snow loads and loads due to initial rainwater or ice.
- Earthquake loads and dynamic load effects in general are not accounted for.
- The user is responsible for the specification of critical combinations of factored loads.

#### Clause A5: Design Basis

##### A5.1 - Required Strength at Factored Loads.

- Design by plastic analysis is not accounted for.
- Moment reductions at rigid connections are not accounted for.

##### A5.2 - Limit States.

##### A5.3 - Design for Strength.

##### A5.4 - Design for Serviceability and Other Considerations.

- Deflection serviceability is alone accounted for, for which the user must specify the appropriate upper-bound deflection values.

## Chapter B. Design Requirements

#### Clause B2: Net Area

- For the net area in tension, the user is responsible for specifying a bolt hole value that incorporates the effect of a chain of holes extending across a part in any diagonal or zigzag line.
- The net area for shear is not accounted for.
- The user is responsible for specifying the actual bolt hole width.

#### Clause B3: Effective Net Area

- The user is responsible for the specification of the reduction coefficient  $U (= A_e./A_n)$ .

#### Clause B4: Stability

#### Clause B5: Local Buckling

##### B5.1 - Classification of Steel Sections.

- Built-up sections are not accounted for.
- For rectangular hollow structural sections, the flat width is taken as the total section width minus three times the wall thickness.
- Tapered flanges of rolled sections are not accounted for.

##### B5.3 - Slender Compression Elements.

- Plate girders are not accounted for.

#### Clause B7: Limiting Slenderness Ratios

## Chapter C. Frames and Other Structures



#### Clause C1: Second Order Effects

- SODA applies formula C1-1 as  $M_u = B_1 * M$  where M is the moment resulting from either a first or second-order analysis. In the latter case M will include  $P\Delta$  moments. The  $P\Delta$  analysis used by SODA accounts for additional bending moments produced by lateral movement of the structure. The  $B_1$  factor accounts for the additional bending moment produced by the column load acting through bending deformations in the member itself. In determining  $P_e$ ,  $K = 1$  is used unless the user has specified a value for K in the input.

#### Clause C2: Frame Stability

##### C2.1 - Braced Frames.

- SODA calculates  $K = 1$  if the user selects "SWAY PREVENTED" in the input.
- The user is responsible for the specification of the lay-out and characteristics of the vertical bracing system for the structure.

##### C2.2 - Unbraced Frames.

- SODA calculates  $K = 1$  if the user selects "SWAY PERMITTED" in the input.
- Stiffness reduction due to column inelasticity is not accounted for.
- Plastic design is not accounted for.

### Chapter D. Tension Members

#### Clause D1: Design Tensile Strength

### Chapter E. Columns and Other Compression Members

#### Clause E1: Effective Length and Slenderness Limitations

##### E1.1 - Effective Length.

#### Clause E2: Design Compressive Strength

#### Clause E3: Flexural-Torsional Buckling

### Chapter F. Beams and Other Flexural Members

#### Clause F1: Design for Flexure

- Hybrid or built-up beams and girders are not accounted for.
- Clauses pertaining to welded shapes are not accounted for.

##### F1.2c - Tees and Double Angles

- SODA conservatively assumes the stem is in compression. Thus B is a negative value in equation. F1-15

#### Clause F2: Design for Shear

##### F2.1 - Web Area Determination.

##### F2.2 - Design Shear Strength.

- The web plate buckling coefficient is taken as  $k=5$ .

### Chapter H. Members under Torsion and Combined Forces

#### Clause H1: Symmetric Members Subject to Bending and Axial Force

##### H1.1 - Doubly and Singly Symmetric Members in Flexure and Tension.

##### H1.2 - Doubly and Singly Symmetric Members in Flexure and Compression.

- Design on the basis of plastic analysis is not accounted for.

### Chapter L. Serviceability Design Considerations

### Clause L3: Deflections, Vibration and Drift

#### L3.1 - Deflections.

- The user is responsible for the specification of upper-bound deflection values under service loads.

#### L3.3 - Drift.

- The user is responsible for the specification of upper-bound lateral deflection values under static loads; seismic and dynamic loads in general are not accounted for.

# SODA Design Constraints (cont'd)

## AISC-LRFD-93 Steel Design Code (cont'd)

### Appendix B. Design Requirements

Note the following exceptions:

B.5.3.b:  $\phi_t = \phi \phi_b F_y Q_s$ , (where  $\phi_b = 0.90$ )

Section B.5.3.a(c) for built-up members is not incorporated.

### Appendix E. Columns and Other Compression Members

Note the following exceptions:

E3:  $K_z = \max(K_x L, K_y L)$

E3: Equation A-E3-7,  $F_e$  for unsymmetric shapes is not accounted for.

### Appendix F. Beams and Other Flexural Members

Web Tapered Members are excluded

## CSA-S16.1-M89 LSD of Steel Structures

The provisions of the CAN/CSA-S16.1-M89 steel design code that are explicitly accounted for by SODA are listed clause-by-clause in the following.

### 5. Material: Standards and Identification

Clause 5.1: Standards

- The user is responsible for acceptable material and product standards and specifications.
- 5.1.3 - Structural Steel.

### 6. Design Requirements

Clause 6.1: General

- The only ultimate limit state explicitly accounted for is "strength".
- The only serviceability limit state explicitly accounted for is "deflections"; the user is responsible for the specification of deflection limit states.
- The user is responsible for the specification of specified loads.
- The user is responsible for the specification of load factors.
- Camber, provisions for expansion and contraction, and corrosion protection are not accounted for.

Clause 6.2: Requirements Under Specified Loads

- 6.2.1 - Deflections.
- 6.2.1.1 - The user is responsible for the specification of acceptable deflection limits.

Clause 6.3: Requirements Under Factored Loads

- 6.3.1 - Strength
- The user is responsible for the application of the factored loads acting in the most critical combination

### 7. Loads and Safety Criterion

#### Clause 7.1: Specified Loads

- 7.1.1 - The user is responsible for the specification of dead loads and live loads and wind loads.
- Earthquake loads and dynamic load effects in general are not accounted for.
- The user is responsible for the specification of temperature changes and differential settlement.
- Shrinkage or creep of component materials is not accounted for.

#### Clause 7.2: Safety Criterion and Effect of Factored Loads

- 7.2.1 - The user is responsible for the specification of factored loads.
- Overturning, uplift, and stress reversal are not accounted for.
- 7.2.2 - The user is responsible for the factored load combinations.
- 7.2.3 - The user is responsible for the specification of the load factors.
- 7.2.4 - The user is responsible for the specification of the load combination factor.
- 7.2.5 - The user is responsible for the specification of the importance factor.

## **8. Analysis of Structure**

#### Clause 8.1: General

- 8.1.1 - Only deflection and strength requirements in Clause 6 are accounted for.
- 8.1.2

#### Clause 8.2: Continuous Construction

#### Clause 8.3: Simple Construction

- 8.3.1 - The user is responsible for ensuring a suitable system of bracing as resistance to lateral loads, including sway effects.

#### Clause 8.4: Elastic Analysis

#### Clause 8.6: Stability Effects

- 8.6.1 - Part (b) is not accounted for.
- 8.6.2 is not accounted for.

## **9. Design Lengths of Members**

#### Clause 9.1: Simple Span Flexural Members

- The effect of any significant moment or eccentricity arising from the manner in which a beam, girder, or truss may actually be connected or supported is not accounted for.

#### Clause 9.2: Continuous Span Flexural Members

#### Clause 9.3: Compression Members

- 9.3.1
- 9.3.2
- 9.3.3

## **10. Slenderness Ratios**

#### Clause 10.1: General

#### Clause 10.2: Maximum Slenderness Ratio

- 10.2.1
- 10.2.2

## 11. Width-Thickness Ratios: Compression Elements

Clause 11.1: Classification of Sections

- 11.1.1
- 11.1.2
- 11.1.3

Clause 11.2: Maximum Width-Thickness Ratios of Elements Subject to Compression

Clause 11.3: Width and Thickness

- 11.3.1 - Part (a) is not accounted for.
- 11.3.2 - Parts (a) and (c) are not accounted for.
- 11.3.3

## 12. Gross and Net Areas

Clause 12.1: Application

Clause 12.2: Gross Area

Clause 12.3: Net Area

- 12.3.1 - The calculation of net width and area of parts containing holes in accordance with Clause 12.3.3 is not accounted for.
- 12.3.2 - The user is responsible for ascertaining the width of bolt holes.
- 12.3.4 - The critical net area of the part is taken to be one of the limit values in (a), (b) or (c).

## 13. Member and Connection Resistance

Clause 13.1: General

- The user is responsible for the determination of factored loads in accordance with Clause 7.2.

Clause 13.2: Axial Tension

- Part (b) is not accounted for.

Clause 13.3: Axial Compression

- 13.3.1 - It is assumed that the capacity of singly symmetric double angle sections may be safely calculated by Clause 13.3.1.
- 13.3.2

Clause 13.4: Shear

- 13.4.1.1 - Elastic Analysis.
- Webs are considered to be un-stiffened, such that  $a/h = \text{infinity}$ ,  $\tau = 1$ ,  $\eta = 0$  and  $k_v = 5.34$ .
- 13.4.1.3 - Maximum Slenderness.

Clause 13.5: Bending - Laterally Supported Members

- Part (c)(i) is not accounted for.
- Part (c) (iii) Calculates  $M_r$  by the 'Alternative' method

Clause 13.6: Bending - Laterally Unsupported Members

- Part (c) and (d) are not accounted for.

Clause 13.8: Axial Compression and Bending

- The detailed methods of determining the resistance of columns subject to biaxial bending given in Appendix F are not considered.

- 13.8.1 - For Part (b) SODA will only set  $K = 1$  if the user has specified 'K = calc' in the input.
- 13.8.2 - For Part (b) SODA will only set  $K = 1$  if the user has specified 'K = calc' in the input.
- For a member divided into two (or more) segments, the user is responsible for ensuring the correct slenderness ratio  $KL/r$  for each segment for use in Clause 13.8; for example, in the SODA Interface, specify the effective length factor  $K=2.0$  for each of the two segments of a member that has been divided in half.
- 13.8.3 -
- 13.8.4 -Part (c) is not accounted for.

Clause 13.9: Axial Tension and Bending

- Part (b) is not accounted for.

## 15. Beams and Girders

Clause 15.4: Reduced Moment Resistance of Girders with Thin Webs

## 20. Stability of Structures and Individual Members

Clause 20.1: General

- 20.1.1 - The user is responsible for ensuring that a complete structural system is provided to transfer the factored loads to the foundations.
- The user is responsible for ensuring adequate resistance to torsional deformations.

## CSA-S16.1-94 LSD of Steel Structures

The provisions of the CAN/CSA-S16.1-94 steel design code that are explicitly accounted for by SODA are listed clause-by-clause in the following.

### 5. Material: Standards and Identification

Clause 5.1: Standards

- The user is responsible for acceptable material and product standards and specifications.
- 5.1.3 - Structural Steel.

### 6. Design Requirements

Clause 6.1: General

- The only ultimate limit state explicitly accounted for is "strength".
- The only serviceability limit state explicitly accounted for is "deflections"; the user is responsible for the specification of deflection limit states.
- The user is responsible for the specification of specified loads.
- The user is responsible for the specification of load factors.
- Camber, provisions for expansion and contraction, and corrosion protection are not accounted for.

Clause 6.2: Requirements Under Specified Loads

- 6.2.1 - Deflections.
- 6.2.1.1 - The user is responsible for the specification of acceptable deflection limits.

Clause 6.3: Requirements Under Factored Loads

- 6.3.1 - Strength
- The user is responsible for the application of the factored loads acting in the most critical combination

## **7. Loads and Safety Criterion**

### Clause 7.1: Specified Loads

- 7.1.1 - The user is responsible for the specification of dead loads and live loads and wind loads.
- Earthquake loads and dynamic load effects in general are not accounted for.
- The user is responsible for the specification of temperature changes and differential settlement.
- Shrinkage or creep of component materials is not accounted for.

### Clause 7.2: Safety Criterion and Effect of Factored Loads

- 7.2.1 - The user is responsible for the specification of factored loads.
- Overturning, uplift, and stress reversal are not accounted for.
- 7.2.2 - The user is responsible for the factored load combinations.
- 7.2.3 - The user is responsible for the specification of the load factors.
- 7.2.4 - The user is responsible for the specification of the load combination factor.
- 7.2.5 - The user is responsible for the specification of the importance factor.

## **8. Analysis of Structure**

### Clause 8.1: General

- 8.1.1 - Only deflection and strength requirements in Clause 6 are accounted for.
- 8.1.2

### Clause 8.2: Continuous Construction

### Clause 8.3: Simple Construction

- 8.3.1 - The user is responsible for ensuring a suitable system of bracing as resistance to lateral loads, including sway effects.

### Clause 8.4: Elastic Analysis

### Clause 8.6: Stability Effects

- 8.6.1 - Part (b) is not accounted for.
- 8.6.2 is not accounted for.

## **9. Design Lengths of Members**

### Clause 9.1: Simple Span Flexural Members

- The effect of any significant moment or eccentricity arising from the manner in which a beam, girder, or truss may actually be connected or supported is not accounted for.

### Clause 9.2: Continuous Span Flexural Members

### Clause 9.3: Compression Members

- 9.3.1
- 9.3.2
- 9.3.3

## **10. Slenderness Ratios**

### Clause 10.1: General

### Clause 10.2: Maximum Slenderness Ratio

- 10.2.1

- 10.2.2

## 11. Width-Thickness Ratios: Compression Elements

Clause 1 1.1: Classification of Sections

- 11.1.1
- 11.1.2
- 11.1.3

Clause 11.2: Maximum Width-Thickness Ratios of Elements Subject to Compression

- The  $h/w$  limits in Table 1 are not applied to webs in pure compression. Instead a limit of  $670/\sqrt{F_y}$  is used as indicated by the commentary in the steel handbook published by CISC. The user may override this interpretation and enforce strict interpretation of Table 1 by changing the StrictTable1 parameter in the SODA.INI file.

Clause 11.3: Width and Thickness

- 11.3.1 - Part (a) is not accounted for.
- 11.3.2 - Parts (a) and (c) are not accounted for.
- 11.3.3

## 12. Gross and Net Areas

Clause 12.1: Application

Clause 12.2: Gross Area

Clause 12.3: Net Area

- 12.3.1 - The calculation of net width and area of parts containing holes in accordance with Clause 12.3.3 is not accounted for.
- 12.3.2 - The user is responsible for ascertaining the width of bolt holes.
- 12.3.4 - The critical net area of the part is taken to be one of the limit values in (a), (b) or (c).

## 13. Member and Connection Resistance

Clause 13.1: General

- The user is responsible for the determination of factored loads in accordance with Clause 7.2.

Clause 13.2: Axial Tension

- Part (b) is not accounted for.

Clause 13.3: Axial Compression

- 13.3.1 - It is assumed that the capacity of singly symmetric double angle sections may be safely calculated by Clause 13.3.1.
- SODA applies  $n = 2.24$  for all WWF shapes.

Clause 13.4: Shear

- 13.4.1.1 - Elastic Analysis.
- Webs are considered to be unstiffened, such that  $a/h = \text{infinity}$ ,  $\tau = 1$ ,  $\eta = 0$  and  $k_v = 5.34$ .
- 13.4.1.3 - Maximum Slenderness.

Clause 13.5: Bending - Laterally Supported Members

- Part (c)(i) is not accounted for.
- Part (c) (iii) Calculates  $M_r$  by the 'Alternative' method



#### Clause 13.6: Bending - Laterally Unsupported Members

- Part (c) and (d) are not accounted for.

#### Clause 13.8: Axial Compression and Bending

- The detailed methods of determining the resistance of columns subject to biaxial bending given in Appendix F are not considered.

- 13.8.1 - For Part (b) SODA will only set  $K = 1$  if the user has specified 'K = calc' in the input.

- 13.8.2 - For Part (b) SODA will only set  $K = 1$  if the user has specified 'K = calc' in the input.

- For a member divided into two (or more) segments, the user is responsible for ensuring the correct slenderness ratio  $KL/r$  for each segment for use in Clause 13.8; for example, in the SODA Interface, specify the effective length factor  $K=2.0$  for each of the two segments of a member that has been divided in half.

- 13.8.3 -

- 13.8.4 -Part (c) is not accounted for.

#### Clause 13.9: Axial Tension and Bending

- Part (b) is not accounted for.

## 15. Beams and Girders

#### Clause 15.4: Reduced Moment Resistance of Girders with Thin Webs

## 20. Stability of Structures and Individual Members

#### Clause 20.1: Structures

- 20.1.1 - The user is responsible for ensuring that a complete structural system is provided to transfer the factored loads to the foundations.

- The user is responsible for ensuring adequate resistance to torsional deformations.

# SODA Databases

The section properties are stored in databases that are used by SODA in conjunction with the corresponding steel design code. The databases are organised into files that contain either standard American or Canadian sections or user-defined custom sections.

## SODA Database Editor

The section properties can be conveniently viewed and edited by using the Database Editor application. For instructions on using the Database Editor, please consult the on-line Help.

## American Section Database

The American section database can be used in conjunction with three steel design codes:

**AISC-ASD-89** : American Institute of Steel Construction, 'Specification for Structural Steel Buildings: Allowable Stress Design and Plastic Design', June 1, 1989

**AISC-LRFD-86** : American Institute of Steel Construction, 'Load and Resistance Factor Design Specification for Structural Steel Buildings', September 1, 1986

**AISC-LRFD-93** : American Institute of Steel Construction, 'Load and Resistance Factor Design Specification for Structural Steel Buildings', December 1, 1993

The properties of the standard shapes in the American section database are from the 'AISC Manual of Steel Construction', Ninth Edition, 1989

The (local buckling) classifications of the steel sections in the American database are in accordance with the provisions of the 'AISC Manual of Steel Construction: Load and Resistance Factor Design', First Edition, 1986 (Table B5.1). These classifications apply for all section shapes when the AISC-LRFD-86 or AISC-LRFD-93 design code is applied. When the AISC-ASD-89 design code is applied, however, these classifications only apply for WT, Single Angle and Double Angle section shapes (for all other section shapes, the local buckling classifications are established internally by SODA at run time).

For computer application, the American section database is defined by the master file AISC.SDA. This file lists the names of the fifteen files that contain the properties of the fifteen different section shapes that comprise the database, i.e.,

AISC.SDA = <W.USA, HP.USA, M.USA, S.USA, C.USA, MC.USA, RHS.USA, SHS.USA, CHS.USA, WT.USA, EL1L.USA, UL1L.USA, EL2L.USA, LL2L.USA, SL2L.USA>

(Note that '.USA' is the file-type extension to each section shape filename in the American database)

Each of the fifteen shape files in the American database contains the properties of the individual steel sections having that shape. The number of sections for each shape is given at the top of the file. The first section listed in each file has the smallest cross-section area for that shape, and the sections thereafter are ordered with increasing area (so as to facilitate the SODA optimization process).

## Notation: American Section Database

The header notation appearing in the fifteen different shape files that comprise the American section database is defined in the following.

AREA = Area of cross section (in<sup>2</sup>)

D = Section depth (in).

WT = Web thickness; wall thickness (in).

B = Flange width; width of angles and tubing (in).

TF = Flange thickness (in).

BTF = Ratio of the flange width to twice the flange thickness.

DTW = Ratio of the section depth to the web thickness.

DAF = Ratio of the depth to the compression flange area for W, M, S, HP, C and MC shapes.

ID = Inner diameter for pipe shapes (in).

OD = Outer diameter for pipe shapes (in).

FYP = Theoretical maximum yield stress  $F_y'$ (ksi), based on the width-thickness ratio of 1/2 the unstiffened compression flange, beyond which a particular shape is not compact. (Specified as 0 if greater than 65 ksi).

FYPPP = Theoretical maximum yield stress  $F_y''$ (ksi), based on the depth-thickness ratio of the web, below which a particular shape may be considered compact for any condition of combined bending and axial stresses. (Specified as 0 if greater than 65 ksi).

RT = Radius of gyration of a section comprising the compression flange plus 1/3 of the compression web area, taken about an axis in the plane of the web, for W, M, S and HP shapes (in).

IX = Moment of inertia about x-x axis ( $\text{in}^4$ ).

SX = Elastic section modulus about x-x axis ( $\text{in}^3$ ).

RX = Radius of gyration about x-x axis (in).

IY = Moment of inertia about y-y axis ( $\text{in}^4$ ).

SY = Elastic section modulus about y-y axis ( $\text{in}^3$ ).

RY = Radius of gyration about y-y axis (in).

RYO = RY for space = 0 in for Double Angle (2L) shapes (in).

RY3/8 = RY for space = 3/8 in for 2L shapes (in).

RY3/4 = RY for space = 3/4 in for 2L shapes (in).

RO = Shear centre co-ordinate (in)

ROO = RO for space = 0 in for 2L shapes (in).

R03/8 = RO for space = 3/8 in for 2L shapes (in).

R03/4 = RO for space = 3/4 in for 2L shapes (in).

H = Flexural Constant.

HO = H for space = 0 in for 2L shapes.

H3/8 = H for space = 3/8 in for 2L shapes.

H3/4 = H for space = 3/4 in for 2L shapes.

TJ = Torsional constant ( $\text{in}^4$ ).

CW = Warping Constant ( $\text{in}^6$ ).

ZX = a) Plastic section modulus about x-x axis ( $\text{in}^3$ ).  
 = b) Centroid of section in x-direction from outer web face for CM and MC shapes (in).

ZY = a) Plastic section modulus about y-y axis ( $\text{in}^3$ ).  
 = b) Centroid of section in y-direction from outer top flange face for WT shapes (in).

= c) Distance from outside face of web to the shear centre for C and MC shapes (in).

Y = Neutral-axis depth from x-x axis for angle shapes (in).

X = Neutral-axis depth from y-y axis for angle shapes (in).

OS36 = Axial stress reduction factor 0, for  $F_y=36$  ksi.

QS50 = Axial stress reduction factor 0, for  $F_y=50$  ksi.

QSC36 = Axial stress reduction factor 0, for  $F_y=36$  ksi for Double Angle(2L) shapes in contact.

OSC50 = Q, for  $F_y = 50$  ksi for 2L shapes in contact.

OSS36 = Q., for  $F_y =36$  ksi for 2L shapes that are separated.

QSS50 = 0, for  $F_y =50$  ksi for 2L shapes that are separated.

RZ = Radius of gyration about z-z axis for Single Angle (1L) shapes (in).

TANA = Tan Alpha for 1L shapes.

T = Thickness of 1L shapes (in).

K = Distance from outer face of flange to web toe fillet of rolled shape (in).

RI = Minimum fillet radius, design value (in).

RA = Maximum fillet radius, detailing value (in); flange toe value for M, S, C and MC shapes.

NT = Tensile grouping number per ASTM A6. (1 through 5)

WGT = Section weight (lb/ft)

F = Foreign section delimiter ( ' ' or '\*\*').- If F = '\*\*', SODA considers the section to be 'Foreign' to the shape file.

DESIGNATION = Section name

SA:36 42 50 = Strong-Axis section classes for three different yield stress values  $F_y = 36,42$  and 50 ksi (36, 42 and 46 ksi. for RHS and SHS shapes)

WA:36 42 50 = Weak-Axis section classes for three different yield stress values  $F_y = 36,42$  and 50 ksi (36, 42 and 46 ksi. for RHS and SHS shapes)

NOTE:

C = Compact, N = Non-compact, S = Slender. - The letter character (C, N or S) under the first digit of the yield stress value defines the Axial section class, while that under the second digit defines the Flexural class.

For CHS, WT, EL1L, UL1L, EL2L, LL2L and SL2L shapes, the Flexural section class has been arbitrarily set equal to the Axial class.

For C, MC, WT, EL1L, UL1L, EL2L, LL2L, and SL2L shapes, the Weak-Axis section class has been arbitrarily set equal to the Strong-Axis class.

The classifications apply for all section shapes when the AISC-LRFD-86 or AISC-LRFD-93 design code is applied, but only for WT, Single and Double Angle section shapes when the AISC-ASD-89 design code is applied (the section classes for other shapes are established internally by SODA at runtime).

## Shapes: American Section Database

Below is a screen capture from the SODA Database Editor of a “W” shape from the American section database.

The screenshot shows the SODA Database Editor window. The interface includes a menu bar (File, Options, Help) and several control elements:

- Select A Database:** A dropdown menu set to "AISC".
- Select A File:** A dropdown menu set to "W.USA", with buttons for "Create New File", "Remove File", and "Import File".
- Select A Section:** A dropdown menu set to "40x655", with a navigation box and the text "Section 305 of 311". A "Foreign Section" checkbox is present and unchecked.
- Shape Type:** A text field containing "W".

Below these controls is a table of section properties:

Designation	Area	$r_x$	$r_y$	$I_x$	$I_y$	$J$
40x655	1.92E+02	1.72E+01	3.86E+00	5.65E+04	2.86E+03	5.96E+02
$d$	$S_x$	$S_y$	$Z_x$	$Z_y$	$NT$	$bf$
4.36E+01	2.59E+03	3.39E+02	3.06E+03	5.41E+02	4.00E+00	1.69E+01
$t_f$	$t_w$	$bf/2t_f$	$d/t_w$	$d/A_f$	$k$	$F_y'$
3.54E+00	1.97E+00	2.40E+00	2.21E+01	7.30E-01	4.94E+00	0.00E+00
$F_y''$	$r_T$	$RI$	$RA$	$C_w$	$R_0$	$H$
0.00E+00	4.43E+00	1.25E+00	1.25E+00	1.14E+06	0.00E+00	0.00E+00

At the bottom of the window are three buttons: "Save Section", "Create New Section", and "Remove Section".

# Canadian Section Database

The Canadian section database can be used in conjunction with the steel design codes:

**CSA-S16.1-M89:** Canadian Standards Association, "Limit States Design of Steel Structures", December, 1989

**CSA-S16.1-94:** Canadian Standards Association, "Limit States Design of Steel Structures", December, 1994

The properties of the standard shapes in the Canadian section database are from the "CISC Handbook of Steel Construction", Fifth Edition, 1991

The (local buckling) classifications of the steel sections in the Canadian database are in accordance with the provisions of the "CISC Handbook of Steel Construction": CSA-S16.1-M89, "Limit States Design of Steel Structures", (Table 1)

For computer application, the Canadian section database is defined by the master file CISC.SDC. This file lists the names of the eighteen files that contain the properties of the nineteen different section shapes that comprise the database, i.e.,

CISC.SDC = < WWF.CDN, W.CDN, WFOR.CDN, WMIX.CDN, HP.CDN, M.CDN, S.CDN, C.CDN, MC.CDN, RHS.CDN, SHS.CDN, CHS.CDN, WWT.CDN, WT.CDN, EL1L.CDN, UL1L.CDN, EL2L.CDN, LL2L.CDN, SL2L.CDN >

(Note that ".CDN" is the file-type extension to each section shape filename in the Canadian database)

Each of the nineteen shape files in the Canadian database contains the properties of the individual steel sections having that shape. The number of sections for each shape is given at the top of the file. The first section listed in each file has the smallest cross-section area for that shape, and the sections thereafter are ordered with increasing area (so as to facilitate the SODA optimization process)

## Notation: Canadian Section Database

The header notation appearing in the nineteen different shape files that comprise the Canadian section database is defined in the following.

- AREA = Area of cross section ( $\text{mm}^2$ ).
- D = Section depth (mm).
- WT = Web thickness; wall thickness (mm).
- B = Flange width; width of angles and tubing (mm).
- TF = Flange thickness (mm).
- DS = Length of back-to-back leg for Double-Angle (2L) shapes (mm).
- BS = Length of outstanding leg for 2L shapes (mm).
- IX = Moment of inertia about x-x axis ( $\text{mm}^4$ ).
- SX = Elastic section modulus about x-x axis ( $\text{mm}^3$ )
- RX = Radius of gyration about x-x axis (mm).
- IY = Moment of inertia about y-y axis ( $\text{mm}^4$ ).
- SY = Elastic section modulus about y-y axis ( $\text{mm}^3$ ).
- SEx = Effective section modulus about x-x axis ( $\text{mm}^3$ ).
- SEy = Effective section modulus about y-y axis ( $\text{mm}^3$ ).
- RY = Radius of gyration about y-y axis (mm).

Ro = RY for space = 0 mm for 2L shapes (mm).  
 R8 = RY for space = 8 mm for 2L shapes (mm).  
 R10 = RY for space = 10 mm for 2L shapes (mm).  
 R12 = RY for space = 12 mm for 2L shapes (mm).  
 R16 = RY for space = 16 mm for 2L shapes (mm).  
 R20 = RY for space = 20 mm for 2L shapes (mm).  
 TJ = Torsional constant ( $\text{mm}^4$ ).  
 CW = Warping constant ( $\text{mm}^6$ ).  
 ZX = Plastic section modulus about x-x axis ( $\text{mm}^3$ ).  
 ZY = Plastic section modulus about y-y axis ( $\text{mm}^3$ ).  
 EO = Distance from outside face of web to the shear centre for C and MC shapes (mm).  
 Y = Neutral-axis depth from x-x axis (mm).  
     = Outside radius of Circular Hollow Sections (mm).  
 X = Neutral-axis depth from y-y axis for Single Angle (1L) shapes (mm).  
 RZ = Radius of gyration about z-z axis for angle shapes (mm).  
 T = Thickness of angle shapes (mm).  
 G = G40.21 group number (refer to Table 6-3, CISC Handbook of Steel Construction, 1991).  
 F = Foreign section delimiter (' ' or '\*'). If F = '\*', SODA considers the section to be 'Foreign' to the shape file.

DESIGNATION = Section name.

SA = Strong-Axis section class for prevailing  $F_y$  value.

WA = Weak-Axis section class for prevailing  $F_y$  value.

NOTE:

The section classifications given in the database are for  $F_y$  300 MPa for WWF, W, HP, M, S, C, MC, WWT, WT, EL1L, UL1L, EL2L, LL2L and SL2L shapes, and  $F_y$  = 350 MPa for RHS, SHS and CHS shapes. (for shapes in combined bending and compression, SODA internally recalculates the section class if a different  $F_y$  than in the foregoing is specified).

1 = class 1, 2 = class 2, 3 = class 3, 4 = class 4.

The digit (1, 2, 3 or 4) under the first letter character of "SA" or "WA" defines the Axial section class, while that under the second letter character defines the Flexural class.

For CHS, WWT, WT, EL1L, UL1L, EL2L, LL2L and SL2L shapes, the Flexural section class has been conservatively set equal to the Axial class.

For C, CHS, SHS, MC, WWT, WT, EL1L, UL1L, EL2L, LL2L and SL2L shapes, the Weak-Axis section class has been arbitrarily set equal to the Strong-Axis class.

For sections having webs in combined flexural and axial compression, SODA internally establishes the governing section class accounting for the prevailing  $C_f/C_y$  ratio (refer to Table 1 in Part 1 of the CISC Handbook of Steel Construction, 1991).

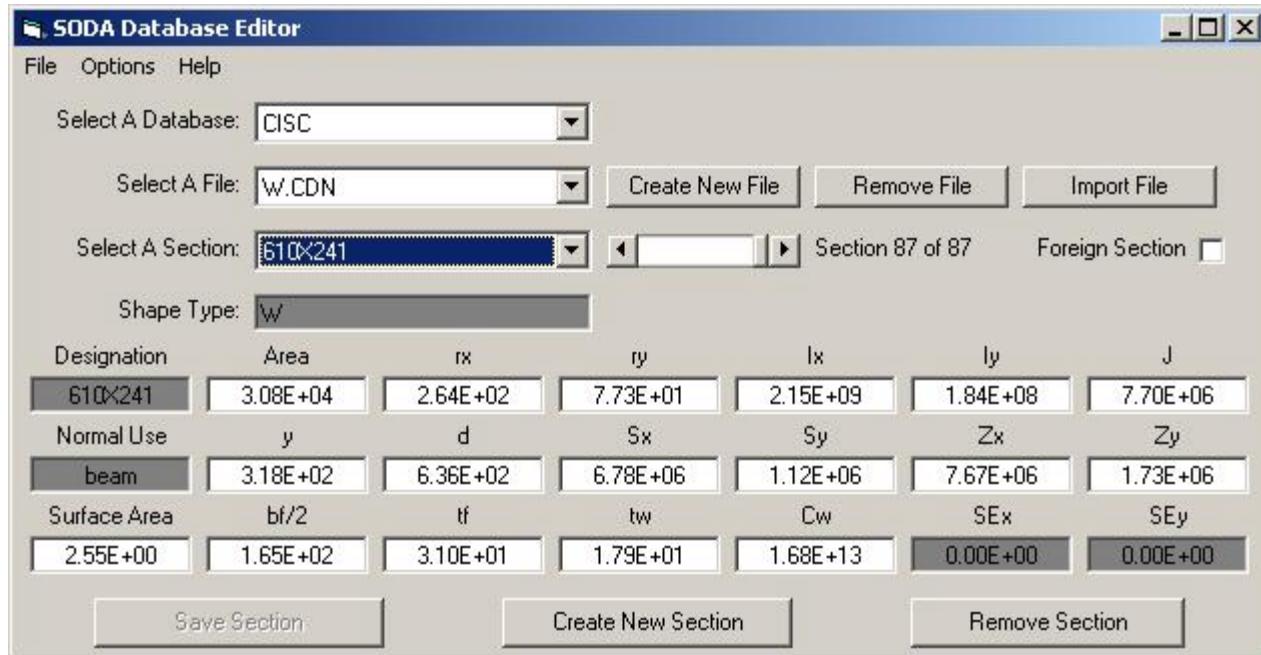
NORMAL USE = Normal use of section

SURFACE AREA = Surface area of section

TALPHA = Tan Alpha for Z-axis orientation of angle sections

### Shapes: Canadian Section Database

Below is a screen capture from the SODA Database Editor of a “W” shape from the Canadian section database.





## User-Created Section Database

Any user-created section database must consist of sections that conform with the requirements of the American or Canadian steel design codes that are supported by SODA (i.e., AISC-ASD-89, AISC-LRFD-86, AISC-LRFD-93, CSA-S16.1-M89, or CSA-S16.1-94).

These databases can be completely created following the instructions below, or by creating the master files only, which then can be imported and the properties edited using the SODA Database Editor.

### Database Guidelines

When creating a custom section database, the standard American and Canadian databases supplied with SODA should *not* be directly modified in any way. Instead, an entirely new section database should be created using the following guidelines:

1. The primary name of the master file that lists the names of the shape files comprising the database should be unique (e.g., INHOUSE), and should NOT be that for the standard American or Canadian section database (i.e. AISC or CISC)
  2. The file-type extension to the primary name of the master file should be '.SDA' if the database is comprised of American shapes, and '.SDC' if the database is comprised of Canadian shapes (e.g., INHOUSE.SDA or INHOUSE.SDC)
  3. The maximum number of sections in a section shape file is limited to 700.
  4. Each of the section shape files comprising the database may be created as follows:
    - 4.1 All sections in a file must have the same shape, and this shape must correspond to one of the 17 different section shapes supported by SODA (WWF, W, HP, M, S, C, MC, RHS, SHS, CHS, WWT, WT, EL1L, UL1L, EL2L, LL2L, SL2L).
    - 4.2 The primary name of the file may be one of the seventeen different shape filenames supported by SODA (WWF, W, etc.). Alternatively, any one to four (1 to 4) letter acronym may be used (e.g., A or AB or ABC or ABCD).
    - 4.3 The file-type extension is at the discretion of the user and may be selected to reflect whether the sections are American (e.g., '.AMR') or Canadian (e.g., '.CAN') shapes. The file-type extensions '.USA' and '.CDN' should not be used so as to avoid confusion with the standard American and Canadian shape files.
    - 4.4 The primary name and extension must appear in the '.SDA' or '.SDC' file. The spacing format used in the AISC.SDA and CISC.SDC files must be adhered to. The name may be added to existing '.SDA' or '.SDC' files or a new file can be created.
    - 4.5 The section having the smallest cross-section area must appear first in the file, and thereafter sections must be ordered with increasing cross-section area.
    - 4.6 The data for each section must be placed in the file in accordance with the format fields defined by the number line appearing in the corresponding standard American or Canadian shape file.
    - 4.7 Ensure that the name of one of the 17 standard shapes appears in the first line of the shape file itself; e.g., the first line of a file WBM.USA could begin 25 W..., indicating that the file contains 25 W-shaped sections. Note that the number of sections must end in column 5 and the shape name must start in column 7.
- Note: The above option allows the user to not only create entirely new shape files, but to also create subsets of the standard shape files so as to impose user-preferences on the design process. For example, WBM.USA could contain just those W-shaped sections suitable for beams, WCOL.USA could contain just column sections, and WBC.USA could contain just beam-column sections, or W14.USA could contain just W-shaped sections with W14 designations.

### Customizing a Section Shape File

The user may customize an existing section shape file using the Database Editor, or manually as follows:

1. Any section in a shape file can be delineated as being 'Foreign' to the file by inserting the character symbol

\*\*) in the appropriate data field for the section in the standard American and Canadian databases. SODA will not consider the section for design if Foreign Sections = NO is then specified in the input data.

2. A new section may be included in a shape file by placing the section data in the file in accordance with the format fields defined by the number line appearing in the corresponding standard American and Canadian shape file. The data for the new section is to be located in the file immediately after that for the existing section that has the next smallest cross-section area (i.e., sections in a shape file are ordered with increasing area).

3. If a section is added or deleted from a file, be sure to revise the number at the top of the file to reflect the remaining number of sections